FEMAP v11.4 New Features and Corrections
Updates and Enhancements

Views

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

Connection Properties, Regions, and Connectors

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

Geometry

- Added Modify, Break, At All Intersections command.
- Updated Modify, Update Other, Surface Normal to work with Parasolid General Bodies.

GUI - Toolbars and Icons

Select Toolbar

- Updated Load Group command to automatically turn on "Select Multiple" switch. Also, if the icon at the top of the drop-down in the toolbar is clicked, the Load Group command will be used again, after being used once.
- Updated Grow command to honor the "Select Related" switch, however if you Grow with "Select Related" enabled, Shrink will no longer remove entities from the selection.

Draw/Erase Toolbar

- Added Draw Mode Select From All to the Select Area icon menu of the Draw/Erase toolbar. When this option is enabled, which is the default, all entities in the model which are not hidden by other visibility methods will be shown when selecting which entities to "Draw". When disabled, entities which are currently being "Drawn" will remain the only visible entities and graphical selection only considers those entities.

GUI - Dockable Panes

Function/Table Editor - New for 11.4!

- Added Function/Table Editor dockable pane to create and edit Functions and Tables. A function always has two columns (XY Data), while the number of columns in a table depends upon the Type of table. Tables are similar to functions, as they also contain values in rows and columns, but a table always have more than two columns.
- Added ability to create a Load Set Combination Table, which has identical functionality to the Load Set Combination Data Surface. The Table will replace the Data Surface in future versions.
- Added ability to create a Result Set Processing Table, which has identical functionality to the Result Set Processing Data Surface. The Table will replace the Data Surface in future versions.

- Added ability to create five Thermal/Flow Vector Tables: Acceleration vs Time, Sun/Planet vs Time, Sun/Planet/Altitude vs Time, Spherical Sun/Planet vs Time, and Spherical Sun/Plant/ Altitude vs Time, which will be used in a future release.

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

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

PostProcessing Toolbox - Freebody Tool
- Added Reverse Freebody Values option to Freebody Tool.
- Added Show icon button to Freebody Tool in the PostProcessing Toolbox to highlight, in the graphics window, both the nodes and elements used by the Freebody currently loaded in the Freebody Tool.

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

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

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

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

Data Table

- Added additional options to the Data Table filter, specifically to filter entities by Text. Previously you could only filter Text as Contains. Now, you can choose Contains, Not Contains, Equals, or Not Equals, with Contains and Not Contains only needing to match a portion, while Equals and Not Equals need an exact match.

Interfaces - FEMAP Neutral

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

Interfaces - Nastran

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

Interfaces - NX Nastran

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

Interfaces - ABAQUS

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

Interfaces - LS-DYNA

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

Interfaces - Geometry

- Added support for Solid Edge with Synchronous Technology 9, CATIA V5-6R2016 SP2, and SolidWorks 2017.
- Removed the "legacy" version of the CATIA V5 geometry translator, which could only be used once a "type-in" preference was specified in the [User] section of the FEMAP.INI file.
- Removed the "legacy" version of the SolidWorks geometry translator, which could only be used once a "type-in" preference was specified in the [User] section of the FEMAP.INI file.

Loads and Boundary Conditions

- Updated the Model, Load, Heat Transfer command by adding an option to enable View Factor Zero Tolerance and enter a corresponding value to the Radiation section of the Heat Transfer Loads dialog box.
- Updated Model, Load, Map Output From Model command by adding Data Conversion drop-down to the Map from Model Output dialog box, which specifies the method of data conversion for the selected output vector.
- Updated the Model, Load, Combine command by removing the From Data Surface option from the Combine To section and the Data Surface drop-down from the Options section. As an alternative, use a Load Set Combination Table in the Function/Table Editor to create more complicated combinations of Load Sets.

Meshing

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

Materials

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

Element - Update Existing Elements

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

Output and Post-Processing

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

Groups and Layers

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

Listing

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

Tools

Vector Manager - New for 11.4!

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

Plane Manager - New for 11.4!

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

Measure, Element Quality

- Added an error message if you attempt to use the Measure Distance Between Geometry command and choose points or nodes that are outside of the Parasolid bounding box

Check, Element Quality

- Added 4 additional NX Nastran Element Quality checks, Tetra EPIA, Pyr EPIA, Penta EPIA, and Hex EPIA, which can be accessed along with the 25 other NX Nastran element quality checks by clicking the NX Nastran tab in the Check Element Quality dialog box.

Model Merge

- Updated File, Merge command to load the table in a sorted order and ask a question to clear the table if you load duplicates multiple times when using the Send Duplicates to Data Table option

User Interface - General

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

Preferences

Views

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

User Interface

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

Interfaces

- Removed the Neutral Digits option, as it is no longer needed. All Neutral files are now written using "Max Precision", which is 16 digits for double-precision real values, such as nodal coordinates, and 8 digits for single-precision real values, such as results.

Results

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

API

New and updated API Objects and Attributes

- Added Plane (fePlane) object to the API. Also, added title, base, vBase, norm, vNorm, axis, and vAxis attributes to the Plane Object.
- Added Vector (feVector) object to the API. Also, added title, base, vBase, dir, vDir, and Length attributes to the Vector Object.

- Added Table Data (feTableData) object to the API. Also, added Title, Type, Subtype, FunctionType, VectorFunctionType, Rows, and Columns to the Table Data Object.
- Added User Defined Graphics (feUserDefinedGraphics) object to the API.
- Added Pt1 and Pt4 attributes to the feAeroPanel Object.
- Added NasCurvatureStrain attribute to the feAnalysisCase Object.
- Added NasCurvatureStrain attribute to the feAnalysisMgr Object.
- Added ComplexPlotLocation, AxisAutoscale2, vAxisAutoscale2, AxisRange2, vAxisRange2, AxisRangePad2, vAxisRangePad2, AxisStyle2, vAxisStyle2, AxisLabelFormat2, vAxisLabelFormat2, AxisLabelDecimal2, and vAxisLabelDecimal2 attributes to the feChart Object.
- Added ConvertMethod, ConvertMethod2, TransformNodalMode, TransformNodalMode2, TransformNodalCSys2, TransformNodalCSys, TransformPlateMode, TransformPlateCSys, TransformPlateDOF, TransformPlateVector, vTransformPlateVector, TransformPlateTolerance, TransformPlateMode2, TransformPlateCSys2,TransformPlateDOF2, TransformPlateVector2, vTransformPlateVector2, TransformPlateTolerance2, ComplexMethod, ComplexPhase, ComplexStart, and ComplexEnd attributes to the feChartSeries Object.
- Added DataConversion attribute to the feDataSurf Object.
- Added DrawModeSelectFromAll attribute to the feDrawErase Object.
- Added NastranTetraEPIAOn, NastranTetraEPIALimit, NastranHexEPIAOn, NastranHexEPIALimit, NastranPenEPIAOn, NastranPenEPIALimit, NastranPyrEPIAOn, and NastranPyrEPIALimit attributes to the feElementQuality Object.
- Added DataConversion attribute to the feMapOutput Object.
- Added b and vb attributes to the feTMGBC Object.
- Added SecondaryRotationAxesOption attribute to the feViewOrient Object.

New and Updated API Methods
- Added SetView, GetView, SetName, Publish, Blank, CollectorPoints, CollectorBitmap, CollectorSymbol, CollectorSymbolREAL8, CollectorText, CollectorTextINT4, CollectorTextREAL8, CollectorLines, CollectorTriangles, CollectorBlank, CollectorLabels, CollectorMarkForDeletion, CollectorAppearance, CollectorAddPointLocations, CollectorAddPointEntityLocations, CollectorAddPointEntityFaceLocations, CollectorAddSymbolLocations, CollectorAddSymbolEntityLocations, CollectorAddSymbolEntityFaceLocations, CollectorAddSymbolEntityFaceNormalLocations, CollectorAddSymbolREAL8Locations, CollectorAddSymbolREAL8EntityLocations, CollectorAddSymbolREAL8EntityFaceLocations, CollectorAddSymbolREAL8EntityFaceNormalLocations, CollectorAddTextLocations, CollectorAddTextEntityLocations, CollectorAddTextEntityFaceLocations, CollectorAddTextINT4Locations, CollectorAddTextINT4EntityLocations, CollectorAddTextINT4EntityFaceLocations, CollectorAddTextREAL8Locations, CollectorAddTextREAL8EntityLocations, CollectorAddTextREAL8EntityFaceLocations, CollectorAddLineLocations, CollectorAddPolyLineLocations, CollectorAddMeshLineLocations, CollectorAddTriangleLocations, CollectorAddMeshTriangleLocations, CollectorAddMeshTriangleNormalLocations, BitmapCreate, BitmapMarkForDeletion, SymbolCreate, SymbolMarkForDeletion, SymbolSetAddInScreen, SymbolAddPoints,

SymbolAddLineStraight, SymbolAddLinePoly, SymbolAddLineCircle, SymbolAddLineBrick, SymbolAddLineCylinder, SymbolAddLineCone, SymbolAddLineSphere, SymbolAddFillTriangle, SymbolAddFillCircle, SymbolAddFillBrick, SymbolAddFillCylinder, SymbolAddFillCone, SymbolAddFillTubeBrick, SymbolAddFillTubeCylinder, SymbolAddFillSphere, Detach, and Attach to the feUserDefinedGraphics Object.

- Added SelectIDWithNew as a Common Entity method.
- Added GetEntitySet to the feConnectionRegion object.
- Added ElementHasThickness to the feElem object.
- Added GetNastranTetraEPIA, NastranTetraEPIA, GetNastranHexEPIA, NastranHexEPIA, GetNastranPenEPIA, NastranPenEPIA, GetNastranPyrEPIA, and NastranPyrEPIA to the feElementQuality object.
- Added Show to the feFreebody object.
- Added ShowAutoscale and SelectIDWithNew to the feSet object. In addition, added new "Saved Set" method, including SetDeveloperID, GetSavedSet, GetAllSavedSets, PutSavedSet, DeleteSavedSet, DeleteAllSavedSets, DeleteOtherSavedSets, ResetSavedSet, NextSavedSet, EmptySavedSet, CountSavedSets, AddToSavedSet, AddSetToSavedSet, RemoveFromSavedSet, and RemoveSetFromSavedSet.
- Added PutAll to the fePlane object.
- Added GetFunction, PutFunction, CellRange, SaveToFile, Initialize, Resize, Clear, SetCellDouble, SetCellInteger, SetCellText, SetCellEquation, SetCellEntity, GetCellDouble, GetCellInteger, GetCellText, GetCellEntity, SetMultiCellDouble, SetMultiCellInteger, SetMultiCellText, GetMultiCellDouble, and GetMultiCellInteger to the feTableData object.
- Added GetAll2 and PutAll2 to the feTMGBC Object.
- Added GetAll2 and PutAll2 to the feTMGCtrl Object.
- Added PutAll to the feVector object.
- Added SnapToAxes, SetRotationAngles, and GetRotationAngles to the feViewOrient object.
- Updated GetLibrary and PutLibrary for the feFunction object.

New and Updated Global Variables
- Added Pref_ElemQualTetEPIA, Pref_ElemQualHexEPIA, Pref_ElemQualPenEPIA, Pref_ElemQualPyrEPIA, Pref_ElemQualTetEPIAVal, Pref_ElemQualHexEPIAVal, Pref_ElemQualPenEPIAVal, Pref_ElemQualPyrEPIAVal, Pref_ShowAutoscale, Pref_SynchronizedRotation, and Pref_DefaultFreebodyReverse to set various preferences.
- Added Info_ViewShowAutoscale, and Info_FreebodyReverse to set various global variables.

The following functions have been added or updated:
- feFileReadIdeas
- feFileReadJT
- feFileWriteJT
- feSurfaceCornersMultiple
- feCurvesBreakAtIntersections
- feSurfaceExtract
- feGenerateCopy2

- feGenerateScale2
- feGenerateRadialCopy2
- feGenerateRotate2
- feGenerateReflect2
- feViewShow2
- feWindowShow2
- feCreateFunction
- feAppModelInfoShow
- feAppSetModelInfoShow
- feSolidSlice2
- feSolidSliceAlongFace2
- feSolidSliceWithSheet2
- feSolidSliceWithCurve2
- feSolidExtractCenterlines

Corrections

Views

- Corrected issue which caused the Coordinate System specified for Rotate Around in the View, Rotate command to not be considered by the icons on the View Orient Toolbar (PR# 7868832)
- Corrected issue in View, Rotate, Model command, which caused nothing to happen when values were entered X, Y, or Z fields in the dialog box.
- Corrected issue which caused Text entities to be considered when the Dynamic Rotate Around Cursor Location preference was enabled. In addition, picking of Text entities was enhanced to highlight when the cursor is over a Text Entity. If over multiple Text Entities, the nearest center is used and if not directly over a Text Entity, the closest center is also used.
- Corrected issue when the middle mouse button was clicked in a different view, to activate the view, which caused an autoscale to occur.
- Corrected issue when the active view was deleted via the View Manager, which could cause FEMAP to unexpectedly exit.
- Corrected issue where multiple tiled views are being displayed, then the title bar of a non-active window/view is clicked, which caused all of windows to be maximized.
- Corrected issue which caused the material orientation to not be displayed properly when solid laminate elements were using a cylindrical coordinate system to define the material orientation (PR# 7883560).

Analysis Manager

- Corrected issue which caused the Add and Update buttons on the Solution Frequencies tab of the NASTRAN Dynamic Analysis dialog box to disappear if the Frequency Value button was clicked, then Cancel was clicked in the Frequency Response Input dialog box.
- Corrected issue which could cause items in the Response Frequencies list on the Solution Frequencies tab of the NASTRAN Dynamic Analysis dialog box to become "empty".

Analysis Monitor

- Corrected issue that caused Femap to use the current directory specifier ".\" if the output directory was the root of a drive (ie, C:\?). When this happened, the current directory was not the same as the model file directory, which caused Femap to not be able to find the NX Nastran log file from a running job and issue an error, even though the run was successful. In addition, the same issue could occur when the output directory specified in the Analysis Set was different than the Output Directory specified on the Interfaces tab of File, Preferences.

Geometry

- Corrected issue finding the intersection of Parasolid curves, which was introduced in V11.3.
- Corrected issue when computing the intersection of a Line and Arc/Circle if the line was perpendicular to the circle (i.e., along the direction of the normal to the plane of the circle).
- Corrected issue in Geometry, Midsurface, Extend command which could issue an an improper error when extending the edge of a surface to a Solid, if the Solid is a single face surface.

Graphics

- Corrected issue when using displaying results using a Cutting Plane, when free edges are displayed, which caused some faces to not be drawn.

Performance Graphics

- Corrected issue when displaying a contour display and the value for % Ambient for Shading View Option was set 25%, which is the default. This could cause colors in the contour (on either side of yellow and cyan) to potentially be saturated in one component, causing the yellow and cyan bands to look too large. The higher the value for % Ambient, the worse it would appear.
- Corrected issue which caused Distributed Loads on line elements to not be displayed properly.
- Corrected issue which caused Distributed Loads to always be offset, even if offset is off.

GUI - Dockable Panes

PostProcessing Toolbox

- Corrected issue where contour arrows with multiple components that were specified using the Advanced Contour Arrow Options dialog box may be overwritten by various functions in the PostProcessing toolbox (PR# 7919757).

Charting Pane

- Corrected issue when using the Delete All command in the Chart Data Series Manager and no active chart was being displayed, which could cause FEMAP to exit unexpectedly.

Entity Editor

- Corrected issue where elemental temperatures could not be edited in the Entity Editor if the elemental temperatures were not in the active Load Set (PR #7526412)
- Corrected issue where Nastran Wapring and Nastran Taper element quality values would not be shown when a linear or parabolic quadrilateral element was loaded in the Entity Editor.

Data Surface Editor
- Corrected issue when Renumbering Load Sets using Load Set Combination Data Surface. Previously, if a Load Set Combination Data Surface had a lower ID than other Data Surfaces, an incorrect and unnecessary error message was displayed for each Data Surface that followed the Load Set Combination Data Surface. The same problem could also occur with Result Set Processing Data Surface or Connection Manager Data Surface.

Data Table
- Corrected issue that caused the "Clear Filter" icon to remain active even if the filter had already been cleared.

Toolbars
View Toolbar
- Corrected issue which caused rotation about the X Axis to be reversed when the X icon was clicked (PR# 7966884).

Select Toolbar
- Corrected issue in Copy to the clipboard that could sometimes cause additional, invalid data to be placed on the clipboard.

Interfaces - FEMAP Neutral
- Corrected issue when importing a neutral file which caused the current visibility settings for Elements By Type and Elements By Shape to not be properly interpreted.
- Corrected issue in previous v11.3 Neutral File converters that caused Parasolid Geometry to be skipped.

Interfaces - Nastran
- Corrected issue when reading RMAXMIN results from Nastran op2 file where the Principal Stress/ Strain vectors would be computed, even though the source data was enveloped transient data. All principal stress/strain computations are now skipped for RMAXMIN results.
- Corrected issue where the combination of PRINT, PLOT, and PUNCH in output requests caused the Output Destination to be set to XDB.
- Corrected issue when reading SET entries for output request when "Create Groups From Includes" is enabled and the Set ID conflicted with a group created for an include file.
- Corrected issue where Model Effective Mass output was skipped when reading the f06 file, if the output table spanned multiple pages.
- Corrected issue when reading Hexahedral Nonlinear Stresses from the f06 file where the data was imported into output vector "60031..Solid Equiv Stress" rather than output vector "60171..Nonlinear Solid Von Mises" as is done in the op2, which is correct.
- Corrected issue when reading and writing SPC entries when some Degrees of freedom where zero and others were non-zero (PR# 7845769).

- Corrected issue which caused MASS normalization to be written instead of MAX for Nastran EIGRL entry (PR# 7875923).
- Corrected issue when reading Nastran INCLUDE statements, when the Preserve INCLUDE Statements preference was enabled. The error caused FMS, Executive, and Case Control INCLUDE statements to be added to the end text of the bulk data section, in addition to their respective sections.
- Corrected issue which caused FEMAP to only store the first line of a multi-line INCLUDE entry when the Preserve INCLUDE Statements preference is enabled.
- Corrected issue where the PRGPST,YES entry would be read in as start text as well as enabling the flag in the Nastran bulk data options, which would cause the entry to be written twice during translation.

Interfaces - NX Nastran

- Corrected issue when reading NX Nastran 11.0.1 composite ply stress/strain results where the new order of the tensor components was not handled properly. This occurs only for NX Nastran 11.0.1 and above, as NX 11.0 results were read correctly.

Interfaces - MSC Nastran

- Corrected issue which caused Glued contact sets to not be written for Nonlinear or Random Response analysis.
- Corrected issue when writing "606..MSC Nastran Fluid Material (MAT10)", which cause the GRID entries to not be written a "Fluid" GRID entries, along with the "Fluid" element flag not being set on the PSOLID entry. Both of these issues have been corrected and some error and warning messages have been added when this material type is specified and the model is being written for a solver other than MSC Nastran.

Interfaces - Autodesk Nastran

- Corrected issue when importing PBEAM when K1,K2 fields were blank, which caused the blank K1,K2 fields to be interpretted as 1.0, which is the correct default for both NX and MSC Nastran. The blank K1,K2 fields are now properly interpreted as 0.0.

Interfaces - ANSYS

- Corrected issue when importing beam elements which caused the elements to be orientated incorrectly.

Interfaces - ABAQUS

- Corrected issue reading *INCLUDE entries if there were spaces around the INPUT parameter.
- Corrected issue which caused *CLOAD entries to not be imported/exported for phase dependent loads (PR #2248269)
- Corrected issue reading axisymmetric elements whose *SOLID SECTION properties were defined via a secondary ELSET, which caused these elements/properties to be stored as linear or parabolic solid elements instead of axisymmetric elements (PR# 7175012).

Interfaces - LS-DYNA

- Corrected issue introduced in v11.3.2, where skipping output cases would not behave as expected when importing d3plot files.

Loads and Boundary Conditions

- Corrected issue when expanding geometric loads when the associated elements were parabolic, but had missing midside nodes.
- Corrected issue when using the Model, Load, Nodal on Face command to properly handle elements with missing midside nodes, as well as, loads on edge faces of planar elements.
- Corrected issue with the Model, Load, From Freebody command, when using the Multi-Model option, which could occur if copying freebody loads to a target model using the "Closest Node" method. This could cause the incorrect closest node to be found if the actual closest node had an ID of 1 (PR# 7893234).
- Corrected issue with Model, Load, Map Output From Model command where nodal pressure on multiple faces of the same element would only be mapped to the last face specified on each element.

Meshing

- Corrected issue in the Mesh, Editing, Element Refine command. Previously, when triangular elements were created from quadrilateral elements the fourth node of the quadrilateral was not zeroed. This did not cause a problem translating to Nastran but did cause potential issues for users who accessed the model via the API or Neutral Files.

Element Update

- Corrected issue when using Modify, Update Elements, Orient Plate Normal/First Edge and Modify, Update Elements, Split Quads commands to work properly with elements that have corner thicknesses defined.

Materials

- Corrected issue with the conversion between Anisotropic (2D) and Anisotropic (3D) material types to directly use terms from the two elastic matrices, rather than converting through an intermediate orthotropic representation.

Output and Post-Processing

- Corrected issue with complex results throughout the various aspects of postprocessing that would allow a user to select a computed vector, such as Total Acceleration, before the necessary phase info had been specified.
- Corrected issue when displaying complex results exapnded on-the-fly, which caused post titles to not have a non zero for phase angle, unless a displacement was selected.
- Corrected issue when Deformed Style is set to Animate-MultiSet and the user rotated the model with a Spaceball (3D Mouse), which caused the wrong title for Group to appear in the View Legend, even if no group is selected.

- Corrected issue when model was animating, then the user tried to perform dynamic rotate, which caused the nodes to stay deformed but elements would become undeformed.
- Corrected issue where contour arrows with multiple components that were specified using the Advanced Contour Arrow Options dialog box may be overwritten by various functions in the PostProcessing toolbox (PR# 7919757).
- Corrected issue when displaying contour vectors plate bending moments by having Plate X Moment in the Y direction, while Plate Y Moment in the X direction, which now match the moment directions shown in the Nastran documentation.
- Corrected issue which could cause the output vectors in an output set, either from an "Attached" output file or "As Needed" created via a Data Surface or Table, to disappear if the title was updated using the Model, Output, Create/Manage Set command.

Groups and Layers

- Corrected issue with the Group, Operations, Booleans command to have it automatically follow the Next Group parameter in Tools, Parameters.

Listing

- Corrected issue where listing freebody interface load calculation details to the Data Table, which would repeat the calculated forces in the calculated moments column for each entry, but the calculations were not affected by this. Additionally, listing to the Message window or Clipboard were also not affected.
- Corrected issue when listing options on the LS-Dyna tab of the Connection Property to replace Penalty Check with Penetration Check to the Data Table, Messages Window, or sent to the Entity Editor.

Tools

- Corrected issue with Tools, Mass Properties, Mesh Properties command where when creating a mass element, only the inertias would be populated (i.e., the calculated total mass was missing).
- Corrected issue which could cause the Tools, Check, Element Quality command to calculate different Jacobian values than NX Nastran, when encountering highly distorted tetrahedral elements.

User Interface

- Corrected issue which allowed solids that are not on a visible layer, but whose surfaces are on the visible layer, to highlight and pick the solid.
- Corrected issue that occurred if the "Middle Mouse Button Click for OK" preference was enabled, then the middle button was used in a tabbed dialog, when the focus was in a control located on a tab. In that case, the dialog did not exit and in some cases caused other problems. For example, in the dialog box for the View, Visibility command it deleted all of the icons and the entries from some of the lists.

API

- Corrected issue in feCurveOffsetCurveWasher method which caused the method to always return FE_FAIL.
- Corrected issue in PutCorrelate2 method of the Analysis Manager Object that caused the realInterp and imagInterp arguments to not be saved properly.
- Corrected issue where the Put method on Analysis Manager Object would not return FE_FAIL, even if user was trying to save superelement reference to a non-existing analysis set or an analysis set which was not for NX Nastran or MSC Nastran.
- Corrected issue when starting Femap via the API, through an external executable, which caused the spaceball to not be initialized, thus sapceball initialization was added to feAppVisible.
- Corrected issue in DeleteRows method of the DataTable object. Previously unless the rows were entered in decreasing, sorted order incorrect/unexpected rows would be deleted. Now, rows can be entered in any order.
- Corrected issue in feCrossSectionReport method that previously caused it to list the property ID in the report incorrectly. The correct property data was reported, the ID was simply mislabeled.
- Corrected issue in the Show method of the Set Object and in the feViewShow method of the Application Object. Previously, if an API deleted or created entities that caused the model size to grow or shrink, then Show was called prior to the model size being reevaluated, the selected entities might not be shown.
- Corrected issue when using the feMeshSweepElem method that could cause invalid results if the method was used multiple times, depending on what other methods were used between the calls. In addition, this problem could have also appeared in the other Sweep, Extrude and Revolve API methods, but was never actually witnessed.
- Corrected issue which could potentially issue the wrong return code for VarEquation, VarOutputMap, VarMeshSurface, VarSpatialUVtable, VarSpatialXYZTable, VarParametric2Pt, and VarParametric4Pt methods on the Data Surface Object.
- Corrected issue where the Put method on Analysis Manager Object would not return FE_FAIL, even if user was trying to save superelement reference to a non-existing analysis set or an analysis set which was not for NX Nastran or MSC Nastran.
- Corrected issue Which could arise if pval(57) on the Connection Property Object, which is the option for Dyna Constraint, was set to a value other than 0, 1, or 2. If an invalid value is used, it simply specifies a 0.

FEMAP v11.3.2 New Features and Corrections

Updates and Enhancements

Interfaces - NX Nastran

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

Interfaces - Geometry

- Added support for Solid Edge with Synchronous Technology 9.
- Added support for NX 11.0.

Element - Update Existing Elements

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

API

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

Corrections

Views

- Corrected issue which caused view orientation to not be copied correctly when view is copied. (ER 7763623)

Performance Graphics

- Corrected issue which was causing the orientation vector on Spring/Damper elements to not be displayed when the element orientation was specified using an Orientation Coordinate System, either on the element of the referenced property.

GUI - General

- Corrected issue which occurred when you set a Startup View other than the default by using the File, Preferences command, then shutdown FEMAP with the PostProcessing toolbox open and visible on top of any other tabbed panes. In that case, the problem prevented FEMAP from reopening unless some data was manually cleared from the Windows Registry.

GUI - Dockable Panes

PostProcessing Toolbox - Freebody Tool

- Corrected issue which could occur when "Display Mode" is set to "Section Cut" and "Entity Selection Mode" is set to "Plane/Normal", "Plane/Vector", or "Vector", which caused the "Location Slider" to become "zeroed out" when the coordinate system in the Plane or Vector definition dialog box was changed to a different coordinate system.
- Corrected issue which could occur when "Display Mode" is set to "Section Cut" and "Entity Selection Mode" is set to "Plane/Normal", "Plane/Vector", or "Vector", which caused the overall "section cut path" to not be properly recalculated when the coordinate system in the Plane or Vector definition dialog box was changed to a different coordinate system. This could then allow the "Location Slider" to move the "section cut" well beyond the extents of the model for one portion/side, while not being able to reach the extents of a different portion/side.

Interfaces - Nastran

- Corrected issue when reading ACCEL1 entries which use a coordinate system ID (CID).
- Corrected issue caused by the addition of support for results in the BOUGV Data Block which could cause issues when attaching to output files containing certain types of output.
- Corrected issue when writing PBEND entry in wide field format which could cause the second half of a continuation line to not be written.

Interfaces - LS-DYNA

- Corrected issue when reading output files which contained DCOMP != 0 which directs LS-Dyna to not include output in d3plot for Rigid Elements.

Properties

- Corrected issue which occurred when creating a beam property which occurred when the beam section is a general surface with reference point.

Output and Post-Processing

- Corrected issue when using thee List, Output, Results To Data Table command. Previously if you used a Results Set Processing Data Surface to create combined Results Sets then attempted to use the List, Output, Results To Data Table command to add output vectors from multiple combined output sets to the Data Table in a single command, the results shown for computed output vectors, such as Von Mises Stress were not correct, but were instead the scaled combinations of the same output vector from the underlying output sets. Now, the values are recomputed from the scaled underlying component output values.
- Corrected issue which could cause the program to become unresponsive when an ABAQUS results file (ODB) was attached, the "Double-Sided Planar Contours" option was enabled, and a "calculated" output vector (for example, von Mises Stress) was selected in the Contour drop-down (PR 7799435)

- Corrected issue which could cause the program to exit unexpectedly when using the "View, Advanced Post, Beam Cross Section" command and an element referencing a Beam or Bar cross-section set to NASTRAN was selected.

Element - Update Existing Elements

- Corrected issue when using Modify, Update Elements, Type command which limited properties available for selection to only properties currently being used by the selected elements, which hindered desired functionality. In addition, made similar correction to Modify, Update Elements, Property ID command which now allows any property in the model to be selected from the screen, then only updates any selected element if property type match individual element's current type.
- Corrected several issues with Modify, Update Elements, Line Element Orientation which allowed updates of Spring/Damper elements with coincident nodes and Spring/Damper to Ground elements using invalid methods. If an attempt is now made to update these type of elements using an invalid method, these elements will not be updated.
- Corrected issue with Modify, Update Elements, Line Element Orientation which could cause incorrect vector orientations to be defined on line elements when using the Radial method.

API

- Corrected issue which caused feTextPut and feTextMultiPut to be artificially limited to 255 characters. Both calls now allow access to the maximum 511 characters.
- Corrected issue which caused feFileWriteAnsys to always open an Open/Save dialog box to specify a file name, regardless if one had been properly specified in the API call.
- Corrected issue which caused feFileWriteFNO to only be able to access output from a single attached Nastran output (OP2) file.

FEMAP v11.3.1 New Features and Corrections
Updates and Enhancements

Connection Properties, Regions, and Connectors

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

Loads and Boundary Conditions

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

Meshing

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

Mesh Associativity

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

GUI - Dockable Panes

Data Surface Editor

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

Interfaces - Nastran

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

Interfaces - Geometry

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

Output and Post-Processing

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

API

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

Corrections

Views

- Corrected issue which caused the current orientation to not be saved to/loaded from the View Library (view.esp) when using the View, Visibility or View, Create/Manage commands.
- Corrected issue in View, Rotate, Model command which caused the coordinate system selected with Rotate Around drop-down to be ignored when the XY Top, YZ Right, ZX Front, Bottom, Left, Back, Isometric, Dimetric, or Trimetric button was pressed, thus the resulting rotation was always in the Basic Rectangular Coordinate System.
- Corrected issue which caused the name of the group specified as the Label Group to appear in the View Legend instead of just the Group ID. This only occurred when Multiple Groups were being displayed in the view.
- Corrected issue which caused different behavior when holding down Ctrl+Shift at the same time and moving the mouse to dynamically manipulate the model in the graphics window when the Shift for Pan, Ctrl for Zoom option is enabled on the User Interface tab of File, Preferences. In previous versions, this holding down Shift+Ctrl would always pan the model instead of zooming. This behavior has been restored for 11.3.1.

- Corrected issue which caused constraint symbols to change from "Pins", "Arrows", or "Triangles" to the "default symbol" (single triangle) if the View, Visibility command was used to toggle Constraint Labels on/off (PR 7748429).

Performance Graphics
- Corrected issue which could cause the independent/dependent symbols on RBE2 and RBE3 elements to be "clipped" and not be displayed.
- Corrected issue which caused the independent/dependent symbols of Rigid elements to be reversed, unless Model Style was set to Free Face using the View, Select command.
- Corrected issue which could cause RSPLINE elements to not be displayed properly when the model was deformed.
- Corrected issue which could cause nodes to not be displayed when Model Style was set to Free Edge using the View, Select command.
- Corrected issue which caused line elements not to be "clipped" when using the Model Clipping Plane (i.e., they would remain visible even when they should have been removed from the display).
- Corrected issue which caused labels on enforced displacements defined as constraints which had values between 0.0 and -1.0 to be drawn incorrectly.

Graphics
- Corrected issue which caused all nodes to not be displayed if any individual solid, sheet solid, of general body is hidden via the Model Info tree, the View, Visibility command, or the Select Toolbar.

Connection Properties, Regions, and Connectors
- Corrected issue which caused normal vectors to be drawn incorrectly when Face 2 of linear or parabolic shell elements (3-noded and 6-noded triangle elements, 4-noded and 8-noded quadrilateral elements) are specified in Connection Regions.

GUI - General
- Corrected issue where an error about FEMAP not being registered as a COM server would incorrectly show up when opening FEMAP by double-clicking a .modfem file on Windows 8 and Windows 10 machines. (IR 7732060)
- Corrected issue when using File, Copy commands which could cause surfaces to "bleed through" elements and vice versa. Problem was due to software mode graphics having a 32 bit depth buffer, which is different than the 24-bit depth buffer used in hardware mode. Now, both hardware and software always use the equivalent of a 24-bit depth buffer.
- Corrected several issues when using Query Picking when Perspective is enabled and the view mode is set to "Roll-Thru" (View, Roll-Thru command). First, entities behind the viewer, which was not possible before "Roll-Thru", would be included in the list of entities. Second, due to the perspective mapping anywhere between 20-50% of entities which should have been in the list of entities to select would not be available for selection.

GUI - Dockable Panes
Data Surface Editor

- Corrected issue which caused vector fields on output map Data Surface to be plotted as scalar values in the graphics window.

Meshing Toolbox - Geometry Editing Tool
- Corrected issue which caused a solid to not be remeshed properly with solid elements after the "Project Curve" operation was used to split a surface on the solid.

Interfaces - FEMAP Neutral
- Corrected issue in V11.3 Neutral Files. If Draw/Erase was not active in the model (nothing erased) when the file was written all model data was written to the file but the v11.3 neutral reader could not read past the "empty" Draw/Erase information causing other information like Results to be lost in the transfer. These fixes correct the issue with writing the file properly and also allow both the "incorrect" v11.3 files and the corrected v11.3.1 and beyond files to be read successfully.

Interfaces - Nastran
- Corrected issue in the Nastran translator that caused Pressure loads on Face 5 of Wedge elements and Face 6 of Brick elements to be improperly written to PLOAD4 loads if varying corner pressures were defined. The proper pressures were written, but were incorrectly applied to the wrong element corners.

Interfaces - NX Nastran
- Corrected issue which prevented results from Complex Modal Analysis from being imported properly due to results being interpreted as results from Rotor Dynamics.

Interfaces - ANSYS
- Corrected issue which caused material orientations for plate elements to not be written correctly.
- Corrected issue which caused some mass elements to be skipped when writing the ANSYS input file.
- Corrected issue which caused ESYS entries to be written with an ID of 0 instead of the correct ID for every other coordinate system.
- Corrected issue which could sometimes cause Real Constants to not be written properly to represent thickness for SHELL63 elements when shell elements with offsets also existed in the model.

Interfaces - ABAQUS
- Corrected issue which caused ODB Results Files which contained output created by the CSTRESSERI output variable to not attach properly (IR 7748825).

Interfaces - LS-DYNA
- Corrected issue which caused shell elements which have both an offset and a material orientation angle defined to be written out as *ELEMENT_SHELL_THICKNESS_OFFSET entry with the

offset information on the line before the material angle, instead of on the line after the material angle (PR 7713960)

Meshing

- Corrected issue with Mesh, Connect, Unzip and Mesh, Connect, Closest Link commands where results for CBUSH elements created by specifying a Coordinate System ID would not be displayed in the correct orientation for Contour Arrow plots.

Loads and Boundary Conditions

- Corrected issue when expanding surface pressures to element face corners of solid elements. If the pressure varied across the element, the proper pressures were computed but in some cases were applied to the wrong face corners. This problem only occurred when using the "At Corners" option when defining the load.
- Corrected issue which caused constraint symbols to change from "Pins", "Arrows", or "Triangles" to the "default symbol" (single triangle) if the View, Visibility command was used to toggle Constraint Labels on/off (PR 7748429).

Aeroelasticity

- Corrected issue which caused "cone" shapes to not be displayed as part of Aero Splines. Also, corrected issue which could cause irregular labeling of Aero Splines.
- Corrected issue which caused ID Labels of Aero Bodies to not be displayed. The ID Labels for Aero Panels were displayed correctly.

Output and Post-Processing

- Corrected issue with contour arrow plots of bar and beam bending moments for NASTRAN results that would show plane 1 and plane 2 moments in the incorrect orientation. Also, added vector pairings for ABAQUS beams.

API

- Corrected issue which allowed you to enter IDs of Points as Stress Recovery Points (pval(46) through pval(49) on Property Object) which did not exist in the "outline" of a beam cross-section on the Property object. This can still be done, but the value of the first point in the outline will be used for any IDs which do not exist.
- Corrected issue which caused ComputeGeneralSection method on Property object to not calculate any section properties.

FEMAP v11.3 New Features and Corrections
Updates and Enhancements

Views

- Added ability to specify a "Label Group" to the Group tab of the View, Visibility command.
- Added and modified several commands on the View, Rotate... menu. These commands are also available on the View and Rotation Center icon menu on the View and View-Simple Toolbars. They may also be accessed using the Rotate View commands on the Quick Access Menu (right-mouse click in the Graphics window).
- Added View, Align By, Eye and Directions command.
- Added the ability to control visibility of individual elements. This can be done using the Visibility menu on the context-sensitive menu for elements when Element is the active entity type in the Select Toolbar or by using the Hide Individual Elements or Show All Individual Elements commands on the "Visibility check box" context-sensitive menu for Elements, By Type and Elements, By Shape in the Model Info tree.
- Added ability to double-click in the mouse wheel or middle mouse button to run the View, Autoscale, Visible command, when not in another command. Holding down Shift and double-clicking the mouse wheel or middle mouse button when not in another command will run the View, Autoscale, All command.
- Added ability to display the name of the element quality check currently specified in the Surface Mesh Quality tool of the Meshing Toolbox as a Post Title, when the Surface Mesh Quality is being displayed.
- Updated the View Legend to have a ":" between the item in the legend and the ID(s). Also, the Label Group has been added the View Legend and is shown as "LG:(Group ID)". Finally, items in the View Legend and the Post Titles can now have up to 2 characters to the left of the ":".
- Updated visibility of loads and constraints to not display them when the entity on which they are applied is also not currently visible. This behaves in this manner for mesh-based and geometry-based loads and constraint. Also, if all nodes of a constraint equation are not visible, the constraint equation will also not be displayed.

Analysis Manager

- Added Renumber button to the Analysis Set Manager dialog box, which allows you to renumber either analysis sets or analysis cases in an individual analysis set. To renumber analysis cases, a subcase must be highlighted, otherwise, it will prompt you to renumber analysis sets.
- Added functionality to Boundary Conditions dialog box in the Analysis Set Manager which allows you to specify "-1..Use Active Constraint Set" for Constraints and/or "-1..Use Active Load Set" for Loads, which will use the boundary conditions specified in the active Constraint Set and/or the loads specified in the active Load Set.
- Updated all Nastran analysis manager dialog boxes to use the same NASTRAN title

Connection Properties, Regions, and Connectors

- Added Preview icon button to Connection Region, Fluid Region, NonStructural Mass Region, Bolt Region, and Rotor Region dialog boxes to allow you to highlight the entities currently referenced by the region in the graphics window.
- Updated all regions which allow selection of element faces to use the new Face Selection dialog box and functionality.

Geometry

- Added Plane to Plane method for Modify, Align... commands to align geometry (Point, Curve, Surface, Solid, and Volume). When this method is used, the behavior is similar to Between Coordinate Systems, only each specified plane (X and Y axes) and each plane's normal direction (Z axis) are used to determine XYZ axes.
- Updated Geometry, Solid Cleanup command, which allows the user to Enable Advanced Cleanup, choose categories of "Advanced Cleanup" to attempt, and/or click the Advanced Cleanup Options... button to display a dialog box to select individual options in four different categories. Also, added option to Remove Gashes.
- Added Angle Tolerance to Geometry, Surface, From Mesh command. Controls the allowable difference, in degrees, between the "average mesh normal" at each node of the original mesh and the normal of the newly created surface, at each nodal location. Making this value larger may create geometry which is smoother, but may also cause the new surface to be further away from the original nodal locations.
- Updated commands on the Geometry, Copy...; Geometry, Scale...; Geometry, Rotate...; and Geometry, Reflect... menus to not copy any attachment or reference to boundary surfaces and/or combined curves if only the underlying geometric entities are copied, scaled, rotated, or reflected.

GUI - Toolbars and Icons

Draw/Erase Toolbar - New for 11.3

- This toolbar offers two different modes, one which will temporarily only "Draw" entities of certain types selected with the toolbar, while the other will temporarily "Erase" the selected entities. The selected entities will remain "Drawn" or "Erased" until cleared using the Clear icon on the toolbar. At any time you can switch from "Draw" mode to "Erase" mode and "swap" what appears in the graphics window. The overall functionality of the Draw/Erase toolbar can also be toggled off at any time to display all entities currently visible in the model, then toggled on again.

View and View - Simple Toolbars

- Added or modified the commands on the View and Rotation Center icon menu. The available commands are Rotate About View Center, Rotate About Rotation Center..., Rotate Around View Axes, Rotate Around Model Axes, Rotate Around Coordinate System..., Rotate Around Vector..., Roll-Thru, Advanced Rotate..., Single Axis Rotation, and Set View Center.
- The View-Simple Toolbar is now the default Toolbar for manipulating views. The View Toolbar still exists and can be turned on using the various methods available for displaying toolbars.

Select Toolbar
- Added Visibility commands to the context-sensitive menu for elements when element is the active entity. This includes the Hide Individual Elements... command which controls visibility of individual elements, which was not available in previous versions.

Post Toolbar
- Added Select a Contour Arrow View icon to the Post Toolbar, which will set Contour Style to Arrow. Also, the icons for Deformed Style (None, Deform, Animate) and Contour Style (None, Contour, Criteria, Arrow) will now highlight to indicate the current mode.

Custom and User Tools Toolbar (formally, Custom Tools Toolbar)
- Added the User Tools menu, which works exactly like the Custom Tools menu, but uses a different directory.

GUI - Dockable Panes

Connection Editor - New for 11.3!
- Added the Tools, Connection Editor command, which will display the Connection Editor dockable pane, which provides you with a tool to interactively manage and/or edit a large number of Connectors using an intuitive table control. Much like the Data Table, each Connector appears as a single row separated into a number of columns when it enters the Connection Editor. Also, once in the Connection Editor, information about the Connectors can be sorted, filtered, and evaluated to help you understand the composition and location of each Connector to a greater degree. Unlike the Data Table, the Connection Editor allows you to modify certain aspects of each Connector in the model, including individual field modification in certain columns.

Model Info Tree
- Updated functionality throughout the Model Info tree to automatically select all items in a particular branch, under certain conditions, even if some entities are not in the tree because the number of entities exceeds the value set for Max Entities on the User Interface tab of File, Preferences. Specifically, if you choose a command from a context-sensitive menu while selecting an entity header (for instance, Properties), all entities will be chosen. Also, if using the "Visibility check box" context-sensitive menu, the Show Selected Only, Show All, Hide All and Show/Hide Reverse commands will change visibility for all entities, not just the ones currently shown in the Model Info tree.
- Added Select Show Only... and Select to Hide... commands to the "Visibility check box" context-sensitive menu for Coordinate Systems, Geometry, Regions, Connectors, Aero Panels/Bodies, Aero Splines, Aero Control Surfaces, Materials, and Properties. In all cases, the selected command displays the standard entity selection dialog box which is used to select entities of the appropriate entity type to "Show Only" or "Hide", respectively.
- Added Show All Individual Elements command to the "Visibility check box" context-sensitive menu for Elements, By Type and Elements, By Shape, which simply sets the visibility of all individual elements to "visible".

- Added Hide Individual Elements command to the "Visibility check box" context-sensitive menu for Elements, By Type and Elements, By Shape, which displays the standard entity selection dialog box to select individual elements to "hide".
- Added Plot Study sub-menu to the context-sensitive menu for Analysis Study. The commands on this sub-menu allow you to create a Chart called "Quick Plot" of "nodal results vs Output Set" in the Charting dockable pane. These commands are only available when multiple output sets reside in a Study and the Charting dockable pane is open. Only Translations (Total, X, Y, or Z), Accelerations (Total, X, Y, or Z), and Nodal Temperatures are currently available.

Meshing Toolbox - All Tools which potentially modify geometry
- Added functionality which tracks "Mesh Points" currently defined on surfaces, then automatically reassigns the mesh points to the appropriate surface when geometry is modified. Reassignment of mesh points only occurs when geometry is modified using commands in the various tools of the Meshing Toolbox, not when using commands on the Geometry menu.

Meshing Toolbox - Feature Removal Tool
- Added ability to select Point as a Feature Type to remove, which will attempt to remove any selected points which are redundant.
- Added Combine Surfaces to Curve Options when Feature Type is set to Curve. This option uses the Parasolid kernel to calculate a new surface through the original two surfaces, which share the selected curve. If successful, all the curves sharing the original two surfaces are removed, as they become redundant. This option will also remove any redundant points on the ends of the removed curves.

Meshing Toolbox - Mesh Surface Tool
- Added Max Quads option to Mesh Surface tool.

Meshing Toolbox - Surface Mesh Quality
- Added option to choose between Femap and Nastran element quality checks. When Quality Source is set to Nastran, Quality Type can be set to Quad Skew, Quad Taper, Quad Warp, Quad IAMin, Quad IAMax, Quad AR, Tria Skew, Tria IAMax, Tria AR. Also, element quality is now only shown on elements which are currently visible.

Meshing Toolbox - Locator
- Added option to choose between Femap and Nastran element quality checks when Search For is set to Elements and Search Method is set to Quality in the Locator. When Quality Source is set to Nastran, Quality Type can be set to Quad Skew, Quad Taper, Quad Warp, Quad IAMin, Quad IAMax, Quad AR, Tria Skew, Tria IAMax, Tria AR.

PostProcessing Toolbox - General
- Updated the default behavior of the PostProcessing toolbox to use the Expand Active Tool Only option.

PostProcessing Toolbox - Contour and Deform Tools
- Updated the Contour tool when Style is set to Contour Arrow to provide access to the new capabilities when displaying an "arrow plot".

PostProcessing Toolbox - Freebody Tool
- Updated the behavior of the icons used for listing in the Freebody tool to use the functionality of the updated List, Output, Force Balance command. Certain options in the List Force Balance dialog box will be pre-selected based on the Display Mode of the freebody entity currently active in the Freebody tool. Also, the List Current Freebody to Data Table and List Current Freebody Summation to Data Table icons have been removed.

Entity Editor
- Added appropriate NX Nastran element quality values when an element is in the Entity Editor. For instance, if a quadrilateral element is in the Entity Editor, you will see Nastran SKEW (Quad Skew), Nastran IAMIN (Quad IAMIN), Nastran IAMAX (Quad IAMAX), and Nastran AR (Quad AR), while the value for Quad Warp is the same as the Nastran Warping FEMAP element quality check and the value for Quad Taper is the same as the Alt Taper FEMAP element quality check, thus they are not shown individually.

Charting Pane
- Added Scale Override option to Chart Settings tab of Charting dialog box. When enabled, the Y values of all Data Series currently displayed in a Chart will be scaled by the Scale Override value, not the value specified for Scale on the individual Data Series. If a Chart does not have this option enabled, then the value specified for Scale on the individual Data Series is used.
- Added Study drop-down on the Vector vs. Output Set and Vector vs. Vector tabs in the Chart Data Series dialog box, which allows you to specify a range of output sets by selecting an Analysis Study.
- Added functionality which causes the mouse cursor symbol to change to a cross (+) when the pointer is positioned over a data point and Tooltips are enabled. This makes it easier to determine when the cursor is over a point when the chart is zoomed

Data Table
- Added additional options to the Data Table filter, specifically to filter entities by Text. Previously you could only filter Text as Contains. Now, you can choose Contains, Not Contains, Equals, or Not Equals, with Contains and Not Contains only needing to match a portion, while Equals and Not Equals need an exact match.
- Updated the Show When Selected capability for Connectors to highlight both the Connector and associated Connection Regions, which now matches the behavior of the Model Info tree.

Interfaces - FEMAP Neutral
- Updated Neutral Read and Write for v11.3 changes
- Updated all FEMAP Neutral file converters from version 10.0 and above to be 64.bit applications.

- Added ability to select the Write Output option in the Entity Options section of the Neutral File Write Options dialog box when Group Only is selected. This will write only output to the neutral file for entities in the group. This option is off by default and must be turned on to write the output for the specified group.
- Added Binary Output File (FNO) documentation to Neutral File docs (neutral.pdf).
- Added the "Automatic Add" group ID to the Neutral File.

Interfaces - Nastran

- Added Additional Command Line Arguments field to the NASTRAN Executive and Solution Options dialog box. This allows the user to include command line arguments which are not explicitly supported by FEMAP, when launching Nastran. If any command line arguments are specified in the Arguments field for NX Nastran, MSC/MD Nastran, and/or Autodesk Nastran on the Solvers tab of File, Preferences, they will also appear in the Additional Command Line Arguments field when a new analysis set is created using the corresponding solver.
- Added functionality which will attempt to use the Nastran Subcase ID as the FEMAP output set ID. If an output set of that ID already exists, it will use the next available output set ID higher than the Nastran Subcase ID. To enable this functionality, turn on the Use Static Subcase IDs option on the Results tab of File, Preferences.
- Added functionality which will add a revision number when creating a new output set which corresponds to a Nastran Subcase ID, when a FEMAP output set corresponding to that Nastran subcase ID already exists in the model. To enable this functionality, turn on the Track Revision option on the Results tab of File, Preferences.
- Added support to read strain output on CBEND elements from the *.op2 file.
- Added support to write NonZero Constraints as SPC entries. An individual SPC entry will be written for each non-zero value specified on a node. When reading SPC entries from a Nastran input file, a single constraint will be created if multiple SPC entries have the same SID value and G1 value.
- Added support to read and write the RANDT1 entry, which specifies values for autocorrelation function time lag. These values can be specified in the Autocorrelation Function Time Lag section of the NASTRAN Power Spectral Density Factors dialog box.
- Added support to read and write XYPRINT\PEAK velocities, which can be selected by checking the appropriate check boxes in the Nodal Output Requests section of the NASTRAN Output for Random Analysis dialog box. Also, added support to create PSDF and/or AUTO XYPRINT\PEAK entries, which can be specified by checking the PSDF and/or AUTO check boxes in the NASTRAN Output for Random Analysis dialog box. Finally, added reading XYPRINT\PEAK AUTO\PSDF results from the *.f06 file.
- Updated the Varying Translational Acceleration body load, which creates an ACCEL entry, to allow the user to define an acceleration where the acceleration direction is aligned with the direction of acceleration variation.
- Updated how the TIME executive control entry is written. When creating a new analysis set, the default value for Max Time (in minutes) in the NASTRAN Executive and Solution Options dialog box will be 0, which causes the TIME entry to not be written. TIME will only be written when the user sets the value above 0.

- Updated Nastran interface so that real values between 100,000 and 1,000,000 come out as a full 8 character wide field as long as you have the Improve Real Number Precision option enabled on the Interfaces tab of File, Preferences. Previously some cases came out as 7 characters and lost the first decimal digit.
- Updated the name of the Advanced Options tab in the NASTRAN Dynamic Analysis dialog box to be Solution Frequencies and modified how list of solution frequencies are defined to allow more than two FREQ or FREQi entries to be written to a Nastran input file or read in from a Nastran input file.

Interfaces - NX Nastran

- Added support to read and write PLASALG and corresponding value for the NXSTRAT entry.
- Added ability to read the analysis type from the CASECC data block when using SOL 601.
- Added support to handle Drilling Grid Point Force output.
- Added support to import and attach to output on Solid Laminate elements found in *.op2 files created by NX Nastran 11.0.
- Added support to write MAT3 entry for plane strain plane stress elements (CPLSTNi, CPLSTSi elements).
- Updated translator to not write nothing in the NORM field of the EIGC entry, as NX Nastran has removed the option. The Normalization Method section in the NASTRAN Modal Analysis dialog box becomes unavailable when any option in the Complex Solution Methods section is selected.
- Updated translator for Rotor Dynamics to always write the EIGRL entry using MASS normalization no matter what the Normalization Method option is set to in the NASTRAN Modal Analysis dialog box.

Interfaces - ANSYS

- Added ANSYS Executive and Solution Control dialog box to the Analysis Set Manager. The ANSYS Executive and Solution Control dialog box contains information about current ANSYS Version setup to run with FEMAP and allows you to specify a number of command line arguments.
- Added support to write non-zero constraints as D, (node ID), (UX, UY, UZ, ROTX, ROTY, or ROTZ), (non-zero value). Also, added support to read non-zero values on constraints from an ANSYS input file.
- Added support to write DOF Spring to Ground elements as COMBIN14 elements. An additional node constrained in all six degrees of freedom will also be written to the ANSYS input file. When COMBIN14 elements are read from an ANSYS input file, DOF Spring elements/properties are created, not DOF Spring to Ground elements/properties.
- Added support to write Spring/Damper elements which reference a Spring/Damper property with Type set to CBUSH and Spring/Damper to Ground elements as MATRIX27 elements.
- Updated support when reading MATRIX27 elements and associated property information from ANSYS input file, which now become General Matrix elements referencing General Matrix properties set to the appropriate Matrix Type in FEMAP.

Interfaces - ABAQUS

- Added support to write DOF Spring elements which have both Stiffness and Damping defined as *ELEMENT, TYPE=SPRING2/*ELEMENT, TYPE=DASHPOT2 combinations. When reading these entries from an ABAQUS input file, when both Stiffness and Damping are defined, two DOF Spring properties will be created (one containing the stiffness values, the other containing the damping values) and two DOF Spring elements will be created (one referencing the property with the stiffness value, the other referencing the property with the damping value).

- Added support to write DOF Spring to Ground elements with only Stiffness values defined as *ELEMENT, TYPE=SPRING1, with only Damping values defined as *ELEMENT, TYPE=DASHPOT1, and with both Stiffness and Damping values defined as *ELEMENT, TYPE=SPRING1/*ELEMENT, TYPE=DASHPOT1 combinations. When reading these entries from an ABAQUS input file, when both Stiffness and Damping are defined, two DOF Spring to Ground properties will be created (one containing the stiffness values, the other containing the damping values) and two DOF Spring to Ground elements will be created (one referencing the property with the stiffness value, the other referencing the property with the damping value).

- Added support to write Spring/Damper elements which reference a Spring/Damper property with Type set to CBUSH and Spring/Damper to Ground elements as *MATRIX INPUT/*MATRIX ASSEMBLE combinations. When read from the ABAQUS input file, General Matrix elements referencing General Matrix properties set to the appropriate Matrix Type will be created in FEMAP.

Interfaces - LS-DYNA

- Added Advanced... button to LS-DYNA Analysis Control dialog box, which allows the user to enter values which will write the *CONTROL_IMPLICIT_AUTO and *CONTROL_IMPLICIT_GENERAL entries when using the implicit solver.

- Added support to write non-zero constraints to the LS-Dyna input file as *BOUNDARY_PRESCRIBED_MOTION_NODE entries.

- Added "98..LS-DYNA Simplified Johnson Cook" material type to Other Types, which writes *MAT_SIMPLIFIED_JOHNSON_COOK.

