FEMAP v2021.1 MP 1 New Features and Corrections

Updates and Enhancements

Interfaces - Simcenter Nastran (formally NX Nastran)

SOL 401/402

- Added support to write the CRSTRN case control entry to the input file if either the Total Strain or Elastic Strain output request is enabled in the NASTRAN Output Requests dialog box.

Corrections

Geometry

- Corrected an issue where curves and surfaces that cannot be properly faceted by the geometry engine was causing faceting to be attempted multiple times.

User Interface - General

- Corrected an issue when using the New Picking option in File, Preferences which allowed entities of the pick type to be selected even though the entities were on non-visible layers (PR# 9960532).

User Interface - Dockable Panes

Meshing Toolbox - Geometry Editing Tool

- Corrected an issue where solids were not being remeshed properly when Operation is set to Washer, the Split Solid option is enabled, and multiple curves on internal holes are selected in a single usage. Solid Washer remeshing now works as expected.

Interfaces - Simcenter Nastran (formally NX Nastran)

- Corrected an issue which caused duplicate PLSTRN output requests to be written to the Case Control section of the input file if an imported input file originally contained the PLSTRN command (PR# 9952909).
- Corrected an issue which prevented the STRMEAS parameter from being written to the MATS1 entry when Analysis Type is set to “27..Multi-Step Structural”.

Materials

- Corrected an issue with material reference counters that could allow materials to be deleted while still referenced by properties. In 2021.1 MP 1, use the File, Rebuild command to rebuild and repair any corrupted material counters.
- Corrected an issue with material references which could cause the “by Material” option on the Method^ menu of the standard entity selection dialog box and/or any of the “Material” commands on the Group menu to fail.

Output and Post-Processing

- Corrected an issue where every attempt to interact with the drop-down in the Show Stress section of the Beam Cross Section Stress Control dialog box caused the beam cross section stresses to be re-evaluated.

API

- Corrected an issue in the feSetToolbarCommandBitmap call that prevented a new icon from being assigned once an icon was already assigned to a toolbar command. Also, corrected similar issues in feAddToolbarUserCommand and feAddToolbarSubmenuUserCommand (PR# 9961857).
- Corrected an issue which caused User Defined Graphics to be regenerated more often than required.

FEMAP v2021.1 New Features and Corrections

Updates and Enhancements

Connection Properties, Regions, and Connectors

- Added Check for Connection to Internal Edges option to Connect, Automatic command, which is only available when Edge - Edge is enabled in the Connection Types section of the Auto Detection Options for Connections dialog box. When enabled, allows connections to be automatically created between any combination of free edges an internal edges (i.e., non-free edges) of surfaces (sheet solids), solids, and general bodies which are considered coincident based on the value specified for Tolerance in the Detection Options section. If disabled, which is the default, no connections will be made between any edge and any internal edge.
- Added Surfaces to the Output section for Connection Regions. Only available when Defined By is set to Surfaces and currently only used by MSC Nastran.
- Updated the Connection Region Options dialog box to have three tabs: Simcenter Nastran, MSC Nastran, and ABAQUS. The options on the Simcenter Nastran and ABAQUS tabs were available in previous versions of FEMAP, but all of the options on the MSC Nastran tab other than Friction Value (FRIC) are new for 2021.1.
- Added Norm Scale Fac (SFNPNLT) and Tang Scale Fac (SFTPNLT) options to Contact section of MSC Nastran Advanced Contact Options dialog box accessed by the Advanced Options button on the MSC Nastran tab.
- Added Cohesive Options button to Glue Break section of MSC Nastran Advanced Contact Options dialog box accessed by the Advanced Options button on the MSC Nastran tab.
- Added Interference Options button to MSC Nastran Advanced Contact Options dialog box accessed by the Advanced Options button on the MSC Nastran tab.

Geometry

- Added Overlap Repair option to the Automatic Midsurface Generation dialog box accessed by the Geometry, Midsurface, Automatic command.
- Enhanced Geometry, Curve - Centerline to better handle more complex beam-like solids, including helical and swept shapes created from complex splines.

Performance Graphics

- Added support for RBE1 elements when using Best Available or Performance Graphics.

User Interface - General

- Added functionality to hide the list of entity ranges in the standard selection dialog box to improve performance when more than 1000 unique ID ranges have been selected. Once more then 1000 “unique ID ranges” are selected, the list of selected entities is replaced by text stating “(number of unique ranges) Ranges Selected” and “Show List>>>”. To display the entire list of ranges, click anywhere in the list area. To hide the list again, click the Hide List button which appears to the left of the list area.

- Added the Match Color^ button menu to the Color Palette, which selects a color by matching the color of an existing entity.
- Improved handling of "repeated error messages" to catch cases where multiple errors were repeated in a pattern. This can dramatically improve import times of invalid or incomplete Nastran models.
- Enhanced performance of picking, especially front picking (up to 15 times faster) and area picking (up to 10 times faster), when using the New Picking option on the Graphics tab in the Preferences dialog box. New picking also takes advantage of OpenMP technology.

User Interface - Dockable Panes

Mesh Control Explorer - New for 2021.1!

- Added Mesh Control Explorer to perform a number of different tasks, including specification of options for mesh propagation, visualization of slaved surfaces for solid meshing and/or mesh approaches on surfaces, and determining if curves are paired by a mesh approach and/or can be automatically paired based on proximity when mesh sizing.

Meshing Toolbox - Overall

- Added icon to open the Mesh Control Explorer dockable pane from within the Meshing Toolbox.

Meshing Toolbox - Mesh Sizing tool

- Added ability to update hex mesh on multiple solids by selecting a single curve and having the mesh propagate correctly through all required solids. Previously, if all curves required to update the hex mesh were not selected, hex meshing would fail.

Meshing Toolbox - Mesh Surface tool

- Added ability to update hex mesh on multiple solids by changing the surface mesh on a single surface and having the mesh propagate correctly through all required solids.

Function/Table Editor

- Added Connect To Excel command which is only available after the Send To Excel command on the Copy to Clipboard menu has been used to send data to Excel. Connect to Excel opens the Excel Connection dialog box. When the Live option is enabled, any additions and modifications done in Excel are automatically reflected in the Function/Table Editor. If the Live option is disabled, then additions and modifications can be committed to the Function/Table Editor by using the Update button.

Interfaces - FEMAP Neutral

- Updated Neutral Read and Write for v2021.1 changes

Interfaces - Nastran

- Added read/write support for MODSEL (Simcenter Nastran) and a subset of the functionality of MODESELECT (MSC Nastran). This can be specified using the Response Mode Selection (a, b, c THRU d) section in the NASTRAN Modal Analysis dialog box in the Analysis Set Manager when

creating an Analysis Set with Analysis Type set to “3..Transient Dynamic/Time History” or “4..Frequency/Harmonic Response”.

Interfaces - Simcenter Nastran (formally NX Nastran)

DDAM

- Added read/write support for SELMODE entry. This is specified using the Mode Selection section in the NASTRAN DDAM Solution Options dialog box in the Analysis Set Manager.
- Added read support for MPC Forces, Element Strains, and Membrane-only Stresses for Beams, Shells, and Laminates. These results are requested by PARAM,SHLMEM,YES.

SOL 401/402

- Added support for post-processing of Creep Strain results in SOL401/SOL402 (PR/ER# 9930657)
- Added support for post-processing of Bolt Relative Shortening results in SOL401/SOL402 (PR/ER# 9913500)

Bolt Preloads

- Added a warning message for Analysis Sets with Cases selecting FEMAP Load Combinations containing Bolt Preloads. A message will now be issued to confirm that the Bolt Preloads will not be included in the Load Combination.

Interfaces - MSC Nastran

SOL 400 Overview

- Added support for Analysis Type “30..Implicit Nonlinear”, which is used to setup analysis jobs using SOL 400. To support SOL 400, new dialog boxes have been created in the Analysis Set Manager to enter options for Nonlinear Behavior (Nonlinear Behavior Options), Automatic Contact Parameters (Automatic Contact Generation), and Control Parameters (NASTRAN Implicit Nonlinear Analysis). In addition, new options were added to Contact Parameters (MSC Nastran Contact Solver Parameters). Finally, two new entity types, Analysis Step (Analysis Step dialog box) and Global Step (Global Analysis Steps dialog box), have been created to perform analysis chaining within a particular subcase.

SOL 400 Elements

- Added read/write support for CIFHEX and CIFPENT (Solid Interface Cohesive Zone Modeling Elements) entries.

SOL 400 Materials

- Added read/write support for MATEP (Elastic-Plastic Material), MATTEP (Thermo-Elastic-Plastic Material), MATF (Material Failure Model), MATORT (Elastic 3D Orthotropic Material), MATG (Gasket Material), and MCOHE (Interface Cohesive Zone Modeling Material) entries.

SOL 400 Properties and Property Extensions

- Added read/write support for PCOMPLS (Layer Solid Composite Property) and PCOHE (Solid Cohesive properties (Interface Cohesive Zone Modeling Property) entries. For both PCOMPLS and PCOHE, some of the items are specified in FEMAP using the Property Extensions dialog box.
- Added read/write support for PRODN1 (Rod Property Extension), PBARN1 (Bar Property Extension), PBEAMN1 (Bar Property Extension), PSHLN1 (Membrane, Bending, Plate, or Laminated Property Extension), PSHLN2 (Plane Strain or Axisymmetric Property Extension), and PSLDN1 (Solid Property Extension).

Contact

- Added read/write support for BCAUTOP, BCDPRP, BCBODY1, BCONECT, BCONPRG, BCONPRP, BCNURBS, BCNURB2, BCRIGID, BCRGSRF, BCPATCH, BCTRIM, BSURF entries. Also, expanded support for BCPARA.
- Added the ability to allow MSC Nastran to automatically compute the "ERROR" parameter on the BCTABLE/BCONPRG entries for contact analysis

Interfaces - ABAQUS

- Added support to maintain the case of names during import of an input file.
- Added support to remove the leading characters of names during export, thus maintaining the names of entities entered by the user.

Interfaces - Geometry

- Added support for Parasolid 33.0, NX 2020 (version 1926), and Solid Edge 2021.

Materials

- Added "607..MSC.Nastran Cohesive Material (MCOHE SOL400)" to Other Types to support the MCOHE entry for MSC Nastran SOL 400.
- Added "608..MSC.Nastran Gasket Material (MATG SOL400, SOL 600)" to Other Types to support the MATG entry for MSC Nastran SOL 400 and SOL 600.

Properties

- Added Define Property Extensions icon button to the Define Property dialog boxes for a number of different property types. When clicked, the Property Extensions dialog box is displayed, which allows specification of solver-specific items for the current property type. The options available change from property type-to-property type. The "Filter" icon button can be used to limit the tabs which are displayed in the Property Extensions dialog box.

Loads and Boundary Conditions

- Added Equation Coord Sys drop-down to Create Loads dialog box for Model, Load, Nodal; Model, Load, Nodal on Face; Model, Load, Elemental; and Model, Load, Elemental on Face commands. The Equation Coord Sys drop-down can be used to specify the coordinate system in which equations are evaluated when defining nodal or elemental loads when Method is set to Variable. By

default, this option is set to the active coordinate system in the model, not to match the Definition Coord Sys.

Meshing

- Added a Size Propagation section to the dialog box for the Mesh, Mesh Control, Size on Curve; Mesh, Mesh Control, Size on Surface; and Mesh, Mesh Control, Size on Solid commands. In each command, there is Propagate Sizing option and an icon button which accesses the Mesh Sizing Propagation Options dialog box. The Propagate Sizing option can also be enabled and the other options specified via the Mesh Control Explorer dockable pane, which was added for 2021.1.
- Added Detect Surface Poles option to "Mapped" approaches in the Surface Mesh Approach dialog box. When enabled, the location of "surface poles" will be detected and the selected points may be reordered to create a higher-quality mesh. When disabled, the points are always used as entered.

Listing

- Added Evaluate Group to insure Entity Selection is valid option to List, Groups command. This option is on by default, but only used when the List Entities in Group option is enabled. Simply performs the Group, Operations, Evaluate command on all selected groups before listing the entities in each group.

Output and Post-Processing

- Added Linear and Parabolic Types option to Contour Options dialog box, which controls if averaging of results occurs across boundaries between linear and parabolic solid elements.

Tools

- Added Interference Factor/Proximity Factor option to the Tools, Check, Mesh Interference command. In addition, consolidated the user interface to use a single dialog box instead of a series of questions. When Interference Factor is selected in the Checking Options section, an element has to overlap at least one other element by more than the specified value for the elements to be considered interfering. When Proximity Factor is selected, it acts in an opposite manner to Interference Factor, meaning if a gap smaller than the specified value exists between and element and any other element, those elements are considered interfering.

Preferences

Graphics

- Added New Picking to Graphics Options section. When enabled, which is the default, New Picking uses new picking introduced in version 2021.1 which takes advantage of "Open MP" to improve performance when selecting a large number of entities from the graphics window, especially when using area picking (Box, Circle, Polygon, or Freehand) and/or Front Pick. Unless there is a problem picking entities, New Picking should remain enabled. Fast Picking uses older picking methods, but for the time being should also remain enabled. Eventually, New Picking will become the only available option and these preferences will be removed.

- Updated OpenMP in the Advanced/Debug Options section.to be enabled by default. In addition, now set a default value of “4” for Number of Threads. This functionality has been implemented to improve performance for a limited number of commands and operations including: New Picking; View, Advanced Post, Dynamic IsoSurface; View, Advanced Post, Dynamic Cutting Plane; and placing a contour on certain types of entities. **Note: If any issues are encountered when using these options, please disable this option, then contact Customer Support to describe the issue.**

Interfaces

- Added Enable Analysis Filtering option and Select Analysis Filters icon button, which accesses the Select Solvers to Filter dialog box. When enabled, this option is used in conjunction with the solvers selected using the Select Analysis Filters icon button to limit solver-specific options found in various locations throughout FEMAP, including dialog boxes and dockable panes. Solvers which can be filtered are MSC Nastran, Autodesk Nastran, LS-DYNA, ABAQUS, ANSYS, SINDA, and MARC. For Simcenter Nastran, certain analysis types can be filtered, including Linear (Contact options including glued contact for most solution sequences), Multistep Structural (SOL 401), Multistep Kinematic (SOL 402), Advanced Nonlinear (SOL 601), and/or Explicit (SOL 701).

API

New and updated API Objects and Attributes

- Added PublishTool (fePublishTool) object to the API. In addition, added PublishTable, which can be used by the PublishTable object to the API. PublishTool and PublishTable have no attributes, only methods.
- Added AnalysisStep (feAnalysisStep) object to the API.
- Added StepTitle, StepLabel, StepID, InternalStepID, StepAnalysisType, UseGlobalStep, GlobalStepID, CaseID, SkipStandard, NasStepAlternateTextLocation, StartText, and EndText attributes to specify options in the MSC Nastran SOL400 Analysis Step dialog box.
- Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlStepOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlStepOptCREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlStepOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEnableCriteria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG, NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlStepOptYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasInlStepOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInlStepOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlStepOptConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptKMETHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOptMAXLS,

NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters in an Analysis Step.

- Added NasModeOn, NasModeEigrSet, NasModeMethod, NasModeSolutionType, NasModeEstRoots, NasModeDesiredRoots, NasModeNormOpt, NasModeNormNode, NasModeNormDOF, NasModeMassForm, vNasModeFreqRange, NasModesSkipEigr, vNasModeImagFreqRange, NasModesComplexConverge, NasModesComplexRegionWidth, and NasModeDampOverall in the NASTRAN Modal Analysis dialog box for an Analysis Step.
- Added BCSet, vBCSet, ContactOption, and ContactSetType attributes to specify options in the Boundary Conditions dialog box for an Analysis Step.
- Added Output, vOutput, Echo, Destination, Imaginary, CornerOutput, IntegrationOutput, NasOmodesOn, NasOmodes, and NasCurvatureStrain attributes to specify options in the NASTRAN Output Requests dialog box for an Analysis Step.
- Added GlobalStep (feGlobalStep) object to the API.
- Added GlobalStepTitle, GlobalStepLabel, globalstepID, GlobalStepAnalysisType, SkipStandard, NasGlobalAlternateTextLocation, StartText, and EndText attributes to specify options in the MSC Nastran SOL400 Global Analysis Step dialog box.
- Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlStepOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlStepOptCREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlStepOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEnableCriteria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG, NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlStepOptYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasInlStepOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInlStepOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlStepOptConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptKMETHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOptMAXLS, NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters in a Global Step.
- Added NasModeOn, NasModeEigrSet, NasModeMethod, NasModeSolutionType, NasModeEstRoots, NasModeDesiredRoots, NasModeNormOpt, NasModeNormNode, NasModeNormDOF, NasModeMassForm, vNasModeFreqRange, NasModesSkipEigr, vNasModeImagFreqRange, NasModesComplexConverge, NasModesComplexRegionWidth, and NasModeDampOverall in the NASTRAN Modal Analysis dialog box for a Global Step.
- Added BCSet, vBCSet, ContactOption, and ContactSetType attributes to specify options in the Boundary Conditions dialog box for a Global Step.

- Added Output, vOutput, Echo, Destination, Imaginary, CornerOutput, IntegrationOutput, NasOmodesOn, NasOmodes, and NasCurvatureStrain attributes to specify options in the NASTRAN Output Requests dialog box for a Global Step.
- Added NasModeSelOn, NasModeSelInc, and NasModeSel attributes to specify options in the NASTRAN Modal Analysis dialog box via an Analysis Manager Object.
- Added NasDdamSelModeMethod and NasDdamSelModeVal attributes to specify options in the NASTRAN DDAM Solution Options dialog box via an Analysis Manager Object.
- Added NasNonlinBehaviorOn, NasNonlinBehaviorCreepExp, NasNonlinBehaviorEnable, NasNonlinBehaviorCreepType, NasNonlinBehaviorCreepImp, NasNonlinBehaviorASSM, NasNonlinBehaviorTSHEAR, NasNonlinBehaviorLRGSTRN, NasNonlinBehaviorPROPMAP, NasNonlinBehaviorPROPBEH, NasNonlinBehaviorDIRECT, NasNonlinBehaviorTHICKOP, NasNonlinBehaviorIPRINT, NasNonlinBehaviorVALD1, NasNonlinBehaviorVALD2, NasNonlinBehaviornINLAM, NasNonlinBehaviorCreepOn, NasNonlinBehaviorCPROJ attributes to specify options in the Nonlinear Behavior Options dialog box via an Analysis Manager Object.
- Added NasAutoCtOn, NasAutoCtEnabled, NasAutoCtBodyVal, NasAutoCtCTYPE, NasAutoCtDISTANCE, NasAutoCtESET, NasAutoCtSeedESet, NasAutoCtSeedGSet, NasAutoCtPropSet, NasAutoCtRigidSet, NasAutoCtBEAMCNT, NasAutoCtBEAMCAP, NasAutoCt2D, NasAutoCtEDGECNT, NasAutoCtFTYPE, NasAutoCtFRIC, NasAutoCtFRICTABLE, NasAutoCtFRICVal, NasAutoCtIGNTHK, NasAutoCtINISTF, NasAutoCtSelfCont, NasAutoCtRegion, and NasAutoCtConProp attributes to specify options in the Automatic Contact Generation dialog box via an Analysis Manager Object.
- Added NasMscCtTHKOFF, NasMscCtLINCNT, NasMscCtSFNPNLT, NasMscCtSFTPNLT, NasMscCtFGCFLG, NasMscCtFGCNST, NasMscCtFGCTST, NasMscCtFGCNSTR, NasMscCtFGCTSTR, NasMscCtFGCNST1, NasMscCtFGCTST1, and NasMscCtFGCRCEN attributes to specify options in the MSC Nastran Contact Solver Parameters dialog box via an Analysis Manager Object.
- Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlStepOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlStepOptCREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlStepOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEnableCriteria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG, NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlStepOptYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasInlStepOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInlStepOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlStepOptConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptKMETHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOptMAXLS,

NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters in the Master Case of an Analysis Set via Analysis Manager Object.

- Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlStepOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlStepOptCREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlStepOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEnableCriteria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG, NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlStepOptYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasInlStepOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInlStepOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlStepOptConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptKMETHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOptMAXLS, NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters via an Analysis Case Object.
- Added SurfaceContactFlag, FrictionType, FrictionValue, FrictionTable, SmoothingControl, ContactCheck, MidNodeProjection, DiscontinuityValue, ControlNode, MotControlType, RotControlNode, AngVelFlag, AngVelTable, AngVelValue, RotAxisFlag, RotAxisTable, RotAxisValue, MotionFlag, MotionTable, MotionValue, AppAngVelVal, AppRotAxisVal, AppVelVal, GrowthComp, and GrowthTable attributes to the Connection Region Object.
- Added ContourElemBreakLinearParabolic attribute to the View Object.

New and Updated API Methods

- Added InitAnalysisStep, DeleteAnalysisStep, and ActiveContactEntities methods to the AnalysisStep Object.
- Added InitGlobalStep, DeleteGlobalStep, and ActiveContactEntities methods to the GlobalStep Object.
- Added Clear, SetFormat, SetParagraphIndent, SetParagraphAlignment, SetParagraphSpacing, AddStyle, AddParagraph, AddPicture, AddView, AddDataTable, AddFunctionTableEditor, AddEntityEditor, AddMessages, AddChart, UserTable, PublishToClipboard, and PublishToFile methods to the PublishTool Object. In addition, added AddColumn, AddColumnGroupTitle, AddRowNumbering, SetRealFormat,SetTableStyle, SetRowTitle, SetCellStyle, SetCellReal, SetCellInteger, SetCellText, SetMultiCellStyle, SetMultiCellReal, SetMultiCellInteger, SetMultiCellText, and Publish methods for PublishTable.
- Added IsNotEmpty, HasOne, and HasMultiple methods to the Set Object.
- Added GetStep, PutStep, and DeleteStep for MSC Nastran SOL 400 to Analysis Case Object.

- Added RemoveDuplicates method to the Connection Region Object.
- Added IsFreeEdge and IsOnSolidRegion methods to the Curve Object.

New and Updated Global Variables

- Added Pref_NewPick to set the New Picking preference on the Graphics tab of File, Preferences.
- Added Pref_Filter_Analysis to set the Enable Analysis Filtering option on the Interfaces tab of File, Preferences. In addition, added Pref_Filter_SC_Nastran_Linear, Pref_Filter_SC_Nastran_MSNL, Pref_Filter_SC_Nastran_MSNLK, Pref_Filter_SC_Nastran_ADVNL, Pref_Filter_SC_Nastran_ADVNLE, Pref_Filter_MSC_Nastran, Pref_Filter_Autodesk_Nastran, Pref_Filter_ABAQUS, Pref_Filter_ANSYS, Pref_Filter_DYNA, Pref_Filter_SINDA, and Pref_Filter_MARC to control the individual options in the Select Solvers to Filter dialog box.
- Added Info_MeshSizePropagation_On to enable the Propagate Sizing option in the dialog boxes for various Mesh Sizing commands. In addition, added Info_MeshSizePropagation_SelMode, Info_MeshSizePropagation_SlavedMapped, Info_MeshSizePropagation_EdgeOption, Info_MeshSizePropagation_SameSolid, Info_MeshSizePropagation_UseTolerance, and Info_MeshSizePropagation_Tolerance to control the options in the Mesh Sizing Propagation Options dialog box.
- Added Pref_API_UseAlternateFileDialog and Pref_API_HonorWorkingDirectory to control specify options needed for improved access for Teamcenter Simulation (TC Sim).

The following functions have been added or updated:

- feAppLoadColorPalette
- feAppLoadContourPalette
- feGroupCondense
- feMeshSizeCurve2
- feMeshSizePropagate
- feSurfaceMidAuto4
- feCheckElemInterference2

Corrections

Connection Properties, Regions, and Connectors

- Corrected an issue that occurred with the Connect, Automatic command. The command should skip over creating regions between surfaces that have already had Matched - Link to Surface meshing approaches assigned. In some cases however, when there were multiple surfaces connecting two solids, connection regions were still being created.

Geometry

- Corrected an issue where the color, layer, loads, constraints, etc. on solids that were being subdivided were not propagating to the new solids.

- Corrected an issue when projecting curves on to Surfaces where tolerances would become mismatched and the projection would fail. Overall, projecting curves on to surfaces is now somewhat more robust then it was in previous versions of FEMAP.

Graphics

- Corrected an issue in Geometry, Solid, Embed Face command where if you cancel out of the dialog to set the offset value, the preview vector is not removed from the graphics window even after a Window, Regenerate. The preview arrow is then drawn in all windows of all models until a different command is used that uses the preview vector.
- Corrected an issue where solids in groups are causing curves and surfaces to be drawn even if Solids are off.
- Corrected an issue where feature lines on drawn elements were being drawn when all elements were off.
- Corrected an issue where highlighting coordinate systems drawn as solid RGB arrows did not work.
- Corrected an issue where picked entity markers are not cleared when canceling out of the standard entity selection dialog box if the user had used the mouse wheel to zoom previously.
- Corrected an issue when picking if there are two views in one model and one view was deformed and the other not. Picking was not handling deformation correctly resulting in incorrect elements selected.

Performance Graphics

- Corrected an issue when using Fast Picking to pick Coordinate Systems. You should be able to pick nodes/points to get the definition coordinate system or pick nodes to get the output coordinate system.
- Corrected an issue where forced convection loads were not drawn correctly.
- Corrected an issue which could occur when Fast Pick Visible was enabled along with front picking. If user dynamically rotates the model while picking, the elements can become all the color white. This only happened if a load set and a constraint set existed in the model (PR# 9933100).
- Corrected an issue in where extra points are drawn when using the Geometry, Solid, Extend command.

User Interface - General

- Corrected an issue that could cause animation frames to not be rendered as expected if Color Diffusion (Dither) option was chosen in for GIF Options in the Picture Save Defaults section of the Preferences dialog box.
- Corrected a number of issues when scaling on high-DPI displays, including in the Visibility dialog list boxes, Model Info tree, Meshing Toolbox, and post-processing toolboxes.
- Corrected an issue in dialog boxes where icons drawn on buttons may not always be scaled according to the display DPI.

- Corrected an issue with font sizes on high-DPI displays in dialog boxes where fixed-width fonts were used which included the Analysis Monitor, Analysis Preview window, and Analysis Manager Start/End Text windows.
- Corrected an issue on freebody page of Visibility Dialog where position of “Show Visible” check box would not update when dialog was resized.
- Corrected an issue where Femap would exit unexpectedly when using the Tools, Mass Properties, Mesh Properties or Tools, Mass Properties, Solid Properties command with the Copy to Clipboard option enabled when using the localized Japanese version of Femap.

User Interface - Dockable Panes

Meshing Toolbox - General

- Corrected an issue the Meshing Toolbox which caused the selection icons to become unavailable when switching between multiple models. Previously, you could be locked out of using certain tools or could use tools without complete specification of required options.
- Corrected an issue which caused FEMAP to exit unexpetedly if you closed a model while the Meshing Toolbox selector was active in either the Feature Editing or Geometry Editing tools.

Meshing Toolbox - Feature Editing Tool

- Corrected an issue in Meshing Toolbox when editing surface features with no value specified in the toolbox. The surface would appear edited even if the user canceled out of the dialog box asking for a feature value.

Meshing Toolbox - Geometry Editing Tool

- Corrected an issue when Operation is set to Slice in the Geometry Editing tool where the curve selector was being set to “Internal Loops Only”, which made it impossible to use Smart Snap or certain options on the Method^ menu on a curve that was not an internal loop.

Meshing Toolbox - Mesh Surface Tool

- Corrected an issue that occurred in some cases if you attempted to apply mapped mesh and mesh size at the same time using the Mesh Surface tool in the meshing toolbox. If the surface had multiple curves along one of its mapped sides the distribution of elements was sometimes skewed incorrectly toward one curve.

Entity Editor

- Corrected issues where items were misspelled in the Entity Editor (PR# 9780911)

Data Table

- Corrected an issue with the List, Output, Contoured Results to Data Table command. While there was nothing wrong with the information that was added to the Data Table, if after using this command the Selector was used to add Element data to the Data Table, the table was not properly cleared before the Element data was added.

Interfaces - FEMAP Neutral

- Corrected issues where Femap 2020.1 models that passed through the external neutral write translator would lose all saved sets and Femap lists, containing large IDs.
- Corrected an issue when reading neutral files with Ansys Analysis Sets which caused the Portion of the Model to Write and Skip Standard items to become corrupt.
- Corrected an issue that caused Parabolic typed laminate plates to not be imported correctly from the neutral file written from version 9.3 and above. In these cases, Femap would create new Layups with invalid data. This also caused the Laminate bottom surface option to be disabled if the Offset Bottom Surface value was equal to 0.0 even if it was enabled in the source neutral file.

Interfaces - Nastran

- Corrected an issue writing and reading shear center offsets on Nastran PBEAM entries. This problem was introduced when the "Write Zeros at End B" flag was created in the User Interface for tapered beams. Shear Center offsets are not tapered properties and as such should not and no longer follow this option. If you specify a 0.0 value for a Shear Center value it is written as a zero, not a blank. You must specify duplicate values to get the same offset at both ends. When reading a Nastran file, FEMAP now properly sets blank values for the End B Shear Center values equal to the End A values.
- Corrected an issue with incorrect line breaks for EXTSEOUT case control command when the line was longer than 72 characters. (PR# 9786335)
- Corrected an issue reading input files with INCLUDE statements when the Preserve "INCLUDE Statements" option is enabled in the Nastran Options section on the Interfaces tab in the Preferences dialog box.

Interfaces - Simcenter Nastran (formally NX Nastran)

- Corrected an issue which caused incorrect parameter values to be exported for the "INERTIA" parameter associated with the "Enable Inertia in Dynamics" check-box in the "Solution and Convergence Options" dialog on the NLCNTL entry for SOL401 (PR# 9875547).
- Corrected an issue which prevented FEMAP from translating the CREEP entry for SOL402 (PR# 9930653)

Interfaces - MSC Nastran

- Corrected an issue which prevented contact lists from being associated with their respective Connection Regions if the list being read from a BSURF entry shared an ID with another Connection Region being reading from a BCBODY or BCBODY1 entry.

Interfaces - Autodesk Nastran

- Corrected an issue reading Autodesk Nastran input files that cause rigid elements to be written as CTE supporting BAR. PARAM,RIGIDELEM2ELAS,ON and PARAM,RIGIDELEMTYPE,BAR.
- Corrected an issue in Autodesk Nastran Bulk Data Option in the Analysis Manager for Rigid Element Method.

Interfaces - ANSYS

- Corrected an issue which occurred when importing ANSYS results file including beam elements which could cause FEMAP to exit unexpectedly.
- Corrected an issue which occurred when importing ANSYS compressed results file which could cause FEMAP to exit unexpectedly.
- Corrected an issue which occurred when running Simcenter Nastran after importing ANSYS input including /TITLE command.
- Corrected an issue with a default value which caused FREQE in MODOPT to be set to 1000 which could cause the modal analysis to find any normal modes.
- Corrected an issue with writing of analysis settings when changing analysis type from “Analysis Set” dialog but not activating the corresponding analysis settings dialog box.
- Corrected an issue when selecting values in drop-down in ANSYS Load Step dialog and ANSYS Output Requests dialog. This issue was introduced in FEMAP 2020.2.
- Corrected an issue which caused FEMAP to exit unexpectedly when reading the SFEDELE command from an input file.
- Corrected an issue when writing shear areas in beam properties to ANSYS input file. The error may cause small differences in beam deflection results.

Interfaces - LS-DYNA

- Corrected an issue where the 2WAY (1=On) field for “54..LS-DYNA Enhanced Composite Damage” in Other Types of materials would not write the correct value to the LS-Dyna input file.

Properties

- Corrected an issue for Nastran Beam Section type Bar when DIM2 is less than DIM1 and PBEAML/PBARL evaluation method used in Femap. If they are written as PBEAML/PBARL, there is not a problem but if the values evaluated by Femap are used in the solver, then it was incorrect.

Loads and Boundary Conditions

- Corrected an issue that could occur if you scaled surfaces and then did nothing else to cause them to be modified prior to applying loads or constraints to the surface. In some of those cases, no loads or constraints would be drawn on the surface until it was modified.
- Corrected an issue where time dependent function IDs not drawn in the graphics for Nodal and Elemental Temperature loads.

Meshing

- Corrected an issue in meshing that could occur if you had a surface that had one or more edges that were imprinted on the surface and they joined a suppressed outer edge.
- Corrected an issue where FEMAP could potentially become unresponsive if you attempted to create a tetrahedral mesh using a growth ratio of 0.8 (PR# 9800998).

Mesh Associativity

- Corrected an issue where midside nodes of certain tetrahedral elements would sometimes not be properly associated with geometry if the corner nodes of an element edge were associated to geometry that was part of a suppressed area (PR# 9910853).

Output and Post-Processing

- Corrected an issue where Femap exits unexpectedly when using Print to PDF when the “Level Mode” for “Contour/Criteria Levels” in View Options is set to “Visible Min/Max”. This issue could also happen in any File, Print; File, Picture, Save; or File, Picture, Copy command if the “Visible Min/Max” option is used with non-screen resolution images (PR# 9906028)
- Corrected an issue which could occur intermittently where post titles have an incorrect displacement line when switching between animation and dynamic rotation.
- Corrected an issue where freebody data was being loaded into memory even when freebody display is disabled and/or no output exists in the model. This created performance bottlenecks during regeneration of views.
- Corrected an issue that could potentially cause corruption to the freebody entity lists that could cause freebodies to display incorrect values when the model is initially loaded (PR# 9699032)
- Corrected an issue that caused Femap to display incorrect results for complex output when a transformation was required (PR# 9762084)

API

- Corrected an issue in the API Curve object ClosestPointToCurve method that could occur if you used it after calling the Get method, but before calling other methods on the same object without calling Get again.
- Corrected an issue with feOutput object where updating output vector components using the vcomponent variant may not properly update the individual component records.
- Corrected an issue where feLicenseGetInfo and feLicensePrintInfo would return incorrect license counts for certain features.

FEMAP v2020.2 MP 2 New Features and Corrections

Updates and Enhancements

Interfaces - Geometry

- Added support for NX (version 1926).

Corrections

User Interface - General

- Corrected issue introduced in 2020.2 where toggling the visibility check box to show or hide a connector set (i.e, a connector that references a number of individual connectors) caused unexpectedly poor performance.
- Corrected issue which arose when copying non-English characters to the clipboard using HTML formatting introduced in 2020.2, as characters from non-English languages (Japanese, Chinese, etc.) were not being sent to the clipboard correctly (PR# 9828549).
- Corrected issue that could prevent icons in dialog boxes from properly scaling to the resolution of the user interface.

User Interface - Dockable Panes

Meshing Toolbox - Mesh Sizing tool

- Corrected issue when using Mesh Sizing tool in the Meshing Toolbox to resize curves on multiple solids in a single command. This caused some surfaces to be remeshed multiple times, leading to poorer than expected performance.

Graphics

- Corrected issue where rotate about screen location did not work if the “Smooth Lines” option was enabled in the Graphics Options section on the Graphics tab of the Preferences dialog box (PR# 9834270).

Performance Graphics

- Corrected issue with the visibility dialog when toggling visibility of property, materials, etc. When using the All On or All Off buttons, the display was not updated until the Visibility dialog box was dismissed, then the Window, Regenerate command (Ctrl+G Hotkey) was used (PR# 9834574).
- Corrected issue which caused redundant points that no longer exist in model to be remain displayed after using the Geometry, Solid, Add command to add more than 2 solids together. Introduced in 2020.2 and points were no longer displayed after using the Window, Regenerate command.

Interfaces - FEMAP Neutral

- Corrected issue migrating models containing Parabolic Laminate Properties to a newer version of FEMAP from all versions of FEMAP from version 9.3 to version 2020.2 MP 1. The error would cause the referenced Layup to become corrupted and could also modify the setting for the Offset Bottom Surface option on the Laminate Property itself.

Interfaces - Nastran

- Corrected issue which caused poorer than expected performance when associating Matrix Input list entries to their Matrix Inputs when importing an analysis model with DMIG bulk data entries.

Interfaces - ANSYS

- Corrected issue which had could cause Femap to exit unexpectedly when reading an ANSYS RST file if the RST file includes results for Beam elements.

Interfaces - LS-Dyna

- Corrected issue when exporting Material Type “54..LS-DYNA Enhanced Composite Damage” which caused the “2WAY (1=On)” parameter to always written with a value of “0” unless “Fail. Citeria (54,55) CRIT” was set to a value of “54”.

Interfaces - Geometry

- Corrected bug for command line option -NX to read NX part files. It conflicts with -NX for Femap with NX Nastran mode so changed it to -UG

Loads and Boundary Conditions

- Corrected issue where Follower Forces and Follower Moments applied to geometry were not scaled correctly when Vector Length was set to “1..Scale by Magnitude” for the Load Vectors option in View Options.

Listing

- Corrected Issue which caused FEMAP to exit unexpectedly when using List, Output, Results to Excel command and the instance of Excel automatically launched by FEMAP was closed before the transfer of data was completed.

Output and Post-Processing

- Corrected issue which caused creation of a new Discrete Value Set using Range Type set to “3..Range by largest gaps” to perform very poorly when attempting to create ranges when Method was set to “From Data” and any of the entity types found on the “Entity ID” branch in the Contour Data From section was selected in the Contour Model Data dialog box.
- Corrected issue where the View, Advanced Post, Beam Cross Section command would not properly display Section Stress results on elements which reference a Bar or Beam properties which have Shape set to Circular Bar or Circular Tube.

API

- Corrected issue with the Facets method on the Surface (feSurface) object which caused it to not return the proper number of xyz coordinates to the xyzFacet output parameter.

- Corrected issue with the IsIdenticalSet method on the Set (feSet) object which caused it to report two Sets as identical, even if they are not. This would only happen if the lowest ID in each Set was different.
- Corrected issue with vcomponent property on the Output (feOutput) object (Deprecated in 2020.1) where updating output vector components using the vcomponent variant may not properly update the individual component records.
- Corrected issue where feLicenseGetInfo and feLicensePrintInfo would return incorrect license counts for certain features.