- Added support to use a combination of beam elements with sections defined in Femap (rectangular or circular bar) with Formulation set to "9..Spotweld" and Material Type "100..LS-DYNA Spotweld" in Other Types. The section properties will be used to determine the required values on *BEAM_SECTION cards. If the cross section is not a rectangular or circular bar and Area>0.0, then the square root of the Area will be used for all required thicknesses. However, thicknesses may also be specified when creating or editing the "100..LS-DYNA Spotweld" material, but will only be used if the beam section property has an Area = 0.0.

- Updated the translator to not always write the *CONTROL_SOLUTION entry.

- Updated default value specified for Termination Time in the LS-DYNA Analysis Control dialog box to be 1.0.

- Updated default value specified for Output Time Interval in the LS-DYNA Analysis Control dialog box to be 0.01.

- Updated "2..LS-DYNA Orthotropic Elastic" material type in Other Types by adding Shear Mod Freq Damp G and Limit Stress SIGF fields.

- Updated "34..LS-DYNA Fabric" material type in Other Types by adding 16 new fields.

- Updated "36..LS-DYNA 3-Parameter Barlat" material type in Other Types by adding Hardening Rule Load Func field.
- Updated "54..LS-DYNA Enhanced Composite Damage" material type in Other Types by adding 2WAY (1=On), Mat Angle MANGLE, Pct Failed Layers PFL, Dmg Init T Shear EPSF, Final Rupture Shear EPSR, Tr Shear Max Damg TSMD, Ortho softening SOFT2, Max->Min Fib T SLIMT1, Max->Min Fib C SLIMC1, Max->Min Mtx T SLIMT2, Max->Min Mtx C SLIMC2, Max->Min Shear SLIMS, Stress Red Cycles NCYRED, Trans Shear Soft SOFTG, Load Curve XC LCXC, Load Curve XT LCXT, Load Curve YC LCYC, Load Curve YT LCYT, Load Curve SC LCSC, and strnRate Avg Opt DT fields.
- Updated "67..LS-DYNA Nonlinear Elastic Discrete Beam" material type in Other Types by adding CST (0,1 or 2) and SCOOR (-3 to 3) fields.
- Updated "68..LS-DYNA Nonlinear Plastic Discrete Beam" material type in Other Types by adding CST (0,1 or 2) and SCOOR (-3 to 3) fields.
- Updated "103..LS-DYNA Anisotropic Viscoplastic" material type in Other Types by adding Fail and NUMINT fields.
- Updated "126..LS-DYNA Modified Honeycomb" material type in Other Types by adding VREF, TREF, and SHDFLG(0,1) fields.

Interfaces - PATRAN

- Added support for reading RBE2 and RBE3 elements (MPCs) from the PATRAN Neutral File Packet 14.
- Updated support when reading PATRAN materials to support Material Types 4 (thermal isotropic) and 5 (thermal anisotropic), and added reading of specific heat and emissivity material properties.
- Added writing of non-zero constraints to PATRAN Neutral File. When reading non-zero constraints, they will come displacement loads.

Interfaces - I-DEAS

- Added support to read and write non zero constraints.

Interfaces - Geometry

- Added support for Parasolid 29.0, Solid Edge with Synchronous Technology 8, Pro/Engineer CREO 3, CATIA V5-6R2015 SP3, ACIS 26 SP1, and SolidWorks 2016.

Loads and Boundary Conditions

- Added ability to modify the color, modify the layer, scale the values, change the function ID, or delete loads of specified type(s) in the active Load Set, all Load Sets, or any number of selected Load Sets. This functionality is found in the Modify, Color, Load; Modify, Layer, Load; Modify, Update Other, Scale Load; Modify, Update Other, Load Function ID; and Delete, Model, Load - Individual commands. Also, updated Modify, Update Other, Load Phase and Delete, Model, Load - Body commands to allow selection of any number of Load Sets.
- Added ability to modify the color, modify the layer, or delete constraints of specified type(s) in the active Constraint Set, all Constraint Sets, or any number of selected Constraint Sets. This

functionality is found in the Modify, Color, Constraint; Modify, Layer, Constraint; and Delete, Model, Constraint - Individual commands.

- Added ability to specify "non-zero constraints" using the Model, Constraint, Nodal command.
- Added a warning message/question prompting the user to save the model if geometric constraints are translated that cause the nodal output coordinate systems to be updated. This can occur when exporting/analyzing Nastran, ANSYS, and ABAQUS models.
- Updated Modify, Edit, Load - Definition to issue a warning message to inform when the user is editing a Load Definition which contains non-uniform load values. Also, updated Modify, Edit, Constraint Definition to issue a warning when editing a Constraint Definition which has non-uniform degrees of freedom specified.
- Updated Model, Load... commands to support creating corner pressures when using Variable or Data Surface.

Properties

- Added Spring/Damper to Ground property type which specifies Stiffness, Damping, Structural Damping, and other options for the Spring/Damper to Ground element.
- Added DOF Spring to Ground property type which specifies Stiffness, Damping, Connect to DOF, and other options for the Spring/Damper to Ground element.
- Updated the name of the Stiffness Matrix property type to General Matrix. There is now a Matrix Type which allows the specified values to be used as a Stiffness Matrix, Damping Matrix, or Mass Matrix. Also, the values can now be entered using a 6x6 Matrix or 12x12 Matrix.
- Updated the Cross Section Definition dialog box, which is accessed by clicking the Shape button in the Define Property... dialog box when creating a Bar, Beam, or Curved Beam property. This now provides useful information to the user and this information can be copied to the Clipboard or saved to a file for use in reports.

Output and Post-Processing

- Added ability to include Max/Min information from the currently displayed output vector in the Post Titles. To display the Max/Min information, use the View, Options command, set Category to PostProcessing, select Post Titles from the Options list, then set Legend Style to either "3..Titles and Min/Max Data" or "4..Titles and Min/Max/Average".
- Added option to display the colors in the Contour/Criteria Legend in reverse order. To display the colors in reverse order, use the View, Options command, set Category to PostProcessing, select Contour/Criteria Legend from the Options list, then set Position to "8..Top Left, Reversed", "9..Top Center, Reversed", "10..Top Right, Reversed", "11..Center Left, Reversed", "12..Center Right, Reversed", "13..Bottom Left, Reversed", "14..Bottom Center, Reversed", "15..Bottom Right, Reversed".
- Updated Model, Output, From Load command to automatically support creation of varying output when selecting Elemental Pressures that vary at the face corners.
- Updated the name of the Contour Style option Vector to Arrow. Also, added options to automatically select the type of arrow plot, arrow style, any additional contour vectors, and other options based on the output vector currently specified in the Contour drop-down of the Select PostProcessing Data dialog box.

Element - Spring/Damper to Ground - New for 11.3!

- Added Spring/Damper to Ground element type. This is a single node CBUSH-Style element which connects the selected node to "Ground" (essentially, connects the selected node to a "phantom" node with the same coordinates, constrained in all 6 DOF). Stiffness, damping, structural damping, and other options are defined on the Spring /Damper to Ground property.

Element - DOF Spring to Ground - New for 11.3!

- Added DOF Spring to Ground element type. This is a single node spring which connects a single DOF of the selected node to "Ground" (essentially, connects the selected node to a "phantom" node with the same coordinates, constrained in all 6 DOF). Stiffness, damping, and other options are defined on the DOF Spring to Ground property.

Element - General Matrix (formally Stiffness Matrix)

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

Element - Update Existing Elements

- Added Increment and Percent methods to the Modify, Update Elements, Adjust Plate Thickness/ Offset command. If using Increment, the specified value can be positive or negative and will simply be added to or subtracted from the existing thickness/offsets. If using Percent, the specified value can be positive or negative, must be entered as a percentage, and is used to modify the thickness/offsets by multiplying the original thickness/offsets by 1/100 of entered value, then adding the value to or subtracting the value from the original thickness/offsets.

Element - Aligning Nodes and Elements

- Added Plane to Plane method for Modify, Align... commands to align nodes and elements When this method is used, the behavior is similar to Between Coordinate Systems, only each specified plane (X and Y axes) and each plane's normal direction (Z axis) are used to determine XYZ axes.

Meshing

- Added Max Quads option to Mesh, Geometry, Surface command. When used, the surface mesher will attempt to create as few triangular elements as possible and in certain cases, will produce a "quad only" mesh.
- Added Mesh, Editing, Element Refine command. This command allows you to dynamically "highlight" shell elements to refine using one of two patterns, "original quad element split into four quad elements" (i.e., the 1 to 4 Pattern) or "original quad element split into nine quad elements" (i.e., the 1 to 9 Pattern). Any elements surrounding the "refined elements" will be automatically split using appropriate "transition patterns" to maintain connectivity between the "newly refined mesh" and the "original mesh".

- Added capability to the Mesh, Editing, Edge Split command to select multiple nodes and automatically split between all of the element edges that they define.
- Added capability to the Mesh, Extrude, Curve; Mesh, Extrude, Element; and Mesh, Extrude, Element Face commands to specify the extrusion direction and number by selecting either element edges or a path of nodes. This allows you to easily extrude parts of an existing mesh to locations that match another portion of a mesh, including irregularly spaced locations.
- Updated Mesh, Editing, Edge Split command to automatically adjust element corner thicknesses when splitting tapered planar elements.
- Updated the Quad mesher and Tri Subdivision mesher to honor the Surface Interior Mesh Growth factor specified in the Automatic Mesh Sizing dialog box.
- Updated the Mesh, Editing, Interactive and Mesh, Editing, Split commands to only merge nodes of elements that have been split or those adjacent to a split. Previously all nodes were merged.
- Updated various commands on the Mesh, Editing... menu to create and maintain associativity to multiple solids for nodes that lie on the boundary of adjoining solids.
- Updated the functionality of the icon buttons in dialog box of the Mesh, Editing, Rigid Connectivity command to simply highlight all of the nodes on the element or element instead of bringing up a single entity selection dialog box. This dialog box was also updated for the Mesh, Connect, Rigid command and the Model, Load, From Freebody command, when using Multi-Model.

Groups and Layers

- Added Group, Operations, Generate Visible command, which automatically creates a group based on the entities currently visible in the active view. The command is "what you see is what you get", so if an entity is not currently visible, it will not be added to the new group.
- Added Group, Operations, Generate Elem Shape command, which automatically creates a single group or a number of separate groups in your model based on element shape.
- Added Group, Region, on Solid command, which will add any regions defined using geometry on selected Solid or mesh associated with selected Solid to the active group.
- Added Group, Coord Sys, on Element command, which will add any coordinate system referenced by a selected element to the active group.
- Added ability to add Loads and Constraints on entity types which do not currently exist in the model to the active group. For instance, you could use Group, Load, Elemental to add Loads on element IDs before any elements exist in the model. Previously, these commands were unavailable until the entity type exists in the model.

Listing

- Updated the List, Model, Load - Individual command to allow you to list loads in the Active Load Set, all Load Sets, or any number of selected load sets. The same capability was add for the List, Model, Constraint - Individual command, only you choose to list from the active Constraint Set, all Constraint Sets, or any number of selected Constraint Sets.
- Updated List, Output, Contoured Results to Data Table command when Contour Style is set to Arrow. When the Select Output from Contour Vector option is enabled, lists the "Element ID" or "Node ID" in the first column, depending on the type of output being displayed. Then, depending

on the option set for Arrow Type, up to 3 additional columns containing output values will be listed.

- Updated the List, Output, Force Balance command to consolidate functionality from various commands previously on the List, Output... menu (Force Balance to Data Table, Force Balance Interface Load, Force Balance Interface Load to Data Table, Freebody Nodal Summations, and Freebody Nodal Summations to Data Table).

Renumber

- Added Modify, Renumber, All command. This command is designed to renumber all individual entities of each entity type currently in the model via a single Renumber All dialog box.

Tools

Measure, Distance Between Geometry

- Updated the command to allow you to measure from any entity type to an arbitrary Plane defined by the user. The Overall Only option is not available when To is set to Plane, therefore only the Minimum distance to the specified plane is reported.

Check, Element Quality

- Added NX 3 additional Nastran Element Quality checks, Quad AR, Tria AR, and TRIA EPLR, which can be accessed along with the 22 other NX Nastran element quality checks by clicking the NX Nastran tab in the Check Element Quality dialog box.

User Interface - General

- Added Add All Connected Elements option to the Pick^ menu of the standard entity selection dialog box when selecting elements.
- Added on Element to the Method^ menu in the standard entity selection dialog box when selecting Coordinate Systems.
- Added on Solid to the Method^ menu in the standard entity selection dialog box when selecting Regions.
- Added ability to use Pick Query or Pick Front when selecting Regions.
- Added ability double-click in the Graphics window, when not in a command, to Autoscale. See Views for details.
- Updated fast picking to improve performance when picking all entity types. It is now about 10X faster.
- Updated font selection in Graphics window to support new fonts loaded with Windows 8 and 10
- Updated icons for Properties to reflect the Property Type. If Bar/Beam Shape is defined, the icon will be shown as the selected shape ("G" icon is for General Section beam).
- Updated Property drop-down controls on many commands to automatically only show properties that are compatible with the current operation. In commands where a property can be created "on-the-fly", it now automatically switches the active Property type to something that is compatible with the operation if the current active Property type is incompatible. Also, the width of the drop-down list will now adjust to show titles wider than the drop-down control.

- Updated usage of the mouse wheel to work on the window/pane the cursor is currently over. Allows multiple graphics views to be manipulated and panes to be scrolled without first clicking in the window to activate the window. In Windows 7 clicking in the window to activate is still required when a dialog box in not open. Only graphics windows from the current model can be manipulated.
- Updated Face Selection in all command which involve selecting Element Faces or Element Edges.

Performance Graphics

- Added support to accelerate graphics for Rigid Elements (RBE2, RBE3, and RSPLINE)
- Updated FEMAP to turn off Performance Graphics automatically when a mode or command is not supported, then return to Performance Graphics after leaving the mode or no longer in that command.

Preferences

Graphics
- Added Fast Pick Visible to Graphics Options section. When enabled, picking is enhanced to allow "Pick Front" to be used in conjunction with area picking (Box, Circle, Polygon, or Freehand). Also, the overall performance of picking when using "Pick Front" is improved.
- Added TDR Protection to Advanced/Debug Options section.

User Interface
- Added Dynamic Zoom Around Cursor Location to Mouse Interface section. When enabled, scrolling the mouse middle mouse wheel or clicking and dragging the mouse with the Shift key held down will zoom in/out around the location of the cursor. When off, which is default, zooming in/out by scrolling the mouse wheel or clicking and dragging the mouse with the Shift key held down will occur around the center of the graphics window.
- Added Dynamic Rotate Around Cursor Location to Mouse Interface section. When enabled, rotates around an automatically selected location on the model near the current location of the cursor. When off, which is default, rotation will occur around the center of the view or rotation center.
- Changed Tooltip Duration to Duration in the Graphical Selection section.

Database
- Added Reset Next ID after Delete All to Database Options section. When enabled, which is the default, this option will return the "Next ID" of certain entity types to "1" after the last entity of that type has been deleted from the model. The entity types tracked by this option are Point, Curve, Surface, Solid, Volume, Coordinate System, Node, Element, Material, Property, Connection Property, Connection Region, Connector, Aero Panel/Body, Aero Property, Aero Spline, Aero Control Surface, Load Set, Constraint Set, Group, Text, View, and Output Set.

Solvers - New for 11.3!

- Added Solvers tab to specify the location of solver programs and optionally enter command line options. The Solvers tab allows you to specify a location of a Program (i.e., finite elements solver executable) and any command line Arguments which should be included when the solver program is automatically launched by FEMAP

Geometry/Model

- Added Allow Solid Boolean to Create NonManifold Geometry option to Geometry Preferences section. When on, allows any "Boolean" command on the Geometry, Solid… menu (Geometry, Solid, Add/Remove/Common/Embed/Intersect commands) to potentially create NonManifold geometry (i.e., a Parasolid "General Body") as a result of the operation. When off, if any operation would create a NonManifold body, you will be asked "Ok to allow this operation to result in a NonManifold Solid?". Answering Yes will create a General Body, while answering No will cause the command to fail.
- Updated the Element Quality Preferences dialog box accessed via the Element Quality button by adding 3 new element quality checks to the NX Nastran tab (Quad AR, Tria AR, and Tria EPLR) and changing all operators on the NX Nastran tab from "<=" or ">=" to "<" or ">", which matches the functionality of GEOMCHECK.

Results

- Added Auto Upgrade Abaqus ODB Database option to File Options section. When enabled, the Abaqus ODB Database will automatically be updated to the most current version of the ABAQUS ODB Database supported by that version of FEMAP.
- Added Nastran Options section and moved two existing options, Output Set Titles (formally Nastran Output Set Titles) and Append Femap Title, from the File Options section into this section.
- Added Use Static Subcase IDs option to Nastran Options section. When this option is enabled, an attempt will be made to create Output Sets using IDs corresponding to results for specific Subcase IDs in a Nastran Output File which has been imported or attached. If an output set with that ID already exists in the FEMAP model, then the next empty ID after the Subcase ID will be used.
- Added Track Revision option to Nastran Options section. When this option is enabled, a Nastran Subcase ID will be stored as the "Case" ID on each Output Set created by importing or attaching to a Nastran Results File. If another Nastran Output File containing the same Nastran Subcase ID is imported or attached, the appropriate "Revision" number will also be stored on the Output Set.
- Added Freebody Defaults section and added the Set Freebody Defaults button. The Set Freebody Defaults button in this section will open the Default Freebody Settings dialog box, which allows you to specify the default settings to be used every time a new Freebody entity is created in the model.

Library/Startup

- Added User Tools Path field to Startup Program File/Basic Script/Executable and Custom/User Tools section. FEMAP contains a toolbar called Custom and User Tools. This toolbar allows you to choose directories on your machine where you can access the "API scripts provided with the latest version of FEMAP" (Custom Tools defaults to the "API" directory shipped with FEMAP) and store any "custom commands and tools" created by you and/or an engineering organization (User Tools).

Files used by Custom Tools and User Tools can be recorded Program Files (*.PRO or *.PRG files), FEMAP Basic scripts (usually *.BAS files), or "other" executable (for instance, a Visual Basic script compiled into a *.EXE file). The Custom Tools and User Tools icon menus on the Custom and User Tools toolbar will take any of those file types it locates in the specified directories and automatically place them into the appropriate menu structure found on the Custom and User Tools toolbar.

API

Overall
- Added checking at startup and warning messages to indicate whether the API/COM server is properly registered for FEMAP.

New and updated API Objects and Attributes
- Added Draw/Erase (feDrawErase) object to the API. Also, added Enable, EraseMode, AutoSelectMesh, ExclusiveDrawMode, EraseGeometry, and EraseMesh attributes to the Draw/Erase Object.
- Added Frequency (feFreq) object to the API. Also, added type, F1, F2, DF, FSPD, NF, and LOG attributes to the Frequency Object.
- Added View Orient (feViewOrient) object to the API. Also, added Center, vCenter, Magnification, ModelAlwaysInFront, AutoAspectRatio, AspectRatio, Perspective, PerspectiveAngle, RotationAxisOption, RotationCenterOption, RotationCoordSys, SingleAxis, RotationCenter, vRotationCenter, RotationVector, and vRotationVector attributes to the View Orient Object.
- Added NasDynFreqID, NasExecGPUOpt, LinkedSolverOption, NasNXStratPlasalg, NasExecUserCmdLine, NasRandCorrOn, NasRandCorrLagInterval, NasRandCorrLagStart, NasRandCorrLagMax, vNasRandXYRequest2, NasRandXYAuto, NasRandXYPsdf, NasNonlinAdapt, AnsCmlOn, AnsCmlVersion, AnsCmlPid, AnsCmlProd, AnsCmlList, AnsCmlJobname, AnsCmlDir, AnsCmlDBmem, AnsCmlWSmem, and AnsCmlUser attributes to the Analysis Manager Object
- Added NonZeroConstraint, value, and vvalue attributes to the BCNode Object.
- Added ScaleInherit and MasterScale attributes to the Chart Object.
- Added StudyID attribute to the Chart Data Series Object.
- Added SpringUseLocation, SpringLocation, SpringUsePropLocation, SpringUseCID, SpringNoOrient, SpringCID, and SpringUsePropCID attributes to the Element Object.
- Added NastranQuadAROn, NastranQuadARLimit, NastranTriaAROn, NastranTriaARLimit, NastranTriaEPLROn, and NastranTriaEPLRLimit attributes to the Element Quality Object.
- Updated SumComponents, vSumComponents, Group, DisplayMode, ShowTotalVec, ShowNodalVec, SumContributions, and vSumContributions attributes to the Freebody Object.
- Added RadViewFactorZTOL and bRadViewFactorZTOL attributes to the Load Set Object.
- Added NumberOfLoads and NumberOfConstraints attributes to the Node Object.
- Added nas_case and nas_revision attributes to the OutputSet Object.
- Added attrVertexLoopsAsHardPoints and attrMaxQuads to the Surface Object.
- Added PerspectiveAngle, vContourVecOn, ContourVecAutoVec, ContourVecAutoOrient, TransformDeformXInput, TransformDeformYInput, TransformDeformZInput,

TransformPlateTolerance, ContourVecMinVecOn, ContourVecMinVec, ContourVecMinLen, ContourVecColor1, and LabelGroup to the View Object.

Removed API Objects and Attributes
- Removed LaunchWithVisQ,vNasDynNoFreq, vNasDynLogInterp, vNasDynFreqType, vNasDynMinFreq, vNasDynMaxFreq, vNasDynSpreadCluster, NasDynNoFreq, NasDynLogInterp, NasDynFreqType, NasDynMinFreq, NasDynMaxFreq, and NasDynSpreadCluster attributes from the Analysis Manager Object.
- Removed PerspectiveDist from the View Object.

New and Updated API Methods
- Added AllFreqOn, AllFreqOff, TurnOnFreq, and TurnOffFreq to the Analysis Manager object.
- Added GetAll and PutAll to the BCEqn object.
- Added SetFixed, SetPinned, SetNoRotation, and SetArbitrary to the BCGeom object.
- Added AddNonZero to the BCNode object.
- Added RemoveAllDataSeries to the Chart object.
- Added BoundingBox and BoundingBoxInCSys to the Connection object.
- Added BoundingBoxInCSys to the Connection Region object.
- Added IsMergeable and BoundingBoxInCSys to the Curve object.
- Added ClearAll, Clear, SaveGroup, LoadGroup, Grow, Shrink, HasErased, HasErasedType, EraseSet, and GetErased to the Draw/Erase object.
- Added GetAllArray3, GetSpringOrient, SetSpringOrient, GetSpringOffset, and SetSpringOffset to the Element object.
- Added GetNastranQuadAR, NastranQuadAR, GetNastranTriaAR, NastranTriaAR, GetNastranTriaEPLR, and NastranTriaEPLR to the Element Quality object.
- Added AddFreq, UpdFreq, AddFreqByFuncID, UpdFreqByFuncID, AddFreq1, UpdFreq1, AddFreq2, UpdFreq2, AddFreq3, UpdFreq3, AddFreq4, UpdFreq4, AddFreq5, UpdFreq5, AddFreq5ByFuncID, UpdFreq5ByFuncID, AllOn, AllOff, TurnOn, and Turn Off to the Frequency object.
- Added AddOpt and AddOpt2 to the Group object.
- Added GetOutputDataSurface and MapOutputDataToLocation to the Map Output object.
- Added GetValueArray, PutValueArray, PutValue, CopyArray, GetLibraryOfType to the Material object. Also, updated GetLibrary.
- Added GetValueArray, PutValueArray, PutValue, CopyArray, GetLibraryOfType to the Property object.
- Added SendToDataTable, SetSubcase, and SetRevision to the Results Browsing object.
- Added AddSolidElementsAlongVector, AddVisible, AddConnectedElements, AddAllConnectedElements, AddConnectedFillets, AddTangentSurfacs, AddConstrained, AddLoaded, AddComponentOutputVectors, AddSimilarOutputVectors, AddComplexOutputVectors, RemoveRule, and RemoveSetRule to the Set object.
- Added GetSet to the Sort object.
- Added BoundingBoxInCSys, RemoveMeshPoint, and RemoveAllMeshPoints to the Surface object.

- Added SetEyeDirection, GetEyeDirection, ViewRotation, ModelRotation, CoordSysRotation, VectorRotation, and RollThru to the ViewOrient object.

New and Updated Global Variables

- Added Pref_UserToolsPath, Pref_NonmanifoldBooleans, Pref_RepeatCreate, Pref_ZoomAroundCursor, Pref_UseSubcaseIdForOutput, Pref_AutoConvertOdb, Pref_RenderFastPickVisible, Pref_FBDefContributions, vPref_FBDefContributions, Pref_FBDefDispMode, Pref_FBDefNodalVectorMode, vPref_FBDefNodalVectorMode, Pref_FBDefTotalVectorMode, vPref_FBDefTotalVectorMode, Pref_FBDefComponents, vPref_FBDefComponents, Pref_ElemQualQuadAR, Pref_ElemQualTriaAR, Pref_ElemQualTriaEPLR, Pref_ElemQualQuadARVal, Pref_ElemQualTriaARVal, Pref_ElemQualTriaEPLRVal, Pref_ResetNextID, Pref_SolNXNastProgram, Pref_SolMSCNastProgram, Pref_SolNEINastProgram, Pref_SolAnsysProgram, Pref_SolAbaqusProgram, Pref_SolLSDynaProgram, Pref_SolNXNastCmd, Pref_SolMSCNastCmd, Pref_SolNEINastCmd, Pref_SolAnsysCmd,Pref_SolAbaqusCmd, Pref_SolLSDynaCmd, Pref_SolEchoCmd, Pref_NasUesTrackRevsion, and Pref_RotateAroundCursor to set various preferences.
- Updated Pref_KeepSolverWindow, Pref_MinimizeDuringSolve, Pref_AnalysisProg, Pref_SolverMemory, Pref_ScrollBackLines, and Pref_WtmassFactor to set various preferences and Info_NextID and vInfo_NextID.

The following functions have been added or updated:
- feAppUndoCheckpoint
- feSurfaceGetUnderlying
- feGroupGenElemShape
- feEntitySetVisibility
- feEntityGetVisibility
- feVectorArrayTransform
- feSurfaceFromMesh2
- feSolidRemoveCurve
- feSurfaceOffset
- feCrossSectionReport
- feGroupGenVisible
- feCurveSolidToFEMAP
- feResultsToDataTable
- feResultsRankingToDataTable
- feDeleteOutputEntry
- feDeleteOutput2

The following functions have been removed:
- feFileReadIdeas

Corrections

Views

- Corrected issue where the View, Autoscale commands may not work correctly in models with a Freebody Entity set to Interface Load when no results were in the model. (PR# 5626239)

Analysis Manager

- Corrected issue where the Analysis Set entity name was improperly set to "Analysis Case". This caused the Analysis Set library file to fail to load from the Analysis Set Manager.

General

- Corrected issue which allowed elements with no properties to be selected when selecting properties or elements with no materials to be selected when selecting materials.
- Corrected issue in Smart Snap that prevented picking curve middle and center locations when a group was active.

Geometry

- Corrected issue that caused duplicate curves to be created by the Geometry, Copy/Scale/Reflect/ Rotate/Radial commands if you canceled in any dialog that was displayed after the Generation Options dialog.
- Corrected issue by changing the approach for finding intersections between spline curves which is used in various commands including Modify, Fillet. This solved issues where the curves were not quite intersecting because they were slightly "out-of-plane" (PR# 7489481).
- Corrected issue by preventing the Modify, Move To; Modify, Move By; Modify, Rotate To; Modify, Rotate By; Modify, Align; and Modify, Scale commands from moving/rotating boundary surfaces which have multiple underlying surfaces. Also updated Modify, Edit, Boundary and Modify, Update Other, Boundary On Surface to prevent editing of boundary surfaces which have multiple underlying surfaces.

Graphics

- Corrected issue where Curve based boundary surfaces were not drawn if filled edges were turned off.
- Corrected issue where certain geometry cleanup operations would cause the geometry from the solid disappear when the solid was in a group, but only a few of the curves and surfaces on that solid are in the group, until using the Windows, Regenerate command.
- Corrected issue where incorrect selection markers would be displayed when Solids were the active entity type in the Select Toolbar and using Pick Front.
- Corrected issue where the border colors for elements when displaying a Criteria Plot were drawn using the wrong color.
- Corrected issue when body loads were beinbg displayed on, where model would appear to be wireframe, which was being caused by the bounding box calculation of the varying acceleration being done incorrectly.

- Corrected issue where constraints displayed as "Pins" were not being drawn correctly in Standard graphics, but were being drawn correctly in Performance Graphics.
- Corrected issue where negative values of Nodal Temperatures, Elemental Heat Generation, Nodal Heat Generation, Nodal Pressure, Nodal Total Pressure, Nodal Heat Flux (per length and per area and at node) and Curve Element Pressure are not displayed correctly, but instead are displayed as absolute values.
- Corrected issues when displaying contour/criteria on elements which are being used by Connection regions.
- Corrected issue where beam element offsets, directions and y directions were drawn even if the element was not drawn in a Criteria plot because the element does not meet the specified criteria limits.

Performance Graphics

- Corrected issue where Element Criteria values were drawn at element centroid of Solid Elements if Fill was turned off. They now draw at the face centroids of solid elements.
- Corrected issue where Planar elements which had there results being transformed into a direction perpendicular to the element would be drawn as filled elements, even when Fill was turned off.
- Corrected issue where the points along a combined curve were still being drawn, which was incorrect, as only the points at the ends of the combined curve should have been drawn.
- Corrected issue where Offsets on Mass Elements where not being drawn in the local coordinate system specified on the Mass Property.
- Corrected issue where non-solid points, displayed as "+", would no longer have the proper shape and appear a "block" or "blob" when symbol size was small and the lines were thick.
- Corrected issue when using Show Selected Only from the Model Info tree, which would cause the whole model to remain displayed.
- Corrected issue to allow undeformed edges to follow filled edge color and not be the undeformed color when Fill is on.
- Corrected issue that caused CBUSH elements to initially be drawn with the wrong symbol after using Model Merge.

GUI - Dockable Panes

Model Info Tree

- Corrected issue that could cause a few entity types, such as Groups, to be displayed with the incorrect icons in the Model Info tree after being renumbered.
- Corrected issue by preventing the use double click in the Model Info tree to activate an entity while you are in another command. Previously, this was allowed, but could lead to model corruption.

Meshing Toolbox - Combined/Composite Curves tool

- Corrected issue when Action is set to Delete, which caused elements to become detached from the geometry and orphaned.

Meshing Toolbox - Feature Removal Tool
- Corrected issue where Feature Type is set to Blends and solids are selected using Box picking which would cause the program to exit unexpectedly.

Meshing Toolbox - Geometry Editing Tool
- Corrected issue where connected surfaces were not being remeshed when Operation was set to Edge to Edge and the Split at Closest option was enabled.
- Corrected issue when curves were projected where modified surfaces had all of their curves' mesh sizes recalculated. Now, only new and updated curves will have their mesh size updated.

PostProcessing Toolbox
- Corrected issue that occurred if you renumbered Output Sets, then immediately chose one of the Output Vector selection buttons. Previously the dialog box shown after pressing the button could have been improperly populated with vectors from the wrong Output Set or with no vectors at all, if the previous Output Set no longer existed. This condition corrected itself if other controls were used first, but now works if done immediately.

Charting Pane
- Corrected issue that could cause FEMAP to exit unexpectedly when using the mouse to pan in a zoomed-in chart and tooltips are enabled.
- Corrected issue when displaying max/min labels using only the X-axis label, only the maximum X value would be labeled.
- Corrected issue where pressing keys on the keyboard may not considered until the mouse is clicked somewhere in FEMAP, if FEMAP was started with the charting pane open.
- Corrected issue where listing Data Series values to the Messages window using the right mouse button would not list data correctly if output overrides were specified on the Chart level. Data copied to the clipboard was not affected.

Data Surface Editor
- Corrected issue where Plot Output Map in Data Surface Editor would show incorrect map and potentially corrupt the Data Map.

Entity Editor
- Corrected issue where elemental temperatures could not be edited in the Entity Editor if the elemental temperatures were not in the active Load Set (PR #7526412)

Interfaces - Nastran
- Corrected issue when indexing composite/laminate results that caused plys to be missing when attached to *.op2 file.
- Corrected issue when reading input files which are written in "free-field" and contain more than 10 fields on a single line (for instance, this can be done for RBE3 entries).
- Corrected issue where no error message was displayed when importing a NASTRAN input file which contained CBUSH elements that references a Coordinate System which does not exist.

During analysis import, error message will now be displayed in the message window, however the element will not be modified and will continue to reference the non-existing Coordinate System. This is beneficial if modifying an include file where the referenced Coordinate System is in another file. No existing Coordinate System will be auto-assigned, as it is impossible to determine the original intent. (PR# 7638131)

Interfaces - NX Nastran

- Corrected issue when reading results from NX Nastran Rotor Dynamics which were in the op2 file format used in NX Nastran 10.0 and above (PR #7458293).
- Corrected issue where contact results from ADINA were missing when attached to .op2 file and post processing anything other than the entire dataset (i.e., post-processing on a group).
- Corrected issue when writing and reading the material angle for the NX Nastran CPLSTNi and CPLSTSi elements, which were incorrectly defined using each element's Element Coordinate system, not the Basic Coordinate System.

Interfaces - ANSYS

- Corrected issue when writing orientation node for Beams and Curved Beams, which are exported as ANSYS BEAM44 (PR# 5932520)
- Corrected issue when writing Spring/Damper elements with the Property set to Other (NASTRAN CROD/CVIS).
- Corrected issue to allow the Default Temperature in the Create Body Loads dialog box is enabled, then it will be written as TUNIF, (Default Temperature Values) and the Reference Temperature on Materials to be written as MP, REFT, (Material ID), (Reference Temperature Value). If the Default Temperature in the Create Body Loads dialog box is not enabled, then TUNIF and MP,REFT will not be written at all.
- Corrected issue where elements were sometimes not written as SHELL181 elements to Nastran input file.

Interfaces - ABAQUS

- Corrected issue where reading values from *SPRING could cause the Stiffness value to be written to the Damping field in the DOF Spring Property.

Interfaces - I-DEAS

- Corrected issue by removing scaling of coordinate system origins when reading I-Deas Universal Files. Previously Coordinate System origins were scaled by a units factor from the file, but Nodal coordinates were not scaled, resulting in an inconsistent model.
- Corrected issue writing and reading nodal coordinates to the I-Deas Universal File when the nodes were defined in any coordinate system other than Global Rectangular

Interfaces - JT Files

- Corrected issue which caused text strings to not be properly converted to Unicode, which would impact an titles or text which were non-ASCII.

Interfaces - VRML files

- Corrected issue where contour colors for VRML files were inverted.

Loads and Boundary Conditions

- Corrected issue in Model, Load, From Output that caused rotational loads to not be created with a Load Definition. (PR #7577929)
- Corrected issue that occurred if you applied Pressure loads to the corners of planar elements using a data surface and chose to apply them to Face 2. Previously the values were computed properly, but were distributed to the wrong element corners.
- Corrected issue that occurred if you created a nodal constraint on a node that was already constrained. Previously the constraints were all handled properly but it left behind a Constraint Definition that did not contain any Constraints. That is now properly deleted.

Meshing

- Corrected issue in when using Mesh, Mesh Control, Size on Surface or Mesh, Mesh Control, Size on Solid, which occurred if you had arc edges that were very near 360 degrees and would have gotten a single element along their length based purely on the mesh size, but which should have been resized based on an Angle Tolerance. In extreme cases, the angle tolerance was missed resulting in a single element along the almost fully circular edge.
- Corrected issues in the Fast Tri mesher that caused the mesher to not be successful in certain cases.
- Corrected issue that could cause some surfaces with Mesh Points on the Surface to mesh with some nodes significantly off of the surface.

Mesh Associativity

- Corrected issue where some elements could lose associativity if the nodes of the element were associated with two surfaces which were meshed using adjacent surface matching.

Elements - Planar Elements

- Corrected issue where shell elements referencing a material coordinate system would not increment the coordinate system counter, which allowed the coordinate system to be deleted even if it was referenced as an MCID on an element.

Elements - Spring/Damper

- Corrected issue where Spring/Damper elements referencing an element coordinate system would not correctly update Coordinate System counters upon a database rebuild

Elements - Rigid

- Corrected issue when editing rigid elements, where the Title of the entity selection dialog box when selecting nodes may have incorrectly specified if Master or Slave nodes were being selected. The dialog box titles have been corrected and also now reflect the Dependent/Independent nomenclature used in the rigid element dialog box.

Element Update

- Corrected issue where Modify, Update Elements, Line Element Orientation command would not correctly update CBUSH orientation if the selected elements were originally oriented using a CID on the element, then reoriented with a different method (node, vector, radial, etc) (PR# 7567417).
- Corrected issue where no error message was written to the Messages window if reorienting line elements using the radial option failed.

Properties

- Corrected issue where meshing within the beam section calculator would fail, resulting in no section properties being calculated. If this happens now, the mesh size is divided by 3 and an additional attempt is made to mesh the cross-section.

Output and Post-Processing

- Corrected issue where the View, Advanced Post, Beam Cross Section command would not work on Bar elements because the results on those Bar elements are actually beam results. This can happen when solving in Nastran.
- Corrected issue where MultiSet Animation and Trace plots were not being displayed correctly when % of Model (Actual) was enabled for Deformed Style.
- Corrected issue where results from Nastran on Solid Laminate elements were not being transformed properly due to an incorrect conversion from 32-bit to 64-bit.
- Corrected issue when displaying a Contour Arrow Plot of results on the Top of Laminate Elements when the results are not ply-by-ply results.
- Corrected issue that occurred if you contoured multiple output vectors, the first output vector had double-sided turned on and had corner results, while at the same time the second or third vectors did not exist in the Output Set being contoured.

Groups and Layers

- Corrected issue in Group, Clipping, Coordinate. Previously when you specified a clipping coordinate system it was properly defined, however if you went back to the command that coordinate system was not reloaded into the dialog box, so you had to choose it again. It is now reloaded properly.
- Corrected issue with Group, Operations, Evaluate Always command where if no groups are selected, the dialog box would not save the current state.
- Corrected issue when using Group, Operations, Add Related on a group containing elements referencing a material coordinate system, which would not add the material coordinate system. Now the material coordinate system will also be added.

Tools

- Corrected issue that occurred when using Tools, Mass Properties, Mesh Properties when computing the mass properties of tapered plate elements. Previously the area was not calculated (listed as 0.0) which then caused the mass to be incorrect if the element/property had nonstructural mass applied.

- Corrected issue that occurred when using Tools, Merge, Coincident Nodes command if Performance Graphics was enabled and multiple nodes were merged on the same element. Previously elements could be corrupted. (PR #7508355)
- Corrected issue which cause the Tools, Measure, Angle Between Nodes command to fail to list the deformed angle if the selected nodes were colinear in their undeformed locations. Also, increased the precision of the various angle measuring commands for angles close to 0.0 and 180.0 degrees. Previously listed as 0 or 180 if within 0.0081 degrees, now only when within 0.00000256 degrees.(PR# 7672168)

Model Merge

- Corrected issue in Model Merge that caused Spring/Damper elements (CBUSH) that were oriented using a coordinate system to transfer improperly without renumbering the coordinate system.

User Interface

- Corrected issue which would occur when using a user defined contour palette to have the number of colors in the user contour palette set the number of levels. Previously, this was not displayed correctly on the user defined levels dialog.

API

- Corrected issue where the range for Pref_ScrollBackLines was defined improperly
- Corrected issue in feFileAttachSave2 that caused the results sets to not be detached when the bDetach argument was set to True.
- Corrected issue in feGetElementFaces that could cause part of the list of selected faces to be lost if you used the "Multiple..." option more than once.
- Corrected issue in Output Object that prevented creation of integer output vectors if you attempted to create the vector by only setting the various parameters and did not use the InitScalarAtNode, InitVectorAtNode, InitScalarAtElem, InitElemWithCorner, or InitScalarAtBeam method.
- Corrected issue where GetElements and GetNodes methods may return too many nodes or elements when walking through the freebody entities in a loop.
- Corrected issue where feCheckCoincidentElem were not using the same settings as the commands in the GUI when doing all checks on all shapes. Now, this API function uses the same settings.

FEMAP v11.2.2 New Features and Corrections
Updates and Enhancements
Connection Properties, Regions, and Connectors
- Added the ability to copy Connections when using the Mesh, Copy...or Mesh, Rotate commands when the "Mesh Sizes, Loads, Constraints..." option is enabled and if both associated Connection Regions are also copied

Interfaces - LS-DYNA
- Added "Max Matrix Strn DFAILM" and "Max T Shear Strn DFAILS" fields to Material Type "54..LS-DYNA Enhanced Composite Damage", which write the DFAILM and DFAILS values for the *MAT_ENHANCED_COMPOSITE_DAMAGE entry.
- Added "R Rot Constrained (1=Yes)", "S Rot Constrained (1=Yes)", "T Rot Constrained (1=Yes)", "CST (0,1 or 2)", and "SCOOR (-3 to 3)" fields to Material Type "71..LS-DYNA Cable Discrete Beam", which write the RRCON, SRCON, TRCON, CST, and SCOOR values to the *MAT_CABLE_DISCRETE_BEAM entry.
- Added "CST (0,1 or 2)", "SCOOR (-3 to 3)", "R Rot Constrained (1=Yes)", "S Rot Constrained (1=Yes)", and "T Rot Constrained (1=Yes)" fields to Material Type "93..LS-DYNA Elastic 6DOf Spring Discrete Beam", which write the CST, SCOOR, RRCON, SRCON, and TRCON values to the *MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM entry.

API
- Updated feSolidRemoveRedundantPoint to make it more useful when cleaning up imported CAD geometry where vertices that simply should not be relevant can now be removed
- Added support for Boundary Surfaces in the API Entity Tracking object.

Corrections
Analysis Manager
- Corrected an issue with regard to Load and Constraint Sets in Subcases after Groups were renumbered. Previously, if any Load Sets or Constraint Sets selected in Subcases had the same ID as one of the Groups that was being renumbered, those Load/Constraint Set references were renumbered.
- Corrected an issue with regard to Groups used to limit entities when making output requests after Groups were renumbered. Previously, references to groups for any Output Selections in the Analysis Manager Subcases were not renumbered.
- Corrected an issue with regard to the Contact Group in the Boundary Condition dialog box after Groups were renumbered. Previously, the Contact Group reference in Analysis Manger Master Case or Subcases was not renumbered.
- Corrected an issue with regard to renumbering Load Sets. Previously, if the renumbered Load Sets were referenced in one or more Analysis Sets as the Temperature Set selection in either the Master Case or Subcases, or as the Bolt Preload Set in Subcases, the reference still contained the original Load Set ID.

Performance Graphics

- Corrected an issue where nodal and elemental thermal loads would cause FEMAP to become unresponsive.
- Corrected an issue where function IDs for loads were not drawn correctly if in component display mode.
- Corrected an issue when curved beams drawn with large radii were not drawn properly.
- Corrected an issue where nodal loads being displayed as components were not being shown in the proper orientation.

Connection Properties, Regions, and Connectors

- Corrected an issue in generation of Automatic Connections that in a few cases caused coincident faces to be missed when using detection strategies that were more aggressive than the defaults.

GUI - General

- Corrected an issue which caused "Smart Snap" snap mode to not allow selection of the midpoint of a curve or center point of a curve when the graphics window was displaying a single group or multiple groups.

GUI - Dockable Panes

Meshing Toolbox - Mesh Surface Tool

- Corrected an issue in the that caused the Mesh Surface tool to resize and remesh surfaces that had not been selected.

Model Info Tree

- Corrected an issue that caused the Renumber command on the context menu of the Model Info tree to fail for Load Definitions and Constraint Definitions unless the definitions were in Load/ Constraint Set 1.

Entity Info

- Corrected an issue where a \ character at the end of an Entity title or a { character or } character anywhere in an Entity title sent to the Entity Info window for any reason (typically during entity selection) could cause FEMAP to become unresponsive.

Interfaces - NX Nastran

- Corrected an issue where PARAM,WMODAL,YES would not be written for Nastran Response Spectrum Analysis (PR 8264897).
- Corrected an issue when reading results for TRIA6 and QUAD8 elements in nonlinear analysis (SOL 601) that caused some Strain Invariant results to not be computed or calculated incorrectly.

Interfaces - Nastran

- Corrected an issue in XDB translator that could cause some Grid Point Force vectors to be skipped.
- Corrected an issue in XDB translator that could cause Grid Point Force vectors to be incorrectly transformed when a local nodal output coordinate system was used.

Interfaces - ANSYS

- Corrected an issue which caused linear or parabolic tetrahedral elements to appear as linear or parabolic pyramid elements when reading elements from an ANSYS input file (*.cdb or *.ans). (PR 7422995)
- Corrected an issue which caused plane strain elements and axisymmetric elements to potentially be assigned a Material Orientation during import of an ANSYS input file.

Interfaces - LS-DYNA

- Corrected an issue where import of d3plot files for Japanese FEMAP was very slow. (PR 7405411)

Elements

- Corrected an issue when using the Modify, Update Elements, Type command to change the "type" of selected elements from Plot Only Planar elements to Laminate Elements, which could cause FEMAP to exit unexpectedly.
- Corrected an issue which caused CBUSH elements created in earlier versions of FEMAP, which had both an orientation Vector specified and referenced a Property which had an Orientation Csys defined, to have their "Orientation" set to "Vector" instead of "From Property" when brought forward into FEMAP 11.2 or 11.2.1.
- Corrected an issue which caused CBUSH elements created in earlier versions of FEMAP, which had both a Node ID specified for orientation and referenced a Property which had an Orientation Csys defined, which would cause "Orientation" set to "Node" instead of "From Property" when brought forward into FEMAP 11.2 or 11.2.1.

Loads and Boundary Conditions

- Corrected an issue which caused Distributed Loads on line elements to not be visible in the graphics window when a Nastran LOAD Combination Load Set was either the Active Load Set or the Load Set Selected for display.
- Corrected an issue when renumbering either Load Definitions or Constraint Definitions. Previously the definition IDs on the Loads/Constraints in the active Load/Constraint Set, not the selected Load/Constraint Set, were renumbered.

Geometry

- Corrected an issue where meshed surface(s) that were being extended were not being remeshed because the extended surfaces where being renumbered, thus not getting remeshed. A different approach is now being used which does not renumber the extended surface(s).

Output and Post-Processing

- Corrected an issue with transforming results that occurred if you attempted to transform into a coordinate system that did not exist. Previously this could happen if you renumbered coordinate systems. Now the coordinate system reference is properly renumbered and transforms are turned off if you specify a nonexistent system any other way. (PR 7455476)

Tools

- Corrected an issue that occurred if you used the Tools, Mass Properties, Mesh Properties command and selected a coordinate system for the mass properties other than Global Rectangular and chose to create a node at the CG. Previously the CG was created at an incorrect location. (PR 7457501)

API

- Corrected an issue which caused the return code for NextEmptyID, PrevEmptyID, NextID, and PrevID to be zReturnCode instead of "INT4" which matches the actual method return.
- Corrected an issue in feSolidFillet and feSolidChamfer that caused both of these methods to fail.
- Corrected an issue where FTO_PYRAMID13 enum was missing for the topology property of the Element Object.

FEMAP v11.2.1 New Features and Corrections
Updates and Enhancements
GUI - General
- Added dynamic face highlighting during Element Face picking and enabled dynamic highlighting of elements in the Connection Region dialog if "Elements-No Faces" is selected
- Added "Plane Element Offset" as an option when using the "Model Data Value" option for "Pick^" in the standard entity selection dialog box
- Added a Cancel button to the "OK to Start New Model (No=Add to Current Model)?" dialog box which appears when you "drag and drop" an analysis model or geometry file into an open instance of FEMAP
- Updated the "Add Connected Elements" option for "Pick^" in the standard entity selection dialog box to dramatically improve performance

Interfaces - NX Nastran
- FEMAP with NX Nastran bundle now includes NX Nastran 10.2
- Added support for NX Nastran 10.2 Thermal and Mechanical Strain (op2 data blocks OSTR1ELC, OSTR1THC, OSTR1EL, OSTR1TH)
- Updated default value for the "Solid Results in Material CSys" option in the "NXSTRAT Solver Parameters" dialog box in the Analysis Set Manager to be Off instead of On, which will write a 0 to the ELRESCS field on the NXSTRAT entry instead of a 1, which is the default value for NX Nastran

Interfaces - LS-Dyna
- Added "20..EQ 13: 1 point nodal pressure tetrahedron" as a formulation option for Solid elements to create ELFO=13
- Updated "Memory (Megawords)" field in the LS-DYNA Analysis Control dialog box, which writes the *KEYWORD MEMORY entry, to allow values up to 99,999,999, as the previous limit was 2,147

Interfaces - Geometry
- Enhanced support for importing geometry files from SolidWorks 2015 by implementing a newer version of the SolidWorks translator

Aeroelasticity
- Added support for randomizing colors of the various Aero entities when modifying the color

Meshing
- Enhanced performance significantly when using the Mesh, Copy, Element; Mesh, Radial Copy, Element; Mesh, Scale, Element; Mesh, Rotate, Element; and Mesh, Reflect, Element commands to create copies of existing elements.

- Enhanced performance significantly of the Mesh, Extrude commands to extrude elements, the Mesh, Revolve commands to revolve elements, and the Mesh, Sweep commands to sweep elements.

Output and Post-Processing

- Updated "View, Advanced Post, Contour Model Data" command to allow display of "Plane Element Offset" on planar elements

API

- Added feAppSetModel32 and feAppGetModel32 methods to supplement the existing feAppSetModel and feAppGetModel methods for programming environments where 64-bit integers are not available
- Added NumberOfLoads and NumberOfConstraints properties to the API Node object
- Added GetAllArray2 and GetCoordArray2 methods to Node Object that returns an extra Variant that indicates which nodes actually exist
- Added GetAllArray2 method to Element Object that returns an extra Variant that indicates which elements actually exist
- Updated the GetAllArray and GetCoordArray methods for the Node object to automatically zero all memory, so if you ask for entities that do not exist the values returned will all be zeroed
- Updated the GetAllArray method for the Element object to automatically zero all memory, so if you ask for entities that do not exist the values returned will all be zeroed

Corrections

Analysis Manager

- Corrected issue when copying Analysis Sets that caused set options to become corrupted (for example, text in the "Direct Output To" field and various "Start Text" and "End Text" dialog boxes would be missing or wrong)
- Corrected issue that caused the MultiSet button in the Analysis Set Manager to not work if you had only Load Sets or Constraint Sets, not both Load Sets and Constraint Sets

Element - CBUSH

- Corrected a number of issues related to element orientation which could occur during creation of the element in FEMAP, importing from a FEMAP neutral file, or importing from a Nastran input file. For more information, please refer to Software Field Bulletin SFB-FEMAP-9054 for more information

Performance Graphics

- Corrected issue where individual Coordinate Systems did not obey the Visibility Check Boxes when Performance Graphics is enabled
- Corrected issue where Coordinate systems were ignored when calculating the bounding box when Performance Graphics is enabled

- Corrected issue where element coordinate system labels were not drawn for solid elements if there were also shell elements in the in model when Performance Graphics is enabled
- Corrected issue where scaling factors were not being generated for geometry loads when Performance Graphics is enabled
- Corrected issue where rounding negative values to specified number of significant figures was not correct when Performance Graphics is enabled, which effected both criteria values and load values
- Corrected issue where beam direction arrows are not drawn when "Normal Style" is set to "2..Backface Shading" for the "Element - Directions" option in the "View, Options" command, when Performance Graphics is enabled

Graphics

- Corrected issue which caused FEMAP to exit unexpectedly when drawing CBUSH elements when the "Show Y Axis" option was enabled for "Element - Beam Y-Axis" option in "View, Options" command
- Corrected issue which caused no arrow representing the element direction to be displayed on solid elements when "Show Direction" is turned on and "Normal Style" set "0..Right-Hand Rule" or "3..Right-Hand Rule First Edge" for the "Element - Directions" option in "View, Options" command and the View Style was set to Solid

GUI - General

- Corrected issue in the Results selection dialog when you attempted to Filter Sets, Studies or Vectors by title. If any of the entities had the string ".." in the title, those entities could only be filtered on text that came after the ".."
- Corrected issue that prevented box/area picking Solids that contained combined curves if you also had the "Pick All Inside" option turned on
- Corrected issue when using the "Model Data Value" option for "Pick^" in the standard entity selection dialog box. Previously the "Plane Element Thickness" Model Data Value selection only worked for thicknesses specified on the Property. Now it also uses the element thickness overrides if selecting elements, while only using property values if selecting properties or other entity types.
- Corrected issue that caused Alt-Pick (automatic Query Selection) to fail if "Smart Snap" was turned on (PR 7382986)

GUI - Toolbars and Icons

Select Toolbar

- Corrected issue that allowed entities to be selected multiple times if you were using the Selector on Multiple. Previously this was correctly prevented

GUI - Dockable Panes

Meshing Toolbox - Geometry Editing Tool

- Corrected issue with the "Project/Move Point" Operation where the "Edge Aligned" option was being considered when "Project/Move Point" was set to "Point", even though that option is not available when set to "Point", which was causing the operation to fail

Data Surface Editor

- Corrected issue that caused a database corruption if you modified a data surface, closed the data surface window then did an undo
- Corrected issue where FEMAP would exit unexpectedly or become unresponsive when trying to create an "Output Map Data Surface" from output on 20-noded hex elements (PR 7330949)
- Corrected issue where the ID field in the Define Connection Manager Data Surface dialog box would be ignored when creating a new Connection Manager Data Surface. Additionally, the ID field is now grayed out when modifying an existing Connection Manager Data Surface.

Data Table

- Corrected issue with "Copy to Clipboard", "Copy Rows", and "Copy Columns". Previously there was an extra trailing tab (column) and trailing carriage return (row). This caused the selection in Excel to be wrong when you pasted in the results

PostProcessing Toolbox

- Corrected issue in Freebody tool where the "Display Freebodies" check box did not change when the "Set the Freebody Style" icon menu was used to change the display state of Freebodies (PR 7367712)

Interfaces - FEMAP Neutral

- Corrected issue in the Neutral File Converters that prevented models created in FEMAP Educational Licenses from being moved forward to later versions of FEMAP, even if you still had an educational license
- Corrected issue with neutral file export that could potentially cause FEMAP to exit unexpectedly when re-importing v11.2 neutral files into FEMAP (PR 7390961)

Interfaces - NX Nastran

- Corrected issue when reading input files containing entities for both linear contact and/or glued contact which would cause only a single Connector to be imported instead of the actual number of Connectors in the input file.

Interfaces - Nastran

- Corrected issue when reading variable length cards with embedded blank fields such as SPC1, which is allowed in Nastran but not typical
- Corrected issue when reading NASTRAN INCLUDE statements if they followed an "open-ended" card, like RBE2 or RBE3
- Corrected issue that caused NASTRAN INCLUDE files to not be read if there were same-line $ comments on the include line
- Corrected issue when reading or attaching to Nastran OP2 and/or XDB files that caused some vectors to be improperly labeled "Nonlinear"

- Corrected issue where composite element failure index vectors were missing when attaching to op2 files (PR 7372455)

Interfaces - MSC Nastran

- Corrected issue that caused *.f06 files from MSC Nastran version 2013 and above to not be imported properly

Interfaces - LS-DYNA

- Corrected issue where translator was writing out comments that exceed the standard 80 char resulting in the solver not being able to properly process the input file (PR 7383233)
- Corrected issue where "Translation Text" specified via the "File, Notes" command was being ignored during export
- Corrected issue where only 2 of the required 7 fields on the first line of the *SECTION_BEAM entry would be written when beam element formulation was set to "9..Spotweld" (PR 7365784)

Loads and Boundary Conditions

- Corrected issue that caused load values to be zeroed if you edited a load definition using the "Edit Load" command and chose to update the faces where the load was applied (or the direction of a Distributed Load). Previously this would cause the load values to be zeroed on all elements other than the first where the load was applied. (PR 7370349)
- Corrected issue that occurred when you updated where geometric constraints were applied. If you removed one or more entities (Points, Curves, Surfaces) linkages in the database were improperly maintained making the entity non-deletable prior to a database rebuild.
- Corrected issue where FEMAP would exit unexpectedly or become unresponsive when trying to use "Model, Load, Map Output From Model" command to map output found on 20-noded hex elements in the "From Model" (PR 7330949)

Geometry

- Corrected issue in display of geometry that caused Solids with large numbers of curves to create the data for display more slowly
- Corrected issue that occurred when a solid got split into multiple components as you deleted surfaces. (PR 7356203, PR 8267969 and PR 8268004)
- Corrected issue with Geometry, Solid, Explode. If you selected multiple solids to explode, each solid lists the number of "exploded faces" that were created but that number was previously a running total of all the exploded faces, not just the ones from each original solid. (PR 7384480)
- Corrected issue which occurred if you tried to break or extend a curve that was non-deletable (meshed, used in connection regions or had applied loads or constraints). Previously, a message was issued that said the curve was skipped, but the reason given was incorrect and sometimes misleading.
- Corrected issue when extending surfaces where the vertex cleanup during the action was creating an invalid solid

- Corrected issue when using the "Geometry, Solid, Stitch" command, or when stitching would occur during a command, where certain bodies would get deleted when the stitching operation failed, but Parasolid said it completed properly
- Corrected issue when undoing, then redoing the "Geometry, Surface, From Mesh" command which could cause the surface to be faceted incorrectly
- Corrected issue when using "Geometry, Surface, From Mesh" command where ruled surface meshes were being missed if they were not the first element batch selected
- Corrected issue when using any command where a curve on a surface could be removed during the command and more than the selected or needed curves would also be removed, causing unexpected results
- Corrected issue when using the Geometry, Scale commands which would cause an unexpected scale factor to be applied to repetitions 2 to "n" when using a value for "Repetitions" other than 1

Meshing
- Corrected issue when using the Mesh, Scale commands which would cause an unexpected scale factor to be applied to repetitions 2 to "n" when using a value for "Repetitions" other than 1

Groups and Layers
- Corrected issue that caused "Group, Operations, Generate with Output" command to fail if you attempted to use results that contained integer values rather than real data.
- Corrected issue when using "Group, Operations, Add Related Entities" command for Groups. Previously connection regions were not added if they referenced geometry but they were added if they referenced nodes or elements. Now they are added in either case.

Output and Post-Processing
- Corrected issue that caused FEMAP to exit unexpectedly when attaching to *.csv files that are not properly formatted
- Corrected issue which caused nothing to be listed when using the "List, Output, Force Balance" command and selecting "No" to "Retrieve nodes from freebody object(s)?" question.
- Corrected issue where some composite element failure index vectors were missing when attaching to op2 files (PR 7372455)
- Corrected issue that caused "Model, Output, Extrapolate" command to fail if you attempted to use results that contained integer values rather than real data
- Corrected issue that caused Linear Combination or RSS combinations results created with "Model, Output, Process" to not be able to be transformed for display (PR 7366572)
- Corrected issue where rod elements with no results are labeled with 0.0 in a criteria display
- Corrected issue that caused some contour vectors to not be drawn if they represented nearly a "no-shear" condition and there was a small nonzero value (on the order of 1.0E-14) in one of the shear values. This condition could also occur when migrating a model where vectors were properly displayed in an existing model then would not be displayed properly after exporting a FEMAP Neutral file and then importing the Neutral file into the same or newer version of FEMAP.

Preferences

- Corrected issue when reloading the value for the "Non-FEMAP Neutral File Version" preference if it was set to a version other than the current version

API

- Corrected issue that caused the Next method of the Results Browser Object to fail if you attempted to use results that contained integer values rather than real data
- Corrected issue with feSurfaceRayFire that caused all surfaces that were hit to be returned with a surface ID of -1

FEMAP v11.2 New Features and Corrections
Updates and Enhancements

Views

- Updated the View, Visibility command by adding a Hide... button to the Coord Sys, Geometry, Material, Property, Aero Panel/Body, and Aero Spline/Control Surface tabs, which allows you to select entities in the graphics window to "hide" (i.e., turn off visibility) using the standard entity selection dialog box for that entity type.
- Updated "Element - Coordinate System" option in "Labels, Entities and Color" Category of View, Options command to with number of different element types. Also, added "2..RGB Line" to Color Mode.

Analysis Manager

- Updated Select Load and Constraint Sets for Analysis Cases dialog box accessed via the MultiSet button to place the Clear Master Case Constraints option beneath the list in the Constraint Sets section and the Clear Master Case Loads options beneath the list in the Load Sets section.

Connection Properties, Regions, and Connectors

- Added Elements - No Faces to the Output section of the Connection Region dialog box to support contact for MSC Nastran, which does not require elements faces. Also, added MSC Nastran section with Friction option to the Connection Region Options dialog box accessed by the Region Options button.
- Added MSC Nastran to Define Connection Property dialog box.
- Added Penalty Autoscale option to the Common Contact (BCTPARM) and Glue (BGPARM) Parameters section of the NX Linear tab of the Define Connection Property dialog box, which creates the AUTOSCAL field on the BCTPARM entry and scales the automatically calculated penalty factors PENN and PENT either up or down and can be used to scale the stiffness of specific contact pairs if convergence issues occur (Default = 1.0).
- Removed Avg Method, Adaptive Stiffness, Penetration Factor, and Min Contact Percentage options from the Common Contact (BCTPARM) and Glue (BGPARM) Parameters section of the NX Linear tab of the Define Connection Property dialog box, as these options are no longer documented in the NX Nastran Quick Reference Guide and should no longer be used.
- Added Connection Manager Data Surface to the Data Surface Editor, which allows you to manage connections in your model using an interactive "contact table". All or any number of selected connection regions can be added to the "contact table", while at the same time any existing connectors referenced by the connection regions can also be added. Once in the data surface, editing can take place.

Geometry

- Added Geometry, Surface, Extract command, which extracts the selected surface(s) from any solid, sheet solid, or general body in which the surface(s) currently reside. When removing surface(s) from a solid, the remaining surfaces of the solid will be changed from solid into a sheet solid or

general body, whichever is more appropriate. If multiple connected surfaces are selected for extraction, then the extracted surfaces will be joined into a sheet solid or general body, whichever is needed.

- Added ability to use Delete, Geometry, Surface to delete individual surfaces from solids. Previously, you would have needed to use Geometry, Solid, Explode before these surfaces could be deleted.

- Added Express option to Geometry, Surface, NonManifold Add command. The Express option in Processing Method attempts to use a single operation in Parasolid to create a "General Body" from the selected surfaces, therefore performance can potentially be greatly improved.

- Added Variable Thickness Processing option to Geometry, Midsurface, Automatic command.

- Added Geometry, Solid, Extend command, which allows you to extend a surface on a solid, sheet solid, or general body to a surface on another solid, sheet solid, or general body.

- For Solids, added Layer, Property, Type (choose from Solid, Sheet, Wire, or General/ NonManifold), Using Curve, and Using Surface options to the Method^ menu in the standard entity selection dialog box.

- Updated the Geometry, Solid, Slice command to be a single command which consolidated the functionality from Geometry, Solid, Slice; Geometry, Solid, Slice Match; and Geometry, Solid, Slice Along Face commands.

GUI - Toolbars and Icons

Solids Toolbar

- Added Solid Slice with Plane icon. Also, added Geometry Solid Slice Match and Geometry Solid Slice Along Face commands to the Additional Commands section on the Commands tab of the Customize dialog box. These commands can be added to any toolbar or menu and provide access to the legacy Geometry, Solid, Slice Match and Geometry, Solid, Slice Along Face commands, respectively.

Select Toolbar

- Added Smart Snap as an option for the Snap Mode icon. Also, added Hide Surfaces and Show Surfaces commands to the Visibility submenu on the context-sensitive menu for Solids. Also, when using the Dialog command in the Selector Actions menu, the various picking methods available in the Method^ menu are now available when a Load or Constraint entity type is set as the active entity type.

Post Toolbar

- Updated Post Data icon

GUI - Dockable Panes

Model Info Tree

- Added Hide Surfaces and Show Surfaces commands to the "visibility check box" context-sensitive menu for Geometry. Hide Surfaces allows you to turn off visibility of the surfaces on the highlighted solids, sheet solids, or general bodies, while Show Surfaces will display any hidden

surfaces again. If a body's overall visibility is turned off, then the body is made visible again, the surfaces on the body will always also be visible again.

- Added Analysis Study and All Results branches under Results. The Analysis Study branch will contain all Analysis Studies currently in the model and any output set(s) currently residing in an analysis study will be listed under the appropriate Analysis Study. The All Results branch simply lists all output sets currently in the model.

- Added context-sensitive menu unique to the top-level Results branch. It contains Attach to Results (displays the Manage Results Files dialog box from the File, Attach to Results command), Delete (does the same thing as Delete, Output, All command), No Deformation (sets Deformed Style to None - Model Only), and No Contour (sets Contour Style to None - Model Only) commands.

- Added context-sensitive menu for Analysis Studies. It contains Manage (opens the Analysis Study Manager, same as Model, Output, Create/Manage Analysis Study command), Copy, Edit, List, Renumber (does the selected operation to any number of selected Analysis Studies), Delete (asks if you want to delete the output set(s) in the selected Analysis Studies along with the Analysis Studies), Remove Study (Deletes the selected Analysis Studies, but not the output set(s) in the Analysis Studies), Animate Study (sets Deformed Style to Animate-MultiSet and uses all output sets in the Analysis Study), and the Envelope Study menu (creates a Max Value, Min Value, or Max Absolute Value envelope of all output vectors in all output sets in the Analysis Study and also includes a Create in Database toggle).

- Added Add to New Study, Add To Study, and Remove From Study to context-sensitive menu for Output Sets. Also, added Create in Database toggle to the Envelope submenu on the context-sensitive menu for Output Sets.

- Added ability to move the vertical scroll bar up and down when a command dialog box is also open.

Meshing Toolbox - Feature Removal Tool

- Added Blends to the Feature Type section. When used, prompts you to select an entire solid, sheet solid, or general body, instead of selecting individual blend (fillet) surfaces, then attempts to remove all blends. Limit Size restricts removal to blends with a radius less than or equal to specified value.

Meshing Toolbox - Feature Editing Tool

- Added Edit to the Operation section. When used, this operation assigns a different size for cylindrical and spherical surfaces on a solid, sheet solid, or general body and is only available when Selection Method is set to Surface.

- Added Offset to the Operation section. When used, this operation offsets surfaces on a solid, sheet solid, or general body and is only available when Selection Method is set to Surface.

Meshing Toolbox - Geometry Editing Tool

- Added Project/Move Point to the Operation section. When using Project/Move To set to Solid, Surface, or Curve, projects a point or points used by a surface, solid, sheet solid, or general body onto an entity of the selected type, thus modifying the original body. Turn on Edge Aligned to have the command attempt to follow the curvature of the edge connected to the point instead of simply projecting the point to the closest location on the target body.

- Added Project Curve to the Operation section.When used, creates a new curve or curves on the selected surface using a normal projection.
- Operation set to Pad or Washer, added ability to select either Factor or Distance in the Offset Type section to set the size of the "Pad" or "Washer".
- Operation set to Extend, added Surface and Surface Auto Curve options to Extend To drop-down. Both Surface and Surface Auto Curve have two modes for selecting surfaces.

PostProcessing Toolbox - General
- Added Select Post Data icon to the toolbar, which displays the Select PostProcessing Data dialog box from the View, Select command.

PostProcessing Toolbox - Contour and Deform Tools
- Added Complex Results Mode option to both the Deform and Contour tools, which allows you to set overall "Model Option(s)" or view-specific "override" values used to convert complex data to real data, on-the-fly, when post-processing complex results. See Output and Post-Processing for more information. Also, The Complex Model Options icon button can be used to set the "Model Option(s)" and the Animation Phase Increment from the active view will be used for Synchronize Phase, when Style in Deform tool is set to Animate.

PostProcessing Toolbox - Freebody Tool
- Added Section Cut as a Display Mode option for the Freebody Tool. Also, added a number of options which are used to control Freebody display when using a Section Cut Freebody. See Output and Post-Processing for more information.
- Added Reverse Values option under Nodal Summation in the Freebody Contributions section, which allows you to treat a solver-calculated nodal imbalance as a contribution. This can be useful when certain forces on a node, such as glue or contact forces, are not included in the grid point force table and result in an imbalance.
- For Total Summation Vector and Nodal Vector(s), added icon buttons to toggle on/off display of "Forces and Moments", "Forces Only", or "Moments Only". Also, added an icon button to toggle between displaying "Component" vectors or a "Resultant" vector, along with an icon button to specify the location of the Total Summation Vector.

Entity Editor
- Added Num Mesh Seeds in the Mesh Attributes section for Curves, showing the number of "mesh seeds" (Number of Elements) on the curve currently in the Entity Editor.
- Added Orientation CSys for Spring/Damper elements, showing the Orientation Coordinate System for any Spring/Damper element referencing a CBUSH property currently in the Entity Editor.
- Added the MSC Nastran Properties section for Connection Properties, showing values on the MSC Nastran tab of the Define Connection Property dialog box for the Connection Property currently in the Entity Editor.

Charting Pane
- Added Relative to Node/Relative to Element option to the Vector vs. Entity tab of the Chart Data Series dialog box. Only available when X-Axis Values is set to Position. When enabled, values be

calculated relative to the X, Y, or Z location of the specified node for a nodal output vector or centroid of the specified element for an elemental output vector.

- Added ability to specify a Color for the "Major Grid Lines". Also, added the Dim for Dark Backgrounds option, which selects a complementary color for the "Major Grid Lines" when using a darker background.

Data Surface Editor
- Added Result Set Processing Data Surface to Create/Load Data Surface icon menu. See Output and Post-Processing for more information.
- Added Connection Manager Data Surface to Create/Load Data Surface icon menu. See Connections (Region, Properties, and Connectors) for more information.

Entity Info
- Added Orientation for a Spring/Damper element which reference CBUSH Property. Depending on the method used for orientation, it will state "Node" with a node ID, "Vector" with XYZ coordinates, or "Coordinate System" with an Coordinate System ID. This information will also appear in the Tooltip for the element.

Data Table
- Added Copy Rows - No Headers command on context-sensitive for rows, which copies the content of the selected rows without including any of the column header information.
- Added Add Nastran Element Checks command to the Show/Hide Columns icon menu, which adds a column for each NX Nastran Element Quality Check available in FEMAP for each element currently in the Data Table.
- Added Orientation CSys and CBUSH Property Orientation columns for Spring/Damper elements being sent to the Data Table.
- Added Num Mesh Seeds column for Curves being sent to the Data Table.

API Programming
- Added Save icon to toolbar to allow you to simply save the file instead of always displaying a dialog box to perform a "save as" operation. If file has not been saved before a dialog box will appear to enter a file name.

Interfaces - FEMAP Neutral
- Updated Neutral Read and Write for v11.2 changes

Interfaces - Nastran
- Added support for reading and writing of the ACCEL1 entry. Also, ACCEL1 entries can be created by applying an acceleration load to a node or geometry, then exporting a Static analysis.
- Added support for reading both Linear and Nonlinear results from a nonlinear analysis into the same output set. Previously, you would have to choose one or the other, but now both are read in during a single import or attach.

- Added support for reading and writing of element corner thickness (TFLAG, T1, T2, T3, and T4 fields) for the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR, CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 elements. Element thickness can be updated via the Modify, Update Elements, Adjust Plate Thickness/Offset and Modify, Update Elements, Midsurface Thickness and Offset commands.
- Added SECOMB option to NASTRAN Bulk Data Options dialog box. When enabled, used to control if output will be combined for a superelement analysis, has a number of caveats.
- Added ALPHA1 and ALPHA2 options to NASTRAN Bulk Data Options dialog box. When enabled, ALPHA1 is the complex scale factor applied to the mass matrix and ALPHA2 to the stiffness matrix. Used in frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to complex zero, then Rayleigh's damping is added to the viscous damping.
- Added Label field in the Master Requests and Conditions and Analysis Case dialog boxes which can be used to write the LABEL entry in Case Control.
- Updated processing of NASTRAN Include to allow the INCLUDE statement to start in any column. Previously it had to start in column 1.
- Updated File, Import, Analysis Results when importing a XDB file. After you run Nastran, choose the File, Import, Analysis Results command, select Nastran, and then choose NX Nastran or MSC/ MD Nastran from the drop-down list. FEMAP will display the standard file access dialog box for you to choose the XDB file you want to read. When you press OK, FEMAP will immediately open the Select Output to Internalize dialog box, which facilitates selection of output sets and vectors. By default, all output sets in the Output Sets section will be selected and the All Output Vectors option in the Output Vectors section will be enabled, thus all output from the XDB file will be imported. To only import a subset of the results from the XDB file, simply select the desired output set(s) and optionally disable the All Output Vectors option to be able to select individual output vector(s) for import.

Interfaces - NX Nastran

- Added GPU Computing option to NASTRAN and Solutions Options dialog box. When enabled, writes GPGPU=ANY to the command line which instructs NX Nastran to automatically determine if a device with GPUs exists and, if so, to use it for during the solve.
- Added SWPANGLE option to NASTRAN Bulk Data Options dialog box. When enabled, allows you to enter the angular increment in degrees at which failure indices and strength ratios are computed and output for laminates in direct frequency (SOL 108) and modal frequency (SOL 111) analysis.
- Added MGRID and MDOF option to NASTRAN Bulk Data Options dialog box. When enabled, used to specify a specific node (MGRID) and degree of freedom (MDOF = 1, 2, 3, 4, 5, or 6) to monitor during a direct frequency or direct transient response, plotted in the NX Nastran Analysis Monitor.
- Added Use NXN v8.5 Elastic Beam Formulation option to NXSTRAT Solver Parameters dialog box. When enabled, (BEAMALG = 1), the algorithm for elastic beam formulation from NX Nastran 8.5 is used instead of the current algorithm for elastic beam formulation.
- Added PYR_DETJ option to GEOMCHECK dialog box, which is the Jacobian Determinant for pyramids. Also, added the Include Parabolic Plate Checks option, which when enabled, includes

parabolic element checks using the same options and values specified for the linear versions of QUAD SKEW (Q8_SKEW), QUAD IAMIN (Q8_IAMIN), QUAD IAMAX (Q8_IAMAX) and TRIA IAMAX (TA6_IAMX).

- Added support for reading ply-by-ply stress and strain output from Direct Transient (SOL 109), Direct Frequency (SOL 108), Modal Transient (SOL 112), or Model Frequency (SOL 111) analysis for laminate elements created. This output was not available until NX Nastran 10.0.
- Added support for reading GPF output for contact from Advanced Nonlinear (SOL 601), which will appear as the Total Contact Force and Total Contact Moment vectors, plus their components.

Interfaces - MSC Nastran

- Added MSC Nastran Contact Solver Parameters dialog box to Analysis Set Manager for Static analysis.
- Added Nonlinear Options dialog box to Analysis Set Manager for Static analysis. This dialog box is available in the Master Requests and Conditions section and individual subcases for static analysis when Analysis Program is set to MSC Nastran, but should only be used if the model contains linear contact. In that case, the Enable NLPARM option must be turned on for the subset of available options to be written to the input file.
- Added Contact/Glue Sets section to Boundary Conditions dialog box, which allows you to select which connectors should be exported along with giving you the ability to specify some special case options in Case Control.

Interfaces - ANSYS

- Added reading and writing of linear and parabolic pyramid elements.
- Added reading Plastic Strain results from nonlinear analysis.
- Updated writing of coordinate systems and nodes to provide a higher level of precision.

Interfaces - LS-DYNA

- Added Old Version Import option to potentially use old translator for legacy pre-v970 results files.
- Added support for Translational Accel/Gravity and Rotational Velocity body loads allowing them to be functionally-dependent.
- Added support for Rate Effect - VP for "24..LS-DYNA Piecewise Linear Plasticity" material type, must be a value between -1 and 1.
- Added support for CST (0, 1 or 2) and SCOOR (-3 to 3) for "66..LS-DYNA Linear Elastic Discrete Beam" material type.
- Added support to properly read d3plot files which contain SPH nodes.

Interfaces - Geometry

- Added support for Solid Edge with Synchronous Technology 7 and NX 10.0
- Added support for Parasolid 27.1
- Added support for SolidWorks 2015
- Added support for CATIA V5-6R2014

- Updated File, Import, Geometry command by adding the Sew Sheets into Manifold Solids option to the Solid Model Read Options dialog box when import ACIS geometry (*.sat files). This option controls if sheets (surfaces) should be stitched into manifold solids. If this option is on, then the surfaces will only be "sewn" together into manifold solids. It will only create "manifold solids", as ACIS does not support "NonManifold Solids" (i.e., General Bodies) like Parasolid. If off, no "sewing" will occur.

Aeroelasticity

- Added PARAM OPPHIPA option to NASTRAN Aerodynamic Data (AEROx, MKAEROx) dialog box, which when checked writes out PARAM,OPPHIPA,1 and will output the real vibration modes at all degrees of freedom, including the aerodynamic degrees of freedom.

Loads and Boundary Conditions

- Added ability to create ACCEL1 entries for Nastran by creating Acceleration loads on nodes or geometry, then exporting a model from the Analysis Set Manager using an Analysis Set with Analysis Type set to "1..Static".
- Added ability to Model, Load, Nodal on Face to apply a "Total Load" for Force, Moment, or Heat Flux loads. When enabled, which is the default, the "Total Load" option applies loads based on the face area, much like a geometry-based load. For parabolic faces, the load is "expanded" using the values and options specified for Load Expansion on Midside Nodes on the Geometry/Model tab of File, Preferences.
- Updated the Model, Load, From Freebody command, when using Multi-Model mode, by adding a Closest Node option and Max Distance field to the Method section of the Create Load(s) from Freebody dialog box. The Closest Node method essentially behaves the same as the Match ID method, but instead of searching the target model for a node with a matching ID, which could be located anywhere, it searches within the specified Max Distance for the "closest node" in the target model. If a node is not within Max Distance, then no load will be created in the target model.
- Updated Constraint Equations to allow up to 6,000 terms (nodal DOF). Previously, the limit was 70 terms.
- Updated the Model, Constraint, Equation command to use a completely new Create Constraint Equation dialog box, which offers a number new options to improve the creation of constraint equations.

Properties

- Added Surface with Reference Point button which is available when Shape is set to General Section in the Cross Section Definition dialog box. The Cross Section Definition dialog box is accessed by clicking the Shape button in the Define Property dialog box when creating or editing Bar, Beam, or Curved Beam properties. If you push the Surface with Reference Point button, you will be asked to specify a coordinate to use for the Reference Point after selecting a surface and specifying a vector, and the Reference Point option will also be automatically enabled.

Output and Post-Processing

- Added ability to show contour plots on line elements, so now it is possible to show results on line elements, planar elements, and solids elements in a single contour plot.

- Added Model, Output, Create/Manage Analysis Study command which creates a new analysis study or activates an existing analysis study.

- Added Result Set Processing Data Surfaces to the Data Surface Editor, which allows you to create to automatically create any number of new output sets based on existing output sets in the model. The scale factors for each existing load set may be entered into the Data Surface Editor directly or pasted in from another program, such as excel.

- Added Section Cut as a Display Mode option for the Freebody Tool in the PostProcessing Toolbox. A user-defined "cut plane" is used to automatically determine the Freebody Nodes and Freebody Elements to essentially create an Interface Load. The "cut plane" and "path" can be defined using one of four methods and the location of the Total Summation Vector typically moves along a specified "path". A number of additional options are available to control the behavior of the Section Cut, including the ability to select elements on the reverse side of the plane, control inclusion of elements "cut" by the "cut plane", etc.

- Added List, Output, Freebody Section Cut command, which is only available when at least one Freebody entity with Display Mode set to Section Cut exists in the model (See Section 7.2.3.3, "Freebody tool"). Creates a listing and sends it to the specified Output Destination, using the selected Freebody entity and additional options specified by the user in the Freebody Section Cuts dialog box:

- Updated the View, Select command by adding the Complex Results button to the Select PostProcessing Data dialog box, which is accessed by clicking the Deformed and Contour Data button. These options enable you to set overall "Model Option(s)" or view-specific "override" values used to convert complex data to real data, on-the-fly, when post-processing complex results.

- Added "3..Material Direction" as an option for Output Orientation when transforming output. This can now be set using the drop-down control for Forces, Stresses, and Strains in Tria3, Tria6, Quad4, and Quad8 sections.

- Added Max Threshold and Max/Min Threshold options to Level Mode for the "Contour/Criteria Levels" option in "PostProcessing" Category of View, Options command. Max Threshold will use the value specified for Maximum as a "threshold" value, thus all results above this value are shown using the color of the uppermost band of the Contour/Criteria Legend, while Max/Min Threshold does essentially the same thing, but uses both the Maximum and Minimum values as upper and lower threshold values.

Element - Spring/Damper, Type = CBUSH

- Added ability to specify an "orientation coordinate system" and "location" for a Spring/Damper element referencing a Spring/Damper property with Type set to CBUSH on the element itself, not via the property. Simply set Orientation to CSys, then select an existing coordinate system from the drop-down control. The option to specify the orientation coordinate system on the property still exists by setting Orientation to From Property. Similarly, select Location in the Offsets section and enter a value to specify spring/damper location or use From Property to use the value for Spring/Damper Loc on the Spring/Damper property.

Element - Plate

- Added ability to specify different thicknesses at each corner for Plate elements on the element itself, not only using the T1, T2, T3, and T4 values on the Plate property. To set the corner thickness values on the element, use the Modify, Update Element, Adjust Plate Thickness/Offset command and choose Element Thickness in the Update section of the Update Plate Element Thickness and Offset dialog box instead of Property Thickness. The only way to edit these thicknesses is via Modify, Update Element, Adjust Plate Thickness/Offset command.

Element - Update Existing Elements

- Added the Modify, Update Elements, Midsurface Thickness and Offset command to attempts to update the element thickness and apply appropriate offsets for elements to better match the original solid used to create the midsurface(s).

- Updated the Modify, Update Elements, Line Element Orientation command by adding the Orientation CSys option to allow you to select an existing coordinate system from a drop-down control (can only be used for Spring/Damper elements referencing a Spring/Damper property with Type set to CBUSH). Also, updated the Radial option. If you select a rectangular or cylindrical coordinate system, the orientation will be updated using a vector extending from the Z-axis of the selected coordinate system to the first node of each element. If you choose a spherical coordinate system, the orientation will be updated using a vector extending from the origin of the selected coordinate system to the first node of each element.

- Updated the Modify, Update Elements, Material Orientation command by adding the -X, -Y, and -Z options for Coordinate Axis in the Material Angle section, which allow you to use the opposite direction of the selected coordinate system axis to define material angle.

- Updated the Modify, Update Elements, Orient Plate Normal/First Edge command by adding four additional options to the Normal section. The Away From Location and Toward Location options require you to specify a location in space, then the normals will be adjusted so they will point away from or toward the specified location. The Align to Vector option will attempt to align the normals of the selected elements to match a specified vector, while Align to CSys Direction option will attempt to align the normals of the selected elements to match the selected axis and coordinate system specified in the Options section. Use the Negative option to align to the opposite direction of the selected coordinate system axis.

- Updated the Modify, Update Elements, Adjust Plate Thickness/Offset command by adding the Element Thickness option to the Update section to specify corner thickness values on the element itself. Also, added the Top At Face and Bottom At Face options to the offset elements so the top face or bottom face is aligned with the nodes. Also, added the Reset Element Thickness button to have selected elements revert to the thickness values on the Plate property and the Reset Element Offset button to set the offset values on all selected elements to 0.0.

- Updated the Modify, Update Elements, Linear/Parabolic Order command by adding the Delete Midside Nodes option, which is turned on by default when Update To is set to Linear. When on, any midside node not used by another element will be deleted. If a midside node is attached to an element which cannot have midside nodes, such as a rigid or interpolation element, then the midside node will be removed from the element where is was a midside node, but remain in the model connected to the element where it was not a midside node.

Meshing

- Added Mesh, Connect, Rigid command, which is used to automatically create rigid or interpolation elements. Simply select any number of "source nodes", then the command will use criteria specified in the Connect Rigid Options dialog box to automatically determine appropriate "target nodes" for each "source node". After creating the new elements, the command will then ask if you would like to edit the elements before finishing the command.

- Added Mesh, Editing, Rigid Connectivity command, which allows you to quickly edit an existing rigid (RBE2) or interpolation (RBE3) element using the Toggle Rigid Element Node dialog box.

- Updated the Mesh, Geometry, Hex Mesh from Elements command to allow you to specify Mesh Spacing options along with number of mesh Layers using the Hex Layers Between Base and Top dialog box. Choose from Equal, Biased, or Geometric Bias. When using Biased or Geometric Bias, specify a Bias Factor and choose a location for the Small Elements (at Start, at End, at Center, or at Both Ends). For instance, a Bias Factor of 2.0 and Small Elements at Start would produce a mesh where the layer of elements closest to the base region is 1/2 the thickness of the layer of elements closest to the top region.

- Updated the Mesh, Geometry, Solids, the Mesh, Geometry, Solids from Surfaces, and the Mesh, Geometry, Solids from Elements commands to allow you to select a value between 2 and 10 when using the Multiple Tet thru Thickness option. This option will create the selected number of tetrahedral elements through the thickness of a solid during tetrahedral meshing. Also, renamed the "Multiple Tet thru Thickness" option found in version 11.1 to Split Tets With All Corners on Exterior and moved it to the Solid Automeshing Options dialog box, accessed via the Options button in the Automesh Solids dialog box. Finally, removed the entire Legacy Meshing Options section (including all the options) from Solid Automeshing Options dialog box.

- Updated the first four commands on the Mesh, Connect... menu (Closest Link, Multiple, Unzip, and Coincident Link) by adding the Line Orientation section, which allows you to select an existing Node, specify a Vector, or select a CSys (Coordinate System, CBUSH elements only) to orient line elements created during the command.

- Updated the Mesh, Editing, Edge Split command to allow you to specify Number of Splits, a Bias Type, a Bias Factor, and a location for Small Elements.

Groups and Layers

- Added Group, Operations, Generate Model Data Value command, which automatically creates groups of elements using the value each element has for a selected type of model data, such as specific type of Element Quality check, a specific type of Material Data, or a specific type of Property Data.

- Added Group, Solid, Type command to add solids of the selected "solid type" (Solid, Sheet, Wire, or General/NonManifold) to the active group.

Tools

Measure, Distance Between Nodes

- Updated command to return the "delta coordinates" between the nodes rather than the vector between the nodes, which makes it consistent with Tools, Measure, Distance and is more useful for non-rectangular coordinate systems.

Measure, Distance Between Geometry
- Updated command to allow you to choose Nodes in the From section, enabling you to be able to measure the distance between a node and geometric entities.

Mass Properties, Mesh Properties
- Updated command by adding the Create Node at Total Center of Gravity option to the Check Mass Properties dialog box.

Check, Element Quality
- Added NX Nastran Element Quality checks, which can be accessed by clicking the NX Nastran tab in the Check Element Quality dialog box. There are 22 separate checks on the NX Nastran tab, which are the same quality checks used by the NX Nastran solver (WARP = Warping, IAMIN = Minimum Internal Angle in degrees, IAMAX = Maximum Internal Angle in degrees, AR = Aspect Ratio, EPLR = Edge Point Length Ratio, DETJ = Jacobian) Quadrilaterals - QUAD SKEW, QUAD TAPER, QUAD WARP, QUAD IAMIN, QUAD IAMAX; Triangles - TRIA SKEW, TRIA IAMAX; Tetrahedrals - TETRA AR, TETRA EPLR, TETRA DETJ; Hexahedrals (Bricks) - HEX AR, HEX EPLR, HEX DETJ, HEX WARP; Wedges - PENTA AR, PENTA EPLR, PENTA DETJ, PENTA WARP; Pyramids - PYR AR, PYR EPLR, PYR DETJ, and PYR WARP.

User Interface - General
- Updated the dialog boxes which are used to select Output Sets and Output Vectors for a number of commands to have a toggle that allows the output sets to be shown in the list using Analysis Studies or simply as a list of all Output Sets. When shown using Analysis Studies, a toggle control exists at the top of each study to toggle on/off all Output Sets in the Analysis Study.
- Updated the By Faces and By Output pick methods for elements to follow the Add/Remove/ Exclude setting and not always Add. Also, grayed the Add Connected Elements, Add Connected Fillets, and Add Tangent Surfaces methods if Add/Remove/Exclude is not on Add.
- Updated various Define Element dialog boxes to return the cursor focus back to the first field used to specify a node after changing the element topology in the dialog box. For example, the first of the Nodes fields will be active after changing from Triangle to Quad (or vice versa) in the Define PLATE Element dialog box.
- Updated the standard Plane Locate dialog box, when using the Surface Normal method, to no longer require you to specify a point for At Point or Axis Point if a planar surface has been selected for On Surface. If neither point is specified, the surface CG is used as the location for At Point.

Model Merge
- Added Add Related and Associated Entities button to Entity Selection section, which adds entities both referenced by other entity types currently in the Entities to Merge list and entities associated to those entities (i.e., mesh associated to geometry or vice versa). For instance, if Entity Selection is set to Group and the selected group only contains elements which have been sent to the Entities to Merge list, then pressing this button will add Node, Material, Property, and Layer entity types, and potentially other referenced entities, along with any associated geometric entities to the Entities to Merge list.

Performance Graphics

- Added support to accelerate graphics for coordinate systems, line elements, single node elements, nodal constraints, nodal loads, and elemental loads along with a number of other entities and options.

Preferences

Graphics

- Added Max Mag to Graphics Options section, which specifies the maximum displayed magnification factor allowed by the graphics window. By default, the maximum magnification factor is 10,000. Once the magnification factor reaches the specified value, the level of magnification in the graphics window cannot be increased via zooming or by scrolling the mouse wheel. An error message will be issued to the Messages window when the specified limit has been reached.
- Updated what is supported by the Performance Graphics options in the Graphics Options section.

User Interface

- Added Snap To option to Graphical Selection section. This option controls which Snap Mode will be used as the default Snap Mode when a model is opened. The available options are "0..Screen" (default),"1..Grid", "2..Point", "3..Node", and "4..Smart". See Section 4.2.3, "Quick Access Menu (Right Mouse Button)" in the FEMAP User Guide for more information on the various snap modes.
- Added Ignore Delimiters if Pasting Tabs option to International Localization/Clipboard section. The Ignore Delimiters if Pasting Tabs option, on by default, allows the user to ignore delimiters, such as a comma (or period in certain regions), when pasting from the clipboard, provided there is a tab between the two values. For example, a spreadsheet contains values in 2 columns, each with a value containing a comma (Row 1, Column 1 = 1,001; Row 1, Column 2 = 5,050), and a tab between the two values when copied to the clipboard. When this option is ON, the values pasted into FEMAP from the clipboard are 1001 and 5050, while when this option is OFF, the values pasted into FEMAP are 1 and 1, while the 5,050 value is completely ignored.

Geometry/Model

- Updated the Element Quality Preferences dialog box accessed via the Element Quality button to be tabbed to allow you to enter default values for the FEMAP element quality checks on the FEMAP tab, while allowing you to enter default values for the NX Nastran element quality checks on the NX Nastran tab. Also, added the All On and All Off icon buttons to make it easier to turn on/off all quality checks on the current tab. Finally, added a Restore Tab Defaults button to restore the default values for the current tab.
- Updated the Current Output Orientation dialog box accessed via the Output Orientation button by allowing you to choose "3..Material Direction" from the drop-down control for Force, Stress, and Strain in the Tria3, Tria6, Quad4, and Quad8 sections.
- Removed the Pre-v11.1 Tet Meshing option from the Meshing and Properties section.

Interfaces
- Removed the Output Set Titles drop-down control from the Nastran Options section, as it has been moved to the File Options section of the Results tab..

Results
- Added Create Studies option to File Options section. When enabled, a Study will be created automatically when results are imported or attached to FEMAP. Each study will include all output data found in a particular results file (i.e., all subcases from a static analysis, all modes from a normal modes analysis, all frequencies or time steps from a dynamic analysis, all load steps from a nonlinear analysis, etc).
- Added Study Titles drop-down control to File Options section.
- Added Nastran Output Set Titles drop-down control to File Options section (previously, this option was called Output Set Titles and was found in the Nastran Options section of the Interfaces tab).
- Added Append Femap Title option to File Options section.When this option is enabled, any value associated with an output set, such as a frequency or time, will be added to the end of the title of each output set during import or attach of the results file.
- Removed the Read Nonlinear Output option from the Auto Answer General Post Read Questions section, as it is no longer needed due to FEMAP being able to import both the Linear and Nonlinear results from a nonlinear analysis into the same output set.

API
New and updated API Objects and Attributes
- Added Analysis Study (feAnalysisStudy) object to the API. Also, added Title, AnalysisProg, AnalysisType, FileTime, Notes, and AnalysisSet attributes to the Analysis Study Object.
- Added CaseLabel, ContactOption, ContactSetType, MSCNasCnlConvergenceFlags, and vMSCNasCnlConvergenceFlags attributes to the Analysis Case Object.
- Added NasExtSEOutAssignForm, NasMscCtOn, NasMscCtEnable, NasMscCtDDULMT, NasMscCtRVCNST, NasMscCtSLDLMT, NasMscCtTAUGMNT,NasMscCtAUGMENT, NasMscCtBEAMB, NasMscCtERRBAS, NasMscCtFTYPE, NasMscCtIBSEP, NasMscCtICSEP, NasMscCtMAXSEP, NasMscCtMETHOD, NasMscCtNLGLUE, NasMscCtNODSEP, NasMscCtSEGSYM, NasMscCtAUGDIST, NasMscCtBIAS, NasMscCtERROR, NasMscCtFNTOL, NasMscCtPENALT, NasMscCtSTKSLP, NasMscCtTPENALT, NasBulkMgrid, NasBulkSwpangle, NasBulkAlpha1, NasBulkAlpha2, NasBulkSwpangleVal, NasBulkMdofVal, NasBulkMgridID, NasBulkAlpha1Val, vNasBulkAlpha1Val, NasBulkAlpha2Val, vNasBulkAlpha2Val, NasAerobOPPHIPA, NasNXStratBeamalg, NasExecGPU, NasBulkEndTextOutsideBulk, CaseLabel, ContactOption, ContactSetType, MSCNasCnlConvergenceFlags, and vMSCNasCnlConvergenceFlags attributes to the Analysis Manager Object. Also, updated NasMCheckDataTol to be a real number instead of an integer and also updated vNasGCheckTest2, vNasGCheckTol2, and vNasGCheckMsg2 to allow you to control the PYR_DETJ option in GEOMCHECK.
- Added AxisMajorColor and AxisMajorColorAutoDim attributes to the Chart Object.
- Added PositionIsRelative attribute to the Chart Data Series Object.
- Added Midpoint, Center, GetPrecision, and ResetPrecision attributes to the Curve Object.

- Added ExtendEdgeValues attribute to the Data Surface Object.
- Added NastranQuadSkewOn, NastranQuadSkewLimit, NastranQuadTaperOn, NastranQuadTaperLimit,NastranQuadWarpOn, NastranQuadWarpLimit, NastranQuadIAMinOn, NastranQuadIAMinLimit, NastranQuadIAMaxOn, NastranQuadIAMaxLimit, NastranTriaSkewOn, NastranTriaSkewLimit, NastranTriaIAMaxOn, NastranTriaIAMaxLimit, NastranTetraAROn, NastranTetraARLimit, NastranTetraEPLROn, NastranTetraEPLRLimit, NastranTetraDetJOn, NastranTetraDetJLimit, NastranHexAROn, NastranHexARLimit, NastranHexEPLROn, NastranHexEPLRLimit, NastranHexDetJOn, NastranHexDetJLimit, NastranHexWarpOn, NastranHexWarpLimit, NastranPenAROn, NastranPenARLimit, NastranPenEPLROn, NastranPenEPLRLimit, NastranPenDetJOn, NastranPenDetJLimit, NastranPenWarpOn, NastranPenWarpLimit, NastranPyrAROn, NastranPyrARLimit, NastranPyrEPLROn, NastranPyrEPLRLimit, NastranPyrDetJOn, NastranPyrDetJLimit, NastranPyrWarpOn, and NastranPyrWarpLimit attributes to the Element Quality Object.
- Added SectionPlaneBase, vSectionPlaneBase, SectionPlaneNormal, vSectionPlaneNormal, SectionCurve, SectionVectorBase, vSectionVectorBase, SectionVectorTip, vSectionVectorTip, SectionRotateSums, SectionIncludeClippedElem, SectionTolerance, SectionSumLocation, SectionLimitToGroup, SectionGroup, ReverseTotalValues, SectionMode, SectionRadius, SectionReverse, SectionLocation, and SectionVectorTip attributes to the Freebody Object.
- Added InitAsInteger attribute to the Output Object.
- Added study and combination_type attributes to the Output Set Object.
- Added AssignForm attribute to the Superelement Reference Object.
- Added ComplexSyncMethod, ComplexSyncPhase, and ComplexSyncIncrement attributes to the View Object.

Removed API Objects and Attributes
- Removed type attribute from the Connection Object.
- Removed TetRecoveryMesher, TetPreV10TetMeshing, and TetMaxElemToAllocate attributes from to the Meshing Object.

New and Updated API Methods
- Added CountOutputSets, OutputSets, HasOutputSet, AddOutputSets, RemoveOutputSets, RemoveAllOutputSets and Delete to the Analysis Study object.
- Added AeroChordXYZ and AeroSpanXYZ to the Aero Panel object.
- Added AddNastranElementChecks to the Data Table object.
- Added SetPlateThickness to the Element object.
- Added GetNastranQuadSkew, NastranQuadSkew, GetNastranQuadTaper, NastranQuadTaper, GetNastranQuadWarp, NastranQuadWarp, GetNastranQuadIAMin, NastranQuadIAMin, GetNastranQuadIAMax, NastranQuadIAMax, GetNastranTriaSkew, NastranTriaSkew, GetNastranTriaIAMax, NastranTriaIAMax, GetNastranTetraAR, NastranTetraAR, GetNastranTetraEPLR, NastranTetraEPLR, GetNastranTetraDetJ, NastranTetraDetJ, GetNastranHexAR, NastranHexAR, GetNastranHexEPLR, NastranHexEPLR, GetNastranHexDetJ, NastranHexDetJ, GetNastranHexWarp, NastranHexWarp, GetNastranPenAR, NastranPenAR, GetNastranPenEPLR, NastranPenEPLR, GetNastranPenDetJ, NastranPenDetJ, GetNastranPenWarp, NastranPenWarp, GetNastranPyrAR, NastranPyrAR, GetNastranPyrEPLR,

NastranPyrEPLR, GetNastranPyrWarp, NastranPyrWarp, GetNastranPyrDetJ, NastranPyrDetJ, and CheckNastranQuality to the Element Quality object.

- Added GetSectionCutSums and SetLocationFromCoord to the Freebody object.
- Added AreDuplicate to the Material object.
- Added PutCoordArray to the Node object.
- Added SetComplexOptions to the Output object.
- Added SetCombination, SetStudyCombination, ExpandCombination, and SetComplexOptions to the Output Set object.
- Added AreDuplicate to the Property object.
- Added ClearSearch to the ReadFile object.
- Added SetComplexOptions to the Results Browsing object.
- Added GetSelectedID and Tooltips to the Selector object.
- Added Show to the Set object.
- Added SheetFacesAsSet, WireCurvesAsSet, and Inside to the Solid object.

New and Updated Global Variables

- Added Pref_CreateResultStudy, Pref_ResultStudyTitle, Pref_TabPasteIgnoreDelimeter, Pref_SnapTo, Pref_RenderMaxMagnification, Pref_StudyAppendFemapTitles to set various preferences.
- Added Info_ViewShowNormal, Info_ViewShowTransparent, Info_ComplexSyncMethod, Info_ComplexSyncPhase, and Info_ComplexSyncIncrement.
- Added SolidAllowNonManifold to allow bodies to become NonManifold after geometry operations.
- Updated Pref_OutputSetTitles to set the Nastran Output Set Title preference.
- Updated Info_SnapTo, Info_SnapStyle, Info_MatlAngleDir, Info_ModelSizeX, vInfo_ModelSizeX, Info_ModelSizeY, vInfo_ModelSizeY, Info_ModelSizeZ, vInfo_ModelSizeZ.
- Added Pref_ElemQualQuadSkew, Pref_ElemQualQuadSkewVal, Pref_ElemQualQuadTaper, Pref_ElemQualQuadTaperVal, Pref_ElemQualQuadWarp, Pref_ElemQualQuadWarpVal, Pref_ElemQualQuadIAMin, Pref_ElemQualQuadIAMinVal, Pref_ElemQualQuadIAMax, Pref_ElemQualQuadIAMaxVal, Pref_ElemQualTriaSkew, Pref_ElemQualTriaSkewVal, Pref_ElemQualTriaIAMax, Pref_ElemQualTriaIAMaxVal, Pref_ElemQualTetAspectRatio, Pref_ElemQualTetAspectRatioVal,Pref_ElemQualTetEPLR, Pref_ElemQualTetEPLRVal, Pref_ElemQualTetDetJ, Pref_ElemQualTetDetJVal, Pref_ElemQualHexAspectRatio, Pref_ElemQualHexAspectRatioVal, Pref_ElemQualHexEPLR, Pref_ElemQualHexEPLRVal,Pref_ElemQualHexDetJ, Pref_ElemQualHexDetJVal, Pref_ElemQualHexWarp, Pref_ElemQualHexWarpVal, Pref_ElemQualPenAspectRatio, Pref_ElemQualPenAspectRatioVal, Pref_ElemQualPenEPLR, Pref_ElemQualPenEPLRVal,Pref_ElemQualPenDetJ, Pref_ElemQualPenDetJVal, Pref_ElemQualPenWarp, Pref_ElemQualPenWarpVal, Pref_ElemQualPyrAspectRatio, Pref_ElemQualPyrAspectRatioVal, Pref_ElemQualPyrEPLR, Pref_ElemQualPyrEPLRVal, Pref_ElemQualPyrWarp, Pref_ElemQualPyrWarpVal, Pref_ElemQualPyrDetJ, and Pref_ElemQualPyrDetJVal to set NX Nastran Element Quality values in the preferences.

The following functions have been added or updated:

- feSurfaceNonManifoldAddExpress
- feCurveOffsetCurveWasher2
- feCurvePad
- feFileReadStepOpt2
- feFileWriteStep2
- feAppMessageClear
- feGetRealLength
- feModifySurfaceNormal
- feMeshEdgeSplit2
- feCurveProjectCurvesOntoSurfaces
- feSolidExtendEdgesToSurfaces
- feSolidRemoveRedundantPoint
- feSolidRemoveBlendsBelowRadius
- feSolidMovePointOntoGeometry
- feSolidRayFire
- feSurfaceRayFire
- feSolidExtendToSurface
- feSolidMidSurfaceManualInput
- feGroupsContaining
- feMeasureDistanceBetweenNodes2
- feMeshUnzip2
- feMeasureDistanceBetweenSolids (Corrected Spelling of function)
- feMeasureDistanceBetweenGeometry

The following functions have been removed:

- feOutputTransform

Corrections

Views

- Corrected issue with model box calculation when there are freebodies in the model but no output sets. Now, freebodies are no longer included in the model box calculation (PR# 7148981).
- Corrected issue which caused the origin to always be included when using View, Autoscale, All when a Freebody entity was visible (PR# 7233495).
- Corrected a number of issues which caused the All Views option in View, Options to be ignored if setting multiple options before clicking OK and in conjunction with using the User Defined Contour Palette (PR# 7248855)

Analysis Manager

- Corrected issue which caused incorrect graying of Constraint Type in the Arc-Length Solution Strategy section found on the AdvancedOptions tab of the Nonlinear Control Options dialog box.

- Corrected issue which caused the state of the Bisection option in the Solution Strategy Overrides section found on the Control Options tab of the Nonlinear Control Options dialog box to not be remembered.
- Corrected issue which could potentially cause the options in the Restart Options section of the NXSTRAT Solver Parameters dialog box to become disabled.
- Corrected issue that occurred when copying Analysis Sets. Previously Start Text and End Text for ANSYS, ABAQUS, DYNA and Marc were not copied but were linked to the text in the original Analysis Set. Now the text is independent and duplicated. (PR# 7328279)
- Corrected issue when Copy button was used in Analysis set manager, as Superelement references were not being copied.

General

- Updated numerous functions throughout FEMAP to use the elemental CG rather than the elemental "center" (average of nodes). This impacts the program file functions XEL(), YEL() and ZEL(), location where a coordinate system axis is used to find the material angle, listing and renumbering of elements, updating element offsets between nodes, evaluation of certain Data Surfaces for heat and temperature loads and updating values on elemental Mesh Data Surface 3D Vector data surfaces. None of these will get the same answers as previously.
- Corrected issue in bmp2raster.exe that caused an error message to be displayed if you saved a picture as JPEG, GIF, TIFF, or PNG and gave the file a 1 character filename. The issue did not prevent the file from being created, it simply showed an error because bmp2raster exited unexpectedly. (PR# 7139366)
- Corrected issue to allow picking of parabolic pyramid elements by shape (20..Pyramid, 13-noded).
- Corrected issue where combined curves could not be selected until they had been drawn, which was an issue when opening a model with curves and combined curves turned off, as only curves could be selected but combined curves could not.

Geometry

- Corrected issue which sometimes allowed you to delete curves that are actually the edges of other solids, which could occur in some cases due to functionality in Parasolid (PR# 7158288).

Graphics

- Corrected issue where direction arrow for parabolic beam was incorrect.
- Corrected issue which caused the number of levels to change when switching the Contour Fill Mode of the Contour Type View Option between Continuous and Level Color. Error was introduced in 10.3 with rewrite of contour legend.
- Corrected issue which caused File, Picture, Save to not work and issue an error when a Trace Deformed plot was being displayed.
- Corrected issue which caused nothing to be displayed if filled edges were turned off when using the Window, Show Entities command with transparent also turned off.
- Corrected issue where if no Constraint Set is active, permanent constraints would disappear when rotating the model and remain invisible.

- Corrected issue which would cause the graphics window not to be updated if the user exited the Load Set Manager or Constraint Set Manager dialog boxes with "none active".
- Corrected issue which caused the mid-side node of a parabolic beam element to not be considered during the View, Autoscale commands.
- Corrected issue which would cause any node used to define weld location on a weld element to remain visible, even if the visibility for the element was turned off.
- Corrected issue which caused extra markers to be shown when picking connection regions by screen area (i.e., Box, Circle, Polygon, or Freehand) (PR# 1976747).

Performance Graphics

- Corrected issue where nodal contours would no be displayed on Face 5 of 8-noded or 20-noded Brick elements, which meant contact stresses would not be drawn if on face 5 of either type of brick element.
- Corrected issue where zero values were not shown, as they were be converted to "Glyphs" but never drawn.
- Corrected issue which caused the Bottom Surface Offset option on the Laminate Property to be ignored.

GUI - Dockable Panes

Model Info Tree
- Corrected issue when renumbering load sets where the type of set changed from Load Set to Load Combination or vice versa. Problem caused the load definitions to disappear from tree and set icons to not match Load Set type.

PostProcessing Toolbox
- Corrected issue that occurred when you used the "{ }" button to select an output set or vector. Previously whatever entity type you had selected prior to that was still active for graphical selection. Now the Output Set or Vector are active.
- Corrected issue in Deformed Transform as it did not previously initialize the component check boxes correctly when you opened the PostProcessing Toolbox.
- Corrected issue in that occurred if you had a Contour Group selected. Previously the contour type was available. Even though it did not do anything, it was somewhat misleading that it looked like it could be changed.
- Corrected issue with freebody validation tool that was reporting missing results if a rigid element was in the list of internal or external elements and one of the selected nodes was a rigid reference node. This was an invalid error because Nastran would never output results for these elements specifically; they'll always be output as F-OF-MPC.
- Corrected issue where freebody toolbox is not updated on deletion of freebody using Delete, Output, Freebody.

Charting Pane
- Corrected issue where Charting pane would draw over other panes when "tabbed" with other panes. This Did not occur if the Charting pane was floating, hidden, or docked.

- Corrected issue where automatic title for vs ID and vs Position data series would only take the title from the first output set.
- Corrected issue where chart data series dialog would repeat error messages if the model had output sets, but no active output set and the first output set was > 1.
- Corrected issue where chart data series would not get titles if the data series is "vs output set" or "vector vs vector", not all output sets are used, and the first output set has different vectors than the selected beginning output set. Now, the standard "1..Untitled" will be used which will prevent the legend from disappearing.

Data Surface Editor
- Corrected issue when using the Plot Output Map command on the Operate on Data Surface menu where the contour legend and post titles were not displayed. They are now displayed and also the output map preview is removed and the corresponding contour legend removed when exiting the Data Surface Editor.
- Corrected issue that caused 4-Point Bilinear data surfaces to be used incorrectly in certain cases because the wrong interpolation plane was chosen.
- Corrected issue when scrolling in the Data Surface editor. Previously, if you were editing a cell and moved out of that cell with an arrow key, and that move caused a scroll of the table, incorrect values were shown in some cells even though the headers scrolled properly. Data was never corrupted, just displayed incorrectly.
- Corrected issue which could possibly cause FEMAP to become unresponsive if you used either a Load Combination data surface that contained a row that referenced the same set as was being created.
- Corrected issue that occurred when pasting list data into the selection dialog or the data surface editor dialog for "Along Coordinates Data Surfaces" if the list delimiter character was other than a comma (i.e., normal for international locales).
- Corrected issue where mapping element pressures to element target using an Output Map Data Surface would create incorrect mapping. This has been fixed.

Entity Editor
- Corrected issue to address memory issues when constraint equations, constraint definitions, and geometry based constraints were loaded in the Entity Editor.

Program File
- Corrected issue that caused program file replay to fail when using the dialog box displayed by the Connect, Connection Region command (PR# 1954846).

Interfaces - FEMAP Neutral
- Corrected issue which was incorrectly setting the default values for the "Performance Graphics" option in the "Tools and View Style" category in View, Options for models brought forward from version 11.0 and below.