FEMAP v2020.2 MP 1 New Features and Corrections

Updates and Enhancements

Interfaces - Geometry

- Added ability to import SolidWorks geometry files using the "-SW" argument when launching FEMAP from a command line prompt.

Loads and Boundary Conditions

- Significantly improved the performance of expanding geometric loads on large models, especially surface-based loads on models meshed with solid elements. Types of Surface-based loads include: Force, Follower Force, Force Per Area, Moment, Follower Moment, Moment Per Area, Bearing Force, Torque, Heat Flux, Heat Flux Per Area, Temperature, and Heat Generation. This improvement not only improves the Model, Load, Expand command, it also improves other commands that need to expand loads to perform their tasks, including commands like Tools, Check, Sum Forces; Model, Analysis, then pressing Preview button; and other commands that write analysis input files.

Output and Post-Processing

- Significantly improved the performance of converting between nodal and elemental results on large models. This improves the performance of contour and criteria plots where the requested type of plot differs from the type of output being displayed. It also improves the performance of the Model, Output, Process command when using the Convert operation.
- Improved performance of View, Advanced Post, Stress Linearization command by using a different algorithm to determine which elements are along the Stress Classification Line (SCL).

Tools

- Significantly improved performance of merging points when using the Tools, Check, Coincident Points command. One model that contained 200,000+ points and 320,000+ curves that formerly took over 16 minutes to merge now takes ~4 seconds. Also grayed the Delete Midside Node option when merging anything but Nodes.

API

New and Updated API Methods

- Added GetCoordArray and GetAllArray to the Point Object.

The following functions have been added or updated:

- feFileReadSolidWorks

Corrections

User Interface - General

- Corrected issue which caused the titles of dockable panes to be incorrect after switching languages using the Language preference in the Global Options section on the User Interface tab of the Preferences dialog box.
- Corrected issue with the status/progress bar that occurred when reading large text files, especially large Nastran files. The issue did not cause any errors reading the file, it simply did not show the progress correctly.
- Corrected issue that occurred when using the Face Selection dialog box to select element faces, such as the Mesh, Edge/Skin Elements, Planar Elements on Faces command to create new elements and Group, Operations, Automatic Add was also enabled. Previously Element IDs of non-existing elements were added to the selected group. (PR# 9784147)

User Interface - Dockable Panes

Meshing Toolbox - Geometry Editing tool

- Corrected issue which could cause solid washer(s) and/or solid pad(s) to not completely extend to the end of the hole in the solid geometry where the washer or pad was being applied. This often occurred when one side of the hole passed through a blend or fillet (PR# 9772138 and 9769750)

Graphics

- Corrected issue in 2020.2 where some options in the View Options dialog box were not applied correctly after clicking Apply or OK. Workaround for OK issue was to use Window, Regenerate command (Ctrl-G hotkey) after exiting View Options dialog.
- Corrected issue in 2020.2.0 where Hide Selected and Show Selected command on the context-sensitive menu for layers in Model Tree did not draw correctly. Often points that should not be drawn were drawn. Work around is to use Window, Regenerate command (Ctrl-G hotkey).
- Corrected issue where selecting a view from a library in the View, Visibility dialog does not do anything until exiting the dialog box and using the Window, Regenerate command (Ctrl-G hotkey)
- Corrected issue with incorrect stresses on back face of plate elements when animating nodal contour results on plates with thickness displayed.
- Corrected issues with highlighting and marking of nodes when using the View, Advanced Post, Stress Linearization command.

Performance Graphics

- Corrected issue where changing the selected output vector did not clear the contours correctly. This issue randomly occurred and using the Window, Regenerate command (Ctrl-G hotkey) was a workaround.
- Corrected Issue where beams with cross-sections drawn are drawn deformed while dynamically rotating during an animation.
- Corrected issue in 2020.2 where the Preview button in the Digit Options and Digit Locale dialog boxes did not work properly.

- Corrected issue where the minimum value of the contour legend was not evaluated correctly when Level Mode is set to 7..Visible Min/Max for the Contour/Criteria Levels option in View Options and the user is displaying a contour arrow plot of nodal or elemental results (PR# 9822843).

Interfaces - FEMAP Neutral

- Corrected issue where Femap 2020.1 models that passed through the external neutral write translator could lose information stored "Saved Sets" and other internal data structures (For more information, see Software Field Bulletin PL8509636).

Interfaces - Autodesk Nastran

- Corrected issue when importing input files that causes rigid elements to be written as CTE supporting BAR elements (i.e., was causing PARAM,RIGIDELEM2ELAS,ON and PARAM,RIGIDELEMTYPE,BAR to be written to the input file). Also re-enabled Rigid Element Method option in the Translator Option section of the NASTRAN Bulk Data Options dialog box in the Analysis Manager.

Interfaces - ANSYS

- Corrected issue writing FEMAP general beam section to ANSYS input file. Previously these were being written with the wrong values for centroid and offset(s).
- Corrected issue importing ANSYS input files created by FEMAP back into FEMAP which significantly reduced import time when the FEMAP model includes a large number of groups.

Interfaces - ABAQUS

- Corrected issue which occurred when displaying or listing computed vectors (von Mises Stress, Mean Stress...) using attached ODB file containing results for Axisymmetric elements (PR# 9805726)
- Corrected issue which could cause FEMAP to exit unexpectedly when attempting to attach to ODB files containing output step or frame titles longer than 79 characters.

Loads and Boundary Conditions

- Corrected issue where the One Equation Per DOF option in the Create Constraint Equation dialog box could become randomly enabled.
- Corrected issue that occurred when creating variable geometric loads using an equation to define the load variation and the equation contained any lower case alphabetic characters. This would cause an error message to be displayed which said the equation was not valid. If upper case characters were used, it worked correctly. Now either are acceptable.

Meshing

- Corrected issue that occurred when using the Mesh, Editing, Element Refine command to split triangular elements. Previously, individual element corner thicknesses could be applied to the split elements even though the elements originally had constant thickness specified by property (PR# 9787696).
- Corrected a number of issues that occurred when using the Surface Mesh Only option in the Meshing Approach section of the Automesh Solids dialog box used by the Mesh, Geometry, Solids command. Non-existent Properties were assigned to the Plot-Only Plate elements which did not matter for most cases, but did cause some issues in the Meshing toolbox (PR# 9766691).

Output and Post-Processing

- Corrected issue when using View, Advanced Post, Stress Linearization command to perform Stress Linearization of Axisymmetric Elements with results recovered from ABAQUS. Results were not in the same Vector IDs as “standard” NASTRAN. The command now attempts to locate the “standard” output vector IDs, and if the proper results are not located, it attempts to locate the results using the “legacy” output vector IDs associated with ABAQUS Axisymmetric results.

Model Update

- Corrected issue that occurred when using Modify, Align, Coordinate System or Modify, Rotate, Coordinate System when the selected coordinate system(s) were referenced by Nodes where Mass Elements were defined. Previously, an error was issued saying that the Mass Property was not selected and the alignment/rotation failed.

FEMAP v2020.2 New Features and Corrections

Updates and Enhancements

Views

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

Connection Properties, Regions, and Connectors

- Added Traction Removal Convergence to Multi-Step Structural (401) tab, which is accessed by clicking the More Options button.
- Updated *Initial Penetration option on Multi-Step Structural (401) tab by adding “1..Ignore Gaps” option.
- Added Include Shell Thickness Offset option to Multi-Step Kinematic (402) tab.

Geometry

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

User Interface - General

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

User Interface - Toolbars and Icons

View - Simple and View Toolbars

- Added Curve Connectivity to the View Style icon menu

User Interface - Dockable Panes

Meshing Toolbox - Feature Removal tool

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

Meshing Toolbox - Geometry Editing tool

- Added Geometry Type option when Operation is set to Extend. When set to Curve on Surface, offers all functionality which existed in previous versions of the tool When set to Surface/Replace Face, mimics the Geometry, Solid, Extend/Replace Face command.

Charting

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

Data Surface Editor

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

Function/Table Editor

- Added Send to Excel command to the Copy Text to Clipboard icon menu, which opens Microsoft Excel, then sends all text, including header information (ID, Title, Type, Column Titles, and Row IDs) directly to Excel.
- Improved the Function/Table editor. If only one Function or Table exists in the model, the Reload commands no longer ask you to select the item to reload, it simply reloads the only one available.
- Updated copying from the Function/Table editor and other Table controls. The copied text format no longer includes numbered row or column headers (RTF still includes them). In addition, blank rows at the end of the table are no longer copied for either format.

Mesh Point Editor

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

Connection Editor

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

Entity Info

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

Data Table

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

API Programming

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

Interfaces - FEMAP Neutral

- Updated Neutral Read and Write for v2020.2 changes

Interfaces - Nastran

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

Interfaces - Simcenter Nastran (formally NX Nastran)

SOL 401 and 402 Only

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

SOL 401 Only

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

SOL 402 Only

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

Interfaces - ANSYS

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

Interfaces - ABAQUS

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

Interfaces - LS-DYNA

- Added Skip CSys, Skip Nodes, Skip Materials, Skip Properties, and Skip Elements to Model Control section of LS-Dyna Model Options dialog box.When enabled, the entity type selected to "skip" will not be written to the LS-Dyna input file. For example, if Skip Nodes and Skip Properties are enabled, then no nodes or properties will be exported.
- Added Skip Groups to Model Control section of LS-Dyna Model Options dialog box. When enabled, groups will not be written as *SET_ entries to the LS-Dyna input file, but entities contained in those groups are still exported unless one of the other Skip options is enabled.
- Added Skip Functions to Model Control section of LS-Dyna Model Options dialog box.When enabled, functions (*DEFINE_CURVE entries) are not written to the LS-Dyna input file. In addition, any loads or material properties which were functionally-dependent in the model no longer reference functions
- Added Skip Misc to Model Control section of LS-Dyna Model Options dialog box.When enabled, comments starting with "$", *KEYWORD, *TITLE with corresponding "title", *CONTROL TERMINATION with corresponding value, *DATABASE_BINARY_D3PLOT with corresponding value, *END, and other entries are not written to the LS-Dyna input file.
- Added LS-Dyna Analysis Monitor to monitor solver progress, review solver files, and optionally import results.

Interfaces - Geometry

- Added support for Parasolid 32.1, NX 2020 (version 1899), SolidWorks 2020, CATIA V5 R8 - V5-6 R2019 SP4, and JT 10.6.
- Added ability to import JT geometry files using the "-JT" argument when launching FEMAP from a command line prompt.
- Added ability to import NX geometry files using the "-NX" argument when launching FEMAP from a command line prompt.

Element - Nastran General Matrix - New for 2020.2!

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

Properties

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

Layups

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

Loads and Boundary Conditions

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

Meshing

- Added Set Default Size button to Mesh, Mesh Control, Default Size command. When clicked, the value for Element Size is calculated based on all geometry currently in the model and the Set Element Size on Next Use option is automatically disabled.
- Updated Duplicate Materials option in Property/Material Options section to be Duplicate Materials/Layups for commands on the Mesh, Copy... menu. For laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. When Geometry option in Other Entities to Include section is enabled, all duplicated laminate and solid laminate properties which have been assigned as a meshing attribute will reference a copy of the referenced layup and copies of all materials used by a referenced layup.
- Updated Duplicate Materials option in Property/Material Options section to be Duplicate Materials/Layups for commands on the Mesh, Rotate... menu. For laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. When Geometry option in Other Entities to Include section is enabled, all duplicated laminate and solid laminate properties which have been assigned as a meshing attribute will reference a copy of the referenced layup and copies of all materials used by a referenced layup.
- Updated Duplicate Materials option in Property/Material Options section to be Duplicate Materials/Layups for commands on the Mesh, Reflect... menu. For laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. When Geometry option in Other Entities to Include section is enabled, all duplicated laminate and solid laminate properties which have been assigned as a meshing attribute will reference a copy of the referenced layup and copies of all materials used by a referenced layup.

- Updated various commands which split elements on the Mesh, Editing... menu to more fully and consistently handle the other entities associated and/or related to the elements being edited
- Improved the Mesh, Geometry, HexMesh Solids command to provide a warning if you attempt to remesh solids that are already meshed. The warning gives the option to delete the existing mesh, skip the meshed solids, or create a duplicate mesh. This workflow now matches the one for tet meshing.
- Improved performance and accuracy when using length-based mesh sizing on the Parasolid curves (i.e., curves on sheet solids, solids, and general bodies). In a large model with many nonlinear curves, this resulted in a 2X increase in performance and more consistent constant length spacing throughout the model. Also, improved accuracy when attempting to size curves by Element Size instead of Number of Elements on curves which were seemingly the same length. Due to numerical accuracy, even a very small difference in the length of two curves could result in the curves having a different number of elements assigned. This improvement should improve sizing on curves, surfaces, and solids, whether they are sized by menu commands or Meshing Toolbox.
- Improved several commands to prevent creation of CBUSH elements that reference coincident nodes when a third node or vector is used for orientation.

Functions

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

Listing

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

Aeroelasticity

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

Output and Post-Processing

- Added List, Output, Results to Excel command, which is similar to List, Output, Results to Data Table, but quickly sends chosen output vector values for specific nodes or elements (from any number of selected output sets) directly to Microsoft Excel using a specified “Report Style”.
- Added View, Advanced Post, Stress Linearization command, which allows you to perform Stress Linearization. The FEMAP Stress Linearization Tool is based on the ASME Boiler and Pressure Vessel Code, 2007 SECTION VIII, DIVISION 2, ANNEX 5.1 LINEARIZATION OF STRESS RESULTS FOR STRESS CLASSIFICATION.

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

Tools

- Updated the Tools, Parameters command to allow specification of the Nodal Output Coordinate System (Output CSys drop-down). While this could be done using the Model, Node command and pressing Parameters button or in other meshing commands by pressing Node Parameters icon button, this is more convenient.
- Updated Tools, Mass Properties, Mesh Properties command to be able to automatically send the mass properties listing to the clipboard using the Send to Clipboard option. In addition, Check Mass Properties dialog box has been reorganized into separate Reporting Options and Mesh Options sections. Finally, improved performance to reduce the amount of time it takes to calculate mass properties for 1.95 million tetrahedral elements from ~38 seconds to ~3 seconds
- Updated the Tools, Mass Properties, Solid Properties command to be able to calculate and output the volume, surface area, center of gravity, and moment of inertia for the selected sheet solid(s), solid(s), and/or general bodies, either as a single assembly or for each individual body. Additional options exist to create a representative mass element and potentially attach it to existing nodes.

Model Merge

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

Preferences

Graphics

- Added Ctrl-G Group Evaluate to Graphics Options section. When this option is enabled, Group, Operations, Evaluate is performed on all groups currently visible in the active view when Window, Regenerate is used. If View, All Views is enabled, then all groups currently visible in all views of the active model will be evaluated. If Show Full Model is specified for Groups in the View, Visibility dialog box, the Model Info tree, or the Grp: menu in Status Bar, no groups are evaluated.

- Added Disable Abort to Graphics Options section. Certain operations can cause the graphics window to be completely redrawn without explicitly using Window, Redraw. In some cases, the user can abort (interrupt) the redraw by clicking in the screen or selecting another command. For this to happen, FEMAP checks for user input during redraw, which can cause the redraw to take additional time to finish compared to not doing any checking. When this option is enabled, no checking is done, thus aborting a redraw is not possible and potentially improving performance.
- Added Center to Include In Dynamic Rotation section. When enabled, the Center option displays a symbol in the graphics window at the "center of rotation" currently being used for dynamic rotation. If the "center of rotation" has been set by View, Rotate, Rotate About Rotation Center, then the "center of rotation" is obvious. Alternatively, if using the "view center" and the Dynamic Rotate Around Cursor Location preference found on the User Interface tab is enabled, then it may be helpful to have a visual representation at the "center of rotation".
- Added OpenMP and Options... button to Advanced/Debug Options section. Enable this option to use OpenMP functionality (i.e. multi-threading) which has been implemented for a limited number of commands and operations. Once enabled, click the Options... button to set Number of Threads, which should not exceed the number of cores available on the machine running FEMAP. **Note: This is initial implementation, thus considered Beta Functionality which is not officially supported. If any issues are encountered, please disable this option, then feel free to contact GTAC to describe the issue.**

User Interface

- Added Language option to new Global Options section. Language displayed by user interface. When set to "0..Default", uses the language which was selected during the initial installation of that version of FEMAP. When set to any other option, that language is used by FEMAP. Available option are "1.English", "2..German", "3..Japanese", "4..Chinese, Traditional", or "5..Chinese, Simplified".
- Added UI Scaling option to new Global Options section. When set to "0..Enabled", which is the default, certain Windows display settings are used to determine how the user interface should be scaled to improve user experience. Scaling the user interface can be especially helpful when using a high resolution display. If scaling is not required for any reason, set this option to "1..Disabled".

Interfaces

- Added Preserve Load/BC SET(s) option to ABAQUS Options section. Sometimes in an ABAQUS input file, *SET entries with unique names are used to specify "groups" of loads or boundary conditions. When this option is enabled, an attempt is made during import of the ABAQUS input file to collect loads or boundary conditions defined by a *SET entry into an appropriate Load Definition or Constraint Definition.

Results

- Added Compute Average Mid Stress/Strain option to General Solver Options section. When enabled, stresses and strains at middle locations for plate elements are calculated as a simple average of the top and bottom values in FEMAP

Color

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

API

General

- Added additional checking to the Read object to make sure the file is open before other methods will work. Also added checking to various parse and string functions to prevent FEMAP from unexpectedly exiting when a NULL string pointer is passed in by mistake.

New and updated API Objects and Attributes

- Added MergeTool (feMergeTool) object to the API. Also, added UsePattern, CreateGroupForMerged, AlwaysCreateParentCSys, CondenseMergedGroups, LimitToMergedEntities, KeepInOriginalSets, and DuplicateFirstInPatternto to the feMergeTool object.
- Added ndID1, ndID2, ouSetID, nElemType, nStressmode, bNonlinearMode, bFullStressTensor, dMembraneStress, dBendingStress, dMaxMemBend, and dMaxStress to the Stress Linearization (StressLinear) Object.
- Added NasMsnlCntRFVAR attribute for Simcenter Nastran SOL 401 to the Analysis Case Object.
- Added NasMsnlCntRFVAR attribute for Simcenter Nastran SOL 401 to the Analysis Manager Object.
- Added AnsModelGroupID, AnsModelSkipCsys, AnsModelSkipNode, AnsModelSkipMatl, AnsModelSkipProp, AnsModelSkipElem, AnsModelSkipGroup, and AnsModelSkipMisc attributes for ANSYS to the Analysis Manager Object.
- Added DynModelGroupID, DynModelSkipCsys, DynModelSkipNode, DynModelSkipMatl, DynModelSkipProp, DynModelSkipElem, DynModelSkipGroup, and DynModelSkipMisc attributes for LS-Dyna to the Analysis Manager Object.
- Added addl_ptID and vaddl_ptID attributes to the LoadGeom Object.
- Added RenderLogDepthOffset attribute to the View Object.
- Added DO_BeamDiagram_Option, DO_BeamDiagram_RoundToZero, DO_BeamDiagram_RoundToZeroValue, DO_BeamDiagram_BeforeDecimalSeparator, DO_BeamDiagram_MinimumExponentSize, DO_BeamDiagram_ExponentDigits, DO_BeamDiagram_SigFigs, DO_BeamDiagram_LeadingZeros, DO_BeamDiagram_TrailingZeros, DO_BeamDiagram_DecimalPlaces, DO_BeamDiagram_MaxDigits, DO_BeamDiagram_BaseExponent, DO_BeamDiagram_OriginalDigits, and DO_BeamDiagram_Exponent attributes to the View Object. These attributes are used by the “Beam Diagram” option in View Options to control the display of digits in the graphics window for Beam Diagrams.
- Added CurveConnectWirebody, CurveConnectFree, CurveConnectManifold, CurveConnectNonManifold, CurveConnectWirebodyColor, CurveConnectFreeColor, CurveConnectManifoldColor, and CurveConnectNonManifoldColor attributes to the View Object. These attributes are used by the “Curve Connectivity” option in View Options to control the visibility and color for the display of curve connectivity.
- Updated RenderPushForward, RenderPushUnit, RenderPushUndeformed, and RenderPushLabel on the View Object. They now have specific values ranges assigned.