Interfaces - Nastran

- Corrected issue where the value in the Print Forces Above field found in the Ground Check section of the Model Check dialog box, which writes RTHRESH, was not written normalized to 0-1 range (PR# 1994718).
- Corrected issue which occurred when writing SPCD loads which used the same coordinate system for the definition and output coordinate system, which could produce small undesirable components when transformed.
- Corrected issue when reading INCLUDE files when INCLUDE statement followed a card that was not ended properly (for instance, on a wide field CORD2R card where the last line was missing). This change causes loss of functionality of reading the second half of a card that is found in an IMCLUDE file.
- Corrected issue which when post processing results from an XDB file for laminates defined with Global Plies.
- Corrected issue when reading PLOAD4 card if 0.0 was defined for P2,P3,P4 or if any of those fields were blank indicating a default of P1 (PR# 7277995).
- Corrected issue when writing out an input file for Nastran DDAM where an error would be written indicating a SUPORT card was needed when a SUPORT1 had actually been defined by the user.
- Corrected issue when reading strain energies from attached op2 files where in some cases the output could not be plotted.
- Corrected issue which caused FEMAP to attach to a op2 file instead of importing the file when the user had chosen to import the f06 file. To be consistent, it now imports the op2 in this case.
- Corrected issue where the incorrect material type would be written if a material was originally defined as one of the NX Nastran Other types (501, 502, 503, 504, 507, 509), then that material was modified to be a standard type (Isotropic, Orthotropic etc.)
- Corrected issue which inadvertently allowed Material Type 509 to be referenced by properties other than ones that create a PSHELL. Added "Error: Invalid Material Type (509..Nastran Equivalent Laminate (Multiple MAT2)) Referenced by Property (ID). Respecify Material." (PR# 1998021)
- Corrected issue which prevented proper writing of the CTE on RBE1 elements (PR# 7194175).
- Corrected issue where pressing the Previous button while in the Boundary Conditions dialog box of a Subcase would take the user to the Master case definition (PR# 1972946).
- Corrected issue when reading op2 files that contain SUBCOM case combinations.
- Corrected issue reading and writing NASTRAN files that contain INCLUDE statements where there were '$' in the name/path, or if the filename was continued to multiple lines and those lines had leading spaces or tabs.
- Corrected issue when importing multi-line NSM1 and NSML1 Nastran cards. Also made read of THRU/BY format more general
- Corrected issue where multi-line INCLUDE statements written using external superelement references were not creating an appropriate checksum to run the bundled version of NX Nastran.
- Corrected issue that would cause FORM=LITTLEENDIAN to be written out even if BIGENDIAN was selected in NASTRAN assign statement creator in the Analysis Set Manager.

Interfaces - NX Nastran

- Corrected issue when reading PLOAD4 pressures applied to Pyramid elements, as the pressure was always applied to face 1 of a Pyramid element.
- Corrected issue writing material orientation to CPLSTS6 elements.

Interfaces - LS-DYNA

- Corrected issue which caused CONTACT_TIED_SHELL_TO_SURFACE_BEAM_OFFSET card to not be exported correctly.
- Corrected issue when reading next * (header) card incorrectly if input file contains *BEAMSECTION that does not have optional 2nd data line (orientation) (PR# 7310500).
- Corrected issue where pre-v9.70 LS-DYNA d3plot result files read in incorrectly.
- Corrected issue where node ID was not being written to *ELEMENT_BEAM card if real node was used to define beam orientation (ER# 7135690).

Listing

- Corrected issue when using format codes with special case 0 for field ID (entity number), error was returned if no extra spaces were inserted into the format. Now, allows just "<0>" instead of requiring "<0 >".
- Corrected issue where List, Output, Force Balance Interface Summary Load was always using the X, Y, or Z coordinates in Basic Rectangular instead of the coordinate system specified for the interface load Freebody (PR# 7138827).
- Corrected a number of issues pertaining to listing of freebody entities that inadvertently used some strings which were changed.
- Corrected issue when listing freebodies with interface load to the data table, the corresponding summation data that was printed to the message window had an incorrect header.

Loads and Boundary Conditions

- Corrected issue which caused load arrows with a value greater than 1.0E19 to not be drawn.
- Corrected issue where nodal pressures were labeled with value when pressure load label set to phase (for element pressure).
- Corrected issue where Element Corner Pressures on pyramid elements were being incorrectly drawn on all 5 faces of the pyramid.
- Corrected issue with distributed load on 3-noded beam which caused them not to be drawn (even though they are invalid). The loads are now drawn and the mid-side nodes are ignored.
- Corrected issue which occurred when editing Load Definitions that contained Elemental Distributed Loads. Previously, if you changed the direction of the load, only the first element was updated and you were incorrectly presented with an option to change the face where the load was applied. Now the option allows you to independently change just the values or just the load direction and is applied to all elements in the definition.

Meshing

- Corrected issue that prevented the Mesh, Connect, Multiple command from showing elements as they were created, which made it difficult to see what had been already done (PR# 7110198).
- Corrected issue when organizing a selected path (i.e., Mesh, Sweep commands) that caused the paths to appear disconnected even though they were not. Whether the issue occurred depended on curve ID ordering and curve orientation (PR# 7158798).
- Corrected issue where internal calls to merging nodes could "over merge" and corrupt a model if the default merge tolerance was large compared to the mesh size of the merged area. This could happen if the overall dimensions of the model were large because some entity was placed at a great distance from the real model. Changed the internal calls to use the "Safe Merge" technology in Tools, Check, Coincident Nodes. (PR# 7148981)
- Corrected issue meshing a solid with multiple voids that also contained part of the solid extending thru the void. Previously, this solid could be meshed if selected as a single solid, but not as part of multiple simultaneously meshed solids (PR 7279461).
- Corrected issue when using Mesh, Extrude, Element Face and Mesh, Revolve, Element Face. Previously, if you canceled one of the final dialogs, extra "coating" elements would remain in the model on the faces being extruded/revolved. These are now properly deleted.

Mesh Associativity

- Corrected issue that occurred when automatically associating mid-side nodes that were created on elements that already referenced nodes that were associated to differing levels of geometry (i.e., a solid element that had nodes associated to both Points and the Solid). Previously, the Solid would be selected and mid-side nodes were associated with it, now the lowest topology level is selected and mid-side nodes are correctly associated with edges/surfaces.

Nodes

- Corrected issue which could occur if you deleted a coordinate system that was the active Nodal Output Coordinate System. Previously, it would still be set and future nodes would attempt to use it. Now it is reset back to Coordinate System "0..Basic Rectangular".

Elements - Solid

- Corrected issue where solid element material direction was using deformed centroid to determine cylindrical/spherical orientation.

Elements - Mass

- Corrected issue with internal counters for entities referenced by weld elements that caused them to be undeletable when using Delete, Model, Mesh or Delete, Model Element.

Elements - Weld

- Corrected issue where offsets on Mass elements would be reflected when using Mesh, Copy, Element, instead of simply copied, if all 3 components of the offset were non-zero.

Element Update

- Corrected issue with Modify, Update Elements, Line Element Orientation when using the Radial option. Previously, this option would take the current vector in Basic Rectangular coordinate system, and transform it from the selected Coordinate System to the Basic Rectangular coordinate system, which yielded somewhat random results. Now, for rectangular and cylindrical coordinate systems, calculates the vector from node A to a point on the coordinate system Z axis. For spherical coordinate systems, the vector is from node A to the coordinate system origin.

Properties

- Corrected issue which caused the Layup not to be assigned when using the Copy button to copy the values of a Solid Laminate property.
- Corrected issue which caused certain general sections to not be drawn correctly in the Cross Section Definition dialog box, which was caused by an oversimplification of the surface boundary for display.

Layups

- Corrected issue in the layup manager dialog. If you chose a material, then went into the Global Ply dialog and exited it without selecting a global ply, then immediately pressed New Ply button an error message would be given that "Material 0 does not exist" even though it appeared you had selected a material for the ply.
- Corrected issue saving layups to the library. Previously, if the material of the first ply was unique (not used on any other ply) it was never written to the library and then could not be reloaded.

Output and Post-Processing

- Corrected issue which caused transformation of results on solid elements to not be able to use the Element option as the Output Orientation.
- Corrected issue in Animate-MultiSet or Trace, with the % of Model (Actual) option turned of for the Deformed Style view option, the contours were being scaled when they should not be scaled.
- Corrected issue which occurred when using Animate-MultiSet, then the user makes a change (i.e., changing from set 1-20 to set 21-40), the minimum and maximum values were not being regenerated correctly.
- Corrected issue which caused beam cross section stresses not to be drawn on bar elements when using View, Advanced Post, Beam Cross-Section, if there any PLOAD1 loads on any of the bar elements in the model.
- Corrected issue when using Vertex Buffer Objects which caused criteria plots to not be displayed correctly.
- Corrected issue with labels when using Deformed Style set to Vector and the vectors were being displayed in component mode. The resultant values were drawn not the component values when labeling.

- Corrected issue when transforming deformed vector components. Prior to this, components were always in the global rectangular coordinate system, but now they are in the transformed coordinate system if results are transformed.
- Corrected issue handling of plate top and bottom fiber thickness when their values cause the thickness values to be swapped (i.e., only top fiber and value is below lower face of plate), which caused the results values not to be properly swapped so were on the wrong face.
- Corrected issue which occurred if you were displaying a contour of "elemental results with averaging turned off" or "nodal results" and you a chose a Contour Group but set the Data Selection option to "All Data/Full Model". Previously, the contours would be displayed on the full model, not just the Contour Group.

Groups and Layers

- Corrected issue in group rules when determining properties used to define weld elements.
- Corrected issue which occurred if Group, Operations, Automatic Add was on and you displayed or used a Combined Constraint or Combined Load Set, the constraints/loads were added into the group. Now they are not.
- Corrected issue which caused the Group, Connector, Property command to incorrectly select connectors instead of connection properties. It now works correctly (PR# 1994721).

Tools

- Corrected issue with Tools, Mass Properties, Mesh Properties command which was always using the "base" rectangular coordinates instead of the cylindrical or spherical coordinates, which caused there to be differences between FEMAP and Nastran (PR# 7236074).
- Corrected issue with formatting of the Tools, Check, Element Quality listing to better line up the columns of numbers.
- Corrected issue where Jacobian check could give erroneous results when mid-side node(s) where missing from Parabolic Wedge and Parabolic Pyramid elements.

Model Merge

- Corrected issue that occurred if you merged combined/boundary surfaces into a new model that already had boundary surfaces causing them to be renumbered. Previously the surfaces were not renumbered properly causing possible issues.
- Corrected issue which caused coordinate systems in the new model to become "non-deletable" when Connection Regions are transferred and the model is transformed. Previously all Connection Region types locked their reference coordinate system even though some types of Connection Regions do not actually use a reference coordinate system. In those cases, the transformed global rectangular coordinate system became "non-deletable". Now, only types that use the reference coordinate system are checked. (PR# 7140167)
- Corrected issue that prevented geometric constraints from being merged if you did not merge geometry. Also handled merging new coordinate systems that are required when merging the expanded constraints.

User Interface

- Corrected issue on high resolution or scaled displays where certain items in the Meshing (or other) toolbox could be clipped.
- Corrected issue which icons assigned to buttons caused "memory leaks" and would eventually cause FEMAP to exit unexpectedly.
- Corrected issue where use of Rotate About... commands on Quick Access menu (right-mouse menu) while in selection dialog box, would clear any previous selections.
- Corrected issue which could cause FEMAP to become unresponsive or exit unexpectedly if a dialog box was open and the icon for the File, New command was clicked on the File Toolbar (or any toolbar).

API

- Corrected issue with BCSet and Output property definitions in the AnalysisCase Object to support STRICT compiler option.
- Corrected issue with GetTitleIDList and GetTitleList methods for the API Entity object where result vector titles could only be retrieved from internalized results (i.e., could not be retrieved from attached results).
- Corrected issue which caused Laminate Solids to not be available via the API zElementType enum.
- Corrected issue which caused the Delete method on the the Solid API object to not delete the solid.
- Corrected issue with API functions feSetToolbarCommandBitmap, feAddToolbarSubmenuUserCommand and feAddToolbarUserCommand. Previously, if you attempted to supply a bitmap that was larger than 16x16 it could overwrite memory and could possibly destroy other menu icons or add undesirable icons to other menus. Now it will scale the bitmap you supply to fit the 16x16 requirement. Also updated handling of the bitmaps that can be placed in the Custom Tools directories.
- Corrected issue in the API Contact region (which also impacted Grouping/Selection of surfaces on regions from the GUI) that prevented surfaces in a region from being selected by the "Surfaces on Region" option after a region had been expanded.
- Corrected issue in feCoordLengthAlong that could return the wrong end of the curve if you specified 0.0 length.
- Corrected issue with BoundingBox on the Connection Region object. Previously, it did not work on regions that were "permanently expanded".
- Corrected issue where enDataType was being set differently depending if API or Model, Load, Map Output From Model command was used, which sometimes led to wrong output to load mapping between models.

Preferences

- Corrected issue where option in the Mouse Interface section found on the User Interface was misspelled, as it should have been "Middle Button Click for OK" instead of "Middle Buton Click for OK".

- Corrected a number of issues to allow proper setting and storing of the default state and values for the various element quality checks using the Element Quality button on the Geometry/Model tab.

Spaceball and Astroid

- Corrected issue which occurred when rotating, which caused the animation to return to the initial orientation you were finished rotating.
- Corrected a number of issues related to magnification limits which improves overall behavior.
- Corrected a number of issues when attempting to use SpatialFreedom Astroid 3-D mouse to dynamically rotate, zoom and pan the model, which was broken for FEMAP 11.1.1.

FEMAP v11.1.2 New Features and Corrections
Updates and Enhancements
Performance Graphics

- Added ability to use the Trace Deformation Style when performance graphics was enabled.

Tools - Element Quality

- Updated how the Jacobian element quality check is calculated for Pyramid Elements. Range varies from 0.0 (Ideal) to 1.0 (Very Poor). If the element returns a value of 2.0, the element has failed the Jacobian element check completely and is likely invalid. To determine the Jacobian value, the pyramid element is first divided into two 4-noded tetrahedral elements by dividing the quadrilateral face at corners 1 and 3 ("Element A" using corners 1, 2, 3, & 5; "Element B" using corners 1, 3, 4, & 5), then the pyramid is divided into two different 4-noded tetrahedral elements by dividing the quadrilateral face at corners 2 and 4 ("Element C" using corners 1, 2, 4, & 5; "Element D" using corners 2, 3, 4, & 5). The Jacobian value is calculated for the 4 tetrahedral elements ("A", "B", "C", and "D"), then the minimum value is subtracted from 1.0 to determine the Jacobian value for the Pyramid Element. The Jacobian value is calculated the same way for parabolic elements as midside nodes are not considered at this time.

Layups

- Added Layup Title, Number of Plies, and Total Thickness to the layup representation displayed in the Layup Viewer.

Corrections
General

- Corrected issue which would cause FEMAP to error when exiting. This could happen if a default view had been defined in which visible layers were being set.

Model Merge

- Corrected issue with File, Merge command which was not transforming the Orientation and Offset Meshing Attributes on Curves when using the Transform Merged Model option.
- Corrected issue with File, Merge command which could cause the FEMAP curves on FEMAP surfaces to be renumbered incorrectly which would then cause a failure to import geometry. The only workaround was to make sure that both FEMAP points and curves are not renumbered during Merge.
- Corrected issue with File, Merge command which would cause the cross-section shape to be lost if the line element property was defined using a General Section.

Analysis Manager

- Corrected issue when using the Scratch Files option in the NASTRAN Executive and Solution Options dialog box of the Analysis Set Manager. Issue only occurred when the same directory was used for both available Scratch Directories. (PR # 7114194)

Connection Properties, Regions, and Connectors

- Corrected issue with Connect, Automatic command to allow location of geometric faces that were adjacent, but not overlapping, but without creation of erroneous connection regions and connectors.
- Corrected issue where faces of pyramid elements were not included in regions which allow selection of element faces.

Element - Planar Elements

- Corrected issue which would allow material orientation on elements to be overwritten when the new orientation is perpendicular to the planar element. This can no longer happen.(PR# 7123115)

Graphics

- Corrected issue which caused Ctrl+C to not execute the File, Picture, Copy command during Animation, Mutli-Set Animation and Trace Deformed Styles.
- Corrected issue which caused material direction of solid and solid laminate elements to not be drawn correctly if aligned to solid element coordinate system. Also, added a square in plane of ply to differentiate between Solid Laminate Material Direction and Solid Material Direction.
- Corrected issue when CBUSH elements referenced node(s) which do not exist and would cause FEMAP to become unresponsive. This could only happen when reading in CBUSH elements in a particular format from a Nastran input file.
- Corrected issue which allowed magnification to go well beyond specified limits and caused graphics issues, especially when using a Spaceball. Magnification limits were increased and may no longer be violated.
- Corrected issue when using the Trace Deformation Style when performance graphics was enabled.
- Corrected issue with Criteria plots where elements which did not meet the specified criteria were not being drawn currently. Only occurred when Vertex Buffer Objects were enabled.
- Corrected issue when rotating model with a Spaceball which would cause FEMAP to become unresponsive.
- Corrected issue when using a Spaceball to rotate the model which occurred during animation of results. When rotating with the mouse after using the Spaceball for rotation, the rotation would revert the display to the position before rotation using the Spaceball.

GUI - Toolbars and Icons

Select Toolbar

- Corrected issue with the Select Toolbar when any of the Load and Constraint options where the Active Entity and the Dialog option was selected via the Selector Actions menu. It would issue an error and then never bring up the dialog box for selection.

GUI - Dockable Panes

Meshing Toolbox

- Corrected issue when using the Feature Removal tool in the Meshing Toolbox, when Feature Type was set to Surfaces and the Limit Size option was enabled. Only surfaces with an area greater than the enter Limit Size could be deleted instead of only surfaces smaller than the Limit Size.

- Corrected issue with the Mesh Sizing tool in the Meshing Toolbox. When mesh sizing with the "Increase" operation, any curve which was previously un-sized but meshed with a default size, would be given a mesh size with one more element than expected. Similarly, if the curve was un-sized and unmeshed, the size initially went to one. It now goes to two since "un-sized" equates to one element along the curve.

Model Info Tree
- Corrected issue which would cause erroneous constraint equation entities to appear above all other entities in the Model Info tree when constraint equations existed in a Nastran SPCADD/MPCADD Combination constraint set.
- Corrected issue which would occur if the Copy or Move commands on the context-sensitive menu for Load/Constraint Definitions in Model Info Tree where used and then the user tried to Copy or Move the selected Load/Constraint Definition(s) to a Nastran LOAD Combination Load Set/ Nastran SPCADD/MPCADD Combination Constraint Set. An error will now be issued instead.

Data Surface Editor
- Corrected issue which caused 4-Point Bilinear data surfaces to be evaluated incorrectly in certain cases because the wrong interpolation plane was used.

Charting Pane
- Corrected issue where the Chart Data Series dialog would repeat error messages if the model had output sets, but no active output set and the first output set was greater than 1.
- Corrected issue with Charting pane which would cause FEMAP to exit unexpectedly if the Chart Title exceeded 77 characters.

Interfaces - NX Nastran
- Corrected issue when writing the NX Nastran Fluid Material (MAT10) entry to Nastran input file. Values in FEMAP of which 0.0 for Density, Bulk Modulus, and Speed of sound would cause values of 0.0 to be written out to MAT10 entry, which are invalid. 0.0 values in those fields now write blank fields to MAT10 entry.
- Corrected issue when reading Search Distance (SDISTi) and Extension Factor (EXTi) fields from either the BGSET of BCSET entries for NX Nastran. Issue would only occur if the shell element entries had corner thicknesses assigned and FEMAP needed to create new properties to represent the corner thicknesses.
- Corrected issue where the dbs command line option for NX Nastran would not be written properly to restart an advanced nonlinear analysis when the path to the restart file contained spaces.

Interfaces - Nastran
- Corrected issue when using File, Attach to Results command to attach to results files from a Nonlinear Transient analysis (SOL 129) which would cause FEMAP to exit unexpectedly if the user chose to not read the nonlinear results. (PR# 1981690)
- Corrected issue which caused FEMAP to exit unexpectedly that occurred when trying to read a Nastran file with an INCLUDE statement that was invalid because it went past 72 characters. In

that case the ending delimiter will never be found. Now gives an error that it cannot find the file. However, in addition to the INCLUDE file, other lines of data will still also be lost in the translation, but a crash will not occur.

Interfaces - MSC Nastran

- Corrected issue when using File, Attach to Results command to attach to XDB files from MSC Nastran which contain Grid Point Force results (HK Method only). Could potentially skip indexing of corners 2..n which could make results less accurate.

Interfaces - ANSYS

- Corrected issue when importing elemental stress results from ANSYS. FEMAP had trouble detecting if the elemental stresses were in the new or old format, which could cause incorrect results at the element corners.
- Corrected issue when writing an ANSYS input file by increasing precision when writing Nodes and Coordinate Systems. FEMAP will now write 16 character fields with 12 digits of precision. (PR# 7113927)

Interfaces - LS-DYNA

- Corrected issue when writing out *ELEMENT_INERTIA in Free Field Format, which needs to be written in Fixed Field Format. This could also have caused problems with many materials and some other entries. The following materials and entries are now written out using Fixed Field: Format: *MAT_ORTHOTROPIC_ELASTIC, *MAT_PLASTIC_KINEMATIC, *MAT_ELASTIC_PLASTIC_THERMAL, *MAT_SOIL_AND_FOAM, *MAT_HIGH_EXPLOSIVE_BURN, *MAT_NULL, *MAT_STEINBERG, *MAT_JOHNSON_COOK, *MAT_PSEUDO_TENSOR, *MAT_ORIENTED_CRACK, *MAT_POWER_LAW_PLASTICITY, *MAT_RIGID, *MAT_ORTHOTROPIC_THERMAL, *MAT_COMPOSITE_DAMAGE, *MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC, *MAT_PIECEWISE_LINEAR_PLASTICITY, *MAT_GEOLOGICAL_CAP_MODEL, *MAT_HONEYCOMB, *MAT_FORCE_LIMITED, *MAT_CLOSED_FORM_SHELL_PLASTICITY, *MAT_FRAZER_NASH_RUBBER_MODEL, *MAT_LAMINATED_GLASS, *MAT_BARLAT_ANISOTROPIC_PLASTICITY, *MAT_FABRIC, *MAT_3-PARAMETER_BARLAT, *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC, *MAT_FLD_TRANSVERSELY_ANISOTROPIC, *MAT_NONLINEAR_ORTHOTROPIC, *MAT_USER_DEFINED_MATERIAL_MODELS, *MAT_BAMMAN, *MAT_BAMMAN_DAMAGE, *MAT_CLOSED_CELL_FOAM, *MAT_ENHANCED_COMPOSITE_DAMAGE, *MAT_LOW_DENSITY_FOAM, *MAT_COMPOSITE_FAILURE_SHELL_MODEL, *MAT_COMPOSITE_FAILURE_SOLID_MODEL, *MAT_ELASTIC_WITH_VISCOSITY, *MAT_KELVIN-MAXWELL_VISCOELASTIC, *MAT_VISCOUS_FOAM, *MAT_CRUSHABLE_FOAM, *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY, *MAT_MODIFIED_ZERILLI_ARMSTRONG, *MAT_LINEAR_ELASTIC_DISCRETE_BEAM,

*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM,
*MAT_SID_DAMPER_DISCRETE_BEAM, *MAT_CONCRETE_DAMAGE,
*MAT_LOW_DENSITY_VISCOUS_FOAM, *MAT_GENERAL_VISCOELASTIC,
*MAT_HYPERELASTIC_RUBBER, *MAT_OGDEN_RUBBER, *MAT_SOIL_CONCRETE,
*MAT_HYSTERETIC_SOIL, *MAT_RAMBERG-OSGOOD,
*MAT_PLASTICITY_WITH_DAMAGE, *MAT_FU_CHANG_FOAM,
*MAT_ORTHOTROPIC_VISCOELASTIC, *MAT_MTS, *MAT_ACOUSTIC,
*MAT_SOFT_TISSUE, *MAT_SOFT_TISSUE_VISCO,
*MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM, *MAT_BRITTLE_DAMAGE,
*MAT_SPOTWELD, *MAT_ANISOTROPIC_VISCOPLASTIC,,
*MAT_MODIFIED_HONEYCOMB, *MAT_SIMPLIFIED_RUBBER/FOAM,
*INITIAL_VELOCITY_NODE, *DEFINE_BOX, *CONTACT_, *ELEMENT_INERTIA,
*PART_COMPOSITE FEMAP Property

- Corrected issue with the LS-DYNA Elastic with Viscosity Material which caused the V0, A, B, and C fields to be exported incorrectly. Also, renamed "Viscosity vs. Time Func" to "Viscosity vs. Temp Func".

- Corrected issue with the LS-DYNA Hysteretic Soil Material which caused the value for Damping Factor - DF to be written to the Yield Constant - A2 field in the LS-Dyna input file and vice versa.

- Corrected issue with the LS-DYNA Temperature Dependent Orthotropic Material which caused the values for Thermal Expansion to be written to the Shear Modulus fields in the LS-Dyna input file and vice versa. (PR# 7140691)

- Corrected issue which caused LS-DYNA Laminated Glass Material to be formatted incorrectly.

- Corrected issue when importing D3PLOT results files which contain results for SPH elements.

- Corrected issue when exporting LS-DYNA Geologic Cap Model. Previously, *MAT_GEOLOGICAL_CAP_MODEL was written, now writes *MAT_GEOLOGIC_CAP_MODEL

- Corrected issue when writing isotropic material with Elasto-Plastic (Bi-Linear) option set on Nonlinear tab to LS-DYNA input file. Shear Modulus, G, was only written when explicitly specified.

- Corrected issue which was causing only a single mass element to be written to the LS-DYNA input file, even though multiple mass elements exist in the model. (PR# 7132470)

- Corrected issue when writing thermal loads on nodes to the LS-DYNA input file. Previously, no thermal loads on nodes were written at all.

- Corrected issue with default formulation for tetrahedral elements written to LS-DYNA input file. Default was -1, but changed to 10 for 4-noded tetrahedral elements and 17 for 10-node tetrahedral elements. (PR# 7140691)

Listing

- Corrected issue which caused "Legacy" XY Plotting information to be included when using the List, View command. Now this information is skipped unless the "Enable Legacy XY Plotting" option is turned on in the Interfaces tab of the File, Preferences command.

- Corrected issue with commands that listed to the Messages window and used a line break, as the line break would now include erroneous text. Was only an issue for FEMAP 11.1.1.

- Corrected issue when listing of freebody interface load results. Summation location was always listed in Basic Rectangular regardless of the Coordinate system used for the summation location.
- Corrected issue when using List, Output, Force Balance Interface Load and no freebodies exist in the database or user chooses to define them manually. Turning off the Reaction option in the Freebody Option dialog box would still include the Reaction contributions in the listing, while turning off MultiPoint Reaction would still include the MultiPoint Reaction contributions, but not include the Reaction contributions.

Loads and Boundary Conditions
- Corrected issue which prevented constraint equations from being displayed when a Nastran SPCADD/MPCADD Combination Constraint Set was the Active or Selected Constraint Set.
- Corrected issue when expanding Total Bearing loads on multiple surfaces. The total load adjustment was only being computed across the final surface where the total load was applied.
- Corrected issue with how FEMAP was determining sheet solids, which slowed performance when expanding geometric loads and exporting analysis files. Before, a model with 2500 surfaces with equation based pressure loads and 230000 elements took over an hour to expand the loads and write a Nastran file, now 11 seconds.

Materials
- Corrected issue when entering values for LS-DYNA Steinberg Material. Previously, there was no way to enter value(s) for Spall Type.
- Corrected issue when entering values for LS-DYNA Force Limited Material. Previously, there was no way to enter value(s) for Yield Mom S2 and Yield Moment T2.
- Corrected issue when entering values for LS-DYNA Frazer-Nash Rubber Model Material. Previously, there was no way to enter value(s) for Strain Limit.
- Corrected issue when entering values for LS-DYNA Composite Failure Solid Material. Previously, there was no way to enter value(s) for Tensile Strength C Dir.
- Corrected issue when entering values for LS-DYNA Modified Zerilli Armstrong Material. Previously, there was no way to enter value(s) for Spall Type. Also, the PC field was not being exported correctly.
- Corrected issue when entering values for LS-DYNA MTS Material. Previously, there was no way to enter value(s) for b1.
- Corrected issue with the LS-DYNA Plasticity with Damage Material which caused the value for Plastic Strain Soft (Log) to be overwritten with other data.

Meshing
- Corrected issue with the Mesh, Mesh Control, Mesh Points on Surface command, which was missing the Use Existing Points option. This option was inadvertently deleted in FEMAP 11.1 and has been restored.
- Corrected issue which occurred when attempting to mesh a boundary surface the referenced curves that no longer existed because the surface had become corrected. This would cause FEMAP to exit unexpectedly. (PR# 7134776)

Functions

- Corrected issue which could cause entries in Functions to be entered twice. Previously, if an entry from the function list was selected and Data Entry method was on Single Value the "Add" button would be available. If either "OK" or "Save to Library" was pressed the values in X and Y would be added back into the function prior to saving the function, thus duplicating the existing entry. Now, "OK" and "Save to Library" do not add any entries, all entries must be in the list prior to pressing them. (PR# 7138043)

Groups and Layers

- Corrected issue with Group, Operations, Generate Solids command which still created groups even if Cancel was pressed on the "Group Operations Generate Solids" dialog box. Now it simply exits.

Output and Post-Processing

- Corrected issue when using the Model, Output, Calculate command with attached OP2 and CSV results files which would prevent the VEC( ) function and the EntityValue( ) method of the Results Browser API object from returning results other than 0.0 (PR# 1977805)
- Corrected issue which cause no results to be displayed when transforming solid element results unless Output Orientation was set to a specific value. Now works with any Output Orientation.
- Corrected issue when using List, Output, Force Balance Interface Load Summary command to create and optionally plot functions. If the freebody summation location was defined in a coordinate system other than Basic Rectangular, it would incorrectly always report the value in Basic Rectangular.

Tools

- Corrected issue when using the Tools, Convert Units command where curve and surface meshing attributes where not converted to new values. Now, meshing attributes such as orientation, offsets, tolerances and target mesh sizes on curves and surfaces are converted. (PR# 7112881)
- Corrected issue with the Tools, Measure, Distance Between Geometry command which would error when basic FEMAP curves were selected instead of solid curves. Now works for all curves (PR# 7133788)
- Corrected issue when calculating Tet Collapse element quality value for pyramid elements.

API

- Corrected issue which would cause an icons that were set on toolbars via the API to be lost when FEMAP was exited.
- Corrected issue with the Results Browser Object in the API when calling VectorComponents method. Would cause FEMAP to exit unexpectedly.
- Corrected issue with the Results Browser Object in the API when calling GetColumns method. Would always return FE_BAD_DATA, even if it succeeded.

FEMAP v11.1.1 New Features and Corrections
Updates and Enhancements
Model Merge

- Updated the File, Merge command to optionally merge geometric entities from one model into another model. In order to create valid geometry, all of the underlying geometric entities must also be selected. For instance, if you want to merge in a solid entity, then all of the surfaces, curves, and points used by that solid must also be selected for merging.

Performance Graphics

- Performance Graphics now displays material direction on planar elements and element directions on planar elements (Right-Hand Rule, Normal Vectors, and Right-Hand Rule First Edge options only).

Analysis Manager

- Added Temperature drop-down in the Boundary Conditions dialog box. This allows you to select a Load Set to and will write TEMP(LOAD) in the Nastran input file.
- Updated MultiSet to use a single consolidated dialog box to automatically created subcases using every combination of the selected Load Set(s) and Constraint Set(s).
- Updated Delete to use a single consolidated dialog box to automatically created subcases using every combination of the selected Load Set(s) and Constraint Set(s).

Connection Properties, Regions, and Connectors

- Updated the Connect, Automatic command to allow selection of Edge Regions Output. This setting controls how edge regions will be exported.
- On the LS-DYNA tab in the Define Connection Property dialog box, added "13..Tied Shell Edge To Surface Beam Offset" option to the Type drop-down in the General section.

Geometry

- Added Geometry, Solid, Fill Hole command.

GUI - Toolbars and Icons

Entity Display Toolbar

- Added an icon that will toggle off both Nodes and Permanent Constraints to replace the Nodes Only icon on the Toolbar. The Nodes Only icon can be added to any Toolbar via Tools, Toolbars, Customize, selecting the Commands tab, choosing Additional Commands, then dragging View Node Visibility.

Custom Tools Toolbar

- Added Stop API Tool command, which will stop an API script which is running. This command also appears on the Quick Access Menu (right-mouse menu) when an API script is running.

GUI - Dockable Panes

Meshing Toolbox - Mesh Surface tool
- Mesh Surface tool - Added "Use Internal Points as Mesh Locations" option to the Advanced Options section.

Meshing Toolbox - Locator tool
- Locator tool - Added "MultiSurface Edges" and "Adjacent Edges" options when Search For is set to Curves. When on and using the MultiSurface option, the Locator will find any curves which are used by multiple surfaces. When on and using the Adjacent Edges option, the Locator will find all curves which overlap, regardless of curve length, and offers both a Tolerance and Angle Tolerance values which may be modified.

PostProcessing Toolbox - Contour Tool
- Added New Element Group icon button beside Show on Groups drop-down. This allows you to quickly create a group containing only elements and is available for when using any Contour Style. When using this command, simply enter a Title and different ID for the new Group (both optional), click OK, then use the standard entity selection dialog box to select elements.

Charting Pane
- Changed default color for minor grid lines to be a darker color.

Data Table
- Added Sort Off option to the context-sensitive menu for Column Headers. This allows you to have entities listed in the Data Table using the original order of entry into the Data Table (i.e., before any sorting was done).

Interfaces - Nastran
- Added support for writing TEMP(LOAD) by selecting a load set using the Temperature drop-down in the Boundary Conditions dialog box. This is very helpful for including temperature loads using a different Load when using Nastran LOAD Combinations for structural loads.
- Added support for reading forces, stresses, and strains for CBUSH elements when importing the .f06 file. Random output (PSDF, RMS, CRMS) is not supported when reading from the .f06 file.
- Improved support when reading a Buckling input files by reading the EIGB entry and limiting the number of subcases created to the appropriate amount.
- Improved support to handle a variety of multi-line, multi-token INCLUDE statements.

Interfaces - MSC Nastran
- Added support for import of contact results from SOL 400 analysis, specifically, for the OFCON3D and OFCON3D0 data blocks.
- Added support for import of results from a Random Response analysis, specifically for the OAGRMS1, OVGRMS1, OAGNO1, and OVGNO1 data blocks.

Interfaces - ANSYS

- Added support for reading TARGE170, CONTA173 and CONTA174 as connection regions. Also creates connections and connection properties from other data in the file.
- Added support for reading NSEL, ESEL, CM entries to create Groups.

Interfaces - LS-DYNA

- Added support to write *PART_COMPOSITE for Laminate properties.
- Added support to write all of the entries needed to run an analysis to run using the implicit solver.
- Added support to write *CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET entry.
- Added support to write EQ 23 and EQ 24 element formulations for plate elements.
- Added support to write EQ -1 and EQ -2 element formulations for solid elements.
- Added support to write negative values for Thickness Overrides (used in Tied Contact).
- Added support to write the G command line option, which specifies where the d3plot files will be written. By default, it will always be the same directory as the input file.

Interfaces - Comma-Separated

- Added support for the Extended Comma-Separated Format. This format must be used in order to attach CSV to FEMAP using the File, Attach to Results command.

Interfaces - Geometry

- Added support for Parasolid 26.1

Loads and Boundary Conditions

- Added Multi-Model option to Model, Load, From Freebody with the intent of streamlining Global-Local modeling between a Global model and a more refined Local model.
- Added the ability to choose Load Definitions to expand loads using the Model, Load, Expand command. A similar option was added for expanding Constraint Definitions when using Model, Constraint, Expand.
- Added capability to Geometry, Copy...; Geometry, Scale..., Geometry, Rotate...; and Geometry, Reflect... commands to properly copy and transform geometric loads with the new geometry.
- Added capability to Geometry, Copy...; Geometry, Scale..., Geometry, Rotate...; and Geometry, Reflect... commands to properly copy geometric constraints that either reference DOF in a specific coordinate system or that allow sliding along a specified direction.
- Loads can now be rotated or otherwise modified along with other entities when using the Modify, Move To, CSys; Modify, Move By, CSys; Modify, Rotate By, CSys; Modify, Rotate To, CSys; or Modify, Align, CSys commands. For this to occur, the Rotate Loads in Modify Rotate/Align CSys options must be turned on in the Geometry/Model tab of the File, Preferences command.
- Added a From Set drop-down control which allows you to select a Load Set when using List, Model, Load Definition; Delete, Model, Load Definition; or Modify, Renumber, Load Definition commands. A similar control was added to allow selection of a Constraint Set when using the List,

Model, Constraint Definition; Delete, Model, Constrain Definition; or Modify, Renumber, Constraint Definition commands.

Materials

- Updated 74..LS-DYNA Hyperelastic Rubber and 76..General Viscoelastic in the Other Types dialog box to work with the current version of LS-DYNA

Element - Rigid

- Added Remove from List icon button to the Define Rigid Element dialog box, which will remove the highlight node(s) from the list. Also, modified the Delete button to bring up the standard entity selection dialog to allow you to pick nodes to remove from the rigid element from the model instead of needing to know the node IDs.

Element - Planar Elements

- Added "19..EQ 23: 8-Node Quadrilateral Shell" and "20..EQ 24: 6-Node Quadratic Triangular Shell" element formulations to the DYNA Options drop-down in the PLATE dialog box.

Element - Solid Elements

- Added "18..EQ -1: Fully Integ S/R for Poor Aspect Ratio, Efficiency" and "19..EQ -2: Fully Integ S/R for Poor Aspect Ratio, Accurate" element formulations to the DYNA Options drop-down in the SOLID dialog box.

Element - Update Existing Elements

- Updated the Modify, Update Elements, Linear/Parabolic Order command by consolidating the different questions asked by the command into a more user friendly dialog box.

Meshing

- Added Use Internal Points as Mesh Locations option to the Mesh, Geometry, Surface command.
- Added On Surface option and a Midside button to all commands on the Mesh, Editing menu (Interactive, Split, and Edge Split).

User Interface

- Updated the multi-select dialog box for "titled" entities which can be accessed via the standard entity selection dialog box. The updated version allows you to check any number of entities as well as filter the list of entities by using the "matching text" filter
- Added Distance icon button to a number of dialog boxes which prompt user for a distance or length.

Modification

- Added Between Coordinate Systems method to all commands on the Modify, Align... menu and also made Between Coordinate Systems the default method.

Preferences

Geometry/Model
- Added preference to Rotate Loads in Modify Rotate/Align CSys. When on, the loads will be rotated along with other entities when using the Modify, Move To, CSys; Modify, Move By, CSys; Modify, Rotate By, CSys; Modify, Rotate To, CSys; or Modify, Align, CSys commands. When off, the loads will not be rotated when using these commands.

Interfaces
- Changed Improve Single Field Precision option to Improve Real Number Precision in Nastran Solver Write Options section. This option now works to improve precision in both Small Field and Wide Field formats.

Results
- Removed the Auto Answer Post Questions button and moved all of options from the Auto Answer Post Read Questions dialog box to the "top-level" of the Results tab.

API

Overall capabilities
- Added ability to stop a API script while it is running with the Stop API Tool command on the Custom Tools Toolbar or the Quick Access Menu (right-mouse menu). This will work even if the interface has been "locked" using the appropriate API calls. Also, the Stop button in the API Programming pane can also now be used when the interface has been "locked" using API calls.

New API Objects and Attributes
- Added TrackData Object
- Added BeamCalculator Object. Also, added Element, Position, MeshFactor, IncludeAxialForce, IncludeShearForceY, IncludeShearForceZ, IncludeMomentY, IncludeMomentZ, and IncludeTorque properties.
- Added read-only ScaleFactor property to Point Object
- Added read-only ScaleFactor property to Curve Object
- Added read-only ScaleFactor property to Surface Object
- Added read-only ScaleFactor property to Solid Object
- Added read-only Visible and Locked properties to Data Table Object

New and Updated API Methods
- Added Clear, Start, StartGeometry, StartMesh, Stop, StopAll, Created, and Deleted to TrackData Object
- Added GetElementProperties, GetMeshInfo, CalculateStress, CalculateStressFromForces, and FindMaxMinStess to BeamCalculator Object

New and Updated Global Variables
- Added Pref_LoadModifyRotate

The following functions have been added or updated:
- feVectorRotate
- fePlaneRotate
- feSolidFillCavity
- feOutputTransform2
- feSolidEmbedMultiple
- feAlignBetweenCSys
- feMeshSizeCurveMatchNodes
- feMeshSizeCurveMatchXYZ

Corrections

Model Merge
- Corrected an issue in Model Merge that caused Nodal Loads defined in user coordinate systems to be handled incorrectly when models were transformed during merge. Occurred if loads were defined in a non-global Coordinate System. Also fixed similar problem for directional elemental pressure loads.

Performance Graphics
- Corrected issues with criteria display. (PR# 6966170)
- Corrected an issue to allow File, Picture, Save to work with animation.
- Corrected an issue where offsets were not drawn following pass/fail for Criteria.

Analysis Manager
- Corrected an issue where the Analyze Multiple button was used, but then canceled. When this was done, FEMAP would start an analysis of the Active set. (PR# 1962447)
- Corrected an issue that caused FEMAP to exit unexpectedly if "MultiSet" was used and the constraint and load set titles that were being combined resulted in a title that was longer than the allowable title length. (PR# 6997791)

Connection Properties, Regions, and Connectors
- Corrected an issue that occurred when reflecting solid elements with connection regions on the faces of those elements. Previously the faces in the region were not renumbered, even though the reflected elements were renumbered. This caused the region to be applied to incorrect faces.

General
- Corrected an issue in rebuild that caused some Functions referenced by materials to become deletable after a rebuild.
- Corrected an issue where Aero entities were not being deleted when using the Delete, All or Delete, Model, All commands.
- Corrected an issue that caused the selected coordinate system to be ignored when renumbering based on coordinates. Previously always renumbered relative to Global Rectangular.

Geometry

- Corrected an issue that occurred if you had combined surfaces and you renumbered the underlying surfaces that were used to define the combined surface. Previously the combined surface was corrupted.
- Corrected an issue where elements lost geometric associativity when Nastran translator auto converts to parabolic triangles. (PR #6979630)
- Corrected an issue where the new curves created by Geometry, Curve - From Surface, Pad and Geometry, Curve - From Surface, Offset Curves/Washer were not getting the correct mesh size when an overall default mesh size was in use. (PR# 1969240)
- Corrected an issue in Geometry, Surface, From Mesh to have it now use Custom Mesh Size on curves to exactly match the nodes that created the surface for mesh sizing.
- Corrected an issue in Geometry, Surface, From Mesh to not use special case code to create a four-sided surface for regions created from mesh which have four edges, but the edges are not all connected.

Graphics

- Corrected an issue when changing Contour Options from the View Options command if the view was animating.
- Corrected an issue where many different nodal loads (Temperature, Heat Flux, Heat Generation, Static Fluid Pressure, Total Fluid Pressure, General Scalar, Steam Quality, Relative Humidity, Fluid Height Condition, Fan Curve, and Periodic Condition) were not following deformation. Previously, were drawn in undeformed location.
- Corrected an issue for element directions for solid elements, so they are only drawn when valid. For instance, if free face is used, direction is not drawn, as it caused arrow heads to appear to be floating in the middle of the solid.
- Corrected an issue when rotating or moving points, curves, surfaces or volumes created with the Standard Geometry Engine where beam cross sections on the curves were not updating the cross section graphically. This also applied to moving/rotating Parasolid solids.
- Corrected an issue where front pick of nodes was not possible for nodes that were not attached to elements when during any element blanking.
- Corrected an issue when using Tools, Undo/Redo where facets on a surface with interior point loops was not undoing correctly.
- Corrected an issue when using the Element - Directions set to "Right-Hand Rule" option for 5-noded and 13-noded pyramid elements.
- Corrected an issue when using the User Defined contour palette where color order is incorrectly reversed. This was created based on changes made in FEMAP 11.1.

GUI - Dockable Panes

Meshing Toolbox

- Corrected an issue in the Meshing Toolbox Locator by hiding the "With Poles" option for everything except Surfaces.

- Corrected an issue when using the Limit Size to suppress curves and surfaces. Now, suppresses curves and surfaces smaller than the specified value. Also suppression of loops computed the wrong size if a box pick was used to select multiple loops.

Entity Editor
- Corrected an issue creating Nastran PLANE_STRAIN Formulation titles for listing and entity editor.

Model Info Tree
- Corrected an issue that occurred when using the context-sensitive menu from the Model Info tree to attempt to list a Body Load. (PR# 1965356)

Charting Pane
- Corrected an issue in chart data series dialog so it does not show an error with output vectors when switching tabs. Additionally, output vector existence is now checked for all output sets that may be used (vs Set and Vector vs Vector)
- Corrected an issue creating Nastran PLANE_STRAIN Formulation titles for listing and entity editor.

Interfaces - FEMAP Neutral
- Corrected an issue writing single line text fields to a Neutral file in FEMAP and neutral file converters which could cause unintentional CR or LF to be written.

Interfaces - Nastran
- Corrected an issue when reading and attaching to RMS stress output set in a model that contains shell and line elements.
- Corrected an issue when reading EXTSEOUT card, EXTID, DMIGOP2 and MATOP4. Previously, values were not being saved correctly.

Interfaces - NEi Nastran
- Corrected an issue in Nei writing response spectrum XYPRINT case control (PR # 6973366)

Interfaces - DYNA
- Corrected an issue where improper translation of rotation prescribed motion occurred. Required input entry format to be updated. (PR# 6973439)
- Corrected an issue where LS_DYNA Material Ogden Rubber parameters were not exported correctly and function IDs specified in dialog do not persist. (PR# 6949123)
- Corrected an issue in translation of MAT_RIGID entry, which requires an extra blank line be written. (PR# 6973798)
- Corrected an issue when writing out small values for *ELEMENT_BEAM_ORIENTATION. Was addressed by writing these values with exponents. (PR# 6976577)

- Corrected an issue where Beam Moment output was imported from d3plot being reversed from NASTRAN standard.

Interfaces - Geometry
- Corrected issue which caused NX translator to not be able to read in NX files from older versions.

Listing
- Corrected an issue when using the List, Model, Load-Individual command that occurred when listing multiple load sets that contained body loads. Previously the body loads from earlier sets were listed again with the body loads from subsequent sets.

Loads and Boundary Conditions
- Corrected an issue that occurred when expanding loads on multiple surfaces using a brick mesh with one element thru the thickness of a solid. (PR# 1962789)
- Corrected an issue where Bearing and Torque loads where not expanding into the correct directions when they were defined relative to user defined coordinate systems. The problem only occurred if the load definition system was a local system, the direction vector could always be defined in any coordinate system and that did not cause any problems. (PR 1971305)
- Corrected an issue drawing direction of vector defined geometric loads when the load coordinate system was not global.
- Corrected issues when using the Between Coordinates Data Surface data surface for creating loads. Because of some issues with interpolation, both the 4-point Bilinear and 8-point Trilinear data surfaces could create loads which were incorrect or missing entirely for some locations, especially in skewed corners.

Meshing
- Corrected an issue that caused FEMAP to become unresponsive when attempting to Locate or Copy length-based spacing in the Custom Mesh Size Along curve command on certain curve types (Splines or Solid curves that were not lines).
- Corrected an issue in surface meshing to properly handle meshing surfaces with imprinted internal points. Previously would not mesh and generated an "Out of Memory" message. (PR 6968157).
- Corrected an issue in the triangle surface mesher by simply aborting when one of the outside Delauney nodes gets hooked to an active triangle.
- Corrected an issue with tetrahedral mesher in version 11.1 where the active element formulation was not being set on the tetrahedral elements being created. Previously, the workaround was to use Modify, Update Elements, Formulation to update the formulation after meshing. PR# 6992847
- Corrected an issue with tetrahedral meshing where a 100x100x100 block meshed at a mesh size of 2 (50 elements to an edge) was unsuccessful due to memory issues.
- Corrected an issue in Mesh, Extrude, Element Face; Mesh, Revolve, Element Face; and Mesh, Sweep, Element Face that caused twisted wedges to be created when any parabolic triangular face was used for the operation.

Output and Post-Processing

- Corrected an issue when attaching results using an op2 file that could cause the translator to read the block headers improperly and fail to plot or cause FEMAP to exit unexpectedly.
- Corrected an issue reading PSDF output from an attached op2 when beams, bars, or rods existed in the model.
- Corrected an issue when internalizing results using the Save To Model option in File, Attach To Results command. Would occur if multiple output sets or when nodal and elemental output vectors were selected at the same time.
- Corrected an issue that prevented Model, Output, Transform from working on multiple output sets. No matter how many were selected transforms were only done on the first set.
- Corrected an issue when the Model, Load, Map Output From Model caused FEMAP to exit unexpectedly when 15-noded wedge elements exist in the source model.

User Interface

- Corrected an issue that occurred if Functions were defined when creating material, then the Load or Copy button was used, then the Function Reference tab was selected. Previously the list of functions appeared multiple times in the drop-down for each function reference.
- Corrected an issue where fields no longer update with a value when Measure Angle is used, but Cancel was used before completing the measurement (PR# 6986540)
- Corrected an issue in both graying and sizing of Allow Labels switch in the View, Advanced Post, Contour Model Data command.
- Corrected an issue that caused Modify, Break (and other commands) to fail if you had a coordinate dialog up in Length Along method, then pressed Ctrl+D to measure a distance and chose a different coordinate method like On Point. Previously once you exited the dialog still thought you were in On Point mode even though the Length Along dialog was displayed. Now when going into the shortcut key mode (Ctrl+D in this case) the coordinate, vector, plane methods are preserved and restored after you finish the shortcut.

API

- Corrected an issue getting the AnalysisManager object when using MARC Contact Table(s).
- Corrected an issue with the feSetToolbarCommandBitmap API function. Previously calling the function did not dynamically update the icon of the command on a toolbar. Image was changed but the change was not visible because the toolbar/control was not properly redrawn.
- Corrected an issue in ComputeAverageVector where the Results Browsing Object was not properly counting the number of items being averaged when a column contained both filled and unfilled cells.

FEMAP v11.1 New Features and Corrections
Updates and Enhancements
Model Merge - New for FEMAP 11.1!

- The File, Merge command allows entities from any model currently open in the same instance of FEMAP to be "merged" with the active model. At least two models must be open for this command to be available. To facilitate bringing entities into the active model, a number of overall Renumbering and Duplicates Strategy, Entity Selection, and Model Orientation options are available in the Model Merge Manager dialog box.

Performance Graphics - New for FEMAP 11.1!

- Added "Performance Graphics" mode to improve performance of dynamic rotation and regeneration of large models. Performance Graphics may be turned on in File, Preferences on the Graphics tab. See Preferences for more information.

Views

- Added Performance Graphics Font button to the "Label Parameters" option in "Labels, Entities and Color" Category of View, Options command. The Performance Graphics Font button allows you to select a font when using Performance Graphics. The Font dialog box appears allowing you to select a Font, a Font Style, a Size, along with selecting a Script, if needed.
- Added Performance Graphics option in the "Tools and View Style" Category of View, Options command.
- Added Load and Save buttons to View Manager accessed via View, Create/Manage command. The Save button simply allows to save the view highlighted in the Available Views - Selected View is Active list to the View library, while the Load button allows you to load a view from the View library into the model.

Analysis Manager

- Added Design Optimization Options dialog box for NX Nastran and MSC Nastran. Only available for Design Optimization analysis type. Used to choose either Static and Normal Modes/Eigenvalue and offers a Track Modes option when set to Normal Modes/Eigenvalue.
- Added question to allow user to optionally clear the Loads/Constraints from the Master Case when using MultiSet to create subcases.
- Removed GEOMCHECK and Model Check dialog boxes from the Next/Prev chain for Nastran solvers to streamline creation of analysis sets. These dialog boxes are still available, but now must be selected via the Options branch of an analysis set.

Connection Properties, Regions, and Connectors

- Updated the Connect, Automatic command to allow more flexibility when selecting which type of connections to find as well as a new option when finding connections with edge contact.
- Updated all dialog boxes used to create the different types of regions to use "Add to List" and "Remove from List" icon buttons to add/remove single items from the list of entities. Also, the

Delete button in all of these dialog boxes is now used to select any number of entities using the standard entity selection dialog box to remove from the list of entities.

- Added support for using Solid Elements to define Bolt Regions using Connect, Bolt Region command

Geometry

- Updated Geometry, Midsurface, Automatic command to use Parasolid "face pairing" technology.
- Added Geometry, Curve - Line, Between Geometry command
- Upgraded Geometry, Surface, NonManifold Add command to use new "tolerant modeling" capabilities available to create "General Bodies" when using the Parasolid Geometry modeling kernel
- Added Geometry, Solid, Sweep command.
- Added Geometry, Solid, Sweep Between command.
- Added Add Washer option to Geometry, Curve - From Surface, Pad command.
- Updated Mesh Sizes, Loads, Constraints... option in various Geometry, Copy/Scale/Rotate/Reflect commands to now also include regions of all types.
- Replaced Modify, Project commands for points with single Modify, Project, Point command.

GUI - Toolbars and Icons

View Toolbar
- Added Distance Between Geometry icon to Measure icon menu.

Select Toolbar
- Added Layers/Groups in Tooltips icon to Selector Mode icon menu.
- Added Renumber command to context-sensitive menus for Solids, Connection Properties, Regions, Connectors, Coordinate Systems, Materials, Properties, and Layups. In all cases, a dialog box appears requesting ID and selected entities will be renumbered using the Original ID method of the Modify, Renumber... commands.

Post Toolbar
- Updated "Next Output Vector" and "Previous Output Vector" icons to increment all 3 possible contour vectors if they are defined. Previously only the primary vector was incremented.

GUI - Dockable Panes

Model Info Tree
- Added ability to toggle on/off the current count of various entity types using the Show Entity Counts icon in the Model Info toolbar.
- Added ability to "scroll" through entities using the middle mouse button while the cursor is over the Entity Icons or Visibility check boxes now available while in another command, except View commands.

- Added Renumber command to context-sensitive menus for Coordinate Systems, Geometry, Connection Properties, Regions, Connectors, Aero Panels/Bodies, Aero Properties, Aero Splines, Aero Control Surfaces, Materials, Properties, Layups, Load Sets, Load Definitions, Constraint Sets, Constraint Sets, Functions, Analysis Sets, Output Sets, Groups, and Layers. In all cases, a dialog box will appear starting ID and the selected entities will be renumbered using the Original ID method of the Modify, Renumber... commands.
- Added Attach to Results command to context-sensitive menu for Results, which displays the Manage Results Files dialog box from the File, Attach to Results command

Meshing Toolbox
- Locater tool - Added "With Poles" option when Search For is set to Surfaces. When on, the Locator will find any surface which contains a "pole". Typically only spherical or conical surfaces and planar surfaces of revolution around a point can have a "pole".
- Geometry Editing tool - Added "Split at Closest" option to the "Edge to Edge" Operation. When on, will attempt to create the shortest possible curve using the two locations on the selected curves closest to one another. Also, added Pad Alignment and Add Washer options to the "Pad" Operation.

PostProcessing Toolbox - Contour Tool
- Added No Average, Centroid Only option for Data Conversion in the Options section when Style is set to Contour. Simply allows all possibilities currently available through the menus and dialog boxes.

PostProcessing Toolbox - Freebody Tool
- Added ability to display freebody results in nodal output coordinate system. Only nodal vectors and quantities will be displayed in the nodal output coordinate system. Interface loads will still be in the freebody coordinate system.
- Added Allow Alternate Vector option in the Freebody Contributions section for the Applied, Reaction, and MultiPoint Reaction contributions. When on, attempts to use alternate vectors if the Grid Point Force Balance vectors are not available for a particular contribution.
- Added option to List Nodal Sums when using the listing commands in the Freebody Tools section. When used, summed values for Fx, Fy, Fz, Mx, My, and Mz at each node will be added to the listing using **SUM** as the Source.
- Added Freebody Validation icon to Freebody Tools section. Model debugging tool that can be used to help determine if results requested for a given freebody exist in the database for the selected set of elements and/or nodes. It does not indicate that the results of a freebody calculation are a valid idealization of the structure being analyzed, as that is up to the individual user to determine.

Charting Pane
- Reorganized the Chart Data Series dialog box into a tabbed format and added the Vector vs. Vector option as a tab.
- Added ability to set the Font Size for the Legend, Chart Axis Labels, and Labels in the Chart.
- Changed Show Tooltips icon into icon menu and added several options for syncing up the active graphics window to the data point currently showing the Tooltip and ability to control what is shown in the Tooltip.

- Changed Copy Chart to Clipboard icon into icon menu offering three options, Copy Chart Image, Copy Chart Data, or Copy Chart Image and Data. The most recently used option will remain the default option for the current instance of FEMAP.
- Moved the Title field from the Chart Title tab to the top of the in the Charting dialog box.
- Added List Data command to Data Series context-sensitive menu to list the data from the Data Series to the Messages window
- Added Show (Element/Node ID) in Active View command to Marker context-sensitive menu to highlight the entity in the active view using the current Show When Selected options, except always displays label.

Data Surface Editor
- Added Load Set Combination Data Surface to Create/Load Data Surface icon menu. See Loads and Constraints for more information.

Data Table
- Added Significant Digits options to Show/Hide Columns icon menu. Simply allows you to specify the number of significant digits to be displayed for values in the Data Table. The number of significant figures will persist until the Data Table is closed.
- Added Sum Selected Rows command to context-sensitive menu for column headers. Displays a dialog box with the Minimum Value, Maximum Value, and Sum using data from the rows currently highlighted.

Interfaces - FEMAP Neutral
- Updated Neutral Read and Write for v11.1 changes
- Added capability to include element connectivity definitions in the FNO file to create plot-only elements. The elements must reference existing nodes in the model and can only be lines, triangles or quads.

Interfaces - Nastran
- Added support for the pyramid element and now read/write the CPYRAM entry.
- Added read support for obsolete ELSTRESS, ELFORCE, ELSTRAIN Case Control entries. Turn on Stress, Force, and/or Strain output requests in the Nastran Output Requests dialog box.
- Added enhanced support for Restarts. New options include the ability to specify a Read Only Restart along with specifying a Version Number for the file and a Starting Subcase for Nonlinear analysis.
- Updated default behavior for auto reading of results when the analysis case is set to "3..Print and PostProcess" which will skip f06 results and only read output from op2 file.
- Updated automatic filename convention. Input file name now matches the entire length of model name instead of only using the first 5 characters of the filename. It also automatically converts characters that are known to be invalid in Nastran command lines (=, $, #) and converts them to '_'. Also added Base Filename for Analyze (Blank to Match Model) field in the NASTRAN Executive and Solution Options dialog box so you can specify a different automatic name, which will be followed by a 3-digit number which automatically increments.
- Added support for buckling setup where LOAD and METHOD are in the master case

Interfaces - NX Nastran

- Added support to write BEDGE entries when edges of axisymmetric elements, CTRAX3, CQUADX4, CTRAX6, CQUADX8 are used in a Connection Region in Edge-Edge contact.
- Added support for ESOPT field on BGPARM entry and PREVIEW field on BGPARM and BCTPARM entries. Removed subdivide method, RTSUBD, from NXSTRAT as it was removed from version 8.5.
- Added support to read ITER and ELEMITER commands from the NASTRAN statement which will select the appropriate Iterative Solver option in the NASTRAN Executive and Solution Options dialog box.
- Added support to request Grid Point Force results in Advanced Nonlinear (SOL 601).
- Added support to write BOLT entry using element IDs for Advanced Nonlinear (SOL 601).
- Added support to set up the PYR_AR, PYR_EPLR, and PYR_WARP element checks in GEOMCHECK dialog box for pyramid elements.
- Added option to request Relative Enforced Motion Results in Nastran Output Requests dialog box for dynamic analysis. Adds REL to DISPLACEMENT, ACCELERATION, and/or VELOCITY Case Control entires.
- Added support for KDAMP and KUPDATE fields on TSTEPNL entry. For Transient Heat Transfer analysis these are specified via the Method drop-down and the Include Differential Stiffness in Damping options in the Stiffness Updates section of the Nonlinear Control Options dialog box. For Nonlinear Transient analysis KUPDATE is specified via the Method drop-down in the Stiffness Updates section of the Control Options tab of the Nonlinear Control Options dialog box, while KDAMP is specified via the Include Differential Stiffness in Damping option in the Additional Transient Options section of the Advanced Options tab of the Nonlinear Control Options dialog box.
- Added support to read RMS von Mises Stress from a Random Response analysis with RMS output requested.
- Updated the NASTRAN Output for Random Analysis dialog box to allow selection of a specific option (Power Spectral Density Functions, Autocorrelation Functions, Root Mean Square), None, or All for both Nodal and Elemental results.
- Added Bulk Mod Ratio - GAMMA, Bulk Modulus Function, Density Function, Damping Coeff Function, and Bulk Mod Ratio Function fields to the "506..Nastran Fluid Material (MAT10)" Material Type in Other Types. These fields write the GAMMA, TIDBULK, TIDRHO, TIDGE, and TIDGAMMA fields to the MAT10 entry.
- Added support for reading output vector from XY plane and matching Radial, Axial, Azimuth directions for axisymmetric elements.

Interfaces - MSC Nastran

- Added read/write support for the SMETHOD Case Control command. When Iterative Solver in NASTRAN Executive and Solution Options dialog box is set to "1..On", writes SMETHOD=MATRIX. When set to "2..Elemental Iter", writes SMETHOD=ELEMENT.
- Added support for the BBBT formatted XDB files and XDB files created by 64-bit version of MSC Nastran via File, Attach to Results command only.

- Added read support for Acceleration and Velocity results in alternate output data blocks (OAG1 and OVG1).
- Added support for the MAT10 entry. Specified using "606...MSC.Nastran Fluid Material" when property type is set to "Other Types" in the Define Material dialog box.

Interfaces - NEi Nastran

- Added read support for plot only triangle and quad elements representing contact regions from FNO file.
- Added option to request Relative Enforced Motion Results in Nastran Output Requests dialog box for dynamic analysis. Adds REL to DISPLACEMENT, ACCELERATION, and/or VELOCITY Case Control entires.
- Updated modal and dynamic analysis methods to always write the EIGRL entry instead of EIGR, unless Real Solutions Methods is set to Auto and Normalization Method is set to Point in the NASTRAN Modal Analysis dialog box.
- Updated Advanced Options tab of Nonlinear Control Options dialog box to have all fields blank by default in the Analysis Set Manager.

Interfaces - ANSYS

- Updated Ansys to support V14.5 and 15.0 with new 64 bit pointers and new stress result block format

Interfaces - LS-DYNA

- Added support for Memory (Megawords) and Processor Count to LS-DYNA Analysis Control dialog box. These fields write *KEYWORD "Memory value" NCPU = "Processor Count value".

Interfaces - Comma-Separated

- Added support for the Extended Comma-Separated Format. This format must be used in order to attach CSV to FEMAP using the File, Attach to Results command.

Interfaces - Geometry

- Added support for Solid Edge with Synchronous Technology 6 and NX 9.0
- Added support for Parasolid 26.0
- Added support for ACIS 24, SP1
- Added support for CATIA V5-6R2013 SP2

Listing

- Added listing of UM DOF for RBE3 Elements

Loads and Boundary Conditions

- Enhanced Load creation to allow the use of Data Surfaces when defining loads of the following types: Force on Curve, Force Per Length on Curve, Moment on Curve, Moment Per Length on

Curve, Force on Surface, Force Per Area on Surface, Moment on Surface, Moment Per Area on Surface, Nodal Heat Flux on Curve, Nodal Heat Flux Per Length on Curve, Nodal Heat Flux on Surface and Nodal Heat Flux Per Area on Surface. For the load types that support it, this capability also works with or without the "Total Load" option.

- Enhanced load expansion from geometry to a mesh for elemental face-based loads on surfaces (Pressure, Heat Flux, Convection and Radiation). Previously on large models these could an extended period of time.
- Updated Model, Load, Combine command to allow a number of new options.
- Added Load Set Combination Data Surface to Data Surface Editor. This Data Surface has one very specific use, to create new load sets based on existing load sets. Each selected existing load set will appear in the Data Surface Editor as an individual column, while each new load set to create will appear as a row

Materials

- Added 606..MSC.Nastran Fluid Material (MAT10) to Other Types.

Properties

- Added Nastran Elbow/Pipe Options section to Curved Tube property.

Output and Post-Processing

- Updated the File, Attach to Results command to allow attaching to XDB files from NX Nastran and MSC Nastran (both formats) and Comma-Separated files using a specific format.
- Updated Model, Output, Transform command to use a dialog box similar to the one used to specify the "on-the-fly" transformations with View, Select and PostProcessing Toolbox. Also, this command now works with output in attached results files.
- Added List, Output, Freebody Nodal Summations and List, Output, Freebody Nodal Summations to Data Table commands.
- Updated "on-the-fly" transformations via the Select PostProcessing Data dialog box and PostProcessing Toolbox to support transformation of Laminate and Solid Laminate Stresses and Strains using the same options available for transformation of Plate Stresses and Strains.
- Updated "Next Output Vector" and "Previous Output Vector" icons on the Post Toolbar to increment all 3 possible contour vectors if they are defined. Previously only the primary vector was incremented.
- Added option when saving output to database from attached results file to optionally detach results file.

Element - Pyramid

- Added Pyramid as an Element Shape for Solid Element Type.

Element - Update Existing Elements

- Added Modify, Update Elements, Rigid DOF Command

- Added Along/Between Path option to Modify, Update Elements, Material Orientation command.

Meshing

- Added several new options to the Mesh, Geometry, Solids command to improve tetrahedral meshing
- Added Copy Method section to Mesh, Copy, Element command, with the default Copy Method being Along Vector which was previously the only option. The Normals and Normals with Thickness Correction options were added for planar elements only and work in the same manner as when extruding or offsetting elements.
- Added Along Vector to Surfaces option to Mesh, Extrude, Curve; Mesh, Extrude, Element; and Mesh, Extrude Element Face commands.
- Replaced Modify, Project commands for nodes with single Modify, Project, Node command. Has all the same options as the Modify, Project, Point command, described in the Geometry section.

Tools

Check, Coincident Nodes

- Added Merge Across Output CSys option to allow nodes with different output coordinate systems to be merged. Either the output coordinate system of the kept node or a selected coordinate system will be used after the merge.
- Added Label with ID option to Preview Coincident dialog box for Tools, Check, Coincident Nodes and Tools, Check, Coincident Points. This options is always on by default and will simply show the IDs of the nodes or points being highlighted, regardless of the options currently set for Window, Show Entities or the Show When Selected icon menus in the Model Info tree or Data Table

Check, Element Quality

- Added All On and All Off buttons and check boxes for Listing the details of elements that failed the checks as well as listing a new summary of the failed elements. Also added a "Show" button that shows all elements that fail the checks. Finally, added option to send the detailed listing to the Data Table instead of the Message Window.

Check, Sum Forces

- Updated Tools, Check, Sum Forces to allow you to select whether to Include Body Loads or not, as well as a Sum option to Sum All Loads, Loads on Selected Nodes and Elements, or Selected Load Definitions.

User Interface

- Enhanced the performance of Combo and List Boxes with lists of entities that have a large number of items. Also, enhanced performance of the Visibility dialog box. Previously, the dialog box could take longer than expected to appear when models had large numbers (50K-100K) of properties, materials, etc.
- Implemented Query and Front picking for Coordinate Systems. Only works for coordinate systems themselves, not selecting nodes or points referencing coordinate systems.

- Added Layers/Groups in Tooltips option to include Layer and Group information in Tooltips and Rotate View About submenu to specify View Center options from the quick access menu (right-mouse click menu).

General

- Enhanced Modify, Renumber... commands which allow Coordinate renumbering to allow the user to specify the Order using the +/- X, Y, or Z locations of each entity.

Preferences

Views
- Added individual fields for Line and Font to the Resolution Options dialog box accessed via the Resolution button in the Picture Save Defaults section.

Graphics
- Added Performance Graphics to Graphics Options section.

User Interface
- Added Show Angles as 0 -> 360 option to Menus and Dialog Boxes section
- Added Middle Button Click for OK option to Mouse Interface section
- Added Wheel Factor for Dynamic Planes option to Mouse Interface section
- Added Use Region Decimal/List Symbols option to new International Localization section

Database
- Updated default Max Cached Label value in Database Performance section to be 99,999,999 on 64-bit operating systems. Remains 5,000,000 for 32-bit operating systems.
- Updated Timed Save options. Saving a model now resets the Timed Save timer and command counter so that you will get a full interval until the next timed save

Geometry/Model
- Added preference to allow optimizing geometry on import.
- Replaced Pre-V10 Tet Meshing and Pre-V10 Surface Meshing options with single Pre-V11.1 Tet Meshing option in Meshing and Properties section.

Results
- Changed Automatically Attach to OP2 Files option to Automatically Attach to Result Files in File Options section.
- Changed Use Memory Mapped OP2 Files option to Use Memory Mapped Files in File Options section.

API

New API Objects and Attributes

- Added MapData Object
- Added NumberOfElements on Node Object
- Added ContourDataSelection, ContourVector1, ContourVector2, ContourVector3, ContourCornerData and ContourDataConversion to View Object
- Added NasNonlinKDAMP, NasNonlinKUPDATE, NasCnlkDAMP, NasRelEMotion, NasExecAnalyzeFilename, NasExecRestart, NasExecRestartVersion, NasExecRestartSubcase, NasOptimAnalysisType, NasOptimTrackModes, NasGCheckTest2, vNasGCheckTest2, NasGCheckTol2, vNasGCheckTol2, NasGCheckMsg2, and vNasGCheckMsg2 to Analysis Set Manager Object.
- Added NasCnlkDAMP to Analysis Case Object.
- Added LabelFontSize, AxisValueFontSize, LegendFontSize, AxisMinorCountLog, vAxisMinorCountLog, AxisRangePad, vAxisRangePad, AxisMinorCount, and vAxisMinorCount to Chart Object
- Added OutputVector2, Location2, and SortData to Chart Data Series Object
- Added AlternateApplied, AlternateSPC, AlternateMPC, and UseNodalOutputCS to Freebody Object

New and Updated API Methods
- Added Clear, Initialize, Count, GetIDs, Remove, RemoveAll, DefineReal, DefineInt, DefineBool, DefineString, DefineVariant, SetReal, SetInt, SetBool, SetString, SetVariant, GetReal, GetInt, GetBool, GetString, and GetVariant to MapData Object
- Added Reload2, AddAllSavedSetIDs, AddCoordinate2, AddAroundPoint2, AddAroundVector2, and AddAroundPlane2 to Set Object
- Added SetReal, GetReal, SetInt, GetInt, SetDataType, and GetDataType to Sort Object
- Added GetColumn, GetColumns, GetRowsAndColumns, GetRowsAndColumnsByID, and GetColumnSum to Results Browsing Object
- Added GetModel and SetModel methods to the Output Object
- Added IsGeneral to Curve Object
- Added IsBlend to Surface Object
- Added GetNodalSums and GetSumAtNode to Freebody Object
- Updated PutCombination for LoadSet and BCSet object to allow combinations using combinations

New and Updated Global Variables
- Added Pref_PreV111TetMesh, Pref_Show0To360Angles, Pref_PlaneWheelFactor, Pref_MiddleMouseButtonOK, Pref_GeomOptimize, Pref_UseRegionDecimalChar, Pref_ResFontScale, and Pref_PerformanceGraphics
- Updated Info_ActiveID to allow you to set the values to 0 to set no active entity

The following functions have been added or updated:
- feElemCountOnProp
- feMeasureDistanceBetweenGeometry
- feFileWriteIdeas
- feOutputProcessEnvelopeFromSets

- feViewAutoscaleAll
- feViewAutoscaleVisible
- feWindowSetSize
- feCheckSumForces (updated summedForces array to provide all values)
- feCheckSumForces2
- feCheckCoincidentNode3
- feRenumberOpt2
- feFileWriteFNO2
- feMeshExtrudeElement2
- feMeshTetSolid2
- feSurfaceOffsetTangent
- feSurfaceMidAuto2
- feMeshSmooth2
- feGroupGenerate2
- feFileAttachSave2
- feSurfaceNonManifoldAdd3
- feSolidSweep
- feSolidSweepBetween
- feProjectOnto
- feCompute3DInvariants
- feCompute2DInvariants
- feSetFree
- feSetFreeNotInSet
- feAppColorCreate
- feAppColorGetBaseColor
- feAppColorGetPattern
- feAppColorGetLinestyle
- feAppColorGetRGB

Corrections

Views

- Corrected issue using relative deformations. The z translation component was not taken into account. Additional bug when deforming relative to.
- Corrected issue where boundary Surfaces were not obeying the back face culling view options, giving bright lines at edges of back faces of solids
- Corrected issue where Freebodies were not included in min/max box calculation so they could get clipped by the near/far clipping volume.
- Corrected issue where freebody nodes not in the active group were not being included when calculating bounding box used by the View, Autoscale, Visible command.
- Corrected issue where CBUSH orientation vector did not use Reference Coordinate System, if active

- Corrected issue which occurred when animating and the left mouse button was clicked without moving the mouse, which caused the animation to stop.
- Corrected issue with Some nodes on the parabolic-element are not shown when the view style is "solid".
- Corrected issue if displaying a contour and then set the contour legend shrink percentage to 0.0 either thru View, Options, the PostProcessing Toolbox or the API. Corrected code and limited the available values for shrink to 1.0 thru 100.0 (PR# 6948674).
- Corrected issue in the View Visibility dialog box that caused the currently selected View and Constraint set to not be properly reloaded when the dialog box was opened again.
- Corrected an issue that caused saved, multi-view layouts to not be restored properly. The results were slightly wrong if using File, Open, the recent model list or drag-and-drop to open the file. They were significantly wrong when the file was double-clicked in Windows Explorer.
- Prevented repeated errors if logo or background bitmaps were selected using View, Background. Now gives error once and then disables logo or background bitmap to prevent further errors.

Connection Properties, Regions, and Connectors

- Corrected issue that prevented NonStructural Mass Regions defined on Curves from expanding to Beam, Bar, and other Line elements.
- Corrected issue in the NonStructural Mass Region dialog box that did not properly enable/disable the Surfaces and Curves options when you selected "Total Mass on Area" or "Total Mass on Length".
- Corrected issue When defining a connection region using the positive side of a surface. If the surface contained planar elements that had their normals reversed compared to the surface normal, the wrong element face was chosen. Expansion of the connection regions to element faces from surfaces is now done correctly regardless of the element normal.
- Updated the choice of positive/negative sense of surfaces on sheet bodies when creating connection regions to insure that all surfaces on the same sheet body use the same convention. Previously if a planar sheet body was basically coincident with its neighboring region, numerical roundoff could cause some surfaces to come out positive and others negative, which is always invalid. Everything is now consistent, however in this "coincident surface" case, there is no way to determine whether the sheet should be positive or negative.

General

- Corrected issue where Femap issues a "Unable To Save Model: Access is Denied" using File, Save when saving a model for the first time and you have attached to results using File, Attach To Results. Workaround is to use File, Save As.
- Corrected issue where quad faces were not being written to a JT file.
- Corrected issue When using File, Picture Copy or File, Picture, Save with a different number of texture levels for screen and hard copy, textures were wrong (PR# 6888214).
- Corrected issue when deleting from a Material or Property library by pressing the "Delete" button in the Library dialog box, while in "Load" during creating a Material or Property. Previously deleted the wrong entry in some cases even though it showed the correct one removed from the list box.

- Added a message to prevent a bad picture being saved when using File, Picture, Save and another window (for example, Paint) is brought up over it (PR# 6888214).
- Changed Undo/Redo file naming conventions and usage so that opening models with very long filenames (100+ characters) will not cause FEMAP to exit unexpectedly.

Geometry

- Corrected an issue where the underlying surfaces of a composite surface could get added to the selection set with "Add Tangent Surfaces" picking method.
- Corrected an issue where Geometry, Surface, NonManifold Add could create disconnected (Parasolid Disjoint) bodies. Although FEMAP was instructing Parasolid not to create these bodies, it could occur anyway. Added functionality to check for and then split up any disjoint bodies.
- Corrected an issue in Geometry, Curve - Line, Offset command to keep the section perpendicular to the drive curve so that the offset value was maintained.
- Corrected an issue which now deletes edge of wire bodies. Previous FEMAP deleted the whole wire body since wire bodies are really not supported, but in certain instances they can make it into the program.

Graphics

- Corrected issue where mesh size on point was not written in the same format as dialog box displays it. For example, if 1234567890 was entered, the graphics would display 1234568326 and the dialog when modifying would show 1.23457E9. The graphics now matches the dialog box.
- Corrected issue where display of stress in beam cross sections when using View, Advanced Post, Beam Cross Section command did not work if the default element type was not set to Plate, Linear or Plate, Parabolic.
- Corrected issue where surface normals were not being drawn on combined surface. This issue was introduced in v10.3.
- Corrected issue When displaying freebody with no contour or deformation. This caused the output set title not to be written to the post titles, only the ID.
- Corrected issue when using the View, Advanced Post, Contour Model Data to display Combined Quality as the element quality, as none of the required values were being calculated.
- Corrected issue where the width of contour legend was controlled by font size. This could make it really thin in some cases. Implemented limits based on 4% of screen dimensions so it would never get too thin.
- Corrected issue that caused Layers to not highlight from the Model Info tree.
- Corrected issue where the boundary label was incorrectly tied to the surface label switch. This has problem first arrived in 11.0.
- Corrected issue where if you reset mesh definition on a curve, the default mesh size is shown instead of nothing. This is now fixed so mesh size is no longer visible and no need to do a Ctrl-G to get correct image.
- Corrected issue drawing Aero Panels when the aerodynamic coordinate system was not aligned with the panel definition system. Updated to use coordinate system from Active Analysis Set and refresh graphics for Aero panels when changing analysis sets.

- Corrected issue for View, Advanced Post, Dynamic Isosurface; View Advanced Post, Dynamic Cutting Plane; or View, Advanced Post, Streamline commands, which caused the load set, constraint set etc stay as they were when the command was completed. Before this fix, if they were set to View Active, they were set to View Selected after completing the command (PR# 1960017).

GUI - Dockable Panes

Meshing Toolbox
- Corrected issues displaying quality when using blanking. Also added warning message for mixed Solid/Shell displays.

Model Info Tree
- Corrected issue when copying a Load Definition using the "Copy to Set" command on the context menu. If the load was a bolt pre-load on a region, previously it would be moved to an incorrect region in the new load set. It is now copied correctly (PR# 1946467).
- Corrected issue when using the Model Info Tree to edit the entities inside a Load or Constraint definition then editing the definition vales from the entity editor before reselecting/reloading the definition in the editor (PR# 6956839).

PostProcessing Toolbox
- Corrected issue where the "Continuous Colors" option was missing from the Contour Levels group when set to "User Palette". It is now available.
- Corrected issue that prevented Additional Output Vectors from showing up in the Contour tool if the Contour Type was set to Nodal

Charting Pane
- Corrected issue where chart may not update when switching between multiple models if the tab started off pinned and was made floating or started off floating and was made pinned.
- Corrected issue where messages printed out when data series are added to / removed from the current chart when using the add/remove button in the data series manager were printed as error messages. Changed to normal messages since it is not an error.
- Corrected issue when chart value max/min are the same, the axis does not get drawn properly. Offsets max + 1 and min - 1 for these situations.
- Corrected issue when editing a chart via the Chart Manager, using Edit Selected button). If editing the chart currently displayed, changes would not get drawn.
- Corrected issue where the Chart Title in the drop-down would not be in sync with the chart title right after it changed when editing through the Chart Settings dialog box.

Data Surface Editor
- Corrected issue when copying a Load Definition using the "Copy to Set" command on the context menu. If the load was a bolt pre-load on a region, previously it would be moved to an incorrect region in the new load set. It is now copied correctly (PR# 1946467).

Interfaces - FEMAP Neutral

- Corrected an issue migrating of RBE3 UM DOF. The data was in the neutral file but the element check stopped FEMAP from reading it. This also fixes possible problems creating RBE3 elements with UM DOF from the API as well as File, Merge command (PR# 6879268).
- Corrected a problem with the FEMAP V11.0 Model/Neutral File Converter that prevented the active view from being transferred properly. All model information was transferred, however the correct view was not activated when the old model was opened.
- Corrected a problem writing a FEMAP Neutral File that caused the last property value (which isn't used by FEMAP, but is accessible via the API) to always be written as 0.0 no matter what its actual value was. This correction is made in FEMAP and all Neutral File converters back to V9.3 where the problem began.

Interfaces - Nastran

- Corrected issue reading FREQ1 and storing maximum frequency and interval. Also fixed FREQ2 default NF for number of Number of logarithmic intervals.
- Corrected issue when writing NASTRAN command where the field separator was sometimes not written between options.
- Corrected issue reading Time Increment for Nonlinear Transient analysis where it was read and written as a negative time step.
- Corrected issue where FEMAP could write an empty load set in the LOAD card. It was just ignored by Nastran and wouldn't cause any problems.
- Corrected issue reading bar output in XDB file using the File, Import, Results command.
- Corrected issue writing TEMPD too many times for dynamic solution.
- Added warning when a SUBCOM references subcases with thermal loads which are not supported by Nastran. The loads need to be in the SUBCOM case (PR# 6670306).

Interfaces - NX Nastran

- Corrected issue reading Hyperelastic Axisymmetric Stress/Strain output (PR# 6869164).
- Corrected SOL 601 LOAD to match standard Nastran LOAD and place SPCD in the LOAD set rather than a referenced load set.
- Changed to explicitly write all local BGPARM fields.

Interfaces - NEi Nastran

- Corrected graying of Dynamic Loads using LOADSET/LSEQ switch for NEi Nastran (PR# 2220278)

Interfaces - ANSYS

- Corrected issue when translating BEAM188 elements, where if previously used material was not the same material as on the element, no material entry was written. Now MAT command gets written out when the previous material written out is not the same as the current material or if a SECDATA card is written.

- Corrected issue when translating offset plates, where the material entry would not get written if the SECDATA card didn't get written (could happen if the last plate was a non-offset plate with a different material). Now MAT command gets written out when the previous material written out isn't the same as the current material or if a SECDATA card is written.

Interfaces - DYNA

- Corrected issue where Femap was writing *MAT_ISOTROPIC_ELASTIC_PLASTIC but should have been writing *MAT_PLASTIC_KINEMATIC for isotropic hardening for shell elements. Also fixed problem writing *MAT_PLASTIC_KINEMATIC for kinematic hardening where Femap was writing the beta Hardening.
- Corrected issue when writing material 181. An extra blank space was added to the beginning of the last line which is now corrected (PR# 6886383).
- Corrected issue where RBE2 was not translated correctly (PR# 6886621).
- Corrected issue when exporting pressure loads on axisymmetric elements (PR# 6823743).

Interfaces - Geometry

- Removed checking of "translator codes" for STEP and IGES interfaces. Previously the "FEMAP Neutral Only" versions had failed to read/write STEP and read IGES. Now there are no "translator code" restrictions on any geometry translators.

Listing

- Corrected issue saving newly created formats from GUI.
- Corrected issue that caused the header for List, Model, Nodes; List, Geometry, Points; and List, Model, Coord Sys to come out on multiple lines if you specified a coordinate system for the listing.
- Corrected an issue in List, Output, Compare. If two sets were being compared and one or more of the vectors did not exist in the second compare set, it was incorrectly listed as not existing in the first compare set.
- Corrected an issue with the List, Model, Element command when listing Rigid Interpolation (RBE3) elements. Previously the listing was incorrectly labeled, where dependent nodes were labeled as independent and vice versa. (PR# 6899564).

Loads and Boundary Conditions

- Corrected an issue that caused reflection of loads to be incorrect in cases where the reflection plane was not aligned with an axis of the load coordinate system. Also changed reflection of Moments, Rotational Displacements/Velocity/Acceleration to reflect in a manner that caused a symmetric loading, and no longer reflect in the same manner as a Force/Displacement/.../ i.e. not as a vector.

Materials

- Updated Material libraries to correct values of specific heat.

Meshing

- Corrected an issue that caused FEMAP to stop functioning when meshing certain surfaces with Quad Edge Layers when the mesh size was very large compared to the surface being meshed.
- Corrected an issue that could cause FEMAP to exit unexpectedly if attempting to mesh surfaces that only had 2 curves on their outer boundary (like a circle), contained similar circular interior holes and the mesh size was set so large that there was only one element per curve.
- Corrected an issue related to the removal of Geometry Preparation so that the "un-prepared" solid would have its mesh sizes automatically recalculated if taken directly to meshing without further preparation or sizing.
- Corrected element extrusion to not pick up beam offsets (and potentially other data) from the elements being extruded in the new elements that are created.
- Updated Mesh, Extrude/Revolve/Sweep Elements and Element Faces commands so that if Linear elements/faces are selected, linear elements are created, while if Parabolic elements/face are selected, Parabolic elements are created.
- Updated the Mesh, Editing, Edge Split command to handle nodal permanent constraints. Now only the common permanent constraints on the edge end nodes are applied to the new split node.

Output and Post-Processing

- Corrected incorrect vector titles for solid laminate shear stress/strain output
- Corrected op2 attach issue reading PSDF output
- Corrected op2 attach issue where Ply Vectors not found on the first output table were never indexed (PR# 6945305)
- Corrected op2 attach issue in Buckling where the Output Sets Generated were not labeled the same as when the results are internalized. This could occur when the buckling solution did not use subcases (PR# 6939741)
- Corrected op2 attach issue where Title, Subtitle, Label text were not being written to the Femap Output Set notes.
- Corrected op2 attach issue with Bar EndB results which caused beam diagram to not be drawn.
- Corrected an issue that occurred if a model was reopened that had attached NEiNastran results files (FNO) and the "Use Memory Mapped" files preference was enabled. In this case, the results were opened correctly as a memory mapped file, however all access to the file was done using non-memory mapped techniques. In all cases, the correct results were obtained, it was simply a potential performance issue. This did not occur when the file was originally attached, only if the model file was saved and reopened.
- Corrected Global Ply/ Sold Composite output problems
- Corrected issue when deleting a portion of the output sets from a single result file (PR# 1948208)
- Corrected issue where laminate plate transformation was not working for global plies (including top and bottom).
- Corrected issue where solid laminate transformation was not working for Global Plies.
- Corrected issue where quite a few options were not implemented for Laminate Solid Results in various commands.

- Corrected issue to that could cause clipping with freebodies that have a total sum vector outside of the model box.
- Changed Model, Output, Process to allow "Convert" to work after renaming Output Sets.

Elements - Rigid

- Corrected issue that caused invalid elements if you edited RBE3 UM DOF degrees of freedom on an existing element and deleted all of the DOF (PR# 6960684)

Tools

- Corrected issue that caused the thickness on Global Plys to not be updated when using Tools, Convert Units. All layups were properly converted previously. The Global Ply thickness is only used as the default thickness when a global ply is used to create a new layup ply.

User Interface

- Corrected issue with the Contour Vector dialog box. If "3D Component" is selected and the 3 components were selected for Vector 1, everything worked correctly. However, if the dialog was exited and reentered, the 3rd component of Vector 1 was set equal to the 2nd component of Vector 1. Vectors 2 and 3 did not have this problem.
- Corrected the labeling in the 3D Orthotropic Material Ply/Bond Failure Dialog Box tab for the Tensile Stress/Strain Limits. They were labeled 12, 23, 13, when in fact they should have simply been 1, 2, 3
- Corrected issue with picking in the view when the Model Info tree is floating but not visible (PR# 1948407).
- Corrected issue in External Superelement Creation dialog box to allow for null values in the "Entity ID Range Checks" fields. Previously, if values were in these fields and then cleared out, an error about being less than min label was displayed.
- Corrected issue if adding a Superelement reference, deleting it, and adding an additional one before closing the External Superelement Reference dialog box, the numbering may not be sequential.
- Corrected several graying issues in the External Superelement Reference dialog box. The check boxes at the bottom would not un-gray until the list box was clicked, even if there were already references in the model.
- Corrected issue which would cause the Create Assign button to "float" in the Analysis Text dialog box when the dialog box was resized vertically. This dialog box was accessed by the Start Text button in the NASTRAN Executive and Solution Options dialog box.
- Corrected issue where the Subspace method in the NASTRAN Modal Analysis dialog box was not being grayed for NX Nastran.
- Corrected issue where the Additional Transient Options on the Advanced Options tab of the Nonlinear Control Options dialog box were not being grayed when setting up a nonlinear static analysis.
- Removed unused options from the renumbering commands dialog box for Aero Panels/Bodies, Aero Properties, Aero Splines, and Aero Control Surfaces

API

- Corrected issue that caused Combined Load and Constraint sets to lose their combination sets and factors if you simply called Get() and Put() on a combined set.
- Corrected issue where user created panes could have improper tab titles when using the feAppRegisterAddInPane call.
- Corrected the maximum dimension of the Info_ModelSizeX, Y & Z and vInfo_ModelSizeX, Y & Z API parameters. The correct dimension, as previously documented, is 2 since each contain the minimum and maximum coordinates in the appropriate coordinate direction.
- Corrected issue in the API Curve object that caused the mesh count to be returned incorrectly (actually with encoded length and bias information) if MeshCount() was called on a curve that did not have a custom mesh size. This issue could cause wrong sizes to be applied to curves if a composite curve was split in the Meshing Toolbox, and could cause considerable delays in remeshing because of excessively large numbers of elements added to the curve.
- Corrected the API commands feViewCascade() and feWindowCascade(). Previously these API calls simply did nothing.
- Corrected issue which issued incorrect error message that appears when >100 locations are not mapped using API.
- Enhanced Results Browsing object to better handle the case when attempting to get data from the object but no data was present (now returns FE_NOT_EXIST), or if attempting to call one of the PutXXX functions with no data to add (returns FE_OK, but handled better internally).

FEMAP v11.0.1 New Features and Corrections
Updates and Enhancements
Connection Properties, Regions, and Connectors
- Model, Delete, Mesh now automatically deletes any Connection Regions where all elements and/or nodes were also deleted by the command. It also automatically deletes any Connections that have had one or both of its Connection Regions deleted.

Geometry
- Added multi-profile and multi-path to Geometry, Surface, Sweep command.
- Added Status Bar Update to Geometry, Surface, Non-manifold Add command to provide feedback when command is non-manifold adding together a large number of sheet bodies.
- Enhanced Non-Manifold Add command to propagate Surface Mesh Attributes onto new surfaces created by surfaces being split by other surfaces during the command.
- During mid-surfacing operations, the thickness attribute information should propagate to surfaces created by the operations.

GUI - Dockable Panes

Charting Pane - New for FEMAP 11
- The number of minor tics will default to 8 when using a logarithmic scale instead of using the user-supplied value. This gives a more usable plot by default.
- Data Series of type "vs Function" are now displayed in the legend with the function ID. Format is "Name (#)" instead of "#..Name" to avoid ambiguity with Data Series IDs

GUI - Libraries
- Added disclaimer to the Library dialog that states users should verify supplied data before using.

Interfaces - Nastran
- Enhanced export of CONM2 to skip writing line 2 continuation if all moments of inertia = 0.0

Interfaces - NX Nastran
- Added support for property based-gluing of composite elements. Supported in version 8.5.

Interfaces - MSC Nastran
- Added support for reading the Offset Vector Interpretation Flag (OFFT) for CBAR, CBEAM, BAROR, and BEAMOR. Elements using this flag will now be oriented properly in FEMAP. Also, added warning for unsupported CBAR, CBEAM, BAROR, BEAMOR OFFT field.

Meshing

- Added the ability to revolve parabolic shells that have an edge on the axis of revolution into parabolic wedges or parabolic tets. Revolving linear shells still does not support creation of tets if a single node lies on the axis.

API

- Added SetAdd2, SetAddOpt2, and Add2 methods to Group API object. Work exactly like SetAdd, SetAddOpt, and Add methods, except use Group Data Type instead of Entity Type for the first argument.

Corrections

Licensing

- Corrected issue in license borrowing where Femap would not write checksum in Nastran file when using a borrowed license.

Views

- Corrected issue where Boundary Surfaces were being drawn on internal surface edges as part of filled edges. The issue also affected highlighting while picking and has also been fixed.

General

- Corrected an issue which caused meshes and connections to be deleted without warning if the Delete, Geometry, Solid command was used, multiple solids selected, and the first (lowest ID) solid did not have a mesh and/or connections.

Geometry

- Corrected issue in Geometry, Surface, From Mesh where the command tries to identify a simple four-sided surface and could potentially create the wrong surface. If a four-sided surface is identified, the command creates a boundary surface, then converts that boundary to a surface. Command was rewritten so the conversion portion of the command works for more cases.
- Corrected issue when Mesh sizing on curves created by the Geometry, Surface, From Mesh command was incorrect.
- Corrected issue in Geometry, Midsurface, Automatic where the original solid being mid-surfaced would be intersected with the newly created mid-surfaces, thus created additional extraneous surfaces.

Graphics

- Corrected and issue where no free faces are found when a solid mesh is "skinned" with shells. For version 11.0.1, change has been made to only process solid elements. Full implementation for "mixed" models expected in a future release.
- Corrected issue where solids were not being faceted properly after using Geometry, Solid, Thicken command (PR# 6845658)

GUI - Dockable Panes

Charting
- Corrected issue in Neutral File import that caused charts to be read incorrectly when multiple Charts in a model reference the same Data Series.
- Corrected issue that caused the charting pane to sometimes be blank after using the Window, Tile or Window, Cascade commands and there are multiple models open.
- Corrected issue where Charting pane may not update when deleting output sets. Charting Pane will now refresh if the has the Update Views option on, which is the default.
- Corrected issue where Data Series that have a scale factor may sometimes report a title that has the scale factor as "ERROR:LIM OVERRUN.
- Corrected issue with Justification of the Y-axis title. This was a limitation of the Toolkit used to create the Charting pane, which has since been addressed.
- Corrected issue which caused FEMAP to exit unexpected when choosing "Chart Options" from the context-sensitive menu while running on 32-bit VMWare. (PR# 1934324)

Meshing Toolbox
- Corrected a memory leak when combining and remeshing already meshed surfaces using the meshing toolbox.
- Corrected issue in Mesh Sizing Tool where the "Bias Factor" option was not available when "Spacing" was set to "Biased, Use Pick Location".

PostProcessing Toolbox - Freebody
- Corrected issue where freebody vectors on a deformed model were using deformed location for transformation, not underlying undeformed locations.
- Corrected issue where freebody vectors were incorrect when model was displayed with element shrink enabled. Coordinates of "shrunk" corners were being used to do transformations, instead of actual coordinates.
- Corrected issue where freebody data printed to the Messages window or sent to the Data Table from Freebodies which referenced a cylindrical or spherical coordinate system would only return results in the rectangular system. Did not affect displayed vectors.

GUI - Entity Selection
- Corrected issue when picking solids using selection area (box, circle, polygon, freehand) when Pick All Inside option was being used. Previously, if one surface of a solid was completely inside the selection area it would be selected, now all surfaces must be in the selection area in order to be selected.

Interfaces - FEMAP Neutral
- Corrected issue writing "Custom" Divisions for Aero Panel/Body and Aero Property to the neutral file that caused some of the list to be skipped.

Interfaces - Analysis Manager

- Corrected issue where pressing the "Next" button when Case was set to SUBCOM (MSC or NX Nastran only) did not bring up the SUBCOM factor dialog box.

Interfaces - Nastran

- Corrected issue writing ACCEL where improper extra line could be written.
- Corrected issue when reading input file with EXTSEOUT card. The ASSIGN statement would be duplicated on the subsequent translation if the original input file that was read in had no other text in the executive section start text other than the ASSIGN statement.
- Corrected issue where editing the files in the External Superelement Reference dialog box may cause the EXTID to be set to zero

Interfaces - DYNA

- Corrected issue where the translator would stop writing user-defined coordinate systems once a non- rectangular coordinate system was encountered. (PR# 6842347)
- Corrected issue where writing of cylindrical constraints in an equivalent local rectangular system was not working properly. (PR# 6842347)

Interfaces - STAAD Interface

- Corrected issue in STAAD Write, fix uninitialized "append" option.

Interfaces - Geometry

- Corrected issue when exporting VRML file where post-processing data (contours, deformations, titles, legends, etc) was not being properly included (PR# 6858293)

Loads and Boundary Conditions

- Corrected issue in Create Constraints on Geometry dialog box where the first item in the "Arbitrary in CSys" drop-down was displayed as "-1..Select Output Vectors to Delete" instead of "-1..Use Nodal Output CSys". (PR# 6864272)

Elements

- Corrected issue where the twist checking/fix-up of elements check was disabled during their creation of elements. This check was changed for V11.0 and the new check has been fully enabled.

Materials

- Corrected material databases to correct Stainless Steel densities and change specific heat values to consistent units in the SI and mm-N-tonne databases

Properties

- Corrected inconsistency when picking a surface to use as a General Section in the Cross Section Definition dialog box in a Bar/Beam property. When using the "Alternative" option, user was prompted for the Y vector using (0,1,0) as the default, while "Original" prompted with a default of (1,0,0). "Alternative" now matches "Original".
- Corrected issue where in the General Section was selected, a surface was picked etc, but then canceled out of the dialog. If the section dialog box was then reentered, the section would be set to none and if you selected general section again, if you selected general section again, it draw the old section which you had canceled from.

Meshing
- Corrected issue that occurred in Mesh, Editing, Edge Split command if a mesh was split which went across multiple geometries (for example, a hex mesh across multiple solids). Previously associativity to some elements adjacent to the transition between the different geometries was lost because the boundary nodes were not associated with both geometries.
- Corrected memory leak in Tet Meshing which occurred if the tet quality table was to be sent to the Data Table. All tets that were not added to the Data Table leaked memory.
- Corrected issue where cross-sections were not being shown correctly on curves when a Curve Meshing Attribute was set and cross-sections were being displayed.

Output and Post-Processing
- Added hard-coded error message in Model, Output, Transform command to warn that it does not support results in attached files and to inform the user they can use the Transform buttons in Select PostProcessing dialog box display transforms.
- Cleaned up output to deal with situations when there were vectors in both attached files and the database. Previously caused FEMAP to go become non-responsive if a user vector was created in an attached set, then it tried to create a second. Also fixed dialog so the first user vector immediately appears in the list when it is created.
- Corrected issue where Contour Legend was reversed when the user palette was selected
- Corrected issue when using View, Advanced Post, Beam Cross Section command where the Y moment was being used as the Z moment and vice versa. Only caused an issue when displaying output where only the Y or Z moment were being used individually, not both. (PR# 1923682)
- Corrected an issue that caused internalized (not attached) vectors that computed elemental total vectors (For Nastran, Total Temperature Gradient and Total Heat Flux) to display incorrectly.
- Corrected issue in 2D/3D Contour vectors. The second (Y) component was being computed incorrectly.
- Corrected an issue in Contour Vectors that caused the vectors to be incorrect if two vectors were chosen that required loading the same underlying data multiple times and the results were in an attached file. Specifically, if attempting to display Max Prin and Min Prin stress the resulting directions were incorrect. Only occurred for attached files.
- Corrected inconsistency in the Contour Vector dialog where the "0..None-Ignore" option was lost in the vector selection combos if the type of vector plot was changed. Also, corrected issue that caused it to look like vectors were selected after the type was changed even though they were not.

- Corrected issue reading solid composite failure indices and strength ratios. Issue only affected solid composite results, not shells.
- Corrected an issue which caused FEMAP to exit unexpectedly if a model was contoured and all but one element was deleted from the model.
- Corrected issue which caused very poor performance if user switched from a Full Model display of nodal contour of elemental data to displaying an empty group. On 500,000 plate model took ~7 minutes, now instantaneous
- Corrected issue when renumbering output in attached output files that caused the renumbered cases to overwrite other cases. Also, cleaned up delete of individual Femap output case in attach so a set cannot be accessed once deleted by Femap.
- Corrected issue problem where the Femap output set would still exist when reopening a model that was saved after a subset of output cases were deleted from an attached file. Also added logic to clear the undo file when renumbering or deleting attached output sets.
- Added correction to sort modal output cases by Femap set ID, instead of by frequency.

User Interface
- Corrected issue that caused FEMAP to exit unexpectedly if a user attempted to open an invalid library, then tried to pick from the original library.
- Corrected issue which caused poor scaling of icons when using 125% or 150% text scaling in Windows 7. Issue was addressed with new version of toolkit used by GUI.
- Corrected an issue which could occur when trying to solve a Heat Transfer Analysis using Nastran and the user was asked to save the model. Occurred sporadically based on timing issues in the NX Nastran Analysis Monitor.
- Corrected the "Tip of the Day" to automatically cycle to next tip at each startup. Previously stayed on same tip.

API
- Corrected an issue in NextVector() of the Results Browsing Object that would skip over other user vectors if starting with a user vector and checking for the next existing vector below that one.
- Corrected issue in GetEntityData where Results Browsing Object would choose the wrong output processor due to a improper check for number of vectors in the table.
- Corrected an issue when using the API Property object with Spring or DOF Spring properties that could cause the database counters to be invalidated.
- Corrected Entity API method GetTitleIDList(). For CSys it did not get CSys 0 if specified a start/ stop range of 0,0. Also got an extra coordinate system if only the global coordinate systems existed in the model.
- Corrected issue getting the element face normal of the triangular faces of parabolic wedge elements. Broken since v10.0.
- Corrected issue with vcomponent in the Femap Output object that would cause Femap to become unresponsive if called.

FEMAP v11.0 New Features and Corrections
Updates and Enhancements
Views
- Updated View Select dialog box of View, Select command
- Added "Element - Material Direction" option in "Labels, Entities and Color" Category of View, Options command. Used to display the material orientation on shell or solid elements. Previously, material orientation on shell elements could be viewed by using the "Show Orientation" option of "Element - Orientation/Shape", but the functionality has been replaced by the "Element - Material Direction" option.
- Added "Reverse" option to "Load Vectors" option in "Labels, Entities and Color" Category of View, Options command. Allows Force and Moment loads to be shown with the head of the load vector on the node, point, or curve instead of the tail.
- Changed "Render Options" to "Graphics Options" in "Tools and View Style" Category of View, Options command.
- Added "Contour Options" button to "Contour/Criteria Style" option in "PostProcessing" Category of View, Options command to access the Select Contour Options dialog box. Also, removed the Data Conversion section as all functionality can be found in the Select Contour Options dialog box.
- Removed "Auto-Group" Level Mode from "Contour/Criteria Levels" option in "PostProcessing" Category of View, Options command. Also, added "Auto Max Min" button to automatically retrieve the Minimum and Maximum values from the currently displayed Contour Output Vector to facilitate use of the "Max Min" Level Mode.
- Removed all "XY" options from the "PostProcessing" Category of View, Options command. All XY plotting is now done using the Charting pane.

Analysis Manager
- Added External Superelement Creation dialog box for NX Nastran and MSC Nastran. Only available for Static and Normal Modes/Eigenvalue analysis types. Used to create the EXTSEOUT entry. Also, the dialog box offers an alternate location to designate the Master (ASET) and QSET DOF sets and optional checking capability.
- Added External Superelement References dialog box for NX Nastran and MSC Nastran. Available for all analysis types. Used to create references for a Superelement "Assembly" model by selecting External Superelement file (Type = .OP2, .OP4, or .PCH), setting Superelement ID, choosing Matrices options (.PCH file only), and choosing ASM file (.OP2 and .OP4 only).
- Added Create ASSIGN button to the Analysis Text dialog box accessed via Start Text button in NASTRAN Executive and Solution Control dialog box. Allows for streamlined creation of ASSIGN statements.

Connection Properties, Regions, and Connectors
- Added Connect, NonStructural Mass Region command.
- Added support for using nodes of Solid Elements to define Bolt Regions using Connect, Bolt Region command.

Functions
- Added 1 new function type, Acceleration vs. Location, for use with Varying Translational Acceleration Body Load.

Geometry
- Added Geometry, Surface, From Mesh command.
- Added "Non-Manifold Add Tolerance" to Geometry, Surface, NonManifold Add command

GUI - Toolbars and Icons
File Toolbar
- Added Attach to Results icon.

Panes Toolbar
- Added Charting icon.

GUI - Dockable Panes

Charting Pane - New for FEMAP 11
- Added the Charting pane for XY plotting of Data Series.

Model Info Tree
- Added ability to use Visibility check boxes while dialog boxes from other commands are open.
- Added ability to "scroll" through entities using the middle mouse button while the cursor is over the Entity Icons or Visibility check boxes.
- Added NonStructural Mass Region command to context-sensitive menu for Regions.
- Added Copy to Set and Move to Set context-sensitive menus for Load Definitions and Constraint Definitions.
- Added Envelope context-sensitive menu for Output Sets. Creates a pair of output sets, one a results envelope, the other set info, based on the selected output sets and Type (Max Value/Min Value/Max Absolute Value)
- Added icon in Results section to differentiate between Output Sets which are attached ("file in folder") vs. internalized ("file with gear in the lower left corner").
- Updated "Compare" context-sensitive command on Results in Model Info Tree to do a full compare of all set combinations.

Meshing Toolbox
- Mesh Sizing tool - Added "Biased, Use Location" option to allow spacing on selected curves to be biased towards the point on each curve closest to a specified Bias Location (XYZ coordinates).

PostProcessing Toolbox
- Added Data Selection drop-down to Contour tool.

- Added Additional Vector(s) check box to Contour Tool, along with two additional Output Vector drop-downs to select the Additional Vector(s).

GUI - Entity Selection

- Added "Add Connected Elements" Pick option when selecting Elements

Interfaces - FEMAP Neutral

- Updated Neutral Read and Write for v11.0 changes

Interfaces - Nastran

- Added support for ACCEL entry defined using Varying Translational Acceleration in Body Load
- Added support for PCOMPS entry defined using Solid Laminate Property Type
- Added support for MATFT entry for Solid Laminate elements and PCOMPS defined using Ply/ Bond Failure tab of Isotropic, Orthotropic (3D), and Anisotropic (3D) Material types.
- Added support for NSM1, NSML1, and NSMADD entries defined using NonStructural Mass Regions. Also, added support for import of NSM and NSML entries to create NonStructural Mass Regions.
- Added support for PARAM,BAILOUT,-1 entry by click BAILOUT in the NASTRAN Bulk Data Options dialog box of the Analysis Set Manager.
- Added support for EXTSEOUT entry defined using External Superelement Creation dialog box in the Analysis Set Manager for Static and Normal Modes/Eigenvalue analysis types.
- Added support for PARAM, SECOMB entry defined using External Superelement Reference dialog box in the Analysis Set Manager. In addition, the DPBLKTOL=#.# is also written when Duplicate Node Tolerance is turned on and a value is provided in the field.

Interfaces - NX Nastran

- Added support for PBUSHT entries for Advanced Nonlinear Static and Advanced Nonlinear Transient Analysis Type.
- Added support for Linear Contact entries for Buckling Analysis
- Added support for using the nodes of Solid Elements to create Bolt Regions. Writes the BOLT entry with ETYPE=2 along with appropriate values for CSID, IDIR, and G(i) fields.

Interfaces - NEi Nastran

- Changed default for Real Solution Methods in NASTRAN Modal Analysis dialog box from Lanczos to Auto.

Interfaces - ANSYS

- Added support for "Standard" and "Reduced Integration" formulations when exporting plate elements.
- Added support to always write KEYOPT(3) = 3 for all beam elements using the "BEAM188/ Section Shape" or "BEAM188/ASEC" formulations.

- Added Skip Preprocessor Command option to ANSYS Model Write dialog box.
- Added Write All Groups as Components option to ANSYS Model Write dialog box.

Interfaces - LS-DYNA
- Added support for initial velocity in rigid body material definition (*INITIAL_VELOCITY_RIGID_BODY)

Interfaces - Geometry
- Added support for NX 8.5, Solid Edge with Synchronous Technology 5, and Solid Works 2012
- Added support for Parasolid 25.1
- Added support for ACIS 23, SP1
- Added support for CATIA V5-6R2012 SP3
- Added support for Pro/Engineer Creo 2

Loads and Boundary Conditions
- Added "Varying Translational Acceleration" to Model, Load, Body.

Materials
- Added Ply/Bond Failure tab for Isotropic, Orthotropic (3D), and Anisotropic (3D) Material Types.

Properties
- Added Solid Laminate Property Type

Layups
- Added AutoCreate check box to Layup Editor. When on, automatically creates a Global Ply for every New Ply added to the Layup.

Output and Post-Processing
- Added the File, Attach to Results command.
- Updated the View Select and Select PostProcessing Data dialog box of the View, Select command.
- Added "Multiple" to the List, Output, Compare command to provide multi-set and/or multi-vector comparisons
- Added Freebody Options dialog box to List, Output, Force Balance Interface Load and List, Output, Force Balance Interface Load to Data Table commands. Used to select freebody contributions when listing freebody data and not using freebody objects.

Element - Solid Laminate
- Added Solid Laminate Element Type

Meshing

- Added the Mesh, Editing, Edge Split command.
- Added the Set Element Size on Next Use option to Mesh, Mesh Control, Default Size command.

User Interface

- Updated FEMAP to take advantage of Vertex Buffer Objects (VBOs) to improve performance of dynamic rotation in larger models. VBOs need to be turned on in File, Preferences on the Graphics tab using the "Vertex Array" drop-down. Once on, define the Max VBO MB and Min VBO B settings to maximize performance.
- Updated the dialog boxes which are used to select Output Sets and Output Vectors for a number of commands.
- XY Plotting has been removed from Views and is now done using a combination or Chart and Chart Data Series entities in the Charting dockable pane.

General

- Added [DriveMap] to FEMAP.INI file. Used to map Unix/Linux style include paths during Nastran Read to local Windows paths. Format is UnixPath=WindowsPath. Can have as many mappings as needed, and simply searches in order until it finds a file.

Preferences

Views
- Added Save Iconified Views to Picture Save Defaults section
- Added Dynamic Min/Max in Post Data Dialog to Options section
- Removed Alternate Fill Mode from Options section

Graphics (previously Render)
- Added Trailing Zeroes to Graphics Options section
- Added "Vertex Buffer Objects" option to Vertex Arrays drop-down in Graphics Option section, along with Max VBO MB and Min VBO B which are used only when "Vertex Buffer Objects" has been selected.
- Changed "Print Debug Messages" and "All" check boxes found in the Advanced/Debug Options section into one drop-down list containing "0..No Debug Messages" (previously "Print Debug Messages" turned off), "1..Main Debug Messages" (previously "Print Debug Messages" turned on), and "2..All Debug Messages" (previously "Print Debug Messages" and "All" both turned on).
- Added Frame Rate option to Advanced/Debug Options section.
- Added VBO Messages drop-down to Advanced/Debug Options section.

User Interface
- Added Enable Legacy XY Plotting option to Dockable Panes section

Database
- Added Cleanup During Save (immediate) option to Database Options section

Interfaces
- Added Preserve INCLUDE Statements option to Nastran Solver Write Options section.
- Moved Compute Principal Stress/Strain and Assume Engineering Shear Strain options from General Solver Options section to the General Solver Options section of the Results tab.
- Moved Read DirCos for Solid Stress/Strain option and Auto Answer Post Questions button from Nastran Solver Write Options section to the Nastran Options section of the Results tab.