New and Updated API Methods

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

New and Updated Global Variables

- Added Pref_RenderCtrlGGroupEvaluate, Pref_RenderDisableAbort, Pref_OpenMP, and Pref_OpenMPThreads to set preferences on the Graphics tab of File, Preferences. Also, updated Pref_RenderRotate to be able to control the Center option in the Include In Dynamic Rotation section.
- Added Pref_GlobalLanguage to set Language option and Pref_GlobalUIScaling to set UI Scaling option in the Global Options section of User Interface tab of File, Preferences
- Added Pref_Abaqus_PreserveCompressedLoads to set Preserve Load/BC SET(s) on Interfaces tab of File, Preferences
- Added Pref_ComputeAverageMidResults to set Compute Averaged Mid Stress/Strain preference on Results tab of File, Preferences
- Added ModelFromPreviousVersion, ModelMigratedFromVersion, and ModelMigratedFromDB as new Global Parameters which can determine from which previous version the model was migrated from and other information.
- Updated Pref_RenderRotate and vPref_RenderRotate to be able to control Center option in Included In Dynamic Rotation section of Graphics tab.
- Updated Pref_EntityColor and vPref_EntityColor to be able to control Mesh Point and Monitor Point color in Entity Colors section of Color tab.

The following functions have been added or updated:

- feAppGetModelByName
- feAppRunningApplicationInfo
- feAppGetRunningApplication
- feEdgesOfFreeFaces

Corrections

Analysis Manager

- Corrected issue that prevented Analysis Sets from loading from the Library in V2020.1 (PR#9718319).

Connection Properties, Regions, and Connectors

- Corrected issue where value for Shell Z-Offset on the Linear tab of Define Connection Property would not be written as an override when exporting to Simcenter Nastran.

Geometry

- Corrected issue with tolerance values which could cause certain surfaces to not be properly offset or cause the offset operation to fail.
- Corrected issue with tolerance values which could cause certain curves to not be properly offset or cause the offset operation to fail.

Graphics

- Corrected various issues with Elements with no results. Options are to draw elements with no contour/criteria, hide the elements, or show the elements with a contour/criteria of value 0.0. Addressed issues include instances where RBE2 and RBE3 not contoured with zero correctly if no results and Line elements not contoured when having a value of zero.
- Corrected issue where Model Clipping Plane is not included in the Reset All Visibility Option button in the Visibility dialog (View, Visibility command) or the Model Info tree.
- Corrected performance issue when displaying curves so calculation of curve mesh graphics only happens for curves that need the curve mesh displayed. The default only processes curves that have an element density set and does not display curve mesh for curves with default mesh size (PR#9666376)
- Corrected issue where element coordinate system axis labels are drawn at the element centroid and not at the end of the coordinate system axes. This did not occur in Wireframe but did occur when Fill was on.

Performance Graphics

- Corrected various issues with Elements with no results. Options are to draw elements with no contour/criteria, hide the elements, or show the elements with a contour/criteria of value 0.0. Addressed issues include instances where no plate or solid elements criteria with zero if no results; RBE2, RBE3, and Mass elements not hiding with contour or criteria; RBE2, RBE3, and Mass elements not showing criteria of zero when no results; RBE2 and RBE3 elements not being contoured with zero when no results; and Line elements not showing criteria when having zero value.

User Interface - General

- Corrected issue where Auto Max Min button for the Contour/Criteria Levels option in View Options dialog box was replaced visually by a “color block” palette button. The button still functioned correctly but was not labeled correctly.
- Corrected issue where Filled Edge Offset factor value for the Performance Graphic option in View Options dialog box, which has not been used since Femap 12.0, was never removed from the user interface. The underlying functionality has been replaced with Filled Edge Depth Offset Factor option in the Advanced Depth Control dialog box accessed via Graphics Options in the View Options dialog box.
- Corrected issue where limiting the element selection set with element face or element edge selection could cause the By Pick method in the entity selection dialog to become disabled until Femap is restarted.

User Interface - Dockable Panes

Meshing Toolbox - General

- Corrected issue that prevented loops from being removed from solids made from multiple disconnected pieces… i.e. disjoint solids. This type of solid is unusual, however it can be created using Geometry, Solid, Add or by importing geometry.

Meshing Toolbox - Geometry Editing

- Corrected issue which could cause a Solid Washer to not be created as expected (PR#9769750)

PostProcessing Toolbox

- Corrected issue that occurred in v2020.1MP2 that prevented the PostProcessing Toolbox from being able to change the Contour Legend Label Color mode (PR#9728308).

Charting

- Corrected issue which caused the Chart dialog box to take an excessive time to open and be available for use.
- Corrected issue where automatic titling for Vector Combination vs Set Chart Data Series would swap the entity IDs for Vector 1 and Vector 2.
- Corrected issue where scaling factors were being applied to phase values on Charts which showed both magnitude and phase components.

- Corrected issue in Chart Data Series dialog box where output sets would not be saved correctly if the data series was loaded with an Output Study selected.

Data Table

- Corrected issue with the List, Output, Results to Data Table command that caused the columns in the Data Table to be shown in the wrong order. All of the data in the table was correct, the columns were simply not shown in the expected order.

Program File

- Corrected issue when playing program files that are recorded in localized languages (PR#9670936)

Interfaces - FEMAP Neutral

- Corrected issue that prevented FEMAP from warning if you tried to read a Neutral File from a version that was newer than the current version. This error only occurs in the 2020.1 version and its maintenance packs.
- Corrected issue when importing a neutral file which contains weld elements and the user selects to offset the IDs when importing. The entities referenced by many types of welds (nodes, elements and properties) were not renumbered within the weld element.
- Corrected issue which caused Ansys group ID for “Portion of the Model to Write” and Skip Standard option to become corrupt when transferring models through the neutral file. This correction will allow old neutral files to migrate forward into Femap version 2020.2 and above, but does not apply to Femap 2019.1 MPx and 2020.1 MPx, as the issue exists in the Neutral import code within those versions.

Interfaces - Nastran

- Corrected issue reading Nastran F06 files for buckling, frequency and flutter analyses that would not properly identify the solution type of the results if there was an addition in-line comment following a “$” on the executive control SOL command.
- Corrected issue which prevented DMIG bulk data entries from being written in the proper field size if the 'Machine Precision' option was used (PR#9709334).
- Corrected issue with Optimization Limits entities which reference output vector IDs greater than 10,000,000, which may cause them to not be properly translated to the appropriate DRESP1 card.

Interfaces - Simcenter Nastran (formally NX Nastran)

- Corrected issue reading thermal and elastic strains on PENTA solid elements for SOL 401. Previously the output vectors where the results were stored was incorrect. This only occurred for wedge elements, not Bricks, Pyramids or Tetras.
- Corrected issue that caused incorrect reading of stresses/strains for wedge elements in SOL 401/ SOL 402
- Corrected issue that prevented the appropriate ASSIGN statement from being written for Restart Analysis in solutions other than SOL 401/SOL 402 (PR#9703290).

Interfaces - MSC Nastran

- Corrected issue that prevented results from being read for pyramid (CPYRAM) elements. This occurred because of a discrepancy between the value associated with this element type for MSC Nastran's delivery database, as opposed to the one for Simcenter Nastran (PR#9716553).

Interfaces - ABAQUS

- Corrected issue when importing an ABAQUS file containing pressure loads on solid elements which could result in the load being applied to the wrong element face.
- Corrected issue which could cause a memory leak during import if the ABAQUS input file contained assemblies.
- Corrected issue where the titles material title contained a character unsupported by the ABAQUS solver.

Interfaces - ANSYS

- Corrected issue which allowed ill-defined Spring/Damper Elements which reference a Spring/ Damper Property with Type set to CBUSH to be created and later exported.
- Corrected issue which caused beam properties using either the NASTRAN Z or NASTRAN BOX1 shape to be exported incorrectly.
- Corrected issue when importing *DIM entries which could case names for row, column, and plate to be incorrect.

Interfaces - LS-DYNA

- Corrected issues which caused parabolic triangular and brick elements to not be exported correctly in some cases.
- Corrected issue which caused orthotropic materials to not be exported correctly.

Element Update

- Corrected an issue that caused the Modify, Update Elements, Solid Material CSys... command to generate incorrectly oriented Coordinate Systems if the 'Vector Direction' method was used for the alignment of the first axis (PR#9715208).

Loads and Boundary Conditions

- Corrected issue when editing a Load Definition that caused future edits to issue the warning that the Load Definition was non-uniform (different loads on different entities), even though it wasn't.
- Corrected issue that caused loads generated by a “Torque on Surface” load to be incorrect. In cases where the load was not applied normal to the surface and the surface was asymmetrical about the load application point, the expanded nodal forces could be too small and would not result in the expected total torque. (PR#9699116)

Meshing

- Corrected numerous issues with the Mesh Editing commands (Mesh, Editing, Interactive; Mesh, Editing, Split; Mesh, Editing, Element Refine; and Mesh, Editing, Edge Split). The issues were related to the commands incomplete and inconsistent handling of various issues rather than incorrect performance. All of these commands should now consistently handle the updating of Constraints (both Set Based and Permanent), Loads (On Element Faces or Edges, Varying Corner Pressures and Distributed Line Loads), Connection Regions (Nodal and Elemental), Groups, Elemental Data (Tapered Beam Shapes, Beam Offsets, Varying Plate Thickness and Material Directions) and Geometric Associativity (PR#9702230)
- Corrected issue when trying to extrude elements or project points or nodes onto a surface along a vector or normal direction. Previously if the vector/normal just missed the edge of the surface due to numerical tolerances the commands or API methods would fail or generate inconsistent results. Now assuming the vector passes within numerical tolerances of the edge the process should work consistently.
- Corrected issue that prevented suppressing loops on faces of NonManifold solids that did not bound a solid region.
- Corrected issue that caused Curves that already had constant length spacing to be resized in the Mesh, Mesh Control, Size On Solid command even if you turned off the "Replace Mesh Sizes on all Curves" option.
- Corrected a number of issues that could sometimes occur with the Mesh, Editing, Cohesive Meshing command if there were multiple elements along the edge being split or if parabolic wedge elements were being split. Also corrected a more general issue with splitting elements during cohesive meshing that was introduced in 2020.1 MP2 (PR#8443193)
- Corrected issue in mesh sizing along curves that could occur if you specified a number of elements rather than a size. Due to the double precision numerical accuracy of the curve length, depending on the number of elements specified, two seemingly identical length curves could previously end up with a slightly different number of elements (+/-1). Improvements have been made to minimize the chances of these differences occurring. (PR#9636055)

Mesh Associativity

- Corrected issue that occurred when reflecting both mesh and geometry if you specified a nonzero trap width. In this case, if any of the reflected nodes fell within the specified trap width, the reflected mesh was properly connected to them, however any elements using those nodes were not properly associated with the reflected geometry. (PR#9767532)

Listing

- Corrected a formatting issue that occurred when listing Weld Properties.

Output and Post-Processing

- Corrected issue that caused Femap to display incorrect results for complex output when a transformation was required (PR#9762084).

Groups and Layers

- Corrected an infinite loop that could occur in the Group, Operation, Generate With Output command dialog if you typed an invalid number into the “Output Vectors”, “From” field. This did not occur if you simply selected from the drop-down or if you typed a valid ID (PR#9703284).

Tools

- Corrected an issue with the Tools, Mass Properties, Mesh Properties command. Previously, although no error message was issued, Solid Laminate Elements were not supported by this command. Mass Properties of Solid Laminate Elements is now supported.

API

- Corrected issues in the Clear and EraseSet methods of the DrawErase object. Previously they did not work as documented or as you would expect. Documentation has also been updated.
- Corrected several issues with the calls used for the Frequency object with the Analysis Set Manager Object. Previously, TurnOnFreq and TurnOffFreq could not process all frequency records, could not give a set of frequency records, and used a positive ID to access a single frequency record. Also, corrected issue in the corresponding API Put function that meant the frequency record usages were not saved unless a “put, get, put” sequence was done instead of just a put. All of these issues have been addressed.
- Corrected issue with AreDuplicate method on Material Object. Previously, it was returning TRUE when the data was different between two materials. Now it returns False.

FEMAP v2020.1 MP 2 New Features and Corrections

Updates and Enhancements

Interfaces - Simcenter Nastran (formally NX Nastran)

SOL 401 and 402 Only

- Added support for reading plastic strain results at grid points for SOL 401/SOL 402. This includes support for solid and shell elements.

SOL 402 Only

- Added support to write request to recover Elastic Strain output at grid points for SOL 402.

Output and Post-Processing

- Added support for post-processing plastic strain results at grid points for SOL 401/SOL 402. This includes support for solid and shell elements.

Corrections

Analysis Manager

- Corrected issue with the "Boundary Conditions" dialog box in the Master Case of Buckling Analysis for Nastran solvers which could cause FEMAP to exit unexpectedly. (PR# 9682285)

Geometry

- Corrected issue which caused some surfaces to not be offset correctly. In addition, some additional warning messages were implemented to potentially alert the user to review the offset surface(s) before meshing.

Graphics

- Corrected issue which caused a very large amount of memory to be used to display mesh size symbols on curves, which now allows a neutral file to properly imported. Please note, an issue still exists when the default point size is too small, causing a massive amounts of curve mesh evaluation and causing a degradation in performance.

Interfaces - Nastran

- Corrected issue introduced in FEMAP v2020.1 that caused the "BOLTFRC" entry to be written for versions of Nastran (i.e., Autodesk Nastran) where it is unsupported. This issue also prevented the correct entry, "BOLTFOR", from being written for those versions of Nastran.

Interfaces - Simcenter Nastran (formally NX Nastran)

SOL 401 and 402 Only

- Corrected issue that affected the writing of the “BCSET” case control command for SOL 401/SOL 402. Contact cannot be changed in a modal subcase, or set if that subcase is the first in the analysis OR not sequentially dependent, thus the “BCSET” case control command is prevented from being written in these special cases.
- Corrected issue which caused erroneous “TEMP(LOAD)” case control to be exported for SOL 401/ SOL 402 when the “From Load Set” option was selected in the Temperature drop-down, even if the load set selected did not contain any temperature loads.

SOL 402 Only

- Corrected issue which caused an invalid value to be written for the MATSYM parameter on the NLCNTL2 entry for SOL 402. The Simcenter Nastran Quick Reference Guide mentions using “YES” or “NO” for this parameter, but Nastran actually accepts only “1” or “0”.

Interfaces - ABAQUS

- Corrected issue when exporting directional pressure loads as DLOAD, type TRVEC on plate elements which would cause “TRVEC” to be written twice to the TRVEC data line (i.e., TRVECTRVEC) when analysis type is Frequency Response.
- Corrected issue when exporting directional pressure loads as DLOAD, type TRVEC on axisymmetric shells and plane strain elements where 2 TRVEC data lines would be printed for a single load for all analysis types other than Frequency Response.

Properties

- Corrected issue with Beam and Bar properties when Shape was set to NASTRAN T2 in the Cross Section Definition which could cause incorrect values to be calculated for Torsional Constant, Y Shear Area, and Z Shear Area.

Meshing

- Corrected issue which occurred when attempting to extrude elements or project nodes along a vector or using an element's normal direction. Previously, if the vector supplied by the user or the normal vector missed the edge of the surface due to numerical tolerances, the commands or API methods would fail or generate inconsistent results. Now, assuming the vector passes within numerical tolerances of the edge of the surface, the process should work consistently. In addition, corrected similar issues when attempting to project points.

API

- Corrected issue which caused deleted output vectors to be listed, one-at-a-time, to the specified List Destination(s) when using feDeleteOutputV2 or feDeleteOutput2V2.

FEMAP v2020.1 MP 1 New Features and Corrections

Updates and Enhancements

GUI - Dockable Panes

PostProcessing Toolbox - General

- Added buttons to the Digits fields in the Deform, Contour, and Freebody Tools, which control the display of digits in the graphics window via the Digit Options dialog box.

Interfaces - ABAQUS

- Added checking for unsupported *ASSEMBLY and *PART usage, warning the user to exit FEMAP and "flatten" the model in Abaqus CAE before importing.

Corrections

Graphics

- Corrected issue that caused labels for Nodes and Contour Arrow which lay on free edges to not be drawn properly.

Interfaces - Simcenter Nastran (formally NX Nastran)

- Corrected issue that prevented PLSTRN material nonlinearity from being specified when Analysis Type is set to "28..Multi-Step Nonlinear Kinematic" (SOL 402).
- Corrected issue that prevented the appropriate temperature load case control from being exported if the "0..From Load Set" option was selected for temperature loads when Analysis Type is set to "27..Multi-Step Structural" (SOL 401) or "28..Multi-Step Nonlinear Kinematic" (SOL 402). Also, grayed "Initial Temperature" combo box in subcases, as these should only be specified globally.
- Corrected issue reading and attaching to strain output on wedge elements creating by SOL 401.

Interfaces - ABAQUS

- Corrected issue that could cause FEMAP to exit unexpectedly when a reading an NSET entry which was used as the reference node on a COUPLING entry (instead of single node ID).
- Corrected issue which was introduced by checking for assembly conditions during import that caused an entire pass through the input file to be skipped, resulting in no properties being imported.

Loads and Boundary Conditions

- Corrected an issue when creating Pressure loads using the Vector option in the Direction portion of the Create Loads dialog box and a Data Surface (PR# 9512542)

Output and Post-Processing

- Corrected multiple issues when using the Model, Output, Transform command. One issue caused transformed corner output for tetrahedron elements to not be saved when a .op2 file was attached.

The other issue caused transformed corner output to not be saved if all corner vectors for solid elements were not selected for transformation (PR# 9645733).

API

- Corrected issue in the Results Browsing Object API interface that caused the NumberOfRows method to be incompatible with code written for previous versions of the Type Library. The correction makes NumberOfRows again compatible with previous versions, but VectorID is now incompatible with the initial release of v2020.1. The incompatibility only affects code that is trying to use a version of the Type Library that does not match the version of FEMAP being used.
- Corrected issue with “Saved Sets” in the API where renumbering or deleting entities could incorrectly cause certain IDs to be removed from saved sets.