Results - New for FEMAP 11
- Added Preserve INCLUDE Statements option to Nastran Solver Write Options section.
- Moved Compute Principal Stress/Strain and Assume Engineering Shear Strain options to the General Solver Options section from the General Solver Options section of the Interfaces tab. Also, moved Read DirCos for Solid Stress/Strain option to General Solver Options section from the Nastran Solver Write Options section of the Interfaces tab.
- Moved Auto Answer Post Questions button to the Nastran Options section from the Nastran Solver Write Options section of the Interfaces tab.

Library/Startup
- Added Chart library

API

- Added Chart (feChart) object to the API. Also, added Style, Title, TitleVisible, TitleLocation, TitleJustification, TitleFontSize, TitleFontBold, TitleFontItalic, TitleFontUnderline, TitleFontColor, Font, Palette, PaletteFromChart, DarkBackground, LegendVisible, LegendLocation, vLegendLocation, LegendDirection, AxisAutoscale, vAxisAutoscale, AxisStyle, vAxisStyle, AxisRange, vAxisRange, AxisVisible, vAxisVisible, AxisMajorGridlineVisible, vAxisMajorGridlineVisible, AxisMajorInterval, vAxisMajorInterval, AxisMajorTickmarkVisible, vAxisMajorTickmarkVisible, AxisMinorGridlineVisible, vAxisMinorGridlineVisible, AxisMinorCount, vAxisMinorCount, AxisMinorTickmarkVisible, vAxisMinorTickmarkVisible, AxisZoom, vAxisZoom, AxisTitleX, AxisTitleY, AxisTitleVisible, vAxisTitleVisible, AxisTitleJustification, vAxisTitleJustification, AxisTitleFontSize, vAxisTitleFontSize, AxisTitleFontBold, vAxisTitleFrontBold, AxisTitleFontItalic, vAxisTitleFrontItalic, AxisTitleFontUnderline, vAxisTitleFrontUnderline, AxisTitleFontColor, vAxisTitleFontColor, AxisLabelDecimal, vAxisLabelDecimal, AxisLabelFormat, vAxisLabelFormat, MarkerFromChart, MarkerVisible, MarkerSize, MarkerStyle, LabelFromChart, LabelVisible, ShowLabelX, ShowLabelY, ShowLabelMaxMin, OutputSetInherit, OutputSet, OutputSetEnd, and OutputSetAll to Chart Object.
- Added Chart Data Series (feChartSeries) object to the API. Also added Title, SeriesType, OuputSet, OutputSetEnd, OutputSetAll, OutputVector, Interval, FunctionID, Group, Position, PositionCoordSys, Location, Scale, ColorInherit, Color, MarkerInherit, MarkerVisible, MarkerStyle, MarkerSize, LabelInherit, LabelVisible, ShowLabelX, ShowLabelY, and ShowLabelMaxMin to Chart Data Series Object.
- Added Results Browsing Object (feResults) object to the API. Also added ValueForNonExisting to Results Browsing Object.

- Added Layer to Freebody Object.
- Added Superelement Reference (feSEReference) object to the API. Also added ReferenceFile, ReferenceType, ReferenceID, ASMFile, Stiffness, Mass, ViscousDamping, StructuralDamping, and LoadsMatrix to Superelement Reference Object.
- Added NasBulkPARAMBailout, NasExtSEOutOn, NasExtSEOutExtID, NasExtSEOutMatStiffness, NasExtSEOutMatMass, NasExtSEOutMatK4damp, NasExtSEOutMatLoads, NasExtSEOutTo, NasExtSEOutUnitID, NasExtSEOutAssignTitle, NasExtSEOutOptGeom, NasExtSEOutOptAsmbulk, NasExtSEOutOptExtbulk, NasBulkSecomb, NasExecAllowDuplicateNodes, NasExecDuplicateTolerance, and NasExecWriteSERefPath to Analysis Set/Analysis Manager Object.
- Added AnsModSkipPreprocessorCMD and AnsModGroupsAsSets to Analysis Set/Analysis Manager Object.
- Added BoltDir, BoltType, MassType, and MassNSM to Region Object.
- Added MaterialOrientType and Material CSys to Element Object. Also, MaterialOrientType now supersedes MaterialAngleFlag.
- Added BodyVaryingAccelOn, BodyVaryingAccelAxis, BodyVaryingAccelFunction, and vBodyVaryingAccel to Load Object.
- Added Prev, PrevBefore, and PrevID to Set API object
- Added AddAroundPoint, AddAroundVector, and AddAroundPlane to Sort API object
- Added AddDataSeries, RemoveDataSeries, GetDataSeries, and ShowPane to Chart API object
- Added GetXY and Copy to Clipboard to Chart Data Series API object
- Added to Clear, GetModel, and SetModel to Results Browsing object - General Object Methods
- Added to NumberOfSets, NextSetReset, NextSet, SetExists, SetInfo, SetTitle, SetNotes, SetLocation, SetHasVectorsInDB, and Sets to Results Browsing object - Set Methods
- Added to NextVectorReset, NextVector, VectorExists, VectorInfo, VectorTitle, VectorComponents, VectorLocation, VectorEntities, and Vectors to Results Browsing object - Vector Methods
- Added to EntityValue to Results Browsing object - Entity Value Methods
- Added to AddColumn, AddEmptyColumns, AddConversionColumn, AddEnvelopeColumn, DataNeeded, SetTri3Orientation, SetTri6Orientation, SetQuad4Orientation, SetQuad8Orientation, SetSolidOrientation, SetEngrShearStrain, SetNodalTransform, SetPlateTransform, SetSolidTransform, SetColumnGlobalPly, Populate, IsPopulated, DataLocation, NumberOfRows, NumberOfColumns, FindColumn, GetColumnTitle, IsApiColumn, GetColumnVector, GetColumnMinMax, Reset, Next, GetValue, GetInRow, GetMultipleInRow, GetRow, GetRowByID, GetRows, GetRowsByID, SetValue, SetInRow, and SetMultipleRow to Results Browsing object - Review Methods
- Added CalculateSummation2 to Freebody API object
- Added IsMidsideNode to Node object
- Added Pref_RenderTrailingZeroes, Pref_RenderMaxVBOMB, Pref_RenderDebugFrameRate, Pref_RenderVBOUsage, Pref_RenderCaptureIconified, Pref_RenderMinVBOB, Pref_PreserveIncludes, Pref_MemoryMappedFiles, Pref_AutoAttachResults, Pref_ViewPostDynamicMaxMin, Pref_CleanupModel, Pref_LegacyXYPlotting, and Pref_LibChart.
- Added Info_MeshSizeAutoDefault

- Added DialogAutoSkipping
- Added feSelectOutputSets, feSurfaceFromMesh, feModifyElemReverse2Added, feFileReadCatia2, feFileAttachResults, feFileAttachManage, feFileAttachInfo, feFileAttachSave, feFileAttachByOutset, feMeshEdgeSplit, feMeasureDistanceBetweenSolids, feSurfaceNonManifoldAdd2, feFileWriteFNO, feChartPaneGetDisplayed, and feChartPaneSetDisplayed functions
- Updated API to support printing of Charts from feFilePrint
- Updated IsTitledEntity() to include feFreebody, feChart, and feChartSeries objects

Corrections
Views
- Removed Data Conversion from Contour/Criteria Style in View Options and added button to get to Contour Options in its place.
- Corrected issue related to material direction for solids. FEMAP was using the first node as the location for transformations but it should have been using centroid.
- Removed View, Regenerate if cancelling out of preferences dialog.
- Reduced number of View, Regenerate when switching between groups and output sets. Previously any change triggered a View, Regenerate. View is now only regenerated when view needs to be modified or the results are reloaded.

Connection Properties, Regions, and Connectors
- Corrected issue highlighting individual curves and nodes in connection regions
- Corrected issue property based contact regions so they do not include properties from previous contact regions

General
- Corrected issue on Windows 7 when choosing paper size and number of copies when printing.
- Corrected issue printing to PDF files (and possibly other printer types) if the printer type was changed in the FEMAP Printer Setup dialog
- Change Spaceball rotate about axis to use any dominate twist input
- Changed main fonts to all truetype and added ability for user to override any font family

Geometry
- Updated Reflect/Rotate Surfaces, Volumes and Solids to properly Reflect/Rotate attached Curve Attributes (cross sections).
- Corrected an issue where a spline could exceed "max points on spline". Simply creates mutliple splines when limit is reached.

Graphics
- Corrected issue when using Trace, no deformed model, and Model Color

- Corrected issue when showing shell thickness. If any thickness other than the first is zero, all values are set to the first.
- Corrected issue highlighting plates with no thickness when thickness is on - they were being drawn as thick and no XOR was visible due to double draw.
- Corrected issue with free face. If there were coincident shell elements (not in free face) and hidden line draw style was used, the nodes attached only to shells not in free face list would not appear
- Corrected contours disappearing when selector is on and model rotated. Occurred when using match output mode, as it prevented results being obtained from the nodal connectivity object successfully
- Changed OpenGL timing messages to be "Warnings" instead of "Errors" . This means they are blue not red and they do not effect the error counting.
- Corrected issue to check line graphics existence. Previously, FEMAP was incorrectly checking face graphics existence before line graphics draw. This caused problems in wire frame mode and perhaps surfaces that failed to facet
- Corrected issue so min/max of boundary surface evaluates correctly
- Corrected issue due to geometry data being deleted twice as usage count not incremented/ decremented correctly
- Corrected issue in multiset animation (would have happened with enough frames in ordinary animation) where the machine runs out of memory but FMEAP was not terminating the animation. Animation now terminates if enough memory not allocated.
- Corrected a crash in facetting caused when reading a sheet solid that had many circular holes tangent to another larger central hole (Part which showed issues was from Catia via STEP and had duplicate points at the tangencies)
- Updated Undo to redraw all windows and to simply redraw at end of undo/redo rather than undo/ show undone/undo redraw. This Corrected issue where the Post Titles disappeared after an undo because the OGL records were removed in the final undo of the automatic redraw
- Corrected issue missing curves when reading an iges file. Curve data had not been loaded into min/ max box and then frustum culling failed

Groups and Layers

- Corrected issue where Connector, Connection Region and Connection Property references within Groups where not being updated when the entities were renumbered. (PR 1891996)

GUI - Dockable Panes

Data Table and Entity Editor
- Corrected an issue with the Data Table that prevented the Entity Editor from being updated properly if the Data Table was unlocked and entities were added one at a time using the selector.
- Corrected issue labeling column row headers for local coordinate systems

Meshing Toolbox
- Improved Washer command in Geometry Editing toolbox so mesh approaches are set after the face is split. This keeps the outer surface from getting a 4 corner mesh.

- Corrected issue problem making Mesh Quality plot when the view was set to Show Active but the group id was a -1. Now simply does whole model.

Model Info Tree
- Corrected an issue that caused the "Next" item to not be reloaded in the Model Info tree for entities that exceeded the "Max Entities" value if the tree was ever reloaded - either manually or thru a File Rebuild, or any other command that caused a full rebuild (PR 2193991).
- Corrected an issue that occurred when creating/changing loads or constraints in sets that were not in the Model Info Tree because they were beyond the More/Prev limits. Previously they went as orphans to the top of the tree and potentially caused crashes. Solved by simply eliminating the More/Prev limits for Load Sets and Constraint Sets
- Corrected issue where total summation loads created using Model->Load->From Freebody may not appear in the Model Info tree.
- Corrected issue copying aero panels from the tree.

Interfaces - Analysis Manager
- Corrected an issue when editing a child branch of an analysis set.
- Corrected an issue when expanding a branch of the tree when the Alt key was depressed.

Interfaces - Nastran
- Corrected issue importing CBUSH elements where the element Orientation CSys and Spring/Damper Location were lost. The problem is that this bdf contains duplicate property IDs, Femap renumbers the property reference on the spring element but fails to correctly propagate the renumbering to the Orientation Coordinate System (PR # 6789713)
- Corrected issue updating CTE on RBE3
- Corrected issue to calculate the Checksum properly when writing a multi-line INCLUDE statements.
- Corrected issue where Femap in a SOL 101 analysis would write an invalid TEMP(BOTH) command when a Material Reference Temperature, Temperature loads and Initial Conditions where specified (PR 6632876)
- Corrected issue importing INCLUDE file where a nested path was relative to the Nastran run directory.
- Corrected reading frequency spread on FREQ4
- Corrected reading thermal constraint load due to a problem skipping unsupported output. Added check and error message for unsupported XDB formats (PR 6752083)
- Corrected issue writing DLOAD in SOL 145 that caused the Load set identification number to be invalid.
- Improved translator to skip PSDF output data blocks in the f06 file. (PR 6711978)
- Corrected issue where Femap repeatedly issues the same warning message indicating contact is unsupported (PR 6719244)
- Corrected issue with error messages for quad/tri elemental corner thickness.

- Corrected issue where Elemental GPF output with a output coordinate system was not being transformed properly when importing results from the XDB file.
- Corrected issue problem reading modes/buckling results from XDB file.

Interfaces - NX Nastran

- Corrected issue ordering BLSEG edge contact when defined on plot only lines. Also, Improved ordering of element edges in BLSEG when T junctions or duplicate edges are found
- Corrected issue problem writing NXSTRAT TNSLCF default (Blank field was 1 in NXN 6.0) now it is 0 in NXN 8.0   Updated the default when creating new analysis set and when writing and reading the bdf.

Interfaces - MD/MSC Nastran

- Corrected issue writing wrong axisymmetric element for NEI when using a Hyperelastic material. Also, corrected issue problem where Femap was checking for the wrong plane (PR 6777178).

Interfaces - ANSYS

- Corrected issue that caused Femap to crash when writing a model with Bolt Regions that contained no entities (PR 6775187).
- Correct issue which now allows FEMAP to read modal load steps arbitrarily written to the rst file. In some cases if FEMAP is unable to determine the difference between a Modal and Random Response solution, FEMAP will ask that user which solution was performed (PR 6534686)
- Corrected multiple issues setting shell element formulation for SHELL63/SHELL181 (PR 1918174)
- Corrected issue of writing MCOMB,PSD. MCOMB,PSD removed from Ansys in v5.4 and replaced with PSDCOM PSDCOM,PSDd, which is now written.
- Corrected issue where IYY and IZZ values were being stored in the opposite locations for BEAM188/ASEC elements.
- Changed Ansys write of shell181 to always write the new and recommended sections method. This stops Ansys from issuing warning messages and in some cases trying to apply the section to subsequent element types in a mixed model. Also, added support for read of SECBLOCK for shells only.
- Corrected issue problem reading SHELL181 sections when importing Ansys cdb blocked or unblocked format. Also, added support for reading sections (shells and beams when the cdb is in blocked format)

Interfaces - ABAQUS

- Updated translator to send the title from the Analysis Case Manager over to the translator so that the *HEADING gets written out (ER 5579564)

Interfaces - DYNA

- Corrected issue where DYNA plate formulations were overwriting Standard and Reduced Integration plate formulations by moving the Dyna plate formulations to a different location in the FEMAP database. Added a conversion to the neutral file to the new Dyna formulation.

Interfaces - I-DEAS Universal File

- Corrected issue reading time function record from dataset 792

Interfaces - PATRAN

- Changed default extension for reading/writing Patran neutral files. Per ER 6731043, Patran changed from .pat to .out in 2008 or before.

Interfaces - Geometry

- Updated Solid matching/replacing process to better work with parts that have no titles provided. This was done for the benefit of the SolidWorks interface.
- Corrected issue of selected units not being used by JT files.

Listing

- Corrected issue that occurred if the List, Destination had a File specified, did a listing, then the file was opened in Excel (or any other program that opened with exclusive access). If the file remained open in the other program and as a destination, then doing another listing would cause FEMAP to exit unexpectedly.
- Changed terminology in List, Element command to Independent/Dependent instead of Master/ Slave for Rigid elements.
- Corrected List, Element command as it did not list nodes for RBE1 style rigids. Also, updated Tooltips for Rigid and Slide Line elements to properly reflect all nodes, DOF, coefficients and element type.
- Changed format of List, Layer command and the visible layer listing in List, View command
- Changed format of header and detailed listing in List, Output, Compare command.

Loads and Boundary Conditions

- Corrected issue where body loads were not turned on/off with the loads button on visibility toolbar
- Corrected issue where FEMAP was allowing Model, Load, Combine to combine loads into a NXN load combination set. This could cause many extra loads to accumulate (PR 1907493)
- Corrected issue with solid element mapping of output to loads. While not wholly incorrect, the inverse distance weighted algorithm was found to not provide exact results. Updated algorithm to more robust version with greater accuracy.

Properties

- Corrected control of shear/warp/torsion and optimized routine for Nastran Cross-sections

Meshing

- Corrected issue when sizing surfaces that had vertex-only loops, which caused unexpected exit.
- Corrected issue with Mesh, Edge Members sometimes getting the wrong direction for a line element, resulting in bad orientation/offset
- Corrected Mesh, Reflect to automatically reflect planar element material angle and material coordinate system
- Corrected issue with finding the normal of thin walled, convex, curve-only boundaries (like hat sections) where the line used to determine the midpoint used to verify the surface normal cut thru the boundary and the midpoint was on the wrong side of the boundary. This could cause elements to be created with their normals facing the opposite direction as the boundary.
- Corrected issue with Mesh Size when splitting a composite curve at one of its existing internal points.
- Corrected issue when using Mesh Splitting where the element shape counts were incorrect when splitting a quad into triangles.
- Corrected issue with setting Mesh attributes on curves where new attribute could not be specified.

Output and Post-Processing

- Changed Contour Max/Min to never include the centroidal values
- Corrected precision issues with freebody total summation values that may have shown round-off errors when dealing with values less than 1e-6.
- Corrected issue where a freebody summation point specified in a non-global coordinate system would not display correctly. In addition, if the a user specifies a non-global definition coordinate system for the freebody summation location, it is now saved in the model.
- Corrected issue where user-defined levels contour changing when it should not as it should have been identical to min/max option
- Corrected user-defined max level to take account of continuous/level contours and user defined palette
- Corrected issue in the tooltip for stress recovery locations in shells
- Corrected issue where creation of modal contribution functions for Model, Output, Forced Response for "mode against frequency" were being done in the wrong order

Tools

- Corrected issue in Tools, Check, Planar that caused an invalid plane to be selected if plane was not manually specified.
- Corrected an issue in Tools, Check, Coincident Nodes (Safe Merge). If not merging, but only listing and "Make Groups" was selected, the titles of the groups were swapped, "To Keep" was "To Merge" and vice versa.
- Corrected an issue in Tools, Check, Coincident Nodes (Safe Merge). If Preview was used, then changes made to the options like the Move To Location, the original location when was used when OK was pressed, not the updated location. Now uses the updated options.

User Interface

- Corrected a problem with Yes/No dialogs. If Esc was used, the return code from the message was Cancel - it is now No.
- Corrected problem with Entity Selection dialog that caused 0 not to be selected when picking coordinate systems from the entity list icon (PR 1891998)
- Prevented Model, Output, Vector command from allowing the "New Vector" to make new vectors outside of the user vector ranges. Also automatically reset the ID if you change the complex type
- Changed Model, Output, Vector to have "None Active" and "Done" like the other "Set" dialogs - instead of OK/Cancel. Also, corrected issue of graying so that it was consistent with the others and correct
- Changed "Single... button on rigid element dialog to "Single RBE2"
- Corrected issue in "Undo" of "Previous Command". It previously took two undos to undo this command, now only one is needed.
- Corrected issue where highlighting and marking do not work properly after use of ctrl+c
- Corrected issue where if Messages window was not open, "Show Mouse Tracking" will not cause FEMAP to exit unexpectedly.

Preferences

- Changed the group naming conventions if Create Groups from Include Files is on to have case sensitive names, not just uppercase.

API

- Corrected issue in the Inside method of the Surface API object when using it with "curve-only" boundary surfaces
- Corrected problem with 64-bit type library registering incorrectly
- Corrected a problem with APIs that used feSolidFillet() and feSurfaceMidAuto() that did not allow the midsurfaces to be extracted because of an internal set overwrite
- Corrected a problem with the AddOutput() method of the API Data Table object. Previously the 3rd argument (nNewColumnID) was never filled
- Corrected a problem that caused integer arrays to fail unless passed as a Variant to several API methods. Also rewrote the GetOutputListAtSet() and GetOutputListAtID() methods of the Output object to improve performance. Previous performance was poor if the IDs/Set contained IDs that did not exist in the output vector. New method can be 30x-40x faster
- Updated Get on CSys API object to work properly with global coordinate systems
- Made changes so some existing APIs that only modify the CFemapOutput object properties will work as they did pre v11
- Corrected issue feFileRead* methods to prevent a crash if no model is open.

FEMAP v10.3.1 New Features and Corrections
Updates and Enhancements

Views

- Added "Right-Hand Rule First Edge" option to the Normal Style of the "Element - Directions" option in "Labels, Entities and Color" Category of View, Options command. Much like the "Right-Hand Rule" option, except the arrow points from the first node to the second node.
- Added Reverse button to the Coord Sys, Connection, Aero Spline/Control Surface, Material, Property, and Layer tab to View, Visibility command. This will "Hide" all entities of a particular type which are currently visible, while "Showing" the ones which are not currently visible.
- Added additional information to the Post Titles when Contour Style is set to Contour, IsoSurface, Section Cut, or Vector. Depending on the plot, this may include information about the contour being a nodal or elemental contour, the averaging method used, the Vector Type for Contour Vectors, etc. To see some of this additional data, the "Legend Style" for the "Post Titles" option of View, Options should be set to "2..Titles and Average Data".

GUI - Toolbars and Icons
Select Toolbar

- When using the "Create Group..." command from the "Selector Actions" menu of the Select Toolbar, the user is now able to select any existing group when using the "Add to Group", "Remove from Group", or "Exclude from Group" options. Previously, these options only worked with the "active" group in the model.
- When Solid, Region, Connector, CSys, Material, or Property is the "active" entity in the Select Toolbar, the context-sensitive menu now includes a Visibility submenu, which contains 5 commands to change the visibility of selected entities

GUI - Dockable Panes
PostProcessing Toolbox

- In the Freebody tool, added "Select Free Edge Nodes" icon button to Freebody Nodes section. This allows the user to quickly and automatically choose the "free edge" nodes of the selected elements when Display Mode is set to "Interface Load".

Data Table

- Added ability to "Add Output Columns" to allow user to select a specific output set (From Output Set drop-down) for selection of output vectors. Same capability also added for List, Output, Summary to Data Table and List, Output, Results to Data Table commands.

Model Info Tree

- Added "Show/Hide Reverse" command to the "Visibility check box" context-sensitive menu for Coordinate Systems, Geometry, Regions, Connectors, Aero Model (Panels/Bodies, Splines, and Control Surfaces), Elements (By Type and By Shape), Materials, Properties, and Layers. This will

"Hide" all entities of a particular type which are currently visible, while "Showing" the ones which are not currently visible.

Data Surface Editor
- Enhanced Data Surface Editor input dialog boxes to provide real number input with significant digits equal to that of the "grid" in the Data Surface Editor (PR#6588366).

Interfaces - Nastran
- Added support for importing files with truncated INCLUDE statements

Interfaces - I-deas
- Updated I-deas read translator to support new (2002, I10 and beyond) changes to Universal Formats 790 and 791 for nodal constraints, nodal forces and face pressures.

Interfaces - Geometry
- Added support for Parasolid 24.1
- Updated Parasolid capability to properly handle multi-byte (Japanese) characters in filenames. PR 2180402
- Added support for ACIS 22, SP1
- Added support to optionally read or skip "Free Points" during import of an IGES file.

Loads and Boundary Conditions
- Updated Model, Load, From Freebody command. Allows loads to be created from any number of Freebodies in the model across any number of output sets. One additional feature, is that only forces and or moments which would be displayed if the Freebody is active will become loads in the new load set(s).

Tools
Check, Coincident Nodes
- Added "Move Only, No Merge" option to Action drop-down in Tools, Check, Coincident Nodes command.

Meshing
- Added Delete All button to dialog box of Mesh, Mesh Control, Custom Size Along Curve command.
- Added ability to specify a different Property when using the Mesh, Copy/Radial Copy/Scale/ Rotate/Reflect, Element commands. Default is "0..Match Original". Only elements which share a common topology with typical elements of the selected property will be changed. All other elements will retain their original properties.

User Interface

- Added ability to reselect invalid file names by converting them to the short form of the file name which was available until FEMAP 8.3
- Added Random... button to Color Palette dialog box when using the Modify, Color commands for Point, Curve, Surface, Solid, Coord Sys, Node, Element, Material, and Property. Offers 3 different methods for assignment of random colors (Multiple Colors By ID, Multiple Colors By Type, and Multiple Colors By Group).

Output and Post-Processing

- Added ability to "Override Vector View Options" directly from the Contour Vector options dialog box. Previously, this option could only be toggled on/off using the "2D Tensor Plot View Options Override" option in the Views tab of the File, Preferences command.
- Added ability to use Entity IDs (Element, Material, or Property) when plotting a Contour, a Criteria Plot, or a Beam Diagram when using the View, Advanced Post, Contour Model Data command.
- Added ability to "Rank" selected results in the Data Table using the List, Output, Results Ranking to Data Table command.
- Added List, Output, Force Balance Interface Load Summary command to allow comparison of a single Freebody across a number of output sets or a number of Freebodies using the results of a single output set. Optionally, FEMAP can automatically create functions for Forces (FX, FY, FZ, and Total) and Moments (MX, MY, MZ, and Total) across output sets or Freebodies.

Preferences

Database

- Added Recover _DBData File... button to off a different method to use when attempting to recover a corrupted model file.

API

- Added HasList and CountList Methods for Group object
- Added feModifyColorMultiple function
- Added feFileRecoverDBData function
- Rewrote the GetOutputListAtSet and GetOutputListAtID methods of the Output object to improve performance. Previous performance was especially bad if the IDs/Set contained IDs that did not exist in the output vector. New method can be 30x-40x faster.
- Updated the feFileReadIgesAdv function to reads both "free points" and "free curves" if the read_curves option is on.

Corrections

Licensing

- Corrected an issue that caused "Token" licensing to appear for certain features when "Show Users" was selected with Network licensing

General

- Corrected an issue where printing on Windows 7 with a printer which was not the default was causing an number of issues, including not being able to create .pdf file with Adobe PDF Printer.
- Corrected an issue where multi-line edit control in text entry dialog boxes (i.e., pressing Enter key) conflicts with CTRL+M dialog box shortcut key.
- Corrected an issue where List, Model, Aero/Panel command listed Pt1 twice instead of Pt1 and Pt4.
- Corrected an issue where selection of Aero Mesh boxes using the Select Aero Mesh icon button in the Aero Spline dialog box was unavailable unless the Aero Panel/Body had Custom divisions.

Views

- Corrected an issue where the Max/Min values displayed in the graphics window were always shown using the contour colors, thus not following the setting for Label Color for the Contour/ Criteria Legend option in View, Options. This was only an issue in FEMAP 10.3.
- Changed how FEMAP determined the Max/Min values to display in the graphics windows. Previously, only elements in the "Free Face" list were considered, now all elements are considered.
- Corrected an issue that caused the Contour Legend to show the wrong values after a model was dynamically rotated following use of the File, Picture, Save. The issue would only occur when the Level Mode was set to 1..Auto Group for the Contour/Criteria Levels in View, Options. If these same conditions existed and the File, Picture, Copy command was used, the wrong values would appear in the image sent to the clipboard. This issue has also been fixed.

GUI - Dockable Panes

Data Table

- Corrected an issue where copying rows to clipboard could cause FEMAP to because of uninitialized memory.

Messages Window

- Corrected an issue with Message Window that caused it to "hang" if a filename (or any other text) was echoed that contained a { or }.

Program File

- Corrected an issue with Program File record and playback for "ListView" controls, especially multiselect and those with check boxes next to the list items (PR 6619183)

Interfaces - Analysis Manager

- Removed Function icon button next to "Flutter Method" that was not needed in NASTRAN Flutter Parameters dialog box in the Analysis Manager.

Interfaces - Nastran

- Corrected an issue reading include files when INCLUDE was last line in bdf.

- Corrected an issue where duplicate EIGx was written in a flutter model that contained dynamic loads
- Corrected an issue where dynamic PARAMs were written twice when using the "Use Load Set Options" switch on the Dynamic Control Options dialog box in the Analysis Manager.
- Corrected an issue reading Modes/Buckling output in xdb file.

Interfaces - ANSYS

- Corrected an issue reading elastic strain output for shells and solids (PR# 1867277).
- Corrected an issue with BEAM188 elements being created with end releases when keyopt 3/4 is specified
- Corrected an issue with beam 188 offset direction of T-sections.

Interfaces - LS-DYNA

- Corrected an issue when exporting plate elements with a formulation which has a higher value than "10..Belytschko-Wong-Chang" (PR# 6627229).

Output and Post-Processing

- Corrected an issue in List, Output, Force Balance Interface Load command Previously, if no freebody objects were in the database, the "old" force balance method was used, which would erroneously report MPC forces. Now the new "method" is always used.
- Corrected an issue when using the Envelope tab of the Model, Output, Process command, where when enveloping individual vectors to automatically reassign the enveloped component vectors to the centroidal vectors if you choose to envelope all of the original component vectors.
- Changed several of the column headers when using the list commands inside the Freebody tool or the List menu to better correspond with the names of certain options in the Freebody tool.

User Interface

- Corrected an issued with the Stress Wizard where it was not display properly. Had been issue for several releases.
- Corrected an issue with Stress Wizard and Analysis monitor upon closing FEMAP. Previously, if you clicked in client area then hit X, or if you undocked, re-docked then hit X the panes would cause FEMAP to unexpectedly exit.
- Corrected a problem that caused TMG and SAToolkit toolbars to be duplicated if you alternated between opening FEMAP directly and double clicking filenames, if FEMAP was installed in a directory that had a long style (not 8.3) path.

Preferences

Database

- Corrected issue which caused selection of incorrect Open/Save option after adding 16K test in FEMAP 10.3.

API

- Corrected issue in the Surface API Cylindrical() method
- Corrected a problem that caused integer arrays to fail unless passed as a Variant to several API methods.

FEMAP v10.3 New Features and Corrections
Updates and Enhancements

Views
- Added on View, Align by, Surface to align the view normal to a selected surface and View, Align by, Normal to Plane to align view normal to a specified plane.
- Only tabs of entity types which currently exist in the model will be displayed in the View, Visibility dialog box.
- Added Max Only and Min Only options to Contour/Criteria Style option in PostProcessing category of View, Options.
- Added Preview Option to Tools and View Style Section of View, Options.

Analysis Manager
- Updated Preview Analysis Input File dialog box to show 80 characters per line by default.

Connection Properties, Regions, and Connectors
- Added Activation Distance section to Penetration section on the NEiNastran tab. Allows you to specify a value (real or AUTO) for MAXAD or specify values for MAXNAD and/or MAXRAD)
- Added Friction section to LS-DYNA tab to restore ability to set these values for LS-Dyna contact.
- Updated Fluid Regions to not use the PLANE1, PLANE2, RMAX, FMEXACT inputs when NEi Nastran is default solver.

Functions
- Added 2 new function types, Mach Number vs. Freq and vs. Areodynamic Factor, for use with Aeroeleastic Analysis Types in the Analysis Set Manager.

Geometry
- Added capability to embed multiple solids into the base solid when using Geometry, Solid, Embed.
- Improved tolerancing of Modify, Extend command. Previously failed on some lines where they were being extended to intersection locations (like Modify, Join command) when they were at large coordinates.

GUI - Toolbars and Icons
New 10.3 Toolbar
- Added Aeroelasticity Toolbar. Contains overall visibility controls (Draw Entity check box) of the Aero Panel, Aero Mesh, Aero Spline, and Aero Control Surfaces options in the Labels, Entities and Color section of the View, Options command.

Mesh Toolbar
- Added Mesh Geometry Preparation icon.

GUI - Dockable Panes

PostProcessing Toolbox
- Added Freebody tool to control all facets of Freebody display post-processing.

Data Table
- Added an "Skew" column when using the "Add Element Checks" command.

Model Info Tree
- Added Aero Model branch and underlying branches for Panels/Bodies, Properties, Splines, and Control Surfaces, which allow for creation, copying, editing, listing, and deleting of the various aeroelasticity entities. The color and layer may also be changed.
- Added Visibility check boxes (on/off) for Aero Model - Panels/Bodies, Splines, and Control Surfaces.
- Improved performance of reloading Model Info window when it contained a large number of Load or Constraint sets.

Entity Editor
- Added "Skew" field to Element Quality section when an element is loaded in the Editor.

Meshing Toolbox
- Added Add Surface Mesh Point check box to Feature Removal tool (Feature Type = "Loops" only). Will create a point at the "center" of the "loop", then use that point as a "mesh point" on the surface.
- Performance improvements to Propagate by Mapped Approach option in Mesh Sizing tool. Also, if no mesh sizing exists on a curve, now the number of nodes attached is used for the initial mesh sizing.

Data Surface Editor
- Added "Mapping Tolerance" to the "Options" of the Output Map Data Surface.

GUI - Entity Selection
- Added "on Property" and "on CSys" methods when selecting Coordinate Systems.

Interfaces - FEMAP Neutral
- Updated Neutral Read and Write for v10.3 changes
- Improved reading neutral files by check for duplication of Reference file names to overwrite duplicates not add additional references.\
- Improved performance by elimination of writing a number of data blocks when exporting a neutral file of a group. This also prevents these data blocks from going to the clipboard on a Selector Copy

Interfaces - Nastran

- Added Preference to write continuation cards as "+" only.
- Added support for the Automatic Householder Method (AHOU) for modal analysis.
- Added support for multicase SUPORT1 definition.
- Added support for SOL AESTAT (SOL 144), SOL SEFLUTTER (SOL 145), CAERO1, CAERO2, PAERO1, PAERO2, SPLINE1, SPLINE2, AESURF, AEFACT, AEROS, SET1, TRIM, AERO, FLUTTER, FLFACT, FMETHOD, MKAERO1, and MKAERO2 to support Static Aeroelasticity and Aerodynamic Flutter.
- Added support for PARAM,AUNITS to support Static Aeroelasticity.
- Updated MEFFMASS requests to obey results destination switch.
- Updated import default for PSHELL 12I/T3 and Ts/T default to 0.0

Interfaces - NX Nastran

- Added support for BGRESULTS Glue Output results.
- Added support for PLOADE1 entry.
- Added support for "Mean Dilatational Formulation" on the PPLANE entry.
- Added support MATVE and TABVE entries. GFUNC and KFUNC are defined using dimensionless FEMAP functions where x = decay factor and y = bulk or shear modulus. MOD0 is defined by adding decay time = 0 and MOD0 first term.
- Added support for MATHEV and MATHEM to the MATHE material definition for SOL 601/701.
- Added support for PARAM,CNTSET

Interfaces - MSC/MD Nastran

- Added support for nonlinear results on solid elements from versions above 2008. Results from versions 2008 and before are also still supported.

Interfaces - NEi Nastran

- Added support for Laminate Failure Theories: Max Stress (STRESS), NASA LaRC (LAERC02), Puck PCP (PUCK), and Multicontinium (MCT). Specified on Laminate Property.
- Added support for PARAM, RIGIDELEM2ELAS, ON and PARAM, RIGIDELEMTYPE, BAR to support thermal expansion of Rigid elements.
- Added support for EXTRACTMETHOD (options = LANCZOS, AUTO, or SUBSPACE) for Modal Analysis.
- Added support for PARAM,INREL,AUTO.
- Added support for NITINOL material type. Found in dialog box when Type = Other Types.
- Added support for MAXAD and MAXNAD/MAXRAD for contact.

Interfaces - ANSYS

- Added support for ANSYS 13

Interfaces - LS-DYNA

- Added support for ABCD Contact entries.

Interfaces - Geometry

- Added support for NX 8, Solid Edge with Synchronous Technology 4, and Solid Works 2010
- Added support for Parasolid 24.0
- Added support for ACIS 21, SP3
- Updated STL export to be able to export both solid and plate elements at the same time if they are both selected. If some of the plate elements are coincident with solid element faces, those faces are only exported one time

Loads and Boundary Conditions

- Updated Model, Load, From Freebody command to allow selection of a Freebody entity currently in the model.
- Added Map Tolerance field for Model, Load, Map Output from Model command and in Data Surface editor
- Enhanced Model, Constraint, Expand command.

Materials and Properties

- Added Mullins Effect (MATHEM) and Viscoelastic Effect (MATHEV) support for NX Nastran Hyperelastic materials for SOL 601/701 in Other Types. The additional options are accessed using the Next button when defining Mooney-Rivlin, Hyperfoam, Ogden, Arruda-Boyce, or Sussman-Bathe types.
- Added Viscoelastic Material (MATVE) in Other Types for NX Nastran SOL 601.
- Added NITONAL material type in Other Types for NEi Nastran.
- Added Mean Dilatational Formulation option to Plane Strain Property. This option is for NX Nastran only and is for properties which do not reference a hyperelastic material for Plane Strain or Plane Stress Elements. The formulation of the elements also must be set to "1..CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8" (Plane Strain) or "2..CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8" (Plane Stress) in order to export this property type. The "Mean Dilatational Formulation" switch on the property may be used for nearly incompressible materials, but is ignored for SOL 601. Also, Nonstructural mass/are is ignored for SOL 601.
- Added Type in Spring/Damper Property to define if the elements referencing this Property are CBUSH elements or a combination of CROD and/or CVISC elements when exporting to Nastran.
- Added support for NEi Nastran Failure Theories, Max Stress (STRESS), NASA LaRC (LAERC02), Puck PCP (PUCK), and Multicontinium (MCT), on Laminate Property.

Layups

- Updated Layup Manager to have a "copy" capability like Materials and Properties. Changed old Copy/Paste buttons to icons in Layup Manager Dialog.

- Updated Layup Manager to use current sort order when inserting new plys or modifying plys.

Tools

Parameters
- Added Color, Next ID, and Inc values for Aero Panel, Aero Property, Aero Spline, and Aero Surface

Measure, Surface Area
- Added ability to determined surface area of "combined surfaces" to Tools, Measure, Surface Area.

Mass Properties, Solid Properties
- Improved Tools, Mass Properties, Solid Properties by reversing signs of products of inertia to match the normal engineering convention vs a classical mechanics convention.

Check, Coincident Nodes
- Added a summary table to the Merge Node command to give an indication of why nodes were not merged
- Improved the Check Coincident Nodes "Safe Merge" to not merge midside nodes that have output coordinate system differences, even if the nodes at the ends of the element edges were merged. Previously, if the end nodes were merged no checking was done to prevent merging of the midside nodes.

Check, Element Quaility (formally Check, Distortion)
- Added "Skew" Element Check

Meshing
- Added Mesh, Geometry Preparation command
- Enhanced "Suppress Short Edges" option in Mesh, Mesh Sizing, Size on Surface and Mesh, Mesh Sizing, Size on Solid to be a percentage of Mesh Size instead of a percentage of "average curve length" on selected geometry.
- Added Improve Collapsed Tets option to the Solid Automeshing Options dialog box of the Mesh, Geometry, Solid command, which is accessed by click the Options button.
- Renamed the Length Based Sizing option in the Mesh, Mesh Control, Size on Surface and Mesh, Mesh Control, Size on Solid commands to Sizing Type and added the "2..Parametric/Equal Length" option, which is also now the default.
- Improved the Surface Interior Mesh Growth option in the Mesh, Mesh Control, Size on Surface and Mesh, Mesh Control, Size on Solid commands to allow mapped meshing on surface where it was applied. Previously, mapped meshing was not available on these surfaces.
- Improved Mesh, Mesh Control, Custom Size Along Curve command to remove the limitation on number of custom points which can be assigned.
- Improved performance of Mesh Solid command when choosing a solid that had a large number of already meshed surfaces.
- Improved subdivision mesher in case of long thin surfaces with rounded ends

- Improved feedback when tet meshing by only updating the report window with Tet Collapse and Jacobian ratios that exceed the limits specified by Tools, Check, Element Quality or the default Element Quality values. Also, always give worst of each even if it does not exceed limit.
- Updated surface mesher to try subdivision if other 2 meshers both fail.

Output and Post-Processing
- Freebody display has been enhanced and is now managed via the Freebody tool in the PostProcessing Toolbox.
- Added "Select By Vector" options for Nodal and Elemental output in Model, Output, Forced Response. This allows you to limit the amount of output created by this command.
- Increased the length of the equation that can be specified in the Model, Output, Fill and Model, Output, Calculate commands. Was 160, now 1600.

Element - Spring/Damper
- Updated the Spring/Damper element to use the Type, either CBUSH or Other (NASTRAN CROD/CVISC), specified on the Property referenced by the element to determine if a CBUSH or a combination of CROD and/or CVISC elements will be exported to Nastran. Formally, this was done by setting the element formulation. Also, the Define Spring/Damper Element dialog box will now change to show the appropriate inputs based on the Type of the referenced Property.
- CBUSH elements will now use a circular symbol for display, while Other (NASTRAN CROD/CVISC) elements will use a rectangular symbol.

User Interface
- Added Filter Title and Clear Title Filter icon buttons to the Load Set, Constraint Set, Group, Layer, View, Solid, and Freebody Manager dialog boxes.
- Only tabs of entity types which currently exist in the model will be displayed in the View, Visibility dialog box.
- User created Toolbars will now transfer between versions of FEMAP.
- Pressing Ctrl+M while in a dialog box field asking for a length will display the Select Curve to Measure dialog box, which will return the selected curves length.
- Added the Locate Center to the Methods for specifying the a coordinate

Preferences

Views
- Added JT File Version Option
- Added Optimized check box to GIF Options dialog box.

Render
- Added All, Elapsed Time, and OpenGL Errors check boxes to Advanced/Debug Options section.

User Interface

- Added Pick Method drop-down to Graphical Selection section to allow selection of a default "Pick Method".

Database
- Added 16K test to Read/Write Test

Geometry/Model
- Added "Skew" to enter default value in the Element Quality Preferences dialog box.
- Changed Use Length Based Mesh Sizing option to Mesh Sizing drop-down to allow choice of the new default option, "2..Parametric/Equal Length".

Interfaces
- Added Write Alternate Line Continuation option to the Nastran Solver Write Options section
- Added Include Database Files in Scratch option to the Nastran Solver Write Options section
- Check References on Open and Create Geometry References in File Reference Options section are now "Off" by default.

Color
- Added options to set the default color for Aero Panel, Aero Prop, Aero Spline, and Aero Control Surface.

API
- Added Element Quality (feElementQuality) object to the API. Also added AspectRatioOn, TaperOn, AlternateTaperOn, InternalAngleOn, SkewOn, WarpingOn, NastranWarpingOn, TetCollapseOn, JacobianOn, CombinedOn, ExplicitTimeStepOn, AspectRatioLimit, TaperLimit, AlternateTaperLimit, InternalAngleLimit, SkewLimit, WarpingLimit, NastranWarpingLimit, TetCollapseLimit, JacobianLimit, CombinedLimit, and ExplicitTimeStepLimit to Element Quality Object.
- Added Aero Panel/Body (feAeroPanel) object to the API. Also added color, layer, title, propID, defCSys, nSpan, nChord, iIgid, Pt1, Pt4, dChord12, dChord43, nLspanID, nLchordID, and type attributes to the Aero Panel/Body object.
- Added Aero Property (feAeroProp) object to the API. Also added color, layer, title, pdval, pnval, ap_d_width, ap_d_ar, ap_i_orient, ap_i_lrsb, ap_i_lrib, ap_i_lth1, ap_i_lth2, ap_i_thi1, ap_i_thi2, ap_i_thi3, ap_i_thn1, ap_i_thn2, ap_i_thn3, and type attributes to the Aero Property object.
- Added Aero Spline (feAeroSpline) object to the API. Also added color, layer, title, type, icaero, ibox1, ibox2, isetg, dz, meth, nelem, melem, usage, dtor, cid, dthx, and dthy attributes to the Aero Spline object.
- Added Aero Control Surface (feAeroSurf) object to the API. Also added color, layer, title, csys, csys1, aeid, aeid1, eff, ldw, crefc, crefs, pllim, pulim, hmllim, hmulim, tqllim, tqulim, and label attributes to the Aero Control Surface object.
- Added Freebody (feFreebody) object to the API. Also added title, DisplayMode, Group, CSys, NodeMarkerColor, TotalVectorMode, ShowTotalVec, SumComponents, TotalVecColor, x, y, z,

NodalVectorMode, ShowNodalVec, NodalVecColor, and SumContributions attributes to the Freebody object.

- Added Geometry Preparation and Meshing (feMesher) object to the API. This object has been partially added and is for "Future Use" and should not be used.
- Added NasAeroOn, NasAeroCsID, NasAeroRefCsID, NasAeroRefLength, NasAeroRefSpan, NasAeroRefArea, NasAeroSymXY, NasAeroSymxz, NasAeroAeunit, NasAeroAeunitVal, NasAeroVelo, NasAeroRefDens, NasAeroMkFuID, vNasAeroFreqKeep, NasAeroModesKeep, NasAerobPARAMfzero, NasAerodPARAMfzero, and NasAeroDampMethod attributes to Analysis Manager (AnalysisMgr)object for Static Aeroelasticity and Aerodynamic Flutter. Also, added NasBulkCntAset for Bulk Data.
- Added NasCaeOn, NasCaeMachNumber, NasCaeDynPressure, NasCaeRigidTrim, NasCaeWrtieTrim, NasCflOn, NasCflMethod, NasCflDenID, NasCflMachFactID, NasCflRfreqFactID, NasCflFliMethod, NasCflEig, NasCflEps, NasCflWriteFlutter, NasCflSdamp attributes to Analysis Case (AnalysisCase) object for Static Aeroelasticity and Aerodynamic Flutter.
- Added NextExistingInSet to Entity API objects
- Added Clear, SetModelDefaults, GetModelDefaults, CheckQuality, GetAspectRatio, AspectRatio, GetTaper, Taper, GetAlternateTaper, AlternateTaper, GetInternalAngle, InternalAngle, GetSkew, Skew, GetWarping, Warping, GetNastranWarping, NastranWarping, GetTetCollapse, TetCollapse, GetJacobian, Jacobian, Get Combined, Combined, GetExplicitTimeStep, and ExplicitTimeStep to Element Quality object.
- Added GetDivisionList, PutDivisionList, SlenderBodyCount, InterferenceBodyCount, PanelSpanCount, PanalChordCount, and GetBoxSet to Aero Panel/Body object
- Added GetThetaList, PutThetaList, GetRadiList, PutRadiList, ClearSbList, ClearIbList, ClearTheta1List, and ClearTheta2List to Aero Property object
- Added GetNodeSet and GetBoxSet to Aero Spline object
- Added PutSurfaceSet1, PutSurfaceSet2, GetSurfaceSet1, GetSurfaceSet2, ClearSurfaceSet1, and ClearSurfaceSet2 to Aero Control Surface object
- Added GetElements, SetElements, ClearElements, GetNodes, SetNodes, ClearNodes, CalculateNodalCenter, and CalculateSummation to Freebody object.
- Added Axis and TwoAxis to CSys object
- Added ClearMeshLoc and PointsAsSet to Curve object
- Added FindMappedMeshingCorners, AddMeshPoint, CountMeshPoint, and PointLoops to Surface object
- Added CountCommon, CountNotCommon, HasNotCommon, and NextAfter to Set object
- Added SetMultiGroupListFromSets to View object
- Added IsEmpty to SortSet object
- Added ElementsAsSet2 to Solid object
- Added MapFromModelToSet2 to MapOutput object.
- Added GetList to Group object
- Added DeleteAnalysisCase to Analysis Case object.
- Added Pref_JTFileVersion, Pref_GIFOptimized, and Pref_2DTensorPlotOverride

- Added Pref_RenderXORPicking, Pref_RenderMultiModelMem, Pref_RenderDebugElapsedTime, Pref_DebugAllTime, Pref_DebugOGLErrors, Pref_RenderBlockSize, and Pref_DialogRefresh
- Added Pref_PickMethod, Pref_ConfirmDelete, Pref_ShowMode, Pref_ShowLables, Pref_ShowNormals, and Pref_ShowColor.
- Added Pref_PreserveNextID, Pref_DBOpenSaveWindowsIO, and Pref_DBOpenSaveUnblockedIO
- Added Pref_Prev10TetMesh, Pref_Prev10SurfaceMesh, Pref_ElemQualAspectRatio, Pref_ElemQualTaper, Pref_ElemQualAltTaper, Pref_ElemQualIntAngles, Pref_ElemQualSkew, Pref_ElemQualWarping, Pref_ElemQualNastranWarping, Pref_ElemQualTetCollapse, Pref_ElemQualJacobian, Pref_ElemQualCombined, Pref_ElemQualExplicitTime, Pref_ElemQualAspectRatioVal, Pref_ElemQualTaperVal, Pref_ElemQualAltTaperVal, Pref_ElemQualIntAnglesVal, Pref_ElemQualSkewVal, Pref_ElemQualWarpingVal, Pref_ElemQualNastranWarpingVal, Pref_ElemQualTetCollapseVal, Pref_ElemQualJacobianVal, Pref_ElemQualCombinedVal, and Pref_ElemQualExplicitTimeVal. Also, added Pref_OrientSolidIsoOuput, Pref_OrientSolidAnisoOutput, Pref_OrientSolidHyperOutput, Pref_Tria3StressOutput, PrefTria3StrainOutput, PrefTria3ForceOutput, Pref_Tria6StressOutput, PrefTria6StrainOutput, PrefTria6ForceOutput, Pref_Quad4StressOutput, PrefQuad4StrainOutput, PrefQuad4ForceOutput, Pref_Quad8StressOutput, PrefQuad8StrainOutput, PrefQuad8ForceOutput
- Added Pref_NastranScratchLocation, Pref_NasAlternateContinue and Pref_NasDballScratch
- Added Pref_LibLayup
- Added vPref_SpaceballFactors, Pref_SpaceballFactors, Pref_SpaceballSensitivity, and Pref_SpaceballDebug
- Added slots 18-21 for Aero Panel, Aero Prop, Aero Spline, and Aero Control Surface to Pref_EntityColor
- Updated Pref_LengthBasedMeshSize from BOOL to INT4
- Added feFileIsModified
- Added feGroupBoolean2
- Added feSurfaceExtend
- Added feOutputForceBalance2
- Added feLoadFromFreebody
- Added feCoordCenterOfPoints
- Added feSurfaceMidAttrib
- Added feVectorNormalizedDotProduct
- Increased the length of the equation that can be specified in the for feOutputCalculate API method. Was 160, now 1600.

Corrections

General

- Corrected an issue when rebuilding database on nodal heat generation loads that did not get the counters reset properly.

- Corrected an issue where the counters for Geometry based constraints were never rebuilt during a full model rebuild.
- Allow Convert Complex and Expand Complex commands if output exists, not just if a set and vector are active

Geometry

- Corrected an issue problem that resulted in Tangents of Composite Curves not being continuous - the direction of the tangents was correct, but reversed in sign if the underlying curve was reversed relative to the Composite Curve direction.
- Corrected an issue DXF file which had splines with a large number of knots (> 120)

Groups and Layers

- Corrected an issue where Elements selected into a group using the "in Solid/Volume" method were lost from the group if Solids were renumbered.

GUI - Dockable Panes

Entity Editor
- Corrected an issue with labels in the entity editor for plate property (PR# 6561008)

Meshing Toolbox
- Corrected an issue where the Quality display was not updated after a Geometry Editing command in the mesh toolbox when using groups.

Interfaces - Analysis Manager

- Corrected an issue when editing a child branch of an analysis set.
- Corrected an issue when expanding a branch of the tree when the Alt key was depressed.

Interfaces - Nastran

- Corrected an issue when checking for time dependent thermal loads that caused the DLAOD case control to sometimes be skipped
- Corrected an issue reading and writing Random Tria3 item codes
- Corrected an issue reading STRAIN output request
- Corrected an issue creating the analysis set for buckling and stiffened mode. The STATSUB case is automatically created during export so it needed to be removed during the read (PR #1821212)
- Corrected an issue reading RBE3 with UM when no grid or dof followed the UM identifier

Interfaces - NX Nastran

- Corrected an issue with renumbering where the renumber sets for CBAR and CBEAM overwrote BCCONPROP and BGCONPROP table
- Corrected an issue to allow import of MATSMA entry

- Corrected an issue where a rigid edge contact was being reordered. Also improvements to the reorder code so it will not force reordering if the region is simply defined in reverse.
- Corrected an issue reading DDAM output that was caused because of a NX Nastran 7.1 issue that caused the line TIME to not be written. This caused Femap to improperly read the SUMMED responses cases. New method does not require the TIME line and will work when NX Nastran fixes the issue (PR #6477487)
- Corrected an issue reading Rotor Dynamics results in Frequency Response. The modal displacement table was being read then the frequency response output was read in over top (PR#6477487)
- Corrected an issue reading W4 field in ROTORD where Femap would fail to read W4 and overwrite W3 in FEMAP with the W4 value from the ROTORD card.
- Corrected an issue reading XDB file that caused a extra output vector to be created id = 9999
- Corrected an issue writing of BCTPARM contact local definition when defaults were used as the override. Femap will now write those values explicitly.
- Corrected an issue reading time dependency for PLOADX1: Pressure Load on Axisymmetric Element
- Corrected an issue writing SPCD for NXN SOL 601 when a output coordinate system had been defined (PR#1850895)
- Corrected an issue in NXN SOL 601 when a enforced displacement and Force both referenced the same time dependent function (PR#1850897)

Interfaces - MD/MSC Nastran
- Corrected an issue to Stop Femap from writing NXN contact to MSC Nastran
- Corrected an issue reading MSC Nastran Random PSDF, RMS and Zero Crossing output from op2 file.

Interfaces - ABAQUS
- Corrected an issue when checking if element type and property agree. If they were that same except for order then we were updating the element record from parabolic to linear

Interfaces - ANSYS
- Corrected an issue reading output on solid elements that have degenerate capabilities.
- Corrected an issue where Femap was writing SOLID186 instead of SOLID187.
- Corrected an issue where offsets were being exported/imported in the wrong direction.

Loads and Boundary Conditions
- Corrected an issue that if you deleted a solid that had combined surfaces, some combined surface data was left behind. If you then recreated Solid Surfaces with the same IDs as the boundaries, they would not mesh properly (PR#6504722).
- Corrected an issue that prevented loads on curves that were only meshed with the edges of parabolic brick or wedge elements (only certain edges of the elements) from being properly expanded (PR 6542437).

Meshing

- Corrected an issue that caused elements created using the Mesh, Sweep, Element Face command to be "inside-out" (pressures in wrong direction) when you chose faces of solid elements to sweep.
- Corrected an issue that caused a bad mesh in certain cases if you created a curve only boundary formed from standalone curves and curves that were edges of a solid - especially if the common points were not really coincident.
- Corrected an issue in Mesh, Geometry, On Point to properly attach the mass elements to their geometric points. Also updated Modify, Associativity, Automatic to include automatic associativity for mass elements on points
- Corrected an issue to make hex meshing consistent with tet meshing in that it now only automatically deletes plot-only surface elements and leaves other types of surface elements.

Output and Post-Processing

- Corrected an issue that caused a crash if you tried to linearly combine vectors of beam/bar results.
- Corrected an issue that occurred when you changed the Anisotropic or Hyperelastic Output Orientation options for Solid Elements in the Current Output Orientation dialog box. Previously, if you changed these to the "correct" orientations for Nastran, the wrong option would be used. If you never changed these values at all, the correct orientation was used.

Tools

- Corrected an issue that prevented nodes from being merged if they were connected to potentially zero length elements and they were almost, but not quite zero length (in the range 1E-8 to 1E-15 in length)

User Interface

- Modified the List, Output, XY Plot and File, Picture, Copy commands so that when listing/copying an XY plot that has one or more Log axes, both the original X and Y values and the logX logY values are included

Preferences

- Corrected an issue that occurred when you changed the default values for Anisotropic or Hyperelastic Output Orientation options for Solid Elements in the Current Output Orientation dialog box.

API

- Corrected an issue in feSurfaceMidAttrib
- Corrected an issue modifying properties in the Data Surface API object
- Corrected an issue with AddSetRule method of API Set object.
- Corrected an issue with feVectorDotProduct to give true dot product of provided vectors.
- Corrected an issue BoundingBox method of Curve API object when used with combined curves

FEMAP v10.2.1 New Features and Corrections
Updates and Enhancements

GUI - Dockable Panes
Program File
- Added recording of tab changes in dialog boxes. Allows for cases where tab is changed, but nothing is changed within the tab prior to exiting the dialog box (PR 1823474)

Load and Constraints
- Enhanced Geometric Constraint Expansion to better handle situations where geometry-based and node-based constraints were expanded to the same node(s)

API
- Added Axis and TwoAxis methods to CSys API object
- Added NextExistingInSet method to Entity API object
- Added feGroupBoolean2
- Added support for End B shapes in the Get/Put methods of the Prop object
- Enhanced GFX graphics to now obey layer control

Corrections

General
- Corrected Undo files when deleting output with the "Go Fast" option. Previously, some undo files would remain and would not be deleted from the scratch directory after exiting FEMAP (PR 1822685)

Licensing
- Protected RefreshLicense and other functions from failing if the job was lost
- Corrected issue introduced with the addition of HP Itanium licensing that caused a problem when checking license info and using a Node-lock Any- Host license
- Corrected issue in the FlexLM dll that caused FEMAP to unexpectedly exit if "Show Users" was chosen, because it was not properly passing 64-bit time values

GUI - Dockable Panes
Model Info
- Corrected issue with Properties created by copying an existing property in the tree which used a "General" cross-section which referenced a surface. If either property was later deleted, the cross-section outline would be deleted from the property which still exists in the FEMAP model

Views

- Automatically turn off Model Data Contour display if no elements remain in the model (User must refresh screen). Previously, this would remain displayed and could not be turned off until new elements were created.
- Corrected issue which did not access to View Commands while in the "Tools, Measure..." commands and vice versa (PR 6482183)
- Corrected display of "element directions" for RBE3 elements. Arrows point towards "independent" nodes for all rigid elements now, not just RBE2 elements

Interfaces - Nastran

- Corrected issue where Non-Structural Mass was not being imported for PBEAMLs and PBARLs. These values were actually imported into FEMAP, but then overwritten by an internal calculation
- Corrected issue which caused the last line of data to not be read properly if an INLCUDE file ended with a wide-field Nastran entry
- Corrected issue for combined load and constraint sets. Wrong variable type was used causing an artificial limit of 255 referenced set and would cause FEMAP to unexpectedly exit
- Corrected issue where not all SPCADD and MPCADD (Nastran SPCADD/MPCADD Combination) entries would be exported if different constraint sets were specified in different subcases
- Corrected issue reading DLOAD combinations with more than 400 terms. Increased maximum terms to 4,500
- Updated Plate properties to have 12I/T3 and Ts/T values default to 0.0 if fields were blank in imported Nastran input file. Previously, other default values would be calculated from property values.

Interfaces - NX Nastran

- Prevented writing of MAT11 or MATT11 for 3D Orthotropic materials for Solutions 601 and 701 where they are not supported.
- Corrected issue where imported Connection Properties (Linear or glue) with same IDs as CBEAM or CBEND elements could cause renumbering of Connection Regions
- Corrected issue where rigid 2-D Edge connection Regions were being reordered. Also improvements to the reorder code so it will not force reordering if the Connection Region is simply defined in reverse.

Interfaces - Geometry

- Corrected an issue that prevented IGES files from being imported if the name of the Scratch directory contained spaces

Loads and Constraints

- Corrected issue which some geometric loads to be added incorrectly when two loads came together as a common location bet were defined in different coordinate systems.

- Corrected issue where Bolt Preloads that were transferred from pre-V10 models via neutral files were incorrectly defined and could not be edited
- Corrected issue where Combined load sets (Nastran LOAD Combination) containing varying pressure loads on element face with corners and combined with scale factors other the 1.0. Produced incorrect values when using "Tools, Check, Sum Forces", "Model, Output, From Load", or "Model, Load, Combine" commands.
- Corrected issue where curve-based nodal loads would not be expanded correctly if nodes were on in a certain order on specific faces of 8-noded or 20-noded brick elements
- Corrected issue where internal counters where not being reset properly after a "File, Rebuild" for nodal heat generation loads
- Corrected issue with display of Beam distributed loads, which are now drawn at the shear center. Previously, they were always drawn at the neutral access.
- Corrected issue where incorrect elemental heat generation and heat flux loads were being created when using a Data Surface or "Model, Load, Map Output from Model" command

Meshing
- Corrected an issue which caused FEMAP to unexpectedly exit if both the "Fast Tri" and "3-D Tri" meshers failed on a planar surface

Output and Post-Processing
- Corrected issue with "View, Advanced Post, Beam Cross Sections" to include axial cross term due to Iyz in calculation of stress values
- Corrected issue with "View, Advanced Post, Beam Cross Sections" to allow proper display of stresses on "tube" shaped beams
- Corrected issue where contour set to "Auto-Group" would become all one color (red) when toggling element thickness or offset on/off
- Corrected issue where elements were disappearing when "Cutting Plane" was on. Required "Window, Regenerate" command to have elements reappear

API
- Corrected issue with API method feMeshHexSolid() that in certain situations did not properly mesh solids with attributes when passing propID=0
- Corrected issue that prevented the Variant forms of the API Layup properties (vmatlID, vthickness, vangle and vglobalply) from working if you tried to specify more than 100 plies
- Corrected issue with feMeshSurface2 that caused the surface to not be meshed properly if surface did not already have mesh attributes set and user specified not to set default attributes
- Corrected issue with Prop object by copying an existing property which used a "General" cross-section which referenced a surface. If either property was later deleted, the cross-section outline would be deleted from the property which still exists in the FEMAP model
- Corrected issue with MaxNormalDeviation on Surface object to work properly for boundary surfaces

FEMAP v10.2 New Features and Corrections
Updates and Enhancements

Windows 7
- FEMAP is now supported on 32-bit and 64-bit versions of Windows 7.

Views
- Added Connection and Coord Sys tabs to View, Visibility command.
- View Options: Labels, Entities and Color category: Added Curve/Surface Directions option controls the display of Parametric Directions of Curves and/or Surfaces. Replaces the Curve and Surface Accuracy option found in the Tools and View Style category in previous versions.
- View Options: Tools and View Style category: Clipping Planes option renamed Group Clipping Planes to differentiate between the clipping planes used in groups and the new Model Clipping Plane.
- View Options: Tools and View Style category: Added Model Clipping Plane option.
- View Options: PostProcessing category: Contour/Criteria Levels option. Modified and added options under Set Levels for Standard Colors.
- View Options: PostProcessing category: Contour Type option. Added "2..Match Output" option to Contour Type list. When option is selected, nodal output data will be plotted as a Nodal Contour, while elemental output data will be plotted as an Elemental Contour.
- View Options: PostProcessing category: Beam Diagram option. Added Scale % option to scale beam diagrams.

Analysis Manager
- Added "conditional text" to all Start Text and End Text buttons in Manual Control sections throughout the Analysis Set Manager.
- Increased width of Analysis Text window and added 8-character wide "field markers" to the top of the dialog box to aid in entering fixed field Nastran entries.

Connection Properties, Regions, and Connectors
- Added Look For option when using the Connect, Automatic command. By default, option is set to "1..Face-Face Only", which means the command will only automatically find, then create "face-to-face" connections. Other options are "2..Edge-Face Only", which will only automatically find, then create "edge-to-face" connections, while "0..All Connections" will find, then create both "face-to-face" and "edge-to-face" connections.
- Connection Regions defined with Curves or Nodes, using Output set to Nodes can now be used to create "edge" connection regions for an "edge-to-face" Connector..

Functions
- Added 11 new function types which are currently only used for output functions created by the Model, Output, Forced Response command.

- Added ability to choose a particular XY curve from a list when using the Get XY Plot Data command. Only used when multiple curves are displayed on a single XY plot.

GUI - Toolbars and Icons

New 10.2 Toolbars
- Added View - Simple Toolbar. Contains a subset of commands on the View Toolbar.

Panes Toolbar
- Added PostProcessing icon

Select Toolbar
- Changed "Property/Material Value" item on the "Selector Actions" menu of the Select Toolbar to "Model Data Value". This was done because "Element Quality" values may now be used to select entities along with Property and Material values.

View Toolbar
- Added the Measure icon menu. Contains the six commands on the Tools, Measure menu.
- Added Clipping Plane menu item to the View Style icon menu. Submenu contains commands for toggling the "Model Clipping Plane" on/off (Clipping On), toggling which side of defined plane to "remove" from the display (Clip Positive Side), and specifying the "Model Clipping Plane" (Plane).

GUI - Dockable Panes

PostProcessing Toolbox - New for FEMAP 10.2
- Added PostProcessing Toolbox dockable pane. The PostProcessing Toolbox provides a single, consolidated location in the interface from which to postprocess results from an analysis. First, choose a "Style" from either the Deform or Contour tool, then use the unique set of options for that "Style" to create or change what is displayed in the graphics window. The toolbox itself allows changes to be made "on-the-fly" or when directed by the user.

Data Table
- Added an "Explicit Time Step" column when using the "Add Element Checks" command.

Model Info Tree
- Active entities in the Model Info Tree are now shown using "Bold" blue text.
- Added Visibility check boxes (on/off) for Coordinate Systems (User-defined only), Regions, and Connectors.
- Added "Show Selected, Hide Referenced Groups" to Group "Visibility check boxes" context-sensitive menu.

Entity Editor
- Added "Explicit Time Step" field to Element Quality section when an element is loaded in the Editor.

Meshing Toolbox
- Added the Feature Editing tool.
- Added the Geometry Editing tool.
- Added the Mesh Surface tool.
- "Auto Remesh" is set to "on" by default. Can be set to other options in User Interface tab of File, Preferences.
- "Expand Active Tool Only" is "on" by default. Can be turned "off" in User Interface tab of File, Preferences.
- Added button to clear "Show" of Curves or Surfaces in Feature Suppression tool.
- Added "Match Node(s)" option to Sizing Option section of Mesh Sizing tool, which mimics capabilities found in the Mesh, Mesh Control, Custom Size Along Curve command.
- Added "Elements" as a "Search For" option in the Locator.

API Programming
- Updated to new version API Programing tool, which now shows line numbers (which can be turned off) and changes some of the look and feel for more efficient use.

GUI - Entity Selection
- Changed "Property/Material Value" option on the "Pick" menu of the Entity Selection dialog box to "Model Data Value". This was done because "Element Quality" values may now be used to select entities along with Property and Material values.

Interfaces - FEMAP Neutral
- Updated Neutral Read and Write for v10.2 changes

Interfaces - Nastran
- Added Defaults button to Nonlinear Control Options dialog box for Analysis Types "10..Nonlinear Static" and "11..Nonlinear Transient Response".
- Added support for Structural Damping on each DOF for PBUSH (GEi fields) and PBUSHT (TGEIDi fields)
- Added support for PARAM, KDAMP
- Added support for PARAM, FZERO
- Added support for PDAMPT
- Added support for "Fluid Nodes" by setting CD field of GRID entry to -1
- Added support for writing "blank" Z1 and//or Z2 fields to the PSHELL
- Added support for "Nastran Equivalent Laminate" material, which generates multiple MAT2 entries.
- Added read support for GROUNDCHECK and WEIGHTCHECK

- Added read support for FREQ1, FREQ2, FREQ3, and FREQ4 (only reads first 2 FREQi entries in input file)
- Changed "Bulk Data Delete" entry for restarts from "/,1,999,999" to "/,1,9,999,999"
- Changed default Output Requests for Nastran Nonlinear Static Analysis to include Element Forces.

Interfaces - NX Nastran

- Added support for BLSEG and BCPROPS to support edge-to-face glue.
- Added support for the TSTART and ATSMASS options on NXSTRAT entry.
- Added support for Minimum Acceleration (5th Line of NAVSHOCK File), Unit Conversion -Force (10th Line of NAVSHOCK File), and Unit Conversion - Acceleration (11th Line of NAVSHOCK File) options for DDAM analysis.
- Added support for MAT11 and MATT11 entries for 3D Orthotropic Materials when referenced by solid elements.
- Added support PARAM,WMODAL
- Added support for ENFMOTN system cell. Value 0 = "Constraint Mode", 1 = "Absolute", 2 = "Absolute, Viscous Damping".
- Added support for CPLSTN3, CPLSTN4, CPLSTN6, and CPLSTN8 Plane Strain Elements via formulation.
- Added support for CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 Plane Stress Elements via formulation.
- Added support for reading CDDATA from Mode Tracking Method 2. Also, fixed import of CDDATA when using Mode Tracking Method 1.

Interfaces - MSC/MD Nastran

- Added Support for PARAM, ENFMOTN. ABS = "Absolute", REL = "Relative".

Interfaces - NEi Nastran

- Added support for MAT12 and MATT12 entries for 3D Orthotropic Materials when referenced by solid elements.
- Removed default values from Nonlinear Control Options dialog box for Analysis Types "10..Nonlinear Static" and "11..Nonlinear Transient Response",

Interfaces - ANSYS

- Added support for BEAM188 element type. Set using Formulation.
- Added support for SECTYPE, SECDATA, SECCONTROLS, SECOFFSET and SECNUM entries for properties for BEAM188s and plate elements with offsets.
- Added support for PRETS179 element. Created as a Bolt Preload in FEMAP.
- Added ability to write pressures specified from property card for tube elements.

Interfaces - ABAQUS

- Added support for reading *EQUATIONS defined using NSETS.

Interfaces - LS-DYNA

- Added support for membrane, plate, and plane strain elements with offsets via *ELEMENT_SHELL_OFFSET
- Added support for materials "81..LS-DYNA Plasticity with Damage", "89..LS-DYNA Plasticity Polymer", "91..LS-DYNA Soft Tissue", and "181..LS-DYNA Simplified Rubber/Foam" in "Other Types".
- Updated default formulation for beam elements from "2..Belytschko-Schwer Resultant" to "1..Hughes-Liu". Beams with formulation set to "1..Hughes-Liu", may now be oriented with a vector instead of a 3rd node and are exported as *ELEMENT_BEAM_ORINETATION.
- Updated default formulation for 10-noded tetrahedral solid elements from "10..1 Point Tetrahedron" to "17..10 Node Composite Tetrahedron EQ 17".
- Updated material type "66..LS-DYNA Linear Elastic Discrete Beam" to write MAT_LINEAR_ELASTIC_DISCRETE_BEAM instead of MAT_LINEAR_ELASTIC_BEAM
- Updated material type "67..LS-DYNA Nonlinear Elastic Discrete Beam" to write MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM instead of MAT_NONLINEAR_ELASTIC_BEAM

Interfaces - Geometry

- Added support for NX 7.5, Solid Edge with Synchronous Technology 3, and Solid Works 2010
- Added support for Parasolid 23.0
- Added support for ACIS 21
- Added support for reading IGES files with no "Start Section".

Loads and Boundary Conditions

- Added ability to specify a Coordinate System for "body loads" in the Create Body Loads dialog box

Materials and Properties

- Added support for MAT11 and MATT11 for NX Nastran - solid elements which use a 3-D orthotropic material.
- Added support for MAT12 and MATT12 for NEi Nastran - solid elements which use a 3-D orthotropic material.
- Added support for "Nastran Equivalent Laminate Material", which writes multiple MAT2 entries with IDs higher than 99,999,999, can be created for Nastran. When exported, the material ID in FEMAP will have 100,000,000 added to it for "Membrane", 200,000,000 for "Bending", 300,000,000 for "Transverse Shear", and 400,000,000 for "Membrane-Bending Coupling". Typically, these materials created by a Nastran run and are only used on planar elements
- Added support to specify individual "Structural Damping" values for each DOF in the NASTRAN BUSH Property Values section of the Spring/Damper property, instead a single value for the entire property. Also, added the ability to make the "Structural Damping" functionally dependent for each DOF.
- Added support for "Force vs. Frequency" function for Damping in DOF Spring Property.

- Added check boxes for Top Fiber and Bottom Fiber in the Bending Only, Plate, and Plane Strain Properties. When off writes a "blank" to the Z1 and/or Z2 fields on the PSHELL for Nastran.

Tools

Parameters
- Added global Curve Facetting values for Angle Error, Chord Error, and Curve Factor into Tools, Parameters.

Check, Coincident Nodes
- Added options and improved the Tools, Check, Coincident Nodes command.

Check, Element Quaility (formally Check, Distortion)
- Added "Explicit Time Step" Element Check

Measure Submenu
- Added Tools, Measure submenu. Moved Tools, Distance and Tools, Angle commands under Tools, Measure submenu. Also moved Tools, Mass Properties, Measure Curves and Tools, Mass Properties, Surface Area from Tools, Mass Properties submenu to Tools, Measure submenu.
- Added Tools, Measure, Distance Between Nodes and Tools, Measure, Angle Between Nodes commands.

Meshing

- Added ability to highlight points currently selected for 3-corner and 4-corner mesh approaches when using the "Mesh, Mesh Control, Approach on Surface" command.
- Added Merge Nodes drop-down check box to the various Tet Meshing commands.
- Added Allow Mapped Meshing check box to the various Tet Meshing commands.
- Added Allow Void Regions check box to Mesh, Geometry, Solids From Elements command, which allows meshing enclosed volumes which contain internal voids.
- Improved the "Post-Meshing Cleanup" option in the Automesh Surfaces dialog box to be able to recognize more patterns and mesh issues, then update and improve the mesh.
- Increased number of "custom" mesh locations on a curve from 160 to 325.

Output and Post-Processing

- Added View, Advanced Post, Beam Cross Section command.
- Added Model, Output, Forced Response command.
- Updated Model, Output, Process command.
- Added "Include Max/Min Absolute Value" option to the List, Output, Summary To Data Table command. If Include Max/Min Absolute Value is checked, then additional columns will be created displaying max/min values created using the absolute value of the data
- Updated View, Advanced Post, Contour Model Data command to plot "Element Quality" values on elements as a contour or criteria plot.

- Updated Select XY Curve Data dialog box of View, Select command to use drop-down lists to select Output Sets for From and To in the Show Output Sets section instead of entering an integer value.

Element - Rigid

- Updated the Rigid Element dialog box to be "tabbed" and have separate creation options for RBE1, RBE2, and RBE3 element types. Also, now support the UM DOF for RBE1 and RBE3.

User Interface

- Changed extension of FEMAP model files from *.MOD to *.MODFEM. *.MOD file may still be opened.
- Added "-INI filename" option to the command line options. Allows choice of a specific FEMAP .INI.
- Updated the Generation Options dialog box, which is used in many different commands which create copies of an existing entity. Replaced the Parameters Radio Button with the Color and Layer check box and renamed the section Match Original. Also, moved the Match Mesh Sizes, Loads, and Constraints check box into the Match Original section.
- Color of "suppressed" entities is now saved as a global switch. When an entity is suppressed, the specified "suppression" color will be used. When restored, the color will revert to the entity's original color instead of the "active" color for that entity type.
- Added Modify, Update Other, Node Type command. Allows modification of "Node Type" for any number of selected nodes.
- Changed name of Tools, Check, Distortion command to Tools, Check, Element Quality. Updated references from "Distortion" to "Element Quality" or "Quality" several places throughout the program.

Preferences

Messages
- Added Max Repeated Errors (0=All) option.

Views
- Added Include Metafile Format option in Picture Copy section.
- Added Contour Palette option in Options section.
- Added Resolution button to Picture Save Defaults section.

User Interface
- Added Fast Output Delete drop-down to Menus and Dialog Boxes section.
- Added Meshing Toolbox section, which includes Expand Active Tool Only and Auto Remesh.
- Added Alternate Docking Symbols to Dockable Panes section.

Database
- Added Read/Write Test button to Database Performance section.

Geometry/Model
- Changed name of Element Distortion button to Element Quality.

Interfaces
- Added Create Groups from INCLUDE files option to Nastran Solver Write Options section.
- Changed the default for the Direct Output To option in the Nastran Solver Write Options section from "0..Current Directory" to "1..Model File Directory".

API
- Updated the API Programming Window to use new version of WinWrap.
- Added NasModeDampOverall, NasBulkEnfMotn, NasBulkEnfMotnOpt, NasDynFzero, NasDynFzeroVal, NasDynWmodal, NasDdamForceConversion, NasDdamAcellConversion, NasNXStratAtMass, NasNXStratModexOld, and NasNXStratModexNew to AnalysisMgr Object.
- Added AMatrix, BMatrix, DMatrix, AInvMatrix, BInvMatrix, DInvMatrix, InPlaneProp, BendingProp, vAMatrix, vBMatrix, vDMatrix, vAInvMatrix, vBInvMatrix, vDInvMatrix, vInPlaneProp, and vBendingProp to the Layup Object
- Added DataSurface and vDataSurface to the LoadGeom Object
- Added BodyLoadCSys to the LoadSet Object
- Added xyz to the Node Object
- Added BeamDiagramScale, ClipPlaneOrigin, ClipPlaneNormal, vClipPlaneOrigin, and vClipPlaneNormal to View Object
- Added InitAnalysisCase for AnalysisCase Object
- Added InitAnalysisMgr for AnalysisMgr Object
- Added IsNonManifold, IsSmooth, TangentAtEnds, CurvatureAcrossEdge, and CloserPointToSurface for Curve Object
- Added Get and Put for DataSurf Object
- Added GetMinMaxEdgeLength for Elem Object
- Added Compute2 for Layup Object.
- Added GetVectorAtSingleNode for Output Object
- Added SelectIDInSet and AddNewRemoveCommonSet for Set Object
- Added FreeCurvesAsSet, Points, and PointsAsSet for Solid Object
- Added SortRemoveDuplicates for SortSet Object
- Added BoundingSize, ApproximateArea, MinRadiiOfCurvature, and CurveLoops for Surface Object
- Added MsgWndRepeatedErrors
- Added Pref_ResPrintMethod, Pref_ResPrintScale, Pref_ResCopySaveMethod, Pref_ResCopySaveScale, Pref_ResScaleWithWidth, Pref_ResScaleWithHeight, Pref_ResFixedWidth, Pref_ResFixedHeight, Pref_ResPenMethod, Pref_ResPenScale, Pref_ResScreenLogoScale, Pref_ResPrintLogoScale, Pref_ResCopySaveLogoScale, and Pref_DefContourPalette

- Added Pref_TbxExpandActive, Pref_TbxAutomesh, Pref_PaneAltDockSymbols, and Pref_FastOutputDelete
- Added Pref_OpenSaveMethod
- Added Pref_GroupIncludeFiles
- Added Info_FacetAngleTolerance, Info_FacetChordTolerance, and Info_FacetCurveFactor
- Added Info_SuppressedCurveColor and Info_SuppressedSurfaceColor
- Added feModifyRadialOffsets
- Added feAppUpdateModelBox
- Added feOutputProcessCopy
- Added feOutputProcessMerge
- Added feOutputProcessLinearCombination
- Added feOutputProcessRSSCombination
- Added feOutputProcessEnvelope
- Added feOutputProcessErrorEstimate
- Added feTextMultiPut
- Added feFileReadPatran
- Added feFileReadNeutral3
- Added feFileReadCatiaV5
- Added feFilePrint2
- Added feCheckElemDistortion2
- Added feGetElemDistortion2
- Added feCheckCoincidentNode2
- Added feMeasureDistanceBetweenNodes
- Added feMeasureAngleBetweenNodes
- Added feScreenPctPick
- Added feCurveOffsetCurveWasher
- Added feCurveSplitPointToPoint
- Added feCurveSplitPointToEdge
- Added feCurveSplitEdgeToEdge
- Added feConnectAuto2
- Added feSetToolbarSeparator
- Added feSolidCleanupAdvanced
- Added feRunIOTest
- Added DialogAutoSkip

Corrections

Windows 7

- Updated File Open/Save dialog for Vista/Windows 7 to properly set dialog title, and handle default extensions. Changed all default extensions to lower case so they do not add duplicate extensions if you type one manually (still will double if you type an upper case extension). Removed overrides to OK Button text - now always says Save/Open - for consistency across all Operating Systems

Views

- Corrected a problem that caused a hang/crash when you tried to renumber layers and one or more Views had multi-group visibility lists.
- Corrected problem that caused & characters in titles to show up as underlines in text displayed in the graphics window legends.
- Fixed Auto - Group for multi group in post legend listing of max deformation.
- Fixed Autoscale visible to account for entities which are not visible, based on visibility check boxes.
- Fixed problem with boundaries and eliminated surfaces both on, the surface color was the eliminated color - with boundaries off, it is the suppressed surface color. It is now the suppressed surface color.
- Fixed problem with multi group display of double sided contours
- Fixed spring elements to allow beam diagram display.
- Fixed File, Picture, Copy for arrow plots displayed as Deformed Vector Style
- Reduce the maximum number of contour levels from 256 to 255 to eliminate a problem in Render Graphics where the top level in the legend matched the bottom level if using 256 levels.
- Removed curve/surface color change in Feature Suppression - will be controlled in graphics only
- Removed "Reset Clipping" from page setup
- Updated contact and slideline to properly use visibility check boxes.

Functions

- Corrected a problem that caused "Get XY Plot Data" in Model->Function to fail if you had the "Include Text for XY Plots" View Preference turned off.
- Corrected the end of the "Linear Ramp" function to get the last value
- Fixed problem deleting a item from the function, after the item was deleted it was automatically added back since the x field was not cleared.
- Fixed problem with Default title string for Q Damping Function

Geometry

- Corrected a problem where a cylinder would split using a Global Plane, but not one specified by three points
- Corrected a possible crash in Geometry, Solid, Cleanup if slivers were removed and then match model scaling was done.
- Fixed a stitching problem on general bodies that caused a crash - fixed prior to the release of v10.1.1
- Now move surfaces of single-sheet bodies to the No Pick layer or delete them just like Boundary Surfaces when you use them as the base of an extrusion or revolution.
- Updated Modify, Move To; Modify, Move By; Modify, Rotate To; Modify, Rotate By; and Modify, Align by CSys commands to properly handle the situation where points on a solid were defined in the Coordinate System - previously points were moved which is invalid for Solid points

- Updated the Geometry, Rotate, Curve; Geometry, Reflect, Curve; Modify, Rotate To, Curve; Modify, Rotate By, Curve; and Modify, Align, Curve commands to automatically rotate/reflect any meshing attributes (cross section orientation and offsets) that were attached to the curve.
- Updated Geometry - Solid - Cleanup so the user can convert wire bodies they select in FEMAP to curves if they want to.

Graphics
- Fixed OpenGL non XY plot save picture

Groups and Layers
- Corrected missing value of 0..None for the "No Pick" layer in the Visibility dialog box.
- Corrected the "Add Connected Fillets" and "Add Tangent Surfaces" pick option in the standard entity selection dialog when it was used in a Group definition command. Previously worked in normal entity selection, but not in group commands.
- Corrected issue which did not entities to be added, removed, or excluded by color when the entity color of the selected entity was color = 0 (Black).

GUI - Dockable Panes

Entity Editor
- Fixed problem in entity editor when loading Layups.
- Fixed problem in entity editor saving materials where the standard material constants were not saved to the record.
- Fixed problem setting the Surface Bearing Load Orientation Vector in the Entity editor to ensure a unit vector was defined.

Model Info tree
- Changed Drag and Drop of files so that it properly processes like a normal file read/open command. Previously the model info tree was not updated when you dropped a results file.
- Prevented using "Previous Command" after every change in selection in the Model Info Tree. Previously, this could cause problems if you switched entity types and then chose previous. Crashed if you edited a function, switched to property and hit previous.

Meshing Toolbox
- Corrected a problem in Meshing Toolbox Locator that prevented finding Surface Spikes correctly
- Fixed problem using the Move Node Toolbox where mesh based facet projections failed when the element normals were not consistent.
- Immediately update the Model Info tree after the Meshing Toolbox Locator -> Create Group command rather than after the next command

Data Table

- Corrected a problem that caused some load values in the Report Window to be incorrect/zero if the Entity Editor window was not open

Program File
- Corrected playback of program files for single-selection list view controls (lists with icons), like in the Group, Load, BC, ... Create/Manage dialogs.

Messages Window
- Updated how text is written to the message window to minimize flashing

GUI - Entity Selection
- Converted rigid element picking to use lines of rigid element.
- Fixed box picking of femap and parasolid curves if clipping on and all inside option selected
- Fixed problem in area picking solids - they were not being marked but were being picked.
- Fixed highlighting of properties and materials for box picking.
- Fixed problem with picking coordinate systems by coordinate and around point, line and plane.
- Fixed problem picking spline curves with front or query picking. Also fixed issue when fast standard picking was "off".
- Fixed problem where selection was set to boundary only, then deleted all the boundaries in the model. This would make all surfaces not pickable because the switches were grayed in the dialog.

Interfaces - Neutral
- Corrected problem that prevented Neutral Read from reading library files.
- Fixed problem in neutral file for NEi NL Convergence flags. Patched the neu_101.exe shipped with 10.2 to junk the bad flags and add the correct variables to the end of the line.
- Fixed problem in renumbering. It was incorrectly renumbering the data points in a function and also would not stop reading until it read a -1 in another dataset
- Updated Neutral Write to include needed Global Ply information when you are writing just a Group to the Neutral File. Chooses all global plies referenced by included layups. Also works for Copy in Select Toolbar.
- Updated Output Data format (changed from Block 451 to 1051) for version 10.2 and above. On model with large output saw 40% reduction in Neutral file size and 20%-25% reduction in read/write times.

Interfaces - Nastran
- Changed bulk data delete card to 1, 99999999
- Corrected problems with writing SUPORT, SUPORT1, BSET, CSET , ASET, QSET, OMIT and MAT4 in Large Field Format.
- Corrected problem reading Random results files, PSDF output is now in the f06 file and we were not correctly skipping it.
- Fixed problem reading mixed SORT1, 2 output.
- Fixed issues saving the Solution Strategy Overrides in the analysis set.

- Fixed problem in dynamics writing the static portion of the load when using the "Write Dynamic Load using LOADSET/LSEQ" option
- Fixed problem reading a TABLED2 where the function would be set as the FREQ in the Analysis Manager dynamic solution frequencies.
- Fixed problem reading DLOAD scale factor for response spectrum generation.
- Fixed problem reading frequency response output when a 0.0 solution frequency was defined.
- Fixed problem reading LOAD combinations where the load set id was the same as the referenced set.
- Fixed problem reading MAT1 mass if a MAT4 card also existed and WTMASS factor was used. The WTMASS factor was applied when the MAT1 was read then again when the MAT4 was read.
- Fixed problem reading of METHOD and CMETHOD if extra cards were present that were not activated by the case control.
- Fixed problem reading .op2 design optimization output when the design variables did not exist in Femap.
- Fixed problem reading SPC/MPC ADD combinations when the set ids were the same as the referenced sets.
- Updated output to not transform random nodal results unless user pref CalcRandomResults has been set.
- Updated ASSIGN statements that could be too long (> 72 char) so they split onto multiple lines
- Updated Nastran Case Control reader to support reading various commands with or without equals and with varying spacing

Interfaces - NX Nastran

- Fixed problem reading nonlinear stress from SOL 601 when corner output was requested. SOL 601 is writing values for sigma z and causing the Femap Standard vector calculation to use the 3D calculation.

Interfaces - MD/MSC Nastran

- Updated HEXA, PENTA, TETRA OES datablocks for MSC Nastran ONLY. 2008 r1 and later have two extra words in these data blocks.

Interfaces - LS-DYNA

- Corrected problem skipping unsupported ALE materials which caused Femap to get out of sync.
- Removed "implicit solver" check box from dialog box. Now set via an Analysis Type.

Licensing

- Changed Network Licensing to give a more descriptive message when no license file has been specified and automatically transfer to the dialog to specify it.

Loads and Boundary Conditions

- Changed the PartialCurveLength functions to always return the shorter distance around a FEMAP Circle curve, which was causing an error in some cases of Expand Load when element edges hit the node at the 0.0/1.0 parameter
- Corrected a problem that caused total loads on curve-only boundary surfaces to not expand because these surfaces have no area.
- Corrected a problem that caused variable geometric loads along curves that varied using either the function or interpolation multiplier methods and that produced elemental loads (like pressure) to expand to zero load.
- Corrected a problem that could cause bearing loads that expanded to nodes that were very close to, but not at 90 deg., to have a zero value rather than being skipped.
- Corrected a problem that occurred when you edited a load definition of an elemental face-based load, and defined the updated load using a data surface. Previously, the first load in the definition did not get updated to the data surface values. Corrected prior to release of v10.1.1
- Enabled placing Surface-based Convection and Radiation loads on the back faces of plate elements, just like Element-based loads. Previously this was available in the "Advanced Thermal" interface, but not the others. Also corrected a problem that caused these loads to always be expanded to the back faces, whether or not this flag was turned on (problem was only if you were in Advanced Thermal interface).
- Fixed problem in expanding bearing loads on multiple surfaces where small surfaces would pick up a disproportionate amount of load - fixed prior to release of v10.1.1

Meshing

- Changed Surface mesher to not smooth mapped planar curve-only boundaries
- Changed Meshing error when the mesh approach points do not exist to now error and then reset the mesh approach.
- Corrected a problem in Fast Tri meshing where the element normals did not match the surface normal of a curve-only boundary surface.
- Corrected a problem that caused FEMAP to hang if you reflected a mesh using a nonzero trap size, and you used the "Match Loads, Constraints..." option, and there were nodal loads (like forces) on nodes that fell within the trap distance.
- Corrected a problem that caused tet-meshing of a large number of solids to slow down as you progressed through the list of solids
- Corrected a problem that did not properly assign offsets (if offsetting from top or bottom face) when meshing a surface from meshing attributes
- Corrected a problem with the v10 mesher that caused meshing curve-based boundaries to fail if the first and last curves of the boundary did not join at two coincident points rather than the same, single point.
- Corrected numerous problems with Interactive Mesh Editing where it did not properly set the Material Orientation angle for certain split patterns (Quad to Tri+Quad, Tri to 2 Tri, and Tri to 3 Quad). PR 1774578 reported angles of #NAN being set on some elements. That was not reproduced in the development environment, but this should fix it.

- Corrected problem that allowed the Edge Members command to create invalid elements if you picked a Plot Planar element type and had parabolic faces - Plot Planar elements can not be parabolic
- Corrected a problem that caused associativity of some nodes on a surface that is adjacent to another surface that has a suppressed curve with a shared endpoint to be improperly associated with both surfaces.
- Fixed problem when applying quad paving to a boundary surface that caused the paved element normals to be reversed, and potentially free edges to be created for parabolic elements, if inner holes in the surface had the same clockwise/counterclockwise orientation as the outer loop.
- Fixed problem that created a mesh with a zero property if no property attribute had been set.
- Fixed problem copying surface mesh attributes when combining surfaces into a boundary, if the boundary being created/modified has had mesh attributes initialized then don't overwrite them.
- Fixed problem splitting elements that were loaded with a elemental heat generation.
- Updated the 2D Fast-Tri mesher to loop back and tri multiple starting locations if it has a problem recovering all of the mesh edges. Also properly abort if edge recovery failed
- Updated the Geometry, Rotate, Curve; Geometry, Reflect, Curve; Modify, Rotate To, Curve; Modify, Rotate By, Curve; and Modify, Align, Curve commands to automatically rotate/reflect any meshing attributes (cross section orientation and offsets) that were attached to the curve.

Output and Post-Processing
- Changed title format for computed Laminate output from Lam# to Lam Ply#
- Corrected a problem which caused Global and Top/Bottom Ply data to be computed improperly (data from wrong ply) if on a laminate element the property ID was equal to the layup ID of the previous (lower ID) laminate element but the layup IDs of the two elements were different, and the Global Ply represented a different ply in the two layups.
- Corrected problem that showed that you were using the selected ply rather than top/bottom if top/bottom was selected and no global plies existed in the Laminate Options dialog
- Corrected a problem with the List->Output->Results To Data Table that would cause no output to show up if the selected output was complex and transforms were required.
- Fixed a problem in Output From Loads that caused the magnitude to be incorrect for nodal forces and other nodal vector loads when multiple loads were applied to the same node.
- Fixed problem unloading the XY PLOT request dialog box where in some cases you were not prompted to select the element group.
- Fixed Nodal contour using on the fly transformed results to use the corner data correctly.
- Fixed problem with expand complex for beam results for ends A and B.
- Remove check in op2 Read that caused data to be ignored if any "Put" to the database failed.

Tools
- Changed Merge Nodes command and automatic during meshing to merge two nodes across a connection, even if merge across connections is not set, if the two nodes are also both in the same connection region. This prevents self-contact regions from preventing meshing failures.

User Interface

- Changed wording on several dialog boxes from "Toggle Set Selection" and similar to "Toggle Selected Sets" to better reflect what the buttons do.
- Changed underlined characters in several dialogs that were previously &O, which interferes with program files <OK> - this will not be in the translated versions of v10.2
- Corrected a problem that could occur if you got an error message or a question dialog when working in a dialog. After one of these, if you displayed the context menu, the menus and toolbars would be enabled and you could start a second command while still in the previous one.
- Corrected a problem that caused FEMAP to crash if you assigned a program file to User Command (and then assigned it to a toolbar button), but in that command definition you used a lowercase .pro or .prg filename extension (uppercase worked fine and was the default if you picked the file)
- Fixed a problem caused when launching commands/dialog boxes from an undocked pane. If a second level dialog was created when that dialog ended focus was returned to the pane and not the first level dialog. In particular this caused Shift+Ctrl polygon picking to fail and in some cases crash.
- Fixed problem in tabbed dialogs that caused the mouse to be captured if the mouse was in a control when a dialog was initialized. It would stay captured until you left that control. Fixed prior to the release of v10.1.1
- Updated Font Size drop-downs in File Message Preferences and File Page Setup to contain a list of standard sizes.
- Updated Translators (Catia V4, ProE, STEP, IGES) to support reading files with multi-byte filenames.
- Updated using Ctrl+Z to enter a point location into another dialog, previously always entered in Global Rectangular, now enters properly in active CSys.
- Updated Load, Constraint, Output Set and Group Create/Manage made "New..." the default button if none exist.
- Updated Help, About so NX Nastran modules now show up in the licensed options list when using network licensing (previously only with a dongle).

Preferences

Database

- Corrected potential Undo problem where undo files were written directly to the scratch directory if you specified one in File Preferences, rather than being written to the model subdirectory which is in the scratch directory. This could cause conflicts if you had two FEMAP sessions with different models that had the same base filename.

API

- Changed the PartialCurveLength functions to always return the shorter distance around a FEMAP Circle curve.

- Corrected a problem that caused multiple toolbars created from the API function feAddToolbar() to be lost when you exited FEMAP and returned. Only the last toolbar was reloaded, others were lost. Worked properly if toolbars were created from the user interface.
- Corrected loading of API window so that it always loads the correct version of the type library when multiple versions of FEMAP are installed
- Corrected the feFileReadParasolidOpt() method. The assign_color boolean argument previously worked backwards.
- Corrected a problem with the SelectID method of the Set object that prevented it from choosing the 0..Global Rectangular coordinate system.
- Fixed api definition of NasModeMassForm
- Fixed a crash in SortSet.Current() if you called it when the Sort object was empty or the current index had not been set.
- Fixed a problem that prevented ranges of Coordinate Systems added by ID to be removed from a group when you called RangeDeleteAll(-1) which is supposed to clear all ranges.
- Fixed problem in data surface API object.
- Fixed API access to print and save preferences.
- Make sure that no file extensions get registered during startup with /Register if using an API-only license
- Updated SelectAllOnLayer() method so that it does not change the ID of the current object.

FEMAP v10.1.1 New Features and Corrections
Updates and Enhancements

Views

- Added All Views option to View, Rotate, Model command. When All Views is checked, the first action taken in the View Rotate dialog box will "sync" the views, then the views will move in unison until All Views has been unchecked.
- Added Filter and Clear All Filters buttons along with corresponding text field to View, Visibility dialog box. The Filter and Clear Filter buttons are available for use in the Group, Layer, Material, Property, and Geometry tabs of the Visibility dialog box. Simply enter text into the field, then click the Filter button. The list in that tab will be reduced to only those entries that contain the text you specified. You can now enter additional text, and press the Filter icon button again to further reduce the list. Press Clear All Filters icon button to return to the full list and start again. This can be especially useful in models which contain a large number of groups and layers.
- View Options: Labels, Entities and Color category: Force and Bearing - now controls display of Force and Bearing Force loads
- View Options: Tools and View Style category: View Legend - added Legend Style option "3..Titles, Model Name, Date" which will display the current time and date when the option is turned on along with the full model name and directory path.
- View Options: Tools and View Style category: Render Options - removed the Graphics Engine button.
- Added Geometry tab to View, Visibility command.

Analysis Manager

- Improved performance of the Preview window significantly.
- Added ability to hold down Alt key and left click the "expand/collapse" toggle to expand/collapse all of the "sub-branches" under the highlighted branch. Also, pressing the right arrow key while holding down the Alt key will "expand" all sub-branches, while pressing the left arrow with Alt will "collapse" all sub-branches.
- Enhanced Analysis Multiple capability for MSC Nastran to perform as expected.

Connection Properties, Regions, and Connectors

Connection Property - NX Linear tab

- Added "2..NXN 7.0 Method" to the "Refine Source" drop-down in the Common Contact (BCTPARM) and Glue (BGPARM) Parameters section. This is the default for NX Nastran 7.0.
- Updated Auto Penalty Factor option in the Common Contact (BCTPARM) and Glue (BGPARM) Parameters section to be "on" by default. This is the default for NX Nastran 7.0.

Functions

- Modified the Function Definition dialog box to be more intuitive. The Add button replaces More, Copy Function replaces Copy, Load from Library replaces Load, Save to Library replaces Save,

Copy to Clipboard replaces Put, and Paste from Clipboard replaces Get. Also, added the Update button which will take the currently entered values and update the XY pair currently highlighted in the list, as well as the Get XY Plot Data button, which will place the values from a XY Plot currently being displayed in FEMAP into the list.

- Added 4 new function types which allow the user to specify use of the TABLEM1 for Nastran when creating vs. Temperature functions. They are "19..vs. Temp (TABLEM1 Linear, Linear)", "20..vs. Temp (TABLEM1 Log, Linear)", "21..vs. Temp (TABLEM1 Linear, Log)", "22..vs. Temp (TABLEM1 Log, Log)"

Geometry

- Added 3 "Align" options to the Geometry, Curve - From Surface, Pad command

Graphics

- Improved performance of undo/redo of surface facets.

Groups

- Added support for selecting Solids in the Group, Clipping... commands

GUI - Dockable Panes

Data Table

- Added "Select All" command on context-sensitive menu for the Data Table. Selects all rows currently in the Data Table regardless of which rows are currently highlighted.

Model Info Tree

- Added "Auto Create Definition" to context-sensitive menu for "Other Loads". Allows you to highlight any number of loads and will automatically create new load definitions based on load type, load values, and additional load information (i.e., loaded face of an element). A new definition will be created for loads of the same type which have different values and/or different additional load information, which differs from the Create Definition command.
- Added ability to hold down Alt key and left click the "expand/collapse" toggle to expand/collapse all of the "sub-branches" under the highlighted branch. Also, pressing the right arrow key while holding down the Alt key will "expand" all sub-branches, while pressing the left arrow with Alt will "collapse" all sub-branches.
- Added Visibility check boxes for solids and sheet solids under the Geometry branch.

GUI - Entity Selection

- Added the ability to quickly access "Polygon picking" without choosing "Polygon" from the "Pick" menu in the Entity Selection dialog box. Simply hold down both the Shift and Ctrl keys at once and press the left mouse button to specify the first point of the selection polygon, then click additional points on the screen until the appropriate area is within the polygon.

- Added "Filter" and " Clear All Filters" buttons and corresponding text field to the "Select One or More..." dialog boxes displayed when the Select From List button is pressed in the Entity Selection dialog box. Once text is entered into the text field click the Filter icon button to reduce the list to just those entries that contain the text you specified. You can now enter additional text, then press Filter icon button again to further reduce the list. Press Clear All Filters icon button to return to the full list and start again

- Added "Color" option to the "Pick" menu of the Entity Selection dialog box. This option allows you to select a color from the Color Palette, then adds all entities of the current type which are also the selected color to the selection list. Options also exist to Match Color, Match Pattern/ Transparency, and Match Line Style options which may turned on/off to either broaden or narrow the selection criteria. By default, all Options are on.

- Added "Property/Material Value" option to the "Pick" menu of the Entity Selection dialog box. This option allows you to choose entities in the model with values Equal to a specific material/ property value (i.e., Plane Element Thickness, Young's Modulus, BEAM End A Area etc.) or entities which have values within a range (Above or Below a single value; Between or Outside two values) for a particular material/property entry.

- Improved "Copy as List" option on the "Pick" menu of the Entity Selection dialog box. Using this function on "Large" models containing several million nodes has gone from taking hours to taking seconds.

- Added "Filter" and "Clear All Filters" buttons and corresponding text field to all "Select 'single entity' from list" dialog boxes. An example of a command that would display such a dialog box would be Modify, Update Elements, Property ID, which displays "Select Property for Update".

- Added graphical picking of Solids from the Solid Manager dialog box. Also, the displayed Loads Set, Constraint Set, Group (only when single group displayed), or View can be graphically chosen from the screen and then become highlighted in the appropriate Manager dialog box.

Interfaces - FEMAP Neutral

- Updated Neutral Write to use the proper versions of ACIS and Parasolid when exporting neutral files for older version of FEMAP.

Interfaces - Nastran

- Added support for PCOMPG entry
- Added support to read the QVOL, CONV, PCONV, QHBDY, QVECT, QBDY1, CHBDYG, VIEW, VIEW3D, RADM, RADMT, RADCAV, and RADSET entries
- Added support to read and write PARAM,SIGMA and PARAM,TABS
- Added support to write TABLEM1 entries when creating vs. Temperature functions using specific function types in FEMAP.
- Added ability to Skip NLPARM in the Nonlinear Control Options dialog box for the Master and all Subcases when creating an analysis set for a nonlinear analysis.

Interfaces - NX Nastran

- Added Support for BCTADD and BGADD entries for version 7.0

- Updated XDB import to only allow reading of regular stress or nonlinear stress, not both.
- Added support for Sussman-Bathe hyperelastic material
- Added support for Shape Memory Alloy material

Interfaces - Geometry
- Added support for CATIA V5 release 19
- Added support for NX 7
- Added support to optionally read or skip blanked/invisible/hidden parts when importing NX 6 assemblies
- Restored support for ACIS versions 7, 8, and 9, which had been removed in a previous version.

Layups
- Added ability to create PCOMPG instead of PCOMP entries for Nastran by specifying a Global Ply for every Ply in a Layup. If even 1 ply does not have a Globally Ply assigned, the PCOMP will be written instead.
- Added "Ok to Update Material and Thickness of Global Ply # in all Layups?" question to Edit Ply capability in the Global Ply Definition dialog box.

Loads and Boundary Conditions
- Added "Bearing Force" Load type to "Model, Load, On Surface" command.
- Updated "Force", "Bearing Force", "Moment", and "Torque" load types from "Model, Load, On Surface" command to use "Total Load" option by default. Allows you to take a "total load" and spread it across all of the selected surfaces.
- Updated "Total Load" option for "Force" and "Moment" load types to "Model, Load, On Curve" command to use "Total Load" option by default. Allows you to take a "total load" and spread it across all of the selected curves.
- Added "Update Scale Factors" button to Referenced Load Sets for Nastran LOAD dialog box. Allows you to update the "For References Set" scale factor of all load sets currently highlighted in the list of Referenced Sets.

Materials
- Added support for Sussman-Bathe hyperelastic material to Other Types for solution 601 in NX Nastran
- Added support for Shape Memory Alloy material to Other Types for solution 601 in NX Nastran

Meshing
- Added automatically assigning corners to surfaces with more than 4 corners when using the "Mapped - Four Corner" approach of the Mesh, Mesh Control, Approach on Surface command.
- Added "Radial Offset Around Vector" option to Modify, Update Elements, Line Element Offsets command.

- Improved performance of midside node attachment significantly, especially on models with a large number of geometric entities (Surfaces, Curves. Points). This capability is used when creating new mesh on a solid, as well as when using the Modify, Associativity, Automatic command.
- Added Delete, Model, Mesh on Nodes command. Works exactly like Delete, Model, Mesh except nodes are selected instead of elements.

Output and Post-Processing

- Added support for importing of Nastran output files containing PCOMPG results. PCOMPG results are stored in Nastran output files using the Global Ply ID. Results are converted from Global Ply ID to FEMAP Ply ID.
- Updated View, Advanced Post, Contour Model Data command to only show property and material values in the "Other" lists which are actually available in the model.
- View Options: PostProcessing category: XY Curve 1-9 - Updated use of Scale factor for Log plots.

Element - Rigid

- Added "New Node At Center" method to Independent (Reference) section of Define RIGID Element dialog box. When this method is used, FEMAP will automatically create a node at "center" of all the selected Dependent (Nodes to Average) nodes, much like the "Spider" API command.
- Added "Convert" button to Define RIGID Element dialog box. This button is used to convert a rigid element to an interpolation element and vice versa. When converting from rigid element to interpolation, FEMAP will ask "OK to Convert only Translational Degrees of Freedom?".
- Added "Distance Weighting" option to the Update Interpolation Element dialog box displayed after clicking the "Update" button in Define RIGID Element dialog box. This option will create different interpolation factors for highlighted Nodes to Average based on their distance from the Reference node. Multiple nodes must be highlighted in the list for this option to have any effect

User Interface

- Added File, Picture, Copy Desktop command. Works much like File, Picture, Save Desktop, except it copies a picture of the entire FEMAP GUI to the clipboard instead of saving it to a file.
- Added File, Picture, Copy Layout and File, Picture, Save Layout commands. These commands work much like File, Picture, Copy Desktop and File, Picture, Save Desktop, except they only copy to the clipboard or save to a file the contents of the "Graphics Area" instead of the entire GUI.

Preferences

Views

- Added Include Text for XY Plots option in Picture Copy section.
- Added Picture Save Defaults section.

User Interface

- Added Captions Always on Top option to Dockable Panes section.

Interfaces
- Added Auto Answer Post Questions button.
- Added Delete Read Synthetic Load Sets option.

API

- For functions that take input arguments that are Arrays/Variants, you can now pass a single value/constant directly if the entire array is supposed to be filled with the same value.
- Disabled Undo after calling feFileRebuild, feFileSave, feFileSaveAs, and feFileSaveAll from the API.
- Converted UserData to a non-Entity-based object. Implemented numerous methods that are identical in call to Entity-based objects, but work properly with UserData
- Added Length to Element object
- Added AddAllTitle, AddAllColor, and AddMidsideNodes to Set object
- Added ComputeStdShape and ComputeGeneralShape for Property object
- Added CountLoads and IsTotalLoad for LoadDefinition object
- Added Add to LoadMesh object
- Added Add to BCNode object
- Added Preview to AnalysisMgr object
- Added HasFullGlobalPly for Layup object
- Added Pref_PictureCopyTextForXY
- Added Pref_PictureFormat, Pref_AnimationFormat, Pref_GIFColorOpt, Pref_GIFAnimationDelay, and Pref_GIFFrameSeries
- Added Pref_NasQstOn, Pref_NasQstVal, and Pref_DeleteRdScratchLdSets
- Added Pref_CustomToolsPath
- Added feAppMessageStartListing
- Added feAppMessageEndListing
- Added feFilePictureSave2
- Added feFilePictureCopy2
- Added feFormatReal
- Added feTruncateReal
- Added feModifyRadialOffsets
- Added feDeleteMesh

Corrections

Connection Properties, Regions, and Connectors

- Corrected a problem that caused connection regions defined on Curves or Surfaces, to expand to improper element faces

Graphics

- Fixed clearing of XY graph background if it is OpenGL window
- Fixed issue in weld elements if renumbered badly

- Fixed logo so if bitmap was bigger than graphics region it would still draw.
- Fixed issue in plate result transformation that was introduced in 10.1
- Fixed issue if number of lines of text in post legend was too large
- Fixed random issue when elements were blanked based on timing of redraw.
- Fixed issue that Copy Picture of XY Plots does not work on Vista if you are in render mode.
- Corrected an issue that caused output created by Model, Output, Global Ply to be deleted every time you made a plot of the data.
- Modified drawing of edges to get around nVidia graphics issue in drivers newer than 178.46

Groups
- Fixed issue when displaying multiple groups and had automatic add "on" and were adding into one of the groups being displayed. The created entities did not show up unless you turned one of the groups on/off. Now they show up immediately.

GUI - Dockable Panes

Entity Editor
- Fixed problem editing loads in load definitions that had different faces from the Entity Editor.

Model Info tree
- Corrected a problem that caused the Element Type and Shape counts in the Model Info tree to show linear elements when you hex meshed with parabolics.

Interfaces - Neutral
- Fixed issue reading neutral files from 5.0 or earlier into 10.1
- Corrected a problem reading multiple UserData objects from a neutral file.

Interfaces - Nastran
- Fixed issue that caused velocity body loads not to be written out unless a nodal load was also defined
- Fixed issue creating multiple LOAD combinations.
- Fixed problem where MEFFMASS output was skipped when results destination was set to post
- Fixed problems reading complex output where phase angles were being skipped
- Modified writing of loads so SPCD is written to the LOAD case control set rather than as a combination.
- Corrected a problem that caused Nastran to fail to run if the Nastran Scratch directory you specified in File, Preferences had spaces in the path.

Interfaces - NX Nastran
- Fixed problem reading BCPROP
- Fixed problem when writing CQUADX8 in wide field format.

- Fixed issue when writing BCRPARA in wide field format.

Interfaces - NEi Nastran
- Fixed issue that converts nodal contact regions to elemental faces to fail for solid elements

Interfaces - Ansys
- Added ability to specify Ansys results version in API.

Interfaces - Abaqus
- Fixed problem reading complex output.

Loads and Boundary Conditions
- Fixed issue that caused crash when using face selection to create elemental loads
- Corrected a problem expanding loads on nodes along a curve that used the variable type function or interpolate. Previously they did not expand properly - usually gave 0.0 loads

Meshing
- Corrected a meshing problem that caused planar surfaces with poles to not be smoothed properly if you applied a mapped meshing approach.
- Updated Meshers to properly create and project to a sheet body on multi-surface boundaries that have suppressed loops.

User Interface
- Fixed issue in opening models with preference set to Windows I/O with 64K portions
- Fixed issue renumbering entities that referenced a Weld element.