FEMAP v2020.1 New Features and Corrections

Updates and Enhancements

Views

- Added the ability to view element coordinate systems for Spring/Damper elements which reference a property with Type set to CBUSH by using the View, Options command, setting Category to Labels, Entities and Color, choosing Element - Coordinate System from the Options list, then enabling Show Coord Sys.
- Implemented "Unified Label Architecture", which is used for all graphics, regardless if entities are being drawn with Performance Graphics, "Legacy OpenGL Graphics", or a mix of both. This implementation includes enhanced control of digits for real number values being displayed in the graphics window.

Analysis Manager

- Added Monitor Points and Direct Matrix Input items to the tree structure in appropriate locations for Simcenter Nastran and MSC Nastran.

Connection Properties, Regions, and Connectors

- Added Normal Vel Coeff. drop-down, Tangent Vel Coeff. drop-down, Normal Reg. Type drop-down and Value, Frict vs Time drop-down, and Frict vs Temp drop-down to the Multi-step Kinematic (402) tab.
- Added Target KEYOPTs section and all options in the section to the ANSYS tab. This section contains options which are only for target segments (TARGE169 and TARGA170). Of particular interest is Pilot Constraint (4), which can be used to specify constrained degrees of freedom for target segments which reference a particular connection property.
- Added Thermal section to MSC Nastran tab, which is accessed by clicking the Advanced Options... button. Options in the Thermal section are written to the "HHHB" line in the BCTABLE entry.
- Updated *Adaptively Modify Penalty Factor option on Multi step Structural (401) tab, which is accessed by clicking More Options... button. Previously, it was a check box, but now it is a drop-down with three options.
- Updated the title of the tab in the Define Connection Property dialog box used to specify connection property values for Autodesk Nastran (formally NEi Nastran).

Geometry

- Added Coordinate Systems option to Other Entities to Include section for commands on the Geometry, Copy... menu. In addition, created Parameters section, changed Copy to Active Layer to Assign to Active Layer and moved it into this section, along with the new Assign Active Color… and Assign to Active CSys options.
- Added Coordinate Systems option to Other Entities to Include section for commands on the Geometry, Rotate... menu. In addition, created Parameters section, changed Copy to Active Layer to Assign to Active Layer and moved it into this section, along with the new Assign Active Color… and Assign to Active CSys options.

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

GUI - Toolbars and Icons

Loads Toolbar

- Added Nodal on Face and Elemental on Face commands, which will launch the Model, Load, Nodal on Face or Model, Load, Elemental on Face commands, respectively.

Lines Toolbar

- Added Between Geometry command, which launches the Geometry, Line, Between Geometry command.

GUI - Dockable Panes

Model Info Tree

- Added support to show Loads, Constraints and Results in the Model Info tree when they reference Load Definitions, Constraint Definitions, or Analysis Studies that do not exist. In these cases they are shown just as if they did not reference a Definition/Study. Normally this situation can not exist when just using the User Interface, but it can happen with incorrectly written APIs. Showing these entities in the tree, allows them to be manipulated and placed into existing definitions. PR#9532383
- Added Simulation Entities branch and underlying branches for Monitor Points and Matrix Inputs, which allow for management, editing, listing, deleting, and renumbering of monitor points and matrix inputs, respectively. For monitor points, the color and layer may also be changed.
- Added Visibility check boxes (on/off) for Monitor Points. There is also a context-sensitive menu for the Visibility check boxes which offers Show Selected Only, Show Selected, Hide Selected, Select Show Only..., Select to Hide..., Show All, Hide All, and Show/Hide Reverse commands.
- Added Tables branch under the Model branch, which allows a table entity to be reloaded/edited via the Function/Table Editor or highlighted table(s) to be deleted or renumbered.
- Added Elemental on Face command to the context-sensitive menu for Load Definitions, which will launch the Model, Load, Elemental on Face command.

- Added Renumber command to the context-sensitive menu for Data Surfaces, which renumbers the highlighted data surface(s).
- Enhanced functionality of any command which creates a new Load Set or Constraint Set to automatically "expand" the newly created Set in the Model Info tree. This only occurs when the Model Tree is visible.
- Enhanced Copy to Set command on the context-sensitive menus for Load Definition to allow the highlighted Load Definition(s) to be copied into multiple Load Sets. In addition, improved the Copy to Set command on the context-sensitive menu for Constraint Definition in a similar manner.

Meshing Toolbox - Feature Editing tool

- Added Smart Select option, along with Offset, Identical, Concentric, Colinear, and Tangent Edges options, to control automatic addition of surfaces to a set of already selected surfaces for more robust feature editing. In addition, when Operation is set to Translate Surface(s), added the Translate Mode drop-down, which offers multiple workflow options
- Added ability to translate an internal loop (i.e., hole, slot, cutout) within a surface when Selection Method is set to Feature Edges and Operation is set to Translate Surface(s). In addition, added similar functionality to rotate an internal loop within a surface when Operation is set to Rotate Surface(s)

Meshing Toolbox - Geometry Editing tool

- Added Alignment drop-down when Operation is set to Washer, which provides four options, "Automatic", "Vector", "Tangent to Curve", and "Perpendicular to Curve", which can be used to "align" the split lines of a washer to be parallel to a specified vector, tangent to selected curve, or perpendicular to a selected curve

Function/Table Editor

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

PostProcessing Toolbox - General

- Improved the performance of opening the PostProcessing toolbox when the model contains a large number of Coordinate Systems. On a sample machine, opening the toolbox with around 100,000 Coordinate systems went from nearly two minutes to less than two seconds. This improvement will also help with smaller numbers of Coordinate Systems, along with models containing large numbers of other entity types that are shown in toolbox drop-down lists. (PR# 9576229)

Interfaces - FEMAP Neutral

- Added reading/writing of the Active Layer to the Neutral File
- Updated Neutral Read and Write for v2020.1 changes

Interfaces - Nastran

- Added support for "Large IDs" (i.e., IDs above 99,999,999). Any entity with a "Large ID" will automatically be written in large field format. It up to the user to confirm that any type of entity

which has been assigned a “Large ID” is allowed to have a “Large ID” for the specified solution sequence in the specified solver.

- Added Monitor Points item in Analysis Set Manager for a number of different analysis types, which provides access to the Select Monitor Point(s) dialog box.
- Added Direct Matrix Input item in Analysis Set Manager for a number of different analysis types, which provides access to the NASTRAN Matrix Input Selection dialog box.
- Updated writing the Global Cylindrical and Global Spherical coordinate systems in Nastran Large Field format. Previously, default continuation fields were written to the input file, which would create new coordinate systems if the file was later imported into FEMAP. Now the continuation entries are properly designated as FEMAP global coordinate systems, similar to how they were in Small Field format and will import properly as the Global Coordinate systems. (PR# 9301472)

Interfaces - Simcenter Nastran (formally NX Nastran)

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

SOL 401 and 402 Only

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

SOL 401 Only

- Added access to the Multi-Step Global Control Options dialog box. See SOL 401 and SOL 402 Only section.
- Added Disable Mechanical Load (LOADOFF), Enable Inertia in Dynamics (INERTIA) and Thermal Strain Loading (THRMST) to the Solution Parameters section of the Solution and Convergence Options dialog box.
- Added Quasi Newton-Raphson Iterations, Enable Stiffness Matrix Stabilization (MSTB), and Matrix Stabilization Factor (MSFAC) to the Stiffness Parameters section of the Solution and Convergence Options dialog box.
- Added Norm Criteria for Force (NORMP), Norm Criteria for Displacement (NORMU), Force Error Function Denominator (REFP), and Disp. Error Function Denominator (REFU) to the Convergence Parameters section of the Solution and Convergence Options dialog box.
- Added Modified Generalized Alpha Param (RHOINF) option to the Time Integration section of the Solution and Convergence Options dialog box.

SOL 402 Only

- Added Stress-Strain Measure for Output (STROUT) and Stress-Strain Conversion Method (STRCONV) to the newly created Global Parameters (SOL 402) section of the Multi-Step Global Control Options dialog box.
- Added Threshold for Zero Pivots (PRECPIVO), Modes to Save as Output (NKINE), and Version of Param. Defaults (MODEVERS) to the newly created Advanced Parameters (SOL 402) section of the Multi-Step Global Control Options dialog box.

- Added three options to Multi-Step Control Options dialog box, Activate Time Stepping Based on Disp/Temp Error (ERCD) and Allowable Change/Step (PRED) in Time Stepping and Save Data at Beginning of Computation (IAR0) in Analysis Control. Also, the Other Options section was divided into Plasticity and Creep Control and Internal Restart. In addition, the name of the Solution/Convergence button was changed to Iteration and Control.
- Added Rigid Body Motion Convergence (OTRE) and Threshold for Contact Force Variation (PRCF) to Solution and Convergence dialog box, one in Equilibrium and Convergence and one in Contact. Also, removed Time Step Criteria and Allowable Displacement (PRED) from the Analysis Options section. In addition, updated layout of the entire dialog box to make more efficient use of space.

Interfaces - ANSYS

- Added Portion of Model to Write option to Model Control section of ANSYS Command and Model Control dialog box.Allows you to select a previously defined group, then only exports the supported entities in that group to the ANSYS input file. In some cases, exporting elements without associated nodes or material/property entries may be desired, but this creates an input file which cannot be run by ANSYS unless additional entities are added to the file
- Updated dialog box used for specification of PSD information and other options for random response analysis.
- Improved performance in finding an unused entity ID, which reduced the time needed to read a particular large ANSYS model from 1 hour 30 minutes to 5 minutes.
- Improved support for shell elements with variable shell thicknesses defined via *DIM and *SET, SECFUN.
- Improved support for ESEL and NSEL entries by creating FEMAP Groups when these are encountered. Also, Groups in FEMAP can be used to write ESEL and NSEL entries.
- Improved support for loads and constraints using ANSYS element or node components

Interfaces - ABAQUS

- Added read/write support for DLOAD, TRVEC to create/export directional pressure loads.
- Added support to read *CONNECTOR SECTION types BUSHING and BEAM. For type = BUSHING, Spring/Damper elements which reference a corresponding property with Type set to CBUSH are created. For type = BEAM, Spring/Damper elements which reference a corresponding property with Type set to CBUSH, with very large stiffness values in all directions, are created.
- Added support for reading CONTACT PAIR, SURFACE INTERATION, and SURFACE NAMEs and retaining them in FEMAP.
- Added support for reading off-axis terms from ROTARY INERTIA entries, which are stored as Mass elements in FEMAP.
- Updated import to read a maximum of 9 digits for ID fields in the ABAQUS input file.
- Improved export of Contact Surfaces to only use integers for the label prefix instead of alphanumeric label prefix. Previously used 0..9, a..z, so 11th Contact Surface had prefix label of CSb, but user prefers CS11.
- Removed support for import of fixed field format files, as they are now incompatible with solver.

Interfaces - LS-DYNA

- Added Portion of Model to Write option to LS-DYNA Model Options dialog box. Allows you to select a previously defined group, then only exports the supported entities in that group to the LS-DYNA input file. In some cases, exporting elements without associated nodes or material/property entries may be desired, but this creates an input file which cannot be run by LS-DYNA unless additional entities are added to the file.
- Added support for Beam type “13..Timoshenko”. In FEMAP, this is set using an element formulation.
- Improved support for shell elements with variable shell thicknesses defined using the Variable fields on the *ELEMENT_SHELL_THICKNESS entry.

Interfaces - Geometry

- Added support for Parasolid 32.0, NX 2019 (version 1872.15), Solid Edge 2020, CATIA V5 R8 - V5-6 R2018 SP2, ACIS 2019 1.0.1, and JT 10.

Element - Formulations

- Added “13..Timoshenko” to the DYNA Options drop-down of the Element Formulation dialog box for Bar and Beam elements.

Properties

- Corrected issue with creating tapered beam properties. If user selected the tapered switch on the “Define Property - BEAM Element Type” dialog box before creating End A section properties, the “Compute Shear Center Offset” and “Compute Warping Constant” switches for End B on the Cross Section Definition dialog box are not defaulted to the appropriate setting.

Loads and Boundary Conditions

- Added Model, Load, Elemental on Face command, which the same as Model, Load, Elemental, except that instead of directly selecting the elements where the loads will be applied, here only the faces of elements are selected via the standard Face Selection dialog box. Once the faces are selected, the command continues, just like the normal Model, Load, Elemental command, but only Pressure, Heat Flux, Convection, and Radiation loads are available to be applied.
- Updated Model, Load, From Freebody command by make additional parameters available for automatic creation of interpolation elements when using the Multi-Model option. Also, some of the options from the Rigid Element Options section were moved to the new Node Pairing Options section, which contains four options.
- Updated Model, Load, From Output command to allow selection of All Load Sets or Selected Output Sets to automatically create new load sets using output vector(s) converted into the specified type of load. In addition, added some more robust functionality to specify which element faces to use for Elemental Face Loads, along with an option to automatically create matching constraints when creating Displacement Loads.

Simulation Entities - New for 2020.1!

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

Meshing

- Added Model Free Edges option to the standard "Edge Selection" dialog box which is now used in a number of meshing commands. When this option is selected, any free edges of elements being considered by the command will automatically be selected.
- Added Coordinate Systems option to Other Entities to Include section for commands on the Mesh, Copy, Node and Mesh, Copy, Element commands. In addition, created Parameters section, changed Copy to Active Layer to Assign to Active Layer and moved it into this section, along with the new Assign Active Color… and Assign to Active CSys options.
- Added Coordinate Systems option to Other Entities to Include section for commands on the Mesh, Rotate, Node and Mesh, Rotate, Element commands. In addition, created Parameters section, changed Copy to Active Layer to Assign to Active Layer and moved it into this section, along with the new Assign Active Color… and Assign to Active CSys options.
- Added Coordinate Systems option to Other Entities to Include section for commands on the Mesh, Reflect, Node and Mesh, Reflect, Element commands. In addition, created Parameters section, changed Copy to Active Layer to Assign to Active Layer and moved it into this section, along with the new Assign Active Color… and Assign to Active CSys options.
- Added Coordinate Systems option to Other Entities to Include section for Coordinate System, Node, and Element commands on the Modify, Move By...; Modify, Rotate By...; Modify, Reflect...; and Modify, Align... menus.
- Updated Mesh, Connect, Rigid command by moving the Independent DOF (Target) and Dependent DOF (Source) options for specifying degrees-of-freedom for the newly created elements to the top of the dialog box. Also, renamed the Rigid Element Options section to Node Pairing Options, which now contains four options. Finally, the Preview Target Nodes option has been moved into the Target Node Selection section.
- Updated Mesh, Edge/Skin Elements, Planar Elements on Faces command to display the standard "Face Selection" dialog box instead of having the user select elements, then faces, which was often an non-required step. To restrict which elements to "skin", simply use the "From # Elements" button in the standard "Face Selection" dialog box to limit which elements to consider.
- Updated Mesh, Edge/Skin Elements, Line Elements on Edges command to display the standard "Edge Selection" dialog box instead having the user select elements, then edges. To restrict which

elements to “frame”, simply use the “From # Elements” button in the standard “Edge Selection” dialog box to limit which elements to consider.

- Updated Mesh, Geometry, Solids command by adding the “2..Into Model” option to the Merge Nodes drop-down in the Automesh Solids dialog box. This differs from “1..New Nodes”, which would not merge nodes of newly created mesh with nodes of currently existing mesh, and “3..All Nodes”, which would simply perform a node merge for all nodes in the model.
- Updated the commands on the Mesh, Extrude... menu to use the standard Edge Selection dialog box when Method is set to Along Element Edges, which improves workflow and allows selection of solid element edges.
- Updated the commands on the Mesh, Sweep... menu by adding Added Merge Nodes to Adjacent Elements and Limit Sweep to Visible Mesh to the Options section. Both of this options are only available when Method is set to Along Element Edges. To automatically merge the nodes of the new elements which are created by the sweep to existing adjacent elements, use the Merge Nodes to Adjacent Elements option, which is enabled by default. When using Limit Sweep to Visible Mesh, which is enabled by default, elements will only be swept along the mesh which is currently visible in the “active” view.

Listing

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

Aeroelasticity

- Added the ability to preview the Aero Mesh to the for Control Surface 1 and Control Surface 2 to the Create Aero Control Surface dialog box
- Added support for MONPNT1 for Simcenter Nastran and MSC Nastran

Output and Post-Processing

- Added the ability when plotting an IsoSurface which allows a single IsoSurface to be used as a “cap”, which will contour all elements and portions of elements which have a value below the specified value (“2..Negative Capped Isosurface”) or above the specified value (“3..Positive Capped Isosurface”). This option can be controlled by the IsoSurface option in View Options or via the drop-down when using “Dynamic IsoSurface”
- Added the ability when plotting a Section Cut, set to Cut Model, to choose if the display is showing all elements on the side of the plane toward the positive plane normal are removed (“0..Negative Cap”) or all elements on the side of the plane toward the negative plane normal are removed (“1..Positive Cap”). This options can be set via the drop-down in the Section Cut Options dialog box or when using “Dynamic Cutting Plane”
- Updated the Model, Output, Expand Complex command to automatically add the newly created Output Sets into the same Analysis Study as the original Output Set that was used to create them
- Updated the static ranges of Output Vector IDs, as the ranges in previous versions of FEMAP cannot accommodate the large numbers of new output quantities and element types that are found in analyses being run by a large number of FEMAP users. To accommodate the ongoing addition of

output quantities, some of the existing output vectors have been moved to new ID ranges. If opening a model or importing a neutral file from a version of FEMAP prior to 2020.1, the output vector IDs will be updated automatically and there should be no difference in behavior. The following table shows both the old and new output vector ID ranges.

Groups and Layers

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

Tools

- Added the Tools, Check, Mesh Interference command, which examines a selected set of elements to determine if any of the selected elements interferes with any of the other selected elements (i.e., element passes through and/or overlaps another element). This command can be used to determine if interference is occurring between line, planar, and/or solid elements and the cross-section of line elements and the thickness of planar elements are taken into consideration.
- Updated the Tools, Check, Coincident Nodes command by adding the Delete Midside Nodes in Linear/Parabolic Transitions option, which will delete any midside nodes on parabolic elements which are transitioning to linear elements after the corners of the elements have been merged.
- Updated the Tools, Check, Sum Forces command to be able to list the individual load summation for All Load Sets or any number of Selected Load Sets. In addition, added an option to send the individual summation for each load set to the Data Table.

Model Merge

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

User Interface - General

- Added Select from Standard Select Dialog icon to dialog boxes which bring up the “multi-select dialog box with check boxes” by default (for example, List, Model, Load Definition or Delete, Output, Set). This can be helpful if the IDs for entities that need to be selected already exist in a list or spreadsheet, as the Paste functionality on the Pick^ menu can then be used.
- Improved checking for determine if FEMAP is exceeding memory. Prior releases would give errors when less than 25% of memory is left. For machines with large memory, for instance a machine with 64GB of RAM, this error could occur when you have 16GB left. The error was a dialog box that the user had to answer. Initial occurrences of the error are now just warnings to the Messages window and the dialog box is only displayed when available memory goes below 5% and 100MB.
- Updated the Visibility dialog box to allow simply typing an ID into the various combo boxes to update the selected entity. Previously, combo boxes in this dialog box required the user to choose from the drop-down list to update the selections (ER# 9512008)

- Updated FEMAP to support “Large IDs” (i.e., IDs larger than 99,999,999). Not all solvers will support values higher than 99,999,999, so it is up to the user to determine is using “Large IDs” is appropriate.
- Updated all icons throughout the User Interface. This includes icons in the menu structure, on toolbars, in dialog boxes, and used in dockable panes.
- Updated all “Palette” buttons with “color block” buttons which now show the color of the entity, including settings for Transparency. In addition, if any setting for Line Style is specified, it will also be shown below the “color block” in the button.

Preferences

Graphics

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

Interfaces

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

API

To minimize the immediate impact on users with APIs, all existing API properties and Methods will CONTINUE TO WORK as before, BUT will only work when using the pre-2020.1 output vector IDs. This means existing APIs will works as they always have, and can be updated at your convenience to the new output vector IDs. While API scripts will work as they did before, if an API creates an output vector, and the ID of that output vector has changed, then the new ID for that output vector will be displayed in the FEMAP User Interface. If the pre-2020.1 output vector ID is used later in the API, it will NOT be found.

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

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

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

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

- Removed Output Object and all related methods.

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

- feFileAttachSaveV2
- feFileAttachSave2V2
- feFileWriteFNOV2
- feFileWriteFNO2V2
- feFileWriteFNO3V2
- feModelFileVersionV2
- feMeasureDistanceBetweenNodesV2
- feMeasureDistanceBetweenNodes2V2
- feMeasureAngleBetweenNodesV2
- feOutputProcessV2
- feOutputProcessCopyV2
- feOutputProcessMergeV2
- feOutputProcessLinearCombinationV2
- feOutputProcessRSSCombinationV2
- feOutputProcessConvertV2
- feOutputProcessEnvelopeV2
- feOutputProcessEnvelopeFromSetsV2
- feOutputProcessErrorEstimateV2
- feOutputTransform2V2
- feOutputConvertV92VectorIDV2
- feOutputConvertV2019VectorIDToV2020V2
- feOutputConvertV2020VectorIDToV2019V2
- feOutputCompareV2
- feResultsToDataTableV2
- feResultsToDataTable2V2
- feResultsRankingToDataTableV2
- feOutputGlobalPlyV2
- feDeleteOutputV2
- feDeleteOutput2V2
- feDeleteOutputEntryV2
- feSelectOutputV2
- feSelectOutput2V2

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

- Added OutputVectorV2 and OutputVector2V2 attributes to the Chart Data Series Object

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

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

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

New and updated API Objects and Attributes

- Added Results Vector ID Query (feResultsIDQuery) object to the API. The feResultsIDQuery object has no attributes, only methods.
- Added Monitor Point (feMonitorPoint) object to the API. Also, added color, layer, title, Name, type, defCSys, outCSys, NodeGroup, ElemGroup, xyz, vxyz, SumComponents, vSumComponents, SumContributions, vSumContributions, HasNodeList, HasElemList, HasPanelList, CompTitle, NodeSetID, ElemSetID, PanelSetID, HasAeroMeshList, AeroMeshSetID, x, y, and z attributes to the feMonitorPoint Object.

- Added Direct Matrix Input (feMatrixInput) object to the API. Also, added title, Name, InputForm, InputPrecision, OutputPrecision, ComplexForm, colCount, matrixDir, AssignForm, UnitID, FileInputForm, ScaleFactor, and DataSource attributes to the feMatrixInput Object.
- Added AutoPopulate, NextSetStartID, NextVectorStartID, and NextStudyStartID attributes to the Results Browsing Object.
- Added CopyActiveColor and CopyToActiveCSys attributes to the CopyTool Object.
- Added NasMsnlCntINACCN, NasMsnlCntINERTIA, NasMsnlCntLOADOFF, NasMsnlCntMSTAB, NasMsnlCntMSFAC, NasMsnlCntNORMP, NasMsnlCntNORMU, NasMsnlCntREFP, NasMsnlCntREFU, NasMsnlCntRHOINF, NasMsnlkCnt2IAR0, NasMsnlkCnt2OTRE, NasMsnlkCnt2PRCF, NasEnableDMIG, NasDMIGK2GG, NasDMIGM2GG, NasDMIGB2GG, NasDMIGK42GG, NasDMIGP2G, NasDMIGK2PP, NasDMIGM2PP, NasDMIGB2PP, NasCaseMonitorEnabled, NasCaseMonitorAeroEnabled, NasCaseMonitorLoadEnabled, NasMsnlCntQuasiNewtonIter, and NasCaseAlternateTextLocation attributes to the Analysis Case Object.
- Added NasMsNLKGlobalSTROUT, NasMsNLKGlobalSTRCONV, NasMsNLKGlobalPRECPIVO, NasMsNLKGlobalNKINE, NasMsNLKGlobalMODEVERS, NasMsNLKGlobalRSTGEN, NasMsNLKGlobalRSTUNIT, NasMsNLKGlobalRSTFROM, NasMsNLKGlobalEXEFROM, NasMsNLKGlobalMDLVAL, NasMsnlModalMassForm, NasMsnlCntINACCN, NasMsnlCntINERTIA, NasMsnlCntLOADOFF, NasMsnlCntMSTAB, NasMsnlCntMSFAC, NasMsnlCntNORMP, NasMsnlCntNORMU, NasMsnlCntREFP, NasMsnlCntREFU, NasMsnlCntRHOINF, NasMsnlkCnt2IAR0, NasMsnlkCnt2OTRE, NasMsnlkCnt2PRCF, NasCaseAlternateTextLocation, NasEnableDMIG, NasDMIGK2GG, NasDMIGM2GG, NasDMIGB2GG, NasDMIGK42GG, NasDMIGP2G, NasDMIGK2PP, NasDMIGM2PP, NasDMIGB2PP, NasMonitorEnabled, NasMptSetID, NasMonitorAeroEnabled, NasMonitorLoadEnabled, and NasMsnlCntQuasiNewtonIter attributes to the Analysis Manager Object.
- Added Disabled attribute to the Connection Region Object.
- Added PreloadType attribute to the LoadBolt Object.
- Added NumberOfMatrices attribute to the Node Object.
- Added SectionCapMode attribute to the View Object.
- Added DO_Loads_Option, DO_Loads_RoundToZero, DO_Loads_RoundToZeroValue, DO_Loads_BeforeDecimalSeparator, DO_Loads_MinimumExponentSize, DO_Loads_ExponentDigits, DO_Loads_SigFigs, DO_Loads_LeadingZeros, DO_Loads_TrailingZeros, DO_Loads_DecimalPlaces, DO_Loads_MaxDigits, DO_Loads_BaseExponent, DO_Loads_OriginalDigits, and DO_Loads_Exponent attributes to the View Object. These attributes are used by the “Load Vectors” option in View Options to control the display of digits in the graphics window for Load Vectors
- Added DO_BCs_Option, DO_BCs_RoundToZero, DO_BCs_RoundToZeroValue, DO_BCs_BeforeDecimalSeparator, DO_BCs_MinimumExponentSize, DO_BCs_ExponentDigits, DO_BCs_SigFigs, DO_BCs_LeadingZeros, DO_BCs_TrailingZeros, DO_BCs_DecimalPlaces, DO_BCs_MaxDigits, DO_BCs_BaseExponent, DO_BCs_OriginalDigits, and DO_BCs_Exponent attributes to the View Object. These attributes are used by the “Constraint” option in View Options to control the display of digits in the graphics window for Constraints (Non-Zero)

- Added DO_Deformed_Option, DO_Deformed_RoundToZero, DO_Deformed_RoundToZeroValue, DO_Deformed_BeforeDecimalSeparator, DO_Deformed_MinimumExponentSize, DO_Deformed_ExponentDigits, DO_Deformed_SigFigs, DO_Deformed_LeadingZeros, DO_Deformed_TrailingZeros, DO_Deformed_DecimalPlaces, DO_Deformed_MaxDigits, DO_Deformed_BaseExponent, DO_Deformed_OriginalDigits, and DO_Deformed_Exponent attributes to the View Object. These attributes are used by the "Deformed Model" option in View Options to control the display of digits in the graphics window for the model when deformations are being shown as vectors
- Added DO_Criteria_Option, DO_Criteria_RoundToZero, DO_Criteria_RoundToZeroValue, DO_Criteria_BeforeDecimalSeparator, DO_Criteria_MinimumExponentSize, DO_Criteria_ExponentDigits, DO_Criteria_SigFigs, DO_Criteria_LeadingZeros, DO_Criteria_TrailingZeros, DO_Criteria_DecimalPlaces, DO_Criteria_MaxDigits, DO_Criteria_BaseExponent, DO_Criteria_OriginalDigits, and DO_Criteria_Exponent attributes to the View Object. These attributes are used by the "Contour/Criteria Style" option in View Options to control the display of digits in the graphics window for values of a Criteria Plot, when Max/Min Labels are being displayed in a Contour Plot, and/or when Beam Diagrams are displayed with Labels
- Added DO_ContourLegend_Option, DO_ContourLegend_RoundToZero, DO_ContourLegend_RoundToZeroValue, DO_ContourLegend_BeforeDecimalSeparator, DO_ContourLegend_MinimumExponentSize, DO_ContourLegend_ExponentDigits, DO_ContourLegend_SigFigs, DO_ContourLegend_LeadingZeros, DO_ContourLegend_TrailingZeros, DO_ContourLegend_DecimalPlaces, DO_ContourLegend_MaxDigits, DO_ContourLegend_BaseExponent, DO_ContourLegend_OriginalDigits, and DO_ContourLegend_Exponent attributes to the View Object. These attributes are used by the "Contour/Criteria Legend" option in View Options to control the display of digits in the graphics window for the Contour Legend
- Added DO_ContourArrow_Option, DO_ContourArrow_RoundToZero, DO_ContourArrow_RoundToZeroValue, DO_ContourArrow_BeforeDecimalSeparator, DO_ContourArrow_MinimumExponentSize, DO_ContourArrow_ExponentDigits, DO_ContourArrow_SigFigs, DO_ContourArrow_LeadingZeros, DO_ContourArrow_TrailingZeros, DO_ContourArrow_DecimalPlaces, DO_ContourArrow_MaxDigits, DO_ContourArrow_BaseExponent, DO_ContourArrow_OriginalDigits, and DO_ContourArrow_Exponent attributes to the View Object. These attributes are used by the "Contour Arrow Options" option in View Options to control the display of digits in the graphics window for values of Contour Arrows
- Added DO_FreeBody_Option, DO_FreeBody_RoundToZero, DO_FreeBody_RoundToZeroValue, DO_FreeBody_BeforeDecimalSeparator, DO_FreeBody_MinimumExponentSize, DO_FreeBody_ExponentDigits, DO_FreeBody_SigFigs, DO_FreeBody_LeadingZeros, DO_FreeBody_TrailingZeros, DO_FreeBody_DecimalPlaces, DO_FreeBody_MaxDigits, DO_FreeBody_BaseExponent, DO_FreeBody_OriginalDigits, and DO_FreeBody_Exponent attributes to the View Object. These attributes are used by the "Freebody Vectors" option in View Options to control the display of digits in the graphics window for values of Freebody arrows

- Updated NodeMergeOption attribute on the Geometry Preparation and Meshing Object. It now has 4 possible values instead of 3.

New and Updated API Methods

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

New and Updated Global Variables

- Added Pref_DL_UseIntlAll, Pref_DL_DecimalSeparator, Pref_DL_DigitGroupSeparator, Pref_DL_DigitGroupOption, Pref_DL_DigitGroupFlag, Pref_DL_ExponentChar, Pref_DL_TrailingZeros, Pref_DL_ForceSingleZero, Pref_DL_ForcePlusSign, and Pref_DL_NegativeBraces to set Local preferences for Digit Control.
- Added Pref_DO_Option, Pref_DO_RoundToZero, Pref_DO_RoundToZeroValue, Pref_DO_BeforeDecimalSeparator, Pref_DO_MinimumExponentSize, Pref_DO_ExponentDigits, Pref_DO_SigFigs, Pref_DO_LeadingZeros, Pref_DO_TrailingZeros, Pref_DO_DecimalPlaces,

Pref_DO_MaxDigits, Pref_DO_BaseExponent, Pref_DO_OriginalDigits, and Pref_DO_Exponent to set Options preferences for Digit Control.

- Added Pref_SkipDMIGEntries to set preference on Interfaces tab.
- Added Pref_API_WarningLevel and Pref_API_WarningInDialog to set preferences for warning messages about deprecated API calls (can only be set in GUI when the warning dialog is displayed).
- Added SelectOutputSetsFromStudies, which when set to True, will show “Study View” instead of “All Results View” in dialog boxes which display it both ways, such as “Select Output Sets”.
- Added InitializationPath and ConfigurationPath, to set the directory path to the FEMAP Initialization file (femap.ini) or the FEMAP Configuration file (config.ini).
- Updated Pref_RenderTrailingZeroes to set the same value as Pref_DL_TrailingZeros

The following functions have been added or updated:

- feModelFileVersion (works on unopened model files)
- feCheckElemInterference
- feConnectLinearParabolic
- feCheckCoincidentNode4
- feLicensePrintInfo
- feLicenseGetInfo
- feSplineMultipleCurves
- feMeshConnectRigid
- feModifyPlateThickOffsetBetweenNodes
- feModifyPlateThickOffsetValue
- feModifyPlateThickOffsetEquation
- feModifyPlateThickOffsetReset

Corrections

General

- Corrected issue that potentially caused a large number of message dialogs to be displayed. If user created a variable geometric load but had an error in the specified equation when the load was expanded an error was displayed for every node/element where the node was applied. Now, the equation is checked when it is defined and if errors are detected during expansion they are simply written to the message window rather than displaying a large number of message dialog boxes.

Geometry

- Corrected issue where copied curves were not having the their mesh sizing copied from the original curves
- Corrected issue which caused the title of a solid to sometimes be lost when the solid was edited using various commands

- Corrected issue with Geometry, Solid, Thicken command which caused surfaces and/or curves of newly created geometry to not have periodic splits
- Corrected issue when attempting to use the Geometry, Solid, Embed Face command on a sheet solid which could cause FEMAP to become unresponsive
- Corrected issue that occurred in the Geometry, Copy, Surface command if user selected one or more surfaces from a non-manifold solid that contained solid regions. Previously some surfaces that were not being copied were moved along with the copied surface.

Graphics

- Corrected issue where cohesive elements drawn with property thickness were not incorporated correctly when determining Free Faces
- Corrected issue which occurred when feature lines are being displayed and the user has an active entity in the Select Toolbar. This caused incorrect colors of the feature line and incorrect transparency of the model when the model was dynamically rotated.
- Corrected issue where plate offsets prevented the “element split preview” to be shown on both sides of the element when using the Mesh, Editing, Element Refine command
- Corrected issue with Beam Elements drawing End B stress locations in the wrong location if the beam was non-tapered and the neutral axis offsets are different for End A and End B
- Corrected issue where non-zero constraints were not drawn or labeled correctly
- Corrected issue where nodal contour arrows were not labeled in the interior of mesh consisting of solid elements
- Corrected issue where the visibility of mesh points was not being handles correctly, which caused hidden mesh points to remain visible
- Corrected issue where criteria labels for integer values being displayed by Model Data Contour were being displayed as a real number, which placed a decimal point at the end of the value
- Corrected issue where element coordinate systems are not drawn correctly when a criteria plot is being display and Performance Graphics is disabled. The only element coordinate systems drawn were on “elements which pass”, regardless if the elements themselves were visible or hidden.
- Corrected issue where the Beam Diagram option for Contour was not working in Performance Graphics or non-Performance Graphics. Non-Performance Graphics was first broken in Femap 11.2.0.
- Corrected issue where text on the orientation cube was not drawn when using various commands on the File, Picture menu, but only when Performance Graphics was disabled
- Corrected issue when displaying feature lines in models containing planar and cohesive elements mixed with solid elements, which caused additional feature lines to be generated
- Corrected issue where beam element contours were not drawn with two values when two values were available and Performance Graphics was disabled
- Corrected issue where loads displayed in local coordinate systems were not displayed precisely when Performance Graphics was disabled
- Corrected issue where undeformed draws element coordinate system and material direction.

Performance Graphics

- Corrected issue where End A offset values were being used for both ends of beam elements which reference non-tapered PBEAM properties which have different a neutral axis offset at each end.
- Corrected issue where elements with no contour arrow results are labeled with a value of "-1"
- Corrected issue with combined constraint sets and combined load sets which caused a large degradation in performance (PR# 9438554)
- Corrected issue where pressure load arrows are not drawn when Performance Graphics in enabled and graphics acceleration is being handles by Intel hardware
- Corrected issue where elemental loads (temperature, heat generation, radiation, convection and heat flux) were not drawn correctly when Performance Graphics in enabled and graphics acceleration is being handles by Intel hardware
- Corrected issue where nodes are not drawn if using Free Edge with Model drawn. This issue was introduced in 2019.1 MP1 when a change was implemented to correct another issue.
- Corrected issue when Deformed Style is set to Arrow which displays negative components in the wrong direction
- Corrected issue where contour arrows drawn for 1-D elements, such as mass elements, were not drawn in the correct location
- Corrected issue when displaying pressure loads on a mesh containing mixed element topologies. For instance, if user has pressure loads applied on linear triangle and linear quadrilateral elements, the pressure loads on the linear triangle elements were not drawn in the correct location if linear quadrilateral elements were also being drawn.
- Corrected issue where which caused the axis labels for element coordinate systems to be drawn in the incorrect location for solid elements
- Corrected issue where free edges were not being properly clipped

GUI - General

- Corrected issue where various Optimization Relation dialog boxes would not correctly show that no elements were selected when the user had previously selected elements but then cleared them out via the selection dialog.
- Corrected issue in Element Thickness Relation and Topology Relation dialog boxes that could create relations with no elements, if the user selected only invalid elements and chose to remove invalid elements from the selection list.
- Corrected issue in Element Thickness Relation dialog box where no error message was displayed when the lower bound thickness was < 0.
- Corrected issue in various Optimization dialog boxes when optimization entities contained titles longer than 79 characters, which could possibly cause memory corruption
- Corrected issue which could occur when switching between Material Relation and Property Relation dialog boxes, that caused the selection list to not be properly cleared and could result in extra relations being created.
- Corrected issue in Material Relation dialog box when selecting a natively unsupported material and not defining the Type and Name fields; the error message would incorrectly indicate that an unsupported property was selected.

- Corrected issue in Optimization Limit dialog box that would allow incompatible elements to be selected for certain responses when entities were selected by element ID rather than by property.
- Corrected issue in Optimization Limit dialog box that could result in an optimization limit being created with zero elements if only invalid entities were selected.
- Corrected issue which occurred when using the Previous button in the Vector dialog box, when Methods^ is set to Components, which caused the vector to not be transformed into a user defined coordinate system and/or be properly scaled.
- Corrected issue which occurred when using Vector dialog box, when Methods^ is set to Components, which caused the vector to not be properly transformed when switching from a Rectangular coordinate system to a Cylindrical or Spherical coordinate system.
- Corrected issue with the Palette Button in various dialog boxes, including all dialog boxes used to define entities for aeroelasticity. Previously, if user clicked the button and chose a new color, then clicked OK to exit the Color Palette, then clicked the button again without leaving the command dialog box, the default would be the original color, not the color the user just chose.
- Corrected issue that potentially caused a large number of message dialogs to be displayed. If user created a variable geometric load but had an error in the specified equation when the load was expanded an error was displayed for every node/element where the node was applied. Now, the equation is checked when it is defined and if errors are detected during expansion they are simply written to the message window rather than displaying a large number of message dialog boxes.

GUI - Dockable Panes

Model Info Tree

- Corrected issue that prevented Connections, Connection Regions and Connection Properties from working properly when using the Locate in Model Info capability of the Select Toolbar
- Corrected issue which allowed items in multiple branches of the tree to be selected and offer commands on a context-sensitive menu which was only valid for certain entities

Meshing Toolbox - General

- Corrected issue that occurred when using the Meshing Toolbox to remesh solids with tetrahedral element that did not already have meshing attributes defined on their surfaces. If this was done prior to meshing other solids with tetrahedral elements or meshing other surfaces with triangular elements, then the surfaces would be meshed with quadrilateral elements, resulting in a different and possibly inferior tetrahedral mesh in the solid.
- Corrected issue which caused non-manifold solids to not be properly remeshed after using any tool in the Meshing Toolbox to move a point

Meshing Toolbox - Entity Locator

- Corrected issue which occurred when user attempted to search for Surfaces using the Element Quality Search Method they were unable to select the Nastran Quality option, even though it was displayed. Now it is available for selection.

Entity Editor

- Corrected issue that caused FEMAP to exit unexpectedly when a Layup that contains more than 76 plys was loaded into the Entity Editor

Data Surface Editor

- Corrected issues when using the Result Set Processing Data Surface which caused the newly created combined output sets to have FEMAP specified as the program that generated the results instead of the solver used to generate the original output sets. This was an issue when displaying beam diagrams, as the solver specified for an output set is used to plot an appropriate beam diagram.
- Corrected issue which caused the Output Map Data Conversion Options in the Data Surface Editor Options dialog to be available when they should no have been available. Previously it appeared additional options were available for other Data Surface Types. While setting these options did not cause any issues, the should not have been available.

Function/Table Editor

- Corrected issues when using the Result Set Processing Table which caused the newly created combined output sets to have FEMAP specified as the program that generated the results instead of the solver used to generate the original output sets. This was an issue when displaying beam diagrams, as the solver specified for an output set is used to plot an appropriate beam diagram.
- Corrected issue which caused Load Set Combination Tables and Result Set Processing Tables to not be properly renumbered when a table of either of these types was renumbered.

PostProcessing Toolbox

- Corrected issue where Dynamic Isosurface, Dynamic Cutting Plane, and Streamline did not account for actions like switching geometry on/off when they were initiated from the PostProcessing Toolbox

Interfaces - Nastran

- Corrected issue when reading Laminate Stress results from the .f06 output file which could cause the results to be placed into multiple output sets
- Corrected issue which caused a null number of desired roots to overwrite a non-zero number if both real and complex eigenvalue extraction data was specified
- Corrected issue which would cause the NASTRAN read translator to not recognize certain output requests if they had extra characters appended to the output requests, even if the extra characters were valid

Interfaces - Simcenter Nastran (formally NX Nastran)

- Corrected issue with writing out topology optimization input files where weight and volume design responses were not correctly placed into DESGLB case control, but only when previous optimization limits existed in the model which were deleted prior to the creation of the weight/ volume optimization limit.

- Corrected issue which caused duplicate ASSIGN and PARAM statements to be written for Rotor Dynamics solutions. These were for the Rotor Dynamics post files and their associated parameters that designate their unit numbers. If these already existed in the input file being imported, they would be saved as text, and written out again by the translator.
- Corrected issue which prevented enforced displacement (SPCD) entries from being written for a combined load set in SOL 401/SOL 402
- Corrected issue which prevented Body Loads which referenced a “vs Time” function from being exported for SOL 401/SOL 402 (PR# 9567111)

Interfaces - ABAQUS

- Corrected issue which caused orientation coordinate systems to not being properly applied to CBUSH elements when importing a CONN3D2 entry (CARTESIAN, CARDAN or AXIAL type)
- Corrected issue which caused some “S” based loads, such as Heat Flux, to not be imported
- Corrected issue that allowed commas ',' to be written to NAME fields, which was not valid, as ABAQUS interprets a comma as a field delimiter. Commas are now replaced by “_” for NAME fields.
- Corrected issue when reading an ABAQUS input file to abort when reading a Surface interaction if USER appears in GAP CONDUCTANCE and GAP CONDUCTANCE precedes GAP RADIATION, resulting in RADIATION values not being read.
- Corrected issue which caused body loads (DLOAD types: ROTA, CENTRI, and GRAV) to not be imported

Interfaces - ANSYS

- Corrected issue with writing and reading of D and F commands, where different functions are used in different degree of freedom directions
- Corrected issue with writing BF and BFE commands, where a line return is missing
- Corrected issue with writing coefficients of thermal expansion by allowing negative coefficients to be written
- Corrected issue with reading NBLOCK command by fixing format (1i9,3e20.9e3)
- Corrected issue with parameter substitution when reading ANSYS input files

Interfaces - LS-DYNA

- Corrected issue which could cause *CONTACT_AUTOMATIC_GENERAL to be written, even if it is not needed
- Corrected issue where *DEFINE_CURVE is used wrongly for *DEFINE_TABLE_2D

Interfaces - MSC Marc

- Corrected issue when reading nodal results reported in local coordinate systems from the .t16 output file (PR# 9400723)

Element Update

- Corrected issue which caused the Modify, Update Elements, Sold Material CSys command to produce improperly aligned CSys except under limited circumstances
- Corrected issue which allowed the Modify, Update Elements, Property ID command to attempt update of element types, such as rigid elements, that do not reference a property ID.

Loads and Boundary Conditions

- Corrected issue that caused corruption of Geometric Load and Constraint Definitions when using the commands on the Modify, Move menu to move Nodes or Elements without also moving Geometry
- Corrected issue that occurred when Editing Loads. Previously, if user edited a Load Definition that contained multiple Loads and either applied a Data Surface or an equation that caused some of those Loads to have a Zero Magnitude those invalid "zero loads" would still be created, and while they were not drawn, they could be listed. Now any Loads with Zero Magnitude are removed from the model and an error message in a dialog box is displayed.

Meshing

- Corrected issue which allowed multiple coincident Mesh Points to be considered when meshing a Surface. Now, one mesh point is used in meshing and any others are listed as an error and ignored.
- Corrected issue when using the Mesh, Editing, Edge Split command which could cause FEMAP to exit unexpectedly when splitting elements which are included in a NonStructural Mass Region
- Corrected issue in solid element meshing which could allow the nodes of automatically generated pyramid elements to not be properly merged with the nodes of existing hexahedral and wedge elements
- Corrected issue that occurred when meshing a surface that was a very small portion of a much larger cylinder (less than 1/50 of a degree of arc) and specifying a mapped meshing approach. Previously, interior nodes in the mesh were incorrectly collapsed. Now, the surface meshes correctly.
- Corrected issue that occurred when using commands on the Modify, Rotate By menu. Previously, if elements were not explicitly selected, but all of their nodes were rotated by an operation, then attributes like Beam Orientations and Offsets would not be transformed for those elements. This could happen if a user rotated coordinate systems or nodes, or even if they rotated elements that caused all of the nodes of other non-selected elements to rotate.
- Corrected issue that caused the Mesh, Geometry, Solids From Surfaces command to fail if user tried to use surfaces that were not meshed and part of a non-manifold sheet solid, even if they do form a closed volume (PR# 9582471)

Output and Post-Processing

- Corrected issue in Chart Data Series dialog box, when Type is set to "Vector vs Vector", where the quantities specified for Output Vector 1 and Output Vector 2 could swap positions when filtering output sets by Study.

- Corrected issue where the visibility of freebody entities could be reset when changing output sets using the Freebody Tool in the PostProcessing Toolbox when contour results were not displayed.
- Corrected issue when processing topology optimization results which were not written for hexahedral elements because of an issue with the solver
- Corrected issue where freebody information was listed in Post Titles even if there were no results in the model at all or specifically for that freebody

Model Merge

- Corrected issue in File, Merge command that could cause Simcenter Nastran Bolt Sequences in an Analysis Set to be improperly merged
- Corrected issue which caused the setting for the Create Parent CSys for Merged Model option to be ignored in most instances
- Corrected issue which caused global coordinate systems to not be duplicated and transformed when used by certain types of constraints
- Corrected issue which caused the "Offset By" option in the Model Merge Manager dialog box to be unavailable if the "Renumber to Next IDs" option was enabled, even though it is hidden for "Offset Renumbering". Previous workaround to make "Offset By" available, required user to turn off "Renumber to Next IDs" prior to choosing the "Offset Renumbering" option.
- Corrected issue in File, Merge when merging Layup entities from one more into another model. Previously if user was merging Layups and they chose to overwrite existing Layups in the active Model, if the Layup being merged into the model had fewer plies than the one in the active model, the first plies were overwritten but the extra plies were not removed. Now the resulting Layup properly matches the one being merged. (PR# 9231325)

Preferences

Graphics

- Corrected issue where labels being displayed by Model Data Contour were not controlled by the Labels option in the Include in Dynamic Rotation section

Geometry/Model

- Corrected issue that caused load expansion on element edges to be incorrect. Between Version 12 and version 2019.1 the "Along Edge" field on the Geometry Tab of the File, Preferences command was lost from the Preferences dialog box, which caused an incorrect value to be stored in the FEMAP.INI file every time a user used the File, Preferences command. That incorrect value was then used during load expansion, causing it to be incorrect. The field has been restored and now works properly. The field name in the FEMAP.INI for this variable has been changed so that no old incorrect values will ever be used in version 2020.1 or later. (PR# 9531215)

JT Files

- Corrected issues where the JT Units drop-down was not being handled correctly in User Interface and the selected units were not written correctly to JT file.
- Corrected issue where only some surfaces were colored correctly in JT files written by Femap

- Corrected issue where the item selected via the JT Name drop-down in the User Interface for the File, Picture, Save JT command was not being used. Updated and augmented what appears in the drop-down to include no duplicates and offer the model file name, the view name, the group name, and a name typed-in by the user.

API

- Corrected issue which occurred when duplicate output vectors are loaded into a Results Browsing Object and caused some output vector values to not be transformed
- Corrected issue with the PutLibrary API method. Previously if user attempted to use “libINDEX” >= 0 and the library did not exist or was empty, the method failed, but reported a successful outcome. Also, the documentation was incorrect regarding libINDEX=-1. It did not and does not display a dialog box... it appends to the end of the library.
- Corrected issues with the AddOutput method on the DataTable object that caused the output columns to not be visible and could potentially cause an abort.
- Corrected issue with the NextInSet method of the Output object. Previously, the “setID” argument was treated as if it was an Output Set ID, but was actually the ID of a Set object that contained the desired Output Vector IDs.
- Corrected isue with the feCurveProjectOntoSurfaces method. In v2019.1, using a negative number to specify the ID of a single curve to project did not work properly, while using the ID of a Set that contained the curves worked properly. Now, either option will work as expected.
- Corrected issues with the GetMultipleInRow, GetColumns, GetRowsAndColumns, and GetRowsAndColumnsByID methods of the Results Browser object. Previously, if these methods were used to retrieve multiple columns and one or more of those columns was not populated with data, the API call would fail even though other requested columns may have contained valid data. Now, they work properly in this case by returning the valid values and populating the missing values with the values specified for “ValueForNonExisting” property.

FEMAP v2019.1 MP 1 New Features and Corrections

Updates and Enhancements

Interfaces - Nastran

- Added ability to automatically generate buckling subcases in FEMAP using the “MultiSet” functionality of the Analysis Set Manager. The generated subcases are identical to the synthetic Nastran buckling subcases that were exported in previous versions of FEMAP.

Interfaces - ABAQUS

- Added read support for *DISTRIBUTION and *DISTRIBUTION TABLE entries. Currently supported variable are ORIENTATION and ANGLE. ORIENTATION gets translated to material angle on the element and ANGLE becomes the layup angle for the ply. This will create property and layup for each laminate element, as compression or consolidation of layups is not performed.
- Added read support for *COUPLING and *DISTRIBUTING entries, which are translated to RBE3 elements when certain specifications are met on the entries. Independent nodes must be specified as *SURFACE using NODES, while Dependent (reference node) must be specified as a single node, although ABAQUS lets you specify a set (NSET) In addition, the COUPLING type must be CONTINUUM and the Weighting must be UNIFORM (at all nodes = 1.0 for all 6 components).

Interfaces - Geometry

- Added support for NX 2019.1 v1872.

Corrections

General

- Corrected issue that caused FEMAP to exit unexpectedly when saving the model file when the notes section of the Explorer Tooltip text contained more than 9 lines (PR# 9521407).

Geometry

- Corrected issue with the Geometry, Curves - From Surface, Split at Locations command which allows it to properly split a surface, as it did in FEMAP versions prior to 12.0.
- Corrected issue with the Geometry, Solid, Fillet command where redundant solids points were being removed from the entire solid being filleted instead of only the newly created solids points that are redundant.
- Corrected issue where partial surfaces of solids were not considered when stitching, which could make it impossible to automatically include copies of existing surfaces and execute a stitch.

Graphics

- Corrected issue which occurred when using the File, Picture, Copy command when the screen resolution was scaled, which caused text/labels which are not being drawn with Performance

Graphics to not be scaled. In addition, corrected issue where the width of the contour legend when it is vertical and height when it is horizontal are also not being scaled correctly (PR# 9520223).

- Corrected issue where parabolic edges on shell elements were not being displayed correctly when previewing.
- Corrected issue where a connection region is drawn or previewed with extra faces on the elements in the connection region when “Elements with no results” is set to “1..Hide” for “Contour/Criteria Style” in the View Options dialog box.
- Corrected issue when using the Mesh, Editing, Interactive command which caused split elements to not appear immediately.
- Corrected issue where the graphics window may not update after closing the dialog box to complete the Magnify, Pan, or Rotate command on the View menu, when only the value of an edit field was changed.

Performance Graphics

- Corrected issue where nodes are not drawn when the “Elements as Free Edge” option is enabled in the “Include In Dynamic Rotation” section of the “Graphics” tab of the Preferences dialog box.
- Corrected issue which caused the material direction to be displayed incorrectly for quadrilateral elements when contour or criteria were displayed.

GUI - Dockable Panes

Meshing Toolbox - General

- Corrected issues where the mesh was not updating properly when moving points, curves, or surfaces when using various tools.

PostProcessing Toolbox - Contour Tool

- Corrected issue where the colors for Contour Arrow could not be edited in certain circumstances.

Interfaces - Simcenter Nastran (formally NX Nastran)

- Corrected issue that prevented SOL 401/SOL 402 axisymmetric element stress and strain from being read from the OP2 file when imported or attached (PR# 9523758).
- Corrected an issue that would cause SOL 402 nonlinear dynamics subcases to not properly import if “ANALYSIS = DYNAMICS” was used in the case control instead of “ANALYSIS = TRANSIENT”. Also, updated the translator to only write out “ANALYSIS = DYNAMICS” for SOL 401/SOL 402 nonlinear dynamics subcases.
- Corrected an issue that exported the incorrect value of ITEREF on the NLCNTL2 card. This would occur if ITMA was defined on the NLCNTL2 entry of an imported input file with a non-default value and did not also include a value for ITEREF.
- Corrected issue that caused illegal formatting of the NLCNTLG bulk data entry when multiple parameters were specified as non-default values in the Multi-Step Global Control Options dialog box in the Analysis Set manager (Analysis set to “28..Multi-Step Nonlinear Kinematic”).
- Corrected issue that prevented TABLEDi entries from being written for the CFVE contact control parameter for SOL 402 contact properties

- Corrected issue where design objective response specified by DESOBJ case control command may not be read correctly if SCSET or SCFUNC describers were used in the case control entry.
- Corrected issue where DRESP1 cards created by an Element Optimization Response with Category set to “Strain Energy” would not contain the appropriate entry for the PTYPE field.

Interfaces - ANSYS

- Corrected a number of issues when reading .cdb files, including improved performance when reading bf and bfe entries and ability to read different types of fixed formats.

Interfaces - LS-DYNA

- Corrected issue which would cause FEMAP to unexpectedly exit when exporting a model which had a value specified for Wz in the Rotational Velocity section of the Create Body Loads dialog box for the active Load Set (PR# 9516990).

Meshing

- Corrected an issue when using either the Mesh, Reflect, Element or Modify, Reflect, Element command which could cause an extra property to be created, then the extra property would be used by the reflected elements. This issue could occur when reflecting any planar element types, other than Plate or Plane Strain.

Output and Post-Processing

- Corrected issue where certain ply-based Y normal strain output from laminate elements could be marked as non-linearly combinable in linear output sets. This could create misleading messages when trying to combine output sets containing these vectors which were not categorized correctly.
- Corrected issue when plotting transformed results on a combination of pyramid and higher order elements that may possibly print a message about insufficient results to the message window. The message was in error and results were not affected.
- Corrected issue by removing the CSys ID column from the Data Table when using the List, Output, Results to Data Table command to list results for elemental output. This column was not used for elemental output and could be misinterpreted as a column indicating element output orientation.

FEMAP v2019.1 New Features and Corrections

Updates and Enhancements

Views

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

Analysis Manager

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

Connection Properties, Regions, and Connectors

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

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

Geometry

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

GUI - Toolbars and Icons

View and View - Simple Toolbars

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

Select Toolbar

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

GUI - Dockable Panes

Model Info Tree

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

Meshing Toolbox - General

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

Charting Pane

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

Entity Editor

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

Data Table

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

Interfaces - FEMAP Neutral

- Updated Neutral Read and Write for v2019.1 changes

Interfaces - Nastran

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

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

Interfaces - Simcenter Nastran (formally NX Nastran)

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

SOL 401 and 402 Only

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

SOL 401 Only

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

SOL 402 Only

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

Interfaces - MSC Nastran and Autodesk Nastran

- Added support to read and write pyramid elements

Interfaces - ANSYS

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

Interfaces - ABAQUS

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

Interfaces - Geometry

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

Node - Project

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

Element - Formulations

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

Element - Updates

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

Properties

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

Layups

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

Loads and Boundary Conditions

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

Optimization

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

Meshing

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

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

Listing

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

Aeroelasticity

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

Output and Post-Processing

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

Groups and Layers

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

Tools

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

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

Libraries

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

User Interface - General

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

Preferences

Database

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

Geometry/Model

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

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

Library/Startup

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

API

New and updated API Objects and Attributes

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

New and Updated API Methods

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

New and Updated Global Variables

- Added Pref_ModelThumbnail, Pref_ModelTooltip, PickVisibleOnly, Pref_SharedLibPath, Pref_GeometryBooleanTolerance, Pref_GraphicsFont, and Pref_GraphicsFontSize to set various preferences.
- Added Pref_SuppressScreenEntities, which can disable selection of all “Active Screen Entities” (i.e. View Legend, View Axis, Post Titles, and Contour Legend) in the graphics window.
- Added PickVisibleOnly option which enables the Select Visible Only option on the Pick^ menu of the standard entity selection dialog boxes and on Selector Actions icon menu of the Select toolbar.
- Updated Pref_MsgWndFontSize and Pref_MsgWndFontSize to set various preferences.

Removed Global Variables

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

The following functions have been added or updated:

- feGetCurrentLibraryName
- feSetCurrentLibraryName
- feSolidInsideMulti
- feMeshTetSolid3
- feModifyEdit

- feNodesBetweenNodes
- feSolidElementsInARow

Corrections

General

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

Analysis Manager

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

Connection Properties, Regions, and Connectors

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

Geometry

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

Graphics

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

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

Performance Graphics

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

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

GUI - General

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

GUI - Toolbars and Icons

Select Toolbar

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

GUI - Dockable Panes

Model Info Tree

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

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

Meshing Toolbox - Feature Removal Tool

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

Meshing Toolbox - Geometry Editing Tool

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

Charting Pane

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

Data Table

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

Interfaces - FEMAP Neutral File

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

Interfaces - Nastran

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

Interfaces - Simcenter Nastran (formally NX Nastran)

- Corrected an issue that caused GAPVAL for the SOL 401 BCTPARM card to not be exported or imported.
- Corrected an issue that prevented initial temperatures from being written with the ID specified in case control, for SOL 401, and caused certain TEMP entries to not be written as a result.
- Corrected an issue that would write an error message if bolt preloads defined by strain were applied to Type 3 bolt regions in SOL 401.
- Corrected an issue which prevents the BOLTRESULTS case control command from being written if only a bolt sequence had been specified for SOL 401.
- Corrected issue which caused Connection Regions which have Type set to Rigid to not be written out properly to use for SOL 402.
- Corrected an issue that caused the wrong default value for TSCEQ on the NLCNTL card to be exported.
- Corrected issue where “checkerboarding” manufacturing constraint on Simcenter Nastran DMNCON card was not being imported correctly.
- Corrected issue where NX Nastran Total and Elastic Strain output was read improperly.
- Corrected issue writing BGSET when both contact and glue Connectors referenced the same Connection Property (PR# 9305558).

Interfaces - ABAQUS

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

Interfaces - ANSYS

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

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

Interfaces - LS-DYNA

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

Layups

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

Meshing

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

Output and Post-Processing

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

API

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

FEMAP v12.0.1 New Features and Corrections

Updates and Enhancements

GUI - Dockable Panes

Charting Pane

- Improved algorithm for plotting angular values in “0..Vector vs Entity” Chart Data Series, set to Position, when transforming to cylindrical or spherical coordinate systems. Rather than only transforming polar to Cartesian, values are also offset by 360 degrees, as appropriate, to create the most continuous plot possible.

Data Surface Editor

- Added ability to use Criteria option in Output Map Data Conversion Options section of Define Options for Variation dialog box to create Nodal Loads.

Geometry Interfaces

- Updated File, Import, Geometry command in to automatically scale geometry to insure all geometry resides in the valid modeling region, which allows further operations to work properly. An attempt is made to update solids to the preferred modeling scale factor, provided all solids end up inside modeling region.

Loads and Constraints

- Added ability to use Criteria option in Data Conversion section of Map from Model Output dialog box to create Nodal Loads.

API

- Added ElementNoResultMode attribute to the View Object

Corrections

Views

- Corrected issue when blanking of solids that had either combined curves or combined surfaces. The underlying curves/surfaces were still shown if the “Combined-Eliminated Curves” and/or “Combined-Eliminated Surfaces” View Options were enabled.
- Corrected issue where the list of Visible Layers and list of Multiple Groups were not being properly loaded when using the View, Create/Manage command or when initializing Views if a Default View is selected from the View Library.

Connection Properties, Regions, and Connectors

- Corrected issue that caused connectors which were renumbered to no longer be properly referenced by Connector Sets.

General

- Corrected issue that could FEMAP to exit unexpectedly if attempting to open a file that was not a valid model file.
- Corrected issue in the Visibility dialog box that caused Untitled entities to appear only as IDs rather than the ID followed by “Untitled”.
- Corrected issue which caused a double occurrence of the FEMAP feature in the list of available licenses displayed in the Network License Information dialog box.
- Corrected issue where keyboard input may not be recognized in a dialog box after selecting value for a drop-down or other control by picking in a graphics window.

Geometry

- Corrected issue with slicing solids along a vector where the solid may not have been split properly if the curves were more than a certain distance apart. (PR# 9291736)
- Corrected issue when reflecting curves. Previously, if the curve had meshing attributes the cross section orientation and offset were not properly reflected. This occurred for both copying curves to new curves and moving existing curves.
- Corrected issue when rotating existing curves to a new location using the Modify, Rotate By commands. Previously, if the curves had meshing attributes the cross section orientation and offset were not properly rotated.
- Corrected issue reversing existing curves. Previously any biased or custom mesh size along the curve would also swap direction. Now mesh sizes stay in their original locations and only the orientation of the curve changes.
- Corrected issue in the Geometry, Surface, From Mesh command that could prevent assignment of assign mesh sizes to the curves that it created.
- Corrected issue which prevented Modify, Update Other, Reverse Curve to not be able to reverse curves on solids.
- Corrected issue, introduced in FEMAP version 12, where curves being projecting by various menu commands would not split the hidden face(s) of solids.
- Corrected issue which caused a memory leak because there was no composite curve record in the database for the composite curve.
- Corrected issue which caused slower than expected performance when moving entities in a model that contained a large number of Solids.
- Corrected issue that occurred when attempting to delete one or more surfaces from a solid portion of a nonmanifold solid, where the surface would not be properly deleted.
- Corrected issue which caused the Color and Layer information to be missing after geometric entities were split using various methods (i.e., “new” half of split entities went to the active layer, now to the same layer as the original entity).

Graphics

- Corrected issue that occurred when using File, Picture, Save or File, Picture, Save JT, then chose a file, then immediately went to another command that required entity selection without redrawing the screen. Previously, a string of highlighted entities would be drawn as the mouse was moved across the screen, as if the entities were being painting or simply a black screen with yellow highlighting depending on the Resolution settings in File, Preferences. A redraw would then display the view properly.
- Corrected issue that could cause some solid elements in Bolt and NonStructural Mass regions to only be drawn as a single face instead of the entire element.
- Corrected issue which could cause results on only a portion of shell elements (i.e., partial results) to be displayed incorrectly (PR# 9294192).
- Corrected issue where Z element axis arrow head was not being drawn properly if elemental x or y were into screen, as the cross drawn in that case used up the lines needed for the arrow head for z.

Performance Graphics

- Corrected issues which could cause RBE2 and RBE3 elements to not be displayed properly when using a NVIDIA graphics adapter.
- Corrected issue which would not inform users about using Performance Graphics with an Intel graphics adapter. An error message is now issued.
- Corrected issue which could cause a face of Parabolic Pyramid elements to not be processed correctly.
- Corrected issue which cause the labels of cohesive elements to not appear in the expected location(s).
- Corrected issue that caused elements where topology did not match the property type (i.e. 8-noded hexahedral Element with a Parabolic Solid Property) to not be drawn.
- Corrected issue which prevented parabolic shear panel element from being drawn when the elements referenced a linear shear panel property.
- Corrected issue which caused some Rigid elements to not be drawn (PR# 9332383, fixed in 12.0.1a).

GUI - General

- Corrected issue with underlining specific letters in dialog boxes to use as keyboard accelerators. In some coordinate/vector/plane dialog boxes the text changes from X,Y,Z to R,T,Z or R,T,P depending on the type of the active coordinate system. In most cases, these characters were not already used as keyboard accelerators, thus could be used to quickly move to a certain control. In other cases however, those letters are already used as keyboard accelerators for other controls, which could cause issues, especially when trying to replay a Program File.
- Corrected issue which could occur when attempting to pick solid elements when in Front or Query selection mode. In some cases, depending on which face of the solid element was closest to the front of the screen, an element that was not the one closet to the front could be selected.
- Restored the ability to use “Previous Command” following using “Tools, Undo”.

GUI - Dockable Panes

General

- Corrected issue which caused mouse wheel scrolling to not be available in the Messages dockable pane when another dialog box was open. This worked in previous versions and now does again.
- Corrected issue which caused mouse wheel scrolling to not be available in the Data Surface Editor, Function Table Editor, Mesh Point Editor and Entity Info dockable panes.

Model Info Tree

- Corrected issue that would allow users to select/edit connection properties for connector sets from the Model Info Tree. For example, a user would be able to select a "glue" property for a connector set that had been designated as "contact". If the user were to then go highlight all connectors using the glue property, connectors in a connector set that had been designated as "contact" would also be highlighted.
- Corrected issue which caused poor performance when showing/hiding large numbers of Layers when using any of the commands on the context-sensitive menu for the Visibility check boxes.

Meshing Toolbox - Geometry Editing Tool

- Corrected issue which caused the user to never be asked to specify a location when Operation was set to Extend and Extend To was set to Location.
- Corrected issue which could cause the "washer" pattern to not be properly applied around non-circular holes in sheet solids or general bodies when Operation is set to Washer. This would typically only occur when Solid Geometry Scale Factor was set to "2..Millimeters" on the Geometry/Model tab of the Preferences dialog box (fixed in 12.0.1a).

Meshing Toolbox - Surface Mesh Quality Tool

- Corrected issue which caused the contour legend to not appear in the graphics window when displaying mesh quality on plane elements.

Charting

- Corrected issue where renumbered functions, groups, and output sets would not be updated in various types of chart data series.

Entity Editor

- Corrected issue in Entity Editor that caused Untitled entities to appear only as IDs rather than the ID followed by "Untitled".
- Corrected issue which caused results on Tetrahedral and Wedge elements to not be displayed properly (PR# 9103348)

Data Surface Editor

- Corrected issue that occurred when an "Along Coordinates Data Surface" was used and evaluated. Each time it was evaluated it would create and leave behind extra Points in the model. These points were not used by anything and were therefore deletable, but should have been automatically deleted.

Mesh Point Editor

- Corrected issue which occurred when attempting to create a Mesh Point at a Location. Previously, the coordinate dialog showed ID and Parameter fields even though those fields were not applicable, thus, they are now properly disabled.
- Corrected issue when using the “Load From File” command which prevented appropriate access to comma separated value (.csv) files.

Data Table

- Corrected issue which caused results on Tetrahedral and Wedge elements to not be displayed properly (PR# 9103348).
- Corrected issue which could cause poor performance when showing the Summary Table, which was “pinned” to the last row in the Data Table.

Interfaces - FEMAP Neutral File

- Corrected issue where aero panels referenced by control surfaces, freebody entities (versions < 11.2), or optimization entities would not be taken into account when importing a neutral file and saving as a new model file.
- Corrected issue with opening a neutral file or model from a previous version with no other open models where list of visible layers or visible groups in the neutral file were not properly read, thus the view may not display what was displayed in the original model or neutral file.

Interfaces - Nastran

- Corrected issue with optimization limits where a limits written to the input file immediately after the continuation line on the DCONADD entry would be skipped.
- Corrected issue with attachment and post-processing of OP2 files with results referencing an output coordinate system that could cause poor performance.
- Corrected issue when reading NSM and NSML cards which had continuation lines (PR# 9294539).
- Corrected issue which caused the linear thermal gradient (TPRIME field) on the TEMPP1 entry to be written to the wrong field when using large field format.

Interfaces - NX Nastran

- Corrected issue where plate stresses in the positive crossing output set were not being read from op2 files created by NX Nastran 12.
- Corrected issue which caused nodal results with an output coordinate system to be incorrect when read from xdb files created by ILP-64 version of NX Nastran.
- Corrected issue that would swap values for convergence criteria (EPSP, EPSU) when importing or exporting SOL 401 analysis models. Also added functionality to round-trip convergence criteria (PR# 9322521)
- Corrected issue when writing BGSET when both contact and glue existed in the same model and a single Connection Property was used for multiple connections (PR# 9305558)

- Corrected issue which caused SOL106 output sets from imported op2 results to be sorted based on load factor rather than solution order. This created problems when simulating snap-through instability/buckling, where a forward step in the unstable portion of the analysis can cause a drop in load factor (PR# 9299199)
- Corrected issue when reading and writing the contact control options for stabilization damping and associated scale factors for SOL 401.
- Corrected issue which caused Total and Elastic Strain output to not be read properly for some element types.
- Corrected issue where Normalized Mass Density output was not being properly read from op2 files created by Topology Optimization analysis in NX Nastran 12. This would only occur when the number of "Active Elements" in the analysis exceeded 100,000 (PR# 9324713, fixed in 12.0.1a).

Interfaces - ABAQUS

- Corrected issue when importing an ABAQUS input file where lines longer than 160 characters were truncated, even though the max possible line length is 175 characters.
- Corrected issue when reading entries with only one field, when they entries do not end with a comma and exceed 10 characters (PR# 9283497)

Interfaces - ANSYS

- Corrected issue where solid laminate elements and/or properties may be translated with incorrect ply orientations.
- Corrected issue when reading results for laminate elements that only have top and bottom plies, where ply results for the final ply would not be given the correct output vector ID.
- Corrected issue which caused Beam Properties which have their Shape specified by any NASTRAN cross-section to not be written properly to the ANSYS input file.
- Corrected issues with reading the SFE command from an ANSYS cdb file.
- Corrected an issue where T-sections were not correctly translated to FEMAP T-sections when importing ANSYS cdb files.

Loads and Boundary Conditions

- Corrected issue which could cause poor performance when attempting to create variable loads using an Arbitrary 3-D Data Surface.
- Corrected issue which caused the Auto Create Definition command on the context-sensitive menu for individual loads to create excess load definitions for nodal loads that were defined in cylindrical or spherical coordinate systems.
- Corrected issue which occurred when to editing a load definition which had nodal loads defined in a non-rectangular coordinate system on multiple nodes. Previously, it could erroneously give the message "OK to Edit NonUniform Load Definition. All loads will be updated to a single value..." error, even though the loads were a constant/uniform load.

Meshing

- Corrected issue with “Adjacent Surface Matching” during sizing for a Solid Mesh. Some curves that should have been sized with constant length-based spacing were sized parametrically instead potentially resulting in mismatched meshes.
- Corrected issue which caused issues when meshing surfaces/curves in models that were migrated to V12 from previous versions, where the curves had a non-uniform parameterization (like splines) and they were sized using length-based spacing. In that case, the curves meshed as if they were sized parametrically. As a temporary workaround you could resize in V12 and meshing was correct, but this eliminates the need to resize.
- Corrected issue when using the Mesh, Mesh Control, Size on Curve command on curve which had already been sized, which prevented the dialog box from properly indicating that “length based” sizing had previously been specified.
- Corrected issue which would create an additional rectangular Coordinate System when moving nodes referencing that Coordinate System; this is required for cylindrical and spherical coordinate systems and will still occur.
- Corrected issue that occurred in V12.0 when copying a mesh. Previously, when copying/rotating/reflecting elements using the “Use Property” option the resulting elements would take on the linear/parabolic nature of the property even though they might have been topologically the opposite. This could cause elements to be invisible in “Performance Graphics” and could cause them to be handled improperly in other functions as well. Now they maintain their linear/parabolic nature and simply use the property, changing type if necessary.

Materials

- Corrected issues which caused the values of thermal attributes for stainless steel contained in the three mat_eng-SI libraries to be incorrect.

Output and Post-Processing

- Corrected issue which was caused by Nodal results being converted to Elemental results when plotting results in a Criteria plot, which caused the “Max Value” to always be used, even when the “Min Value” Data Conversion method was chosen.
- Corrected issue where results were not successfully Transformed into coordinate system 0, 1, or 2.

Groups and Layers

- Corrected issue when renumbering Groups. Previously if Group, Operations, Automatic Add was set to Select (rather than Active) the group selected in the drop-down was not renumbered and could be set to a different or non-existent group.

Renumber

- Corrected issue where renumbered functions, groups, and output sets would not be updated in various types of chart data series.

API

- Corrected issue where all cylindrical surfaces were always being evaluated as conical surfaces.
- Corrected issue with the feMeshClosestLink and feMeshCoincidentLink API methods that prevented any needed orientation vector from being properly assigned to the new elements.
- Corrected issue with feMeshSizeCurveMatchNodes that could prevent certain curves from receiving a mesh seed.
- Corrected issue where environment variables were not recognized in custom user commands (API).
- Corrected issue which caused the Frequency Data Type to not be recognized by Delete commands of the main application object.

FEMAP v12.0 New Features and Corrections

Updates and Enhancements

Views

- Added View, Autoscale, Fit command, which is similar to the View, Autoscale command. It too bases its calculations on the overall model dimensions. However, this command only considers the current orientation of the model. It projects the overall dimensions into the current view, then adjusts the magnification factor to attempt to fill the screen with the visible entities. This will always result in a larger image than View, Autoscale. If the model geometry is non-rectangular, or has cutouts, this option still might not fill the view. Also, this command will automatically adjust the centering of the model.
- Added “Mesh Point” option in “Labels, Entities and Color” Category of View, Options command, which controls visibility, label mode, and color mode for mesh points.
- Added “Element - Cohesive” option in “Labels, Entities and Color” Category of View, Options command, which controls visibility of property thickness, label mode, and color mode for mesh points.
- Added “Element - Shell” option in “Labels, Entities and Color” Category of View, Options command, which can be used to suppress thick edges of shell elements to improve performance and aid transparency.
- Added “2..DOF - Shrink Always” option to Release Labels section for “Element - Offsets/ Releases” option in “Labels, Entities and Color” Category of View, Options command, which can be used to always display the beam release degrees of freedom away from the End A and End B nodes.
- Added “Local Components” options to Color/Component section for “Load - Force and Bearing”, “Load - Moment and Torque”, “Load - Accleration”, “Load - Velocity”, and “Load - Enforced Displacement” options in “Labels, Entities and Color” Category of View, Options command, which can be used to display these load types as components in the coordinate system in which they are defined instead of as “Global Components”.
- Added “Feature Line” option in “Tools and View Style” Category of View, Options command, which controls if “Feature Lines” should be displayed on the mesh.
- Added Advanced Depth Control button to “Graphics Options” option in “Tools and View Style” Category of View, Options command.
- Added Discrete Values capability to the View, Advanced Post, Contour Model Data command.

Analysis Manager

- Updated Analysis Set dialog box, only when running in “FEMAP with NX Nastran” mode, to check for existence of a Linked Solver when switching Analysis Program from non-NX Nastran to “36..NX Nastran”. If dialog box is already set to “Linked Solver”, and a linked solver for NX Nastran is specified, nothing will change. Otherwise, it will change to Integrated Solver. In all other cases nothing is changed.

Connection Properties, Regions, and Connectors

- Added NX Multi-step Structural tab to Define Connection Property dialog box for NX Nastran SOL 401.
- Added NX Multi-step Kinematic tab to Define Connection Property dialog box for NX Nastran SOL 402.
- Added new ANSYS tab to Define Connection Property dialog box.
- Added an icon button to various “Define Region” dialog boxes to limit elements used for face selection. Previously, the only option was “Add Multiple” which automatically selected all visible elements and while it was possible to limit the elements for face selection from the face selection dialog box, this streamlines the process.
- Added