API
- Automatically set some fields in PutContactList method of the connection region object if you set them to invalid values
- Made Info_GeometryScale a writable property if there are no solids in the model
- Corrected Layup method on the Property object.

FFEMAP v10.1 New Features and Corrections
Updates and Enhancements

Views
- Changed View, Set to View, Create/Manage
- Added View, Visibility command. Replaces a combination of functionality in Model Data, Quick Options, and Layer Management.
- Added Load - Body option to View Options, Labels, Entities and Color category
- Changed Moment option to Moment and Torque in View Options, Labels, Entities and Color category. Now controls display of Moment and Torque loads
- Changed "1..Surfaces Only" option for Surface Hatch to "1..Hatch Wireframe Surfaces" and added "2..Never Hatch Surfaces" option to Render Options in View Options: Tools and View Style category
- Removed Stereo option from View Options, Tools and View Style category
- Removed Quick Options button from View Options
- Removed Model Data button from View Select

Analysis Manager
- Added Previous (Prev...) buttons to many of the Analysis Set Manager dialog boxes when using the Nastran Solvers.
- Enhanced Analyze and Analyze Multiple options to use internal solver queuing system when multiple jobs in one model or jobs from any number of models are sent to the solver. Queuing system now tracks which model the analysis job was launched from and will attempt to return to the correct model and import results before beginning the next analysis job. Also, added "Clear Queue" button to clear the internal queuing system.

Connection Properties, Regions, and Connectors
- Added "Reverse" button to Connection Regions to switch "positive" to "negative" and vice versa for surfaces and "Face 1" to "Face 2" and vice versa for shell elements.
- Updated Connect, Automatic command to properly handle composite surfaces in regions.

Connection Property - NX Linear tab
- Added Adaptive Stiffness and Penetration Factor to the Contact Property (BCTPARM) section. Create the PENAPAPT and PENETFAC fields on the BCTPARM entry.
- Added Glue Type and Glue Factor to the Common Contact (BCTPARM) and Glue (BGPARM) Parameters section. Create the GLUETYPE and PENGLUE fields on the BGPARM entry.
- Added Auto Penalty Factor to the Common Contact (BCTPARM) and Glue (BGPARM) Parameters section. Creates the PENAUTO field on the BCTPARM entry.
- Modified Penalty Factor Units in the Common Contact (BCTPARM) and Glue (BGPARM) Parameters section to have different options depending on what option is set for Connect Type.

Connection Property - NEiNastran tab
- Added 10..Offset Welded Contact option to Penetration Type drop-down list.

Functions
- Added X Axis Log Scale option to Function Definition dialog box.

Geometry
- Implemented the Solid Manager which is used to activate, update, or make no solids active in the model.
- Modified Geometry, Curve - From Surface, Pad command. Entering an Pad Size Factor of 1.0 will extend curves out using the radius of the chosen circular curves, while entering a value of 1.5 would offset the curves 1.5*the radius of the chosen circular curves in all directions.

Groups and Layers
- Implemented the Group Manager for creation, management, and activation of Groups
- Added ability to create "Referenced Groups"
- Updated Group, Operations, Evaluate; Group, Operations, Evaluate Always; and Group, Operations, Renumber Rules to allow selection of multiple groups.
- Updated Group, Operations, Condense to allow selection of multiple groups and "condense" the groups "in place" without creating a copy.
- Added Condense New Group option to Group, Operations, Copy to also condense the active group when copied.
- Implemented the Layer Manager for creation, management, and activation of Layers

GUI - Toolbars
View Toolbar
- Replaced "View Layers" and "Quick Options" icons with "Visibility" icon on View Toolbar.
- Added "Model Data Contour" icon to View Toolbar.

Post Toolbar
- Added "Laminate Options" and "Contour Vectors" options to Post Options drop-down menu on Post Toolbar.

GUI - Dockable Panes

Data Table
- Added "Memb-Bend Coupling" fields for plate elements.
- Added support for "Nastran LOAD Combination Sets" and "Nastran SPCADD/MPCADD Combination Sets"

Model Info Tree

- Added "Reset All Visibility Options" button.
- Added Visibility check boxes (on/off) for Elements (Shape and Type), Properties, Materials, and Layers.
- Added Visibility check boxes (Show/Hide/Clear) for Groups.
- Added Elements object and context-sensitive menu to tree.
- Added "Copy" command to context-sensitive menus for Coordinate Systems, Connections-Properties, Connections-Regions, Materials, Properties, Layups, Functions, and Groups
- Added "Color" command to context-sensitive menus for Materials and Properties.
- Added "Layer" command to context-sensitive menus for Materials and Properties.
- Added "Global Ply" command to Layups context-sensitive menu.
- Added "Referenced Sets" command to context-sensitive menus for Loads and Constraints.
- Added "Edit Where Applied" command to context-sensitive menus for Load Definitions and Constraint Definitions.
- Changed "Edit" command to "Edit Load" on Load Definition context-sensitive menu.
- Changed "Edit" command to "Edit Constraint" on Constraint Definition context-sensitive menu.
- Added "Show Constrained Entities" command to Constraints context-sensitive menu.
- Added "Referenced Groups" command to Groups context-sensitive menu.
- Changed "View Active" command to "Show Active Group" and added "Show Full Model" and "Show Multiple Groups" to Groups context-sensitive menu.
- Changed "Show All Layers" to "View All Layers" and "Show Visible Layers Only" to "View Visible Layers Only" on Layers context-sensitive menu.
- Removed "Make Visible", "Make Hidden", and "Manage" commands from Layers context-sensitive menu. No longer needed due to Visibility check boxes.

Entity Editor
- Added "Memb-Bend Coupling" fields for plate elements.
- Added support for "Nastran LOAD Combination Sets" and "Nastran SPCADD/MPCADD Combination Sets"

Status Bar
- Changed "Set" to "Create/Manage (Set)" for Load Sets, Constraint Sets, Groups, and Output Sets
- Changed "View Active" to "Show Active" for Groups and added "Show Full Model" and "Show Multiple" options

Interfaces - FEMAP Neutral
- Updated Neutral Read and Write for v10.1 changes

Interfaces - Nastran
- Turned off PARAM,MAXRATIO by default
- Added support to read the CVISC and PVISC entries
- Added support to read and write PARAM,RESVINER
- Added support to read and write LOAD, SPCADD, and MPCADD entries

- Added support to set the All Plates as QUADR/TRIAR option when CQUADR and CTRIAR elements are imported
- Added ability to write GEOMCHECK, NONE and read GEOMCHECK entries and populate GEOMCHECK dialog box in Analysis Set Manager
- Added Dynamic Control Options dialog box to Analysis Set Manager for analysis Types 3..Transient Dynamic/Time History, 4..Frequency/Harmonic Response, 5..Response Spectrum, and 6..Random Response
- Added Nonlinear Control Options dialog box to Analysis Set Manager for analysis Types 10..Nonlinear Static and 12..Nonlinear Transient Response
- Added support to read DLOAD, NONLINEAR, TSTEP, TSTEPNL, NLPARM, SDAMPING, FREQUENCY, RANDOM Case Control entries
- Added support to read PARAMs LMODES, LFREQ, HFREQ, W3, W4, G, RSPECTRA, SCRSPEC, OPTION (ABS, SRSS, NRL, NRLO), CLOSE, LANGLE
- Added support to read TSTEP, TSTEPNL, NLPARM, NLPCI, RANDPS, DTI Bulk Data entries

Interfaces - NX Nastran

- Turned the "Loads Change with Deformation" option in the Analysis Options section of NXSTRAT Solver Parameters dialog box "on" by default for SOL 601 and SOL 701.
- Turned the "Constraint Force" option in the Nodal section of Nastran Output Requests dialog box "on" by default for SOL 601 and SOL 701
- Added support to SOL 601 for function dependent acceleration body loads.

Interfaces - NX Nastran

- Added support to read PARAM,OPTION,CQC

Interfaces - DYNA

- Added support 8-noded Quad elements
- Added support for nonstructural mass for Beam and Shell elements
- Added support for the following element formulations for Shell Elements (Fully Integrated DKT triangular, Fully Integrated linear DK quadrilateral, Fully Integrated linear assumed strain C0, 1 point Eulerian Navier-Stokes, 8 point Eulerian Navier-Stokes, and CVFEM Eulerian Navier-Stokes)

Interfaces - Geometry

- Added support for Parasolid 22.0
- Added support for Solid Edge with Synchronous Technology 2
- Added support for ACIS 20, Service Pack 1

Layups

- Updated Global Ply Definition dialog box for Layups

Listing
- Changed listing of model size from Bytes to MBytes when using "List, Model Info" command
- Updated "List, Model, Element" command to list element formulation based on solver set in the "active" Analysis Set in the Analysis Set Manager.

Loads and Boundary Conditions
- Added "Torque" Load type to "Model, Load, On Surface" command.
- Added "Total Load" option for "Force", "Moment", and "Torque" load types to "Model, Load, On Surface" command. Allows you to take a "total load" and spread it across all of the selected surfaces.
- Added "Total Load" option for "Force" and "Moment" load types to "Model, Load, On Curve" command. Allows you to take a "total load" and spread it across all of the selected curves.
- Added Gradient thru the thickness of plate elements on Temperature loads
- Implemented the Load Set Manager for creation, management, and activation of Load Sets.
- Added option to create a Load "Set Type" option which allows you to create a Nastran LOAD Combination and use Referenced Load Sets
- Implemented the Constraint Set Manager for creation, management, and activation of Constraints Sets
- Added option to create a Load Set which represents a Nastran SPCADD/MPCDD Combination and use Referenced Constraint Sets

Meshing
- Added "Use Reference Point" option to Mesh, Mesh Control, Attributes Along Curve command.

Output and Post-Processing
- Added View, Advanced Post, Contour Model Data command
- Added Contour Vectors - 2D Tensor Plot option to View, Select command
- Added Laminate Options to View, Select command
- Added "Exponent" Color Modes, Digits, and Length options to View Options: PostProcessing category: Contour Vector Style

Element - Rigid
- Added Update Button to Define RIGID Element dialog box. Allows you to update the Interpolation Factor and DOFs on any number of highlighted nodes in the Nodes to Average section when using the Interpolation option.

User Interface
- Added Tab Location option to View Windows. Now the "View Tabs" may be places on the Top, Left, Right, or Bottom of a View Window.
- Added Axis of Revolution method to Vector Definition dialog box.

- Added FEMAP.INI and PATH to listing provided by Help, About command
- Added improved precision to transforms with cos/sin functions in degrees

Preferences

Views
- Added 2D Tensor Plot View Options Override option.

Database
- Added Open/Save Method option.

Interfaces
- Added Use ILP-64bit NX Nastran option.
- Added Write All Static Load/BC Sets option.

API

- Added NasBulkDynLdAsLOADSET, NasBulkResViner, NasGCheckNone, NasBulkWriteAllLoadBCSets, NasDynOn, NasDynUseLoadSet, NasDynDampOverall, NasDynDampW3, NasDynDampW4, vNasDynKeepFreq, NasDynTranDT, NasDynFreqTbl, NasDynDampModalTbl, NasDynKeepModes, NasDynTranTimeSteps, NasDynTransOutInt, NasDynDampModalMethod, NasDynRespSpect, vNasDynNoFreq, vNasDynLogInterp, vNasDynFreqType, vNasDynMinFreq, vNasDynMaxFreq, and vNasDynSpreadCluster to AnalysisMgr object
- Added NasCnlIncrements, NasCnlTime_Increment, NasCnlMaxIter, vNasCnlConvergenceFlags, vNasCnlConvergenceValue, NasCnlCtiffnessMethod, NasCnlKstep, NasCnlIntermediateOutput, NasCnlOutputInterval, NasCnlSolutionStrategy, NasCnlSolutionOverrides, NasCnlModnewtonLineSearch, NasCnlModnewtonQuasiNewton, NasCnlModnewtonBisection, NasCnlArcConstraintType, NasCnlArcMinAdjust, NasCnlArcMaxAdjust, NasCnlArcLoadScale, NasCnlArcDesiredIter, NasCnlArcMaxSteps, NasCnlTimeSkipAdjust, NasCnlDominantPeriodSteps, NasCnlBoundsRb, NasCnlStabilityTolerance, NasCnlDivergenceLimit, NasCnlQuasiNewtonVectors, NasCnlMaxLineSearch, NasCnlCreep, NasCnlLineSearchTolerance, NasCnlMaxBisections, NasCnlMaxRotation, NasCnlFstress, and NasCnlMaxAdjust to AnalysisMgr object.
- Added IsCombination to LoadSet object.
- Added Gradient to LoadETemp object.
- Added IsCombination to BCSet object
- Added GlobalPlyLocation to View object.
- Added AddCoordinate, AddAroundPoint, AddAroundVector, AddAroundPlane, AddNodesOnGeometry, and SelectList methods for Set object
- Added RemoveSet, ConvertToBoundarySurfaces, and SelectListmethods for Sort object
- Added ClearAnalysisQueue, GetCorrelate2, and PutCorrelate2 methods forAnalysisMgr object
- Added GetDataSurfType method for DataSurf object
- Added PartialLengthXYZ, PartialLengthNode, and SelectList methods for Curve object

- Added IsBoundingSolidRegion, Mesh, and ResetMeshAttr methods for Surface object
- Added IsGeneral method for Solid object
- Added GetClosest method for Node object
- Added Thickness, Area, and Inertia methods for Elem object
- Added Thickness, Area, and Inertia methods for Prop object
- Added GetCombination and PutCombination methods for LoadSet object
- Added GetCombination and PutCombination methods for BCSet object
- Added ReferencedGroups method for Group object
- Added DefineReal method for Var object
- Added GetMultiGroupList, SetMultiGroupList, and ClearMultiGroupList methods for View object
- Added InitScalarAtBeam and PutScalarAtBeam methods for Output object
- Added Reverse method for Contact and ConnectionRegion objects
- Added Pref_NastranUseILP64, Pref_ConstructionGeometry, and Pref_NastranWriteAllLdbcSets
- Added FLT_SNTORQUE for Load Type
- Added FVD_AXIS_OF_SURFACE for Vector Definition Method
- Added feSurfaceRemoveHole
- Added feModifySolidFacetting
- Added feSolidRemoveFace
- Added feMeshSurface2
- Added feAppRegisterAddInPaneWithFrame
- Added feVectorAxisOfSurface
- Added feMeshSurfaceByAttributes

Corrections

Analysis Manager
- Fixed problem renumbering Analysis Sets that sometimes caused FEMAP to exit unexpectedly.

Connection Properties, Regions, and Connectors
- Corrected a problem when copying surfaces or solids that were used to define contact segments, the contact segment moved to the copy. It now is duplicated into a new connection region if all of the underlying definition is copied.

Groups and Layers
- Corrected a problem when renumbering Layers that caused layers that were not renumbered to be lost from the visible layers list for views.
- Corrected a problem that caused "Evaluate Always" to be turned off if you renumbered any entities referenced by the group.

GUI - Dockable Panes

Model Info tree
- Updated so that reloading the Model Info tree does not change expansion of branches

Data Surface Editor
- Fixed problem that caused element fill to be disabled when clearing a Data Surface

Data Table
- Enhanced Copy to Clipboard and Save of Data Table. Increased speeds by several orders of magnitude.

Interfaces - Nastran
- Fixed PLOAD4 so Femap writes a blank for element faces with only 3 corners.
- Fixed output request in Transient Heat, was missing SORT1 request in THERMAL and HDOT.
- Fixed problem reading XY Summary and optimization from the f06 when results are in the op2 and auto skipping.
- Fixed a problem where Femap would not write a proper DLOAD scale for each dof when more than 1 SUPORT dof was defined in Response Spectrum Application
- Fixed problem where overwrite were occurring if model file name and path were over 160 characters.
- Fixed wide field for BSURFS, BCTADD, PFAST
- Fixed problem reading XYPRINT output for CQUADR NXN3 new formulation. Femap was reading the items using the item codes from the old formulation.
- Fixed problem writing MFLUID in random and response spectrum.
- Fixed problem where if Force load is created after Temperature load then Force is not written out
- Fixed expand Moment load at points is not expanded to the underlying node
- Fixed problem reading QUADR output with corners from f06 when complex output existed.
- Fixed problem recognizing the proper Femap function type for the TABLED2 being read.
- Fixed issue reading SLOAD only the first sload per card was read properly.
- Fixed Problem when reading Nastran files that have Free Format cards with continuation that have blank field 10 and 1, and that don't continue on the same line (i.e. traditional multiline arrangement)

Interfaces - NX Nastran
- Fixed problem where NXN 6.0 would fail to run if Fluid Pressure output was chosen for MFLUID. The Alter used no longer worked or was needed.
- Fixed problem when writing Glue and Contact sets to SOL601at the same time.

Interfaces - NEi Nastran
- Removed SPCD errors for NL trans and SPCD.

Interfaces - Ansys

- Fixed problem when reading ANSYS elements where the element coordinate system is set with a real constant.

Interfaces - Abaqus

- Fixed problem reading *SYSTEM where a default second axis was used.

Interfaces - I-DEAS/NX

- Corrected a problem reading UNV file from NX6 where the Material Table for properties defined by a table were not being read. Looks like NX added a line to the table definition that is not in the spec.

Loads and Boundary Conditions

- Fixed numerous graying issues with creating loads with data surfaces as well as added restricting type of ds created based on the load type being defined.
- Updated expand load to handle nodal loads on curves when only solid elements are connected

Meshing

- Changed boundary mesh error for coincident nodes to write the nodal location rather than the ID.
- Corrected a problem with Mesh Region that prevented it from working in "Nodes Only" mode - still asked for a valid property/shape
- Prevent automatically choosing a mapped mesh if there is a reentrant (>225 deg) corner in the surface - which could lead to an overlapping mapped mesh.

Output and Post-Processing

- Corrected problem that caused Output Sets that did not exist to be added to a selection set when using the standard selection dialog and typing in ID ranges that did not exist.
- Corrected problem in Freebody Display. If you displayed the freebody resultant, but only displayed moments, the moment calculation did not include the components due to the forces that were not displayed.

User Interface

- Corrected a problem on Message Boxes that just had an OK, or just a Yes/No button, that caused them not to close if you pressed Esc or the X in the title bar.
- Enhanced speed when deleting a large number of solids with their associated meshes.

API

- Fixed problem in ComputeShape where the property was being put after the shape was calculated.
- Fixed infinite loop when loading AMgr_Correlate in the Analysis Mgr object.

- Fixed infinite loop when loading AMgr_Contact in the Marc Analysis Case from the API.
- Removed "CaseID" property on the Analysis Set manager object. The master case should always be ID=0, you should not be able to set it.
- Corrected problem in Set object if you called AddSetRule with GDEF_Elem_byShape

FEMAP v10.0.2 New Features and Corrections
Updates and Enhancements

Interfaces - NX Nastran
- Updated version of NX Nastran included with FEMAP with NX Nastran to version 6.1
- Updated Analysis Monitor to be able to handle NX Nastran 6.1 monitor files

Interfaces - Nastran
- Updated reading of OP2 files to support results files larger than 4GByte

Interfaces - TMG
- Updated version of TMG interfaces for FEMAP to "version 6.0, build 470"

User Interface
- Added "guard bytes" around preferences so that if memory becomes totally corrupted it will not overwrite the FEMAP.INI file with values which are not valid
- Modified certain aspects of FEMAP to allow for more complete localization to Chinese, Japanese, and other languages.
- Removed Modify, Update Other, Surface Divisions command

API
- Updated version of WinWrap used by the API Programming window

Corrections

Coordinate Systems
- Corrected problem that filled in wrong defaults for vector if "snap to node" was on and coordinate systems were repeatedly created using one of the Axes creation options.

Geometry
- Fixed problem deleting Composite Curves when the associated Surface or Solid was deleted.

Graphics
- Prevent facetting of curves if a point is missing from Femap curve definition, which is not normal, but can happen in rare occasions.
- Fixed graphics issue related to using the Modify, Scale, Solid command.

GUI - Dockable Panes
Meshing Toolbox

- Fixed problem using the Mesh Locate Toolbox to edit solid meshes (Tet and Hex).

API Programming
- Fixed loss of focus when you hit Ctrl+C to copy the API Programming Window and other panes.

Interfaces - FEMAP Neutral

- Fixed problem reading the property data block from neutral files from Femap 9.2 and earlier which caused Femap to issue errors where they didn't really exist.
- Fixed issue in Tosca code for Neutral file when importing Neutral file. Neutral files were written properly, but could not read past first analysis manager case

Interfaces - NX Nastran

- Added support for NXN 6.0 Advanced nonlinear datablocks for CQUAD8, CTRIA6
- Fixed problem reading the Nastran f06 file when contact separations (SEPDIS) were requested which caused Femap to go into an infinite loop which could end without ending the FEMAP process.
- Fixed BCTPARM NCHG to allow zero to be written.
- Made NXSTRAT ICMODE default solution dependent.
- Fixed problem writing BCTSET when using the Portion of Model to Write option in the NASTRAN Bulk Data Options section by evaluating the group before the case control is written.
- Fixed problem when writing Nastran files when Improve Single Field Precision option was on in File, Preferences. Was writing too many real fields when creating CONM2 entries

Interfaces - Nastran

- Fixed import of NSM for PBEAML and PBARL.

Interfaces - Dyna

- Fixed problem writing LS-Dyna nodal velocities that sometimes caused Femap to write loads improperly
- Fixed improper *SECTION_SHELL ELFORMIDs for formulations of shell elements. If a formulation below "4..C0 Triangular" on the drop-down list was used, the formulation actually written was off by 1.

Interfaces - ABAQUS

- Fixed problem writing node groups using the DEFINE command. Femap was always writing element sets when writing FEMAP groups as sets.

Interfaces - Geometry

- Added missing Solid Edge Moniker code which fixes matching of previous solid/assembly when updating during read

- Enabled "Read Inactive Layers", "Points", "Curves", "Surfaces" and "Bodies" options to CATIA V5 translator. Previously, these were in the translator, but not available to turn on and off.
- Several issues have been resolved in the CATIA V4, CATIA V5, Pro/Engineer, IGES, STEP, and Parasolid geometry interfaces

Meshing

- Fixed reporting of problems when meshing fails in certain instances. Previous versions of Femap reported the Node ID where it failed, but in v10 and above meshing the mesh is not saved unless it actually succeeds. Therefore the node IDs Femap was reporting do not exist, thus now the coordinates where the failure occurred are reported instead.
- Fixed problem when using new meshing to mesh curve-only boundary surface where no mesh sizes were set on the curves. Both mapped and free meshes were not working as expected
- Fixed problem in new meshing when meshing surfaces that had associativity to nodes that did not exist.
- Fixed problem of "Midside Nodes" and "Midside Nodes on Geometry" options not persisting between surface meshing commands.

Materials and Properties

- Fixed issue of shear center offset values swapping on End B of a non-tapered beam after Shape button was pressed, but no changes were made in the Cross Section Definition dialog box.
- Fixed issue of the Reference Point being greyed in the Cross Section Definition dialog box when creating NASTRAN sections (PBEAML and PBARL).

User Interface

- Fixed issue in output transformation dialog boxes where the main dialog boxes were not hidden when trying to select a vector, making it very difficult to choose a vector graphically from the screen

Preferences

Library/Startup

- Corrected a problem that caused FEMAP to unexpectedly close when trying to open a new model if a startup basic script was set to be run on a new model after all current models had been closed.

API

- Fixed API docs for parametric space and added missed methods, HasPole(), RationalParamToXYZ()

FEMAP v10.0.1 New Features and Corrections
Updates and Enhancements

Connection Properties, Regions, and Connectors
- Added ability to have zero (0) as the rigid reference node for NXN SOL601.

Groups and Layers
- Improved "Group, Operations, Generate Solids" to also include elements on the surfaces, curves and points of solids

GUI - Dockable Panes
Data Table
- Added "Save to a File" command
- Added "Save Rows" command to context sensitive menu.

Properties
- Added "Section Evaluation" option in "Cross Section Definition" dialog box for Beam, Bar, and Curved Beam properties.
- Added PBEAML/PBARL option to "Section Evaluation" for use with PBEAML/PBARLs properties.

API
- Added InsideXYZ to Surface Object

Corrections
Connection Properties, Regions, and Connectors
- Fixed issue in BCTPARM REFINE default.

Groups and Layers
- Corrected problem in Group, Operations, Generate Superelements. Previously, the command included elements that touched the boundary of a superelement in the residual group rather than in the superelement group.

GUI - Dockable Panes
Meshing Toolbox
- Added ability to auto compress loads, bcs, connection regions then delete mesh in the Meshing Toolbox.

Program File

- Corrected problem that caused error message if you tried to save a program file with no filename extension, and without changing the file type on Windows XP. Now properly adds .PRO

Interfaces - Nastran
- Fixed issue writing initial conditions from the master case when all loads were in subsequent subcases.

Loads and Boundary Conditions
- Updated "Load, Expand" and "Constraint, Expand" to default to "Compress" if expanded loads/ constraints are in the active set, otherwise defaults to "Expand".

Listing
- Updated the List Destination dialog box to show longer filenames
- Fixed problem in List Surface that caused some surfaces with a "linked" meshing approach to list extra values in addition to the Linked surface ID.
- Updated "List, Geometry, Curve" to list points on all curves connected to solids, not just straight lines.

Meshing
- Fixed problem in merging nodes that randomly caused some solids to fail hex meshing

API
- Updated API Type Library to return specific interfaces rather than generic IDISPATCH interfaces - should help programming in Python and Matlab

FEMAP v10.0 New Features and Corrections
Updates and Enhancements

Windows Vista

- FEMAP is now supported on 32-bit and 64-bit versions of Windows Vista. Many issues from previous "unsupported" versions of FEMAP with regards to Windows Vista, such as entity picking and proper use of the Model Info tree have been addressed.

Analysis Manager

- Added Analyze Multiple option. This accesses a multi-select dialog box which allows you to pick any number of Analysis Sets and run them one after another.

Connection Properties, Regions, and Connectors

- Updated Connection Regions to support 2-D contact in NX Nastran Solution 601.

Connection Property - NX Linear tab

- Moved Normal Penalty Factor and Tangential Penalty Factor from the Contact Property (BCTPARM) section to the Common Contact (BCTPARM) and Glue Parameters (BGPARM) section.
- Moved Shell Z-Offset from Glued Contact Property (BGSET and BGPARM) section to Contact Property (BCTPARM) section.
- Removed Penalty Factor from Glued Contact Property (BGSET and BGPARM) section.
- Replaced Num Allow Contact Changes with Convergence Criteria and Num For Convergence in the Contact Property (BCTPARM) section. Together, these two values create the NCHG field on the BCTPARM entry.
- Added Contact Inactive to the Contact Property (BCTPARM) section. Creates the CSTRAT field on the BCTPARM entry.
- Added Penalty Factor Units to Common Contact (BCTPARM) and Glue Parameters (BGPARM) section. Creates the PENTYP field on the BCTPARM or PGPARM entry.

Connection Property - NX Adv Nonlin tab

- Added Glued Contact Property (BGSET) section with Extension Factor option. Extension Factor enters a value in the EXTi field specified on the BGSET entry for the contact pair "i". Specifies an "extension factor" for the target region.
- Removed the Time Activation section and moved Birth Time and Death Time options to the General section.
- Added Friction Delay option to Standard Contact Algorithm section.
- Moved all options found in the Rigid Target Contact Algorithm section except Normal Modulus to a the Old Algorithm (RTALG=1 on NXSTRAT) section of the NX Adv Nonlin Rigid Target Algorithm dialog box, which is accessed by clicking the Rigid Target Options button. Normal Modulus is found in Common Options.

- Added Penetration Cutback and Max Penetration options to the Old Algorithm (RTALG=1 on NXSTRAT) section of the NX Adv Nonlin Rigid Target Algorithm dialog box.
- Added Max Tensile Contact Force (TFORCE), Max Sliding Velocity (SLIDVEL), Oscillation Check (OCHECK), Contact Gap (GAPBIAS), and Offset Method (OFFDET) options to the Current Algorithm (RTALG=0 on NXSTRAT) section of the NX Adv Nonlin Rigid Target Algorithm dialog box.

Connection Property - NX Explicit tab
- Renamed Rigid Contact Algorithm section to Old Rigid Contact Algorithm section.
- Added Current Rigid Target Algorithm section with Max Sliding Velocity (SLIDVEL), Contact Gap (GAPBIAS), and Offset Method (OFFDET) options.

Entity Select dialog box
- Added "Combined Curves" options (Default, All Points/Curves, Points/Curves Eliminated by Combined Curves, and Combined Curves Only) to the Pick Menu in the standard Entity Selection dialog box. Only one mode can be selected at any given time.
- Added "Boundary Surfaces" options (Default, All Curves/Surfaces, Curves/Surfaces Eliminated by Boundary, and Boundary Surfaces Only) to the Pick Menu in the standard Entity Selection dialog box. Only one mode can be selected at any given time.
- Added "Add Connected Fillets" option to the Pick Menu in the standard Entity Selection dialog box.
- Added "Add Tangent Surfaces" option to the Pick Menu in the standard Entity Selection dialog box.
- Updated direction of mouse wheel for Query Pick list to follow direction of mouse wheel.

Functions
- Added dynamic XY plotting of functions to the Function Definition dialog box.

Geometry
- Added Geometry, Curve - From Surface, Offset Curve/Washer command.
- Added Geometry, Curve - From Surface, Pad command.
- Added Geometry, Curve - From Surface, Point to Point command.
- Added Geometry, Curve - From Surface, Point to Edge command
- Added Geometry, Curve - From Surface, Edge to Edge command.
- Added Geometry, Surface, NonManifold Add command.
- Added Geometry, Surface, Recover NonManifold Geometry command.
- Added Geometry, Midsurface, Offset Tangent Surfaces command.
- Added "Measure Distance" icon button to Geometry, Midsurface, Automatic command
- Added "Ok to Consolidate Properties by Thickness?" question to Geometry, Midsurface, Assign Mesh Attributes command after the material has been chosen
- Added "Cleanup Mergable Curves" option to Geometry, Solid Stitch command

- Added Modify, Update Other, Solid Facetting command.
- Added option to Modify, Project, Point along Vector and Modify, Project, Node along Vector commands to project in both directions along the vector.

Groups and Layers

- Improved Group, Operations, Add Related Entities to include coordinate systems used as definition coordinate systems for Coordinate Systems in the selected group and include reference nodes on beams when the nodes are related to elements, properties, or materials in the selected group.

GUI - Toolbars

Panes Toolbar

- Added Meshing Toolbox icon

Curves on Surfaces Toolbar

- Added Curve Washer, Curve Pad, Split Between Points, Split Point to Edge, and Split Edge to Edge icons.
- Updated Curve Split at Points icon to be Curve Split at Locations icon.

Select Toolbar

- Improved Select Related mode to include coordinate systems used as definition coordinate systems for other selected Coordinate Systems
- Improved Select Related mode to include reference nodes on beams when the nodes are related to elements, properties, or materials

GUI - Dockable Panes

Meshing Toolbox - new for version 10

- Added Entity Locator
- Added Feature Suppression Tool
- Added Feature Removal Tool
- Added Combined/Composite Curve Tool
- Added Combined/Boundary Surface Tool
- Added Mesh Sizing Tool
- Added Mesh Locate Tool
- Added Mesh Quality display options.

Data Table

- Added "Transformed To" capability for listing nodal and elemental output.
- Updated using Show When Selected. Entities already chosen will now highlight when Show When Selected is turned on and un-highlight when turned off.

Model Info Tree

- Updated using Show When Selected. Entities already chosen will now highlight when Show When Selected is turned on and un-highlight when turned off.

Entity Editor
- Added "Transformed To" capability for displaying nodal output and elemental output.
- Added support for Load Definition and Constraint Definition information.
- Added support for Rotor Region information.
- Added support for Layup ID information.

Status Bar
- Added the ability to customize what entity types appear on the Status Bar.

Interfaces - FEMAP Neutral
- Removed option for choosing Binary and Formatted in File Format Section. All Neutral files are Formatted.
- Updated Neutral Read and Write for v10.0 changes

Interfaces - Nastran
- Added support for "-2..Automatic(Statics)" for INREL to the PARAM section of the NASTRAN Bulk Data Options dialog box.
- Added support for SUPORT1 to the Boundary Conditions dialog box.
- Added support for Fastener elements (CFAST) and properties (PFAST).
- Added support for spring/damper elements (CELAS1 and CDAMP1) which use a property (PELAS and PDAMP). Controlled via the Spring/Damper element formulation.
- Added Beam/Bar Cross-Section Dimensions as comments when Nastran input file is written. When a Nastran file with these comments is imported into FEMAP, the Beam/Bar Cross-Section Dimensions will be filled-in.
- Added support for reading Nastran Free-Field Auto Continuation (long entries with or without embedded continuation fields and large-field free field).
- Added support for reading CMETHOD from the case control

Interfaces - NX Nastran
- Added support for triangle and quadrilateral axisymmetric elements (CTRAX3, CTRAX6, CQUADX4, and CQUADX8), which were new for NX Nastran version 6.
- Added option for "Extended Solution Status Monitoring". Writes SYSTEM(442)=-1 to the *.dat file. This option is on by default and the feedback it produces is used by the NX Nastran Analysis Monitor.
- Added BOLTFACT to the PARAM section of the NASTRAN Bulk Data Options dialog box.
- Added "Gaps as Contact" to the "Plate, Beam, and Rigid" section of the NASTRAN Bulk Data Options dialog box. Writes out a BCSET entry in Case Control. Also added support for reading SYSTEM CELL 412 in the System Cell field of the Analysis Manager. This is the override to have gaps written as normal gaps even when using Contact.

- Added Support for CQUADR and CTRIAR Composite Stress and Strain output from the op2.
- Added "Large Strain Form" (ULFORM), "Incompatible Mode for 4 Node Shells" (ICMODE), "Max Disp/Iteration" (MAXDISP), and "Drilling DOF Factor" (DRILLKF) options to the Analysis Options section of NXSTRAT Solver Parameters dialog box.
- Added "Bolt Force Increments" (BOLTSTP), "Convert Dependency to True Stress" (CVSSVAL), and "Allow Element Rupture" (XTCURVE) options to the Other Parameters section of NXSTRAT Solver Parameters dialog box.
- Added "Line Search Lower Bound" (LSLOWER) and "Line Search Lower Bound" (LSUPPER) options to the Line Search Setting section of NXSTRAT Iterations and Convergence Parameters dialog box.
- Added "Do not allow Consistent Contact Forces" (TNSLCF) and "Use Old Rigid Target Algorithm" (RTALG=1) options to the Contact Control section of NXSTRAT Iterations and Convergence Parameters dialog box.
- Changed "Segment Type" (CSTYPE) options from "0..Old" and "1..New" to "0..Linear Contact" and "1..Element based" in the Contact Control section of NXSTRAT Iterations and Convergence Parameters dialog box.
- Added support for 2-D Contact, usually used in analysis with axisymmetric elements.
- Added support for Glued Contact.
- Added Contact Control section to NXSTRAT Solver Parameters dialog box. Added "Segment Type" (CSTYPE) and "Use Old Rigid Target Algorithm" (RTALG=1) to this section.
- Added Other Parameters section to NXSTRAT Solver Parameters dialog box. Added "Convert Dependency to True Stress" (CVSSVAL) and "Allow Element Rupture" (XTCURVE) options to this section.
- Added support for Initial and Final contact separation distance, which were new for version 6.0.
- Added reading of the SVDSPC from the Nastran command.

Interfaces - Ansys
- Added support for MPC184 rigid beam/link elements. Specified using element Formulation.
- Added support for output from rigid elements (Rigid Axial Force, Rigid Y Moment, Rigid Z Moment, Rigid Y Shear Force, Rigid Z Shear Force, and Rigid Torsional Moment)

Interfaces - DYNA
- Added support for 10-noded tetrahedral elements. Also, added "16..10 Node Tetrahedron - EQ 16" and "17..10 Node Composite Tetrahedron" formulations.
- Added support for Rigid and Interpolation elements. Writes *CONSTRAINED_NODAL_RIGID_BODY (Rigid) and *CONSTRAINED_INTERPOLATION (Interpolation) entries.

Interfaces - Geometry
- Added support for direct geometry import of SolidWorks parts and assemblies. Supports from SolidWorks 2000 - SolidWorks 2009.

- Changed CATIA V5 direct geometry translator. CATIA V5 versions R7 to R18 are supported. Reading of CATParts and CATProducts created using versions prior to R7 is not supported
- Added support for Parasolid 20.0
- Added support for Solid Edge with Synchronous Technology (version 21)
- Added support for NX 6
- Added support for Pro/Engineer Wildfire 4
- Added support for ACIS 19, Service Pack 1

Loads and Boundary Conditions

- Modified Directional Pressure loads to no longer be affected by choosing a particular element face.
- Added option to apply nodal constraints using the "-1..Use Nodal Output System" option when choosing a coordinate system.
- Updated Load Definitions. If a geometry load is applied to multiple curves at the same time, a double load will not be created on shared nodes.
-

Meshing

- Added 3 new patterns to Mesh, Editing, Interactive
- Added "Offset from Reference Point" option to Modify, Update Elements, Line Element Offsets.
- Added "Spring Elements" option to the Connection Type section of the Mesh, Connect, Unzip and Mesh, Connect, Coincident Link commands.
- Updated Mesh, Remesh, Convert Facets command to include capability to associate facets/nodes with the original geometry.
- Removed "Quad Mesh Layer Options" option from Mesh, Mesh Control, Size on Solid.
- Added "Suppress Short Edges" option to Mesh, Mesh Control, Size on Surface.
- Removed "Quad Mesh Layer Options" option from Mesh, Mesh Control, Size on Surface. This capability was improved and is now the Quad Edge Layers "mesh attribute" which can be specified before meshing using Mesh, Mesh Control, Attribute on Surface or during the meshing process using Mesh, Geometry, Surface.
- Added and updated many options found in the Mesh, Geometry, Surface command.
- Added new options for meshing surfaces which have already been meshed.
- Added Initial Size Ratio option to the Automesh Solids dialog box.
- Updated Adjust Nodal Precision option is to be on by default.
- Added Recovery Mesher (Use only if Standard Mesher fails) option to the Solid Automeshing Options. This option should ONLY be checked if the standard mesher has already failed.
- Added Update Data Table with Mesh Quality option to the Solid Automeshing Options.
- Updated the feedback sent to the Messages window during tet-meshing. FEMAP will produce status messages while the tetrahedral meshing is occurring and provide feedback on element numbers and quality.
- Added Offset from Reference Point to Modify, Update Elements, Line Element Offsets
- Updated Mesh, Extrude, Element Face command to automatically delete plot-only elements that it creates on the selected element faces.

Mesh Associativity

- Added the Modify, Associativity, Automatic command to attempt to automatically associate existing mesh to geometry.

Output and Post-Processing

- Added Transformation buttons for Deformation Vector and Contour Vector in the Select PostProcessing dialog box of the View, Select command. These allow for "on-the-fly" transformations of current output vectors.
- Added several options to the Model, Output, Transform command.

Properties

- Modified the Weld property to be the Weld/Fastener property.
- Added switch to specify if the property will used with CWELD (Weld) or CFAST (Fastener) elements. All Weld property inputs are the same as before.
- Added property inputs for CFAST (Fastener) elements.

Tools

Check, Coincident Elem...

- Added choice between Quick Check (Just Corners) and Full Check.
- Added Check Rigid Element option.

Check, Distortion...

- Added "Nastran Warping" and "Combined" Element Checks
- Added Permanent and Reset buttons to the Check Element Distortions dialog box.

User Interface

- Implemented support of the Astroid 3D controller from Spatial Freedom.
- Added support to create GIF, Animated GIF, TIFF, and PNG files when using File, Picture, Save command.
- Improved Curve and Surface facetting to more accurately display geometry.
- Renamed Weld Elements and Properties to Weld/Fastener
- Added automatic database recovery from failure during save (same as manual from File Preferences, but asks automatically when you start FEMAP)
- Added capability when reading files to detect that the file is open and locked by another application and then give option to Retry or Cancel the read.
- Added automatic Window Regenerate to end of Model, Load, Expand and Model, Constraint, Expand commands.
- Improved length-based spacing, distance along, and other length-based curve functions to perform better when highly nonlinear parametric domains exist on curves.

Preferences

Views

- Removed preference for Autoplot Created/Modified Geometry. FEMAP needs to do this in order to function properly.

Render

- Added preference for XOR Picking Graphics.
- Added preference for Dialog Refresh.
- Added preference for Block Size.

User Interface

- Updated how Load Layout works when loading a layout from an older version of the software into a newer version. If a *.LAYOUT file is loaded into a newer version of the software, only "Shortcut Keys" and "User Commands" will be updated, while "Menus and Toolbars" and "Panes" will not.

Geometry/Model

- Added "Construction Geometry - when used" preference.
- Added Output Orientation button which accesses the Current Output Orientation dialog box.
- Added Element Distortion button which accesses the Element Distortion Preferences dialog box.
- Added Pre-v10 Tet Meshing and Pre-v10 Surface Meshing preferences.

Interfaces

- Added Improve Single Field Precision option.

Colors

- Added preference for setting the default color of Combined Curves.

Spaceball

- Added preference for Print Debug Messages.

API

- Added NasExecSolutionMonitor, NasBulkInrelVal, NasBulkGapsAsContact, NasBulkBoltFact, and NasBulkBoltFactVal to AnalysisMgr object
- Added NasNXStratMaxDisp, NasNXStratBoltstp, NasNXStratCvssval, NasNXStratXtcurve, NasNXStratRtalg, NasNXStratTnslcf, NasNXStratDrillkf, NasNXStratLslower, and NasNXStratLsupper to AnalysisMgr object.
- Added InternalToBoundary and InCombinedCurve to Curve object.
- Added InternalToBoundary, attrTopology, attrMesher, attrMappedLevel, attrMapSubdivisions, attrMapEqualOnly, attrMapAltTri, attrMapRightBias, attrMapSplitQuads, attrMapAngleDeviation, attrMapMinCornerAngle, attrMidsideGeom, attrMidsideAngle, attrMinBetween, attrMaxAspect, attrQuickCutNodes, attrQuickCutAngle, attrSmoothLaplacian, attrSmoothIter,

attrSmoothTolerance, attrConnectEdgeNodes, attrConnectEdgeNodeTol, attrOffsetFrom, attrInitialized, and attrPostMeshCleanup to Surface object

- Added RotateCSys, TransformDeformMode, TransformDeformCSys, TransformDeformX, TransformDeformY, TransformDeformZ, TransformNodalMode, TransformNodalCSys, TransformPlateMode, TransformPlateCSys, TransformPlateDOF, vTransformPlateVector, TransformPlateVector, TransformSolidMode, and TransformSolidCSys to View object.

- Added Info_OrientSolidIsoOutput, Info_OrientSolidAnisoOutput, Info_OrientSolidHyperOutput, Info_OrientTria3StressOuput, Info_OrientTria3StrainOuput, Info_OrientTria3ForceOuput, Info_OrientTria6StressOuput, Info_OrientTria6StrainOuput, Info_OrientTria6ForceOuput, Info_OrientQuad4StressOuput, Info_OrientQuad4StrainOuput, Info_OrientQuad4ForceOuput, Info_OrientQuad8StressOuput, Info_OrientQuad8StrainOuput, Info_OrientQuad8ForceOuput to the Global Properties of the main FEMAP application object.

- Added Pref_ReadTabSize, PickBoundaryInternalMode, and PickCombinedCurveInternalMode to the Global Properties of the main FEMAP application object.

- Added SelectID, NextInSet, FirstInSet, and Count methods to the Common Entity Properties object

- Added OutputVectors method to the OutputSet object

- Added AnalyzeMultiple method to AnalysisMgr object

- Added GetMeshLoc, GetMeshLocXYZ, IsSmoothEdge, Surfaces, SurfacesAsSet, ElementsAsSet, NodesAsSet, Normal, IsCombinedCurve, GetCombinedCurves, CombineCurves, CombineCurvesAsSet, and Facets methods to Curve object

- Added AddOutput method to DataTable object

- Added GetCentroid, GetEdgeNodes, GetFaceNodes, and IsParabolic methods to Elem object

- Added Add method to Group object

- Added GetPly, SetPly, GetAllPly, and SetAllPly methods to Layup object

- Added InCombinedCurve, NodesAsSet, Curves, CurvesAsSet, and SurfacesAsSet methods to Point object

- Added SharedDelete, JumpToEnd, Size, Time Created, TimeWritten, and TimeAccessed methods to Read object

- Added RemoveNotCommon, RemoveNotCommonToGroup, RemoveGroup, Debug, IsSetAdded, ConvertToAllSurfaces, ConvertToBoundarySurfaces, ConvertToBoundarySurfacesOnly, ConvertToInternalSurfaces, ConvertToAllCurves, ConvertToCombinedCurves, ConvertToCombinedCurvesOnly, ConvertToInternalCurves, IsArrayAdded, HasCommon, and RemoveArray methods to Set object

- Added CurvesAsSet, SurfacesAsSet, ElementsAsSet, and NodesAsSet methods to Solid object

- Added Current method to Sort object

- Added NormalAtXYZ, NormalBox, BoundarySurfaces, AdjacentSurfaces, BoundarySurfacesAsSet, AdjacentSurfacesAsSet, CurvesAsSet, PointsAsSet, EndPointsAsSet, ElementsAsSet, NodesAsSet, and Solid methods to Surface object.

- Updated Curves and Surfaces methods of Solid Object.

- Updated Curves and Points methods of Surface Object.

- Added feAppModelDefragment

- Added feGetElementEdges

- Added feElementFreeEdge

- Added feElementFreeFace
- Added feSurfaceNormalDeviation
- Added feAddToolbarSubmenuSubmenu
- Added feBoundaryAddSurfaces
- Added feCoordVectorPlaneIntersect
- Added feSurfaceConvert
- Added feGroupMoveToLayer
- Added feBoundaryFromPoints
- Added feAutoMeshAssociativity
- Added feSolidStitchNoCleanup
- Added feAppVersion
- Modified feFilePictureSave to support new file types available in File, Picture, Save.
- Modified feOutputTransform to support new options available in Model, Output, Transform.
- Modified feRenumber to allow renumbering of Layups, Connectors, Regions, Connection Properties, Functions, Analysis Sets, and Layers.
- Modified feDelete to allow deleting of Layups, Analysis Sets in the Analysis Manager, Connection Properties, and Connectors.

Corrections

Licensing

- Corrected problem that caused a hidden FEMAP process to remain after you exited with File, Exit command if you were using network licensing and did not have a valid license. FEMAP was checking for a license during exit and hung the process.

Analysis Manager

- Fixed problem when a Nastran Static Analysis Set is created, then the Analysis Type is changed to Normal Modes. FEMAP was not removing the Load Set and Initial Conditions boundary conditions, which are not available for Normal Modes analysis.

Connection Properties, Regions, and Connectors

- Fixed problem migrating Contact properties to Connection Properties. The Contact properties for NX, Sinda, Ansys, Marc were not being migrated properly.
- Fixed problem reading Connection Regions from the neutral file. The ID offsets were ignored.
- Fixed problem when renumbering Coordinate Systems. The reference csys in Connection Regions were not being renumbered.
- Fixed problem when renumbering Materials. The Material references in Connection Properties were not being renumbered.
- Fixed problem when renumbering Load Sets. The Load Set references in Rotor connection regions were not being renumbered.

Groups and Layers

- Corrected problem with Group->Operations->Add Related that added extra entities into each group if you selected multiple groups for a single command

GUI - Dockable Panes

General Pane corrections

- Corrected how entities are deleted from the Model Info tree when pressing the delete key. Previously they were not using the proper procedure so undo did not work when using delete key.

Model Info tree

- Fixed problem deleting multiple Data Surfaces from the tree when one was loaded in the Data Surface Editor. Femap asked if it was "OK to delete" for each Data Surface instead of once for all selected
- Corrected several issues with next/prev in Model Info tree. When deleting entities, did not properly show/hide prev. Changed titles of next/prev from IDs to Next/Previous. Fixed proper hide/show of Next/Prev as you moved up and down list. No longer show CSys 0,1,2 always - just at the beginning. Added functionality of double clicking Next/Prev to move in list, not just right mouse menu.

Data Surface Editor

- Fixed problem deleting multiple Data Surfaces from the tree when one was loaded in the Data Surface Editor. Femap asked if it was "OK to delete" for each Data Surface instead of once for all selected
- Fixed problem interpolating using the arbitrary data surface when using a coordinate system other than Global Rectangular
- Changed setting of local CSys so coordinate picking in dialogs is in that local CSys.

Entity Editor

- Corrections to entries in Editor Help for Nastran.
- Fixed problem editing a RSPLINE element from Entity Editor, where the element lists were being mishandled.
- Fixed problem editing a Geometric Boundary Condition from the Load Definition.

Program File

- Stopped remembering "Previous Commands" while program file is running, so "Previous Command" reruns program if run from toolbar.
- Multi-select list boxes did not properly record/playback if a pick was made to clear the box after a selection was made and focus changed. This occurred in commands like Model, Load, Combine where more load sets and factors where repeatedly picked without leaving the dialog.

Interfaces - FEMAP Neutral

- Fixed error FEMAP v9.3+ unable to read neutral files from versions between v4.1 and v5.0 if they contained laminate properties.
- Only write TMG records to neutral file when writing the analysis model (not geometry model) and only if no group

Interfaces - Nastran

- Fixed problem requesting Random output. Added support for NX5.0 and MSC 2004 NORPRINT, RPRINT, RPUNCH. This caused problems in FEMAP since random output was written to the f06 file, which causes Femap to skip reading of the op2 file completely.
- Fixed problems reading the op2 file when unsupported composite output existed. FEMAP sometimes could skip supported output in addition to the unsupported output.
- Enhanced FEMAP to support reading up to 50,000 time steps from the f06 and now issues a error when exceeding the number of supports steps.
- Changed entry length limit from MAX_STR_LEN to 1000.
- Fixed problem when skipping the UM field on RBE3's.
- Fixed problem reading AUTOSPC,NO that caused Femap to write out two AUTOSPC entries when file was exported out again.
- Corrected reading of PCOMP if all plies are specified in a single column. Previously aborted reading as soon as it encountered a missing ply.
- Corrected several issues with checksum on Nastran files when using INCLUDE files - had a problem with spaces at the front or back of a line, tabs and blank lines.
- Corrected reading of Nastran OP2 file from Design Optimization analyses. Previously some results data could be missed.
- Fixed problem when writing only entities in a group to Nastran. No geometry based BC we being written.
- Fixed problem writing Design Optimization constraints for CTRIA elements.
- Corrected problem reading op2 files with time steps smaller than 1E-7. Changed to 1E-15, so FEMAP can read the time steps it can write.
- Fixed reading of PWELD elements which were reading properly, but issuing error messages indicating they had been skipped.
- Fixed problem writing LOAD card. When applying only a GRAV load an extra load field was written on the LOAD entry.
- Fixed problem where density for MAT4, MAT5, MATHP, MATHE, MAT10 were not being converted with WTMASS during import of Nastran files.
- Corrected problem where Femap was incorrectly reading End B of a PBEAML.
- Added error if Initial Yield Stress was zero for a plastic material using Von Mises or Tresca criterion.

Interfaces - NX Nastran

- Fixed problem where Femap was incorrectly finding ADINA messages C O R R E S P O N D I N G D I S P L A C E M E N T and L O A D V E C T O R M U L T I P L I E R in the f06 and causing the op2 to fail to read.
- Fixed problem in SOL 701, where TSTEP was not written when only an Initial Conditions boundary conditions set was chosen.
- Fixed problem writing BOLTFOR in SOL 601. FEMAP was using the dynamic loads set rather than the one specific to bolt load, which is setup when writing the case control.
- Fixed problem writing BCTPARM entry. REFINE and INIPENE were being written to the glue set. Only affected BCTPARM when no other options were written.
- Changed reading of NXN results. The output destination defined in the analysis case will now be used to determine where FEMAP should read results from. Warnings will still be read from f06 but if PRINT is not explicitly selected then results will be read from other output files regardless if any valid output exists in f06 file, expect XY PRINT data, which will still be read.
- Suppressed writing of METHOD field of TSTEPNL.

Interfaces - NEi Nastran

- Fixed the problem where FEMAP was not writing out the proper DPHASE entries for frequency response analysis when translating to NEi Nastran

Interfaces - Ansys

- Corrected problem writing Transient, NLTransient, Transient heat transfer and Frequency response. A solve command was being written at the end of these solutions which cased Ansys to sometime overwrite the good results that were calculated from the analysis.
- Corrected a problem writing elemental convection loads, where the bulk temperature was written to the wrong face.
- Fixed problem reading Ansys elements when no real constants were required for that element type.

Interfaces - Abaqus

- Fixed problem reading element continuation lines when the data line contained a single fixed format item.

Listing

- Corrected listing of Geometry loads to list in definition CSys instead of global CSys
- Corrected "Curve using Point" listing method to work properly for all solid curves. Previously, only selected curves that referenced points in the point list.
- Corrected problem that caused listing of Constraint Definitions to fail if you had the List, Destination set to Printer.

Loads and Boundary Conditions

- Fixed problem setting nodal output Csys to 1 or 2 for constraint expansion when Arbitrary in CSys option is used.
- Fixed problem when multiple Constraint Definitions were defined on the same geometric entity. Constraints are deleted but the Constraint Definition was not updated.
- Fixed problem expanding nodal temperatures with a data surface. If load evaluated to zero it was not being saved properly.
- Fixed problem editing face of a Surface Load from the Load Definition.

Meshing

- Fixed problem where a mesh consisting of parabolic beams is created, then converted to linear elements. The converted linear beams would not be written to NX Nastran.
- Corrected an issue introduced in v9.3.1 that prevented Modify, Move By, Offset Element from working.

Tools

Check, Coincident Curves

- Updated Tools, Check, Coincident Curves command to properly renumber boundary surfaces and update the reversed state when they contain curves that are being merged. Previously the boundaries were deleted.

Check, Sum Forces

- Corrected issue where pressure loads were being summed incorrectly. This error would occurred when applying corner pressures to the triangular faces of solid tetrahedral or wedge elements.

User Interface

- Corrected a problem that could leave the progress bar displayed after aborting a mesh on a bad surface/boundary.
- Fixed Element checking to automatically zero extra nodes if fixup is allowed - previously prevented copying rigid elements that had a second node set.
- Expanded width of strings allowable in error, Print... so long errors like in Measure Distance do not get truncated.
- Removed error messages for zero length elements that are valid for that element type
- Stop ESC key from ending Message Boxes that dont have the Cancel button - previously ended Yes/No boxes with Yes.
- Modified custom tools menu so that it processes like a regular command and the tree gets updated

Preferences

- Fixed a problem when a user chooses a new library. If the library fails to load because it is the wrong type Femap was still saving the bad library path to the preferences.

API

- Corrected a problem with API method feRenumber and feRenumberOpt when you tried to renumber Solids or Volumes that would corrupt the database (did not renumber the Solid_Volume records)
- Corrected GetTitleIDList so that it can retrieve the global coordinate system IDs and titles.
- Corrected problem in DataTable API that created extra rows if you called AddColumn with duplicate IDs in the array that you passed.
- Corrected problem with feMoveTo. New coordinates were previously required to be in Global Coordinates, not in the specified coordinate system as documented.
- Fixed API problem where the Set object did not persist in some cases when using the select or add methods.
- Fixed several problems which caused the Outline property (shape of the beam property when using a General Cross-Section) of the Property object to not work.
- Fixed problem in feMoveOffset that caused it to fail if you did not use Set 1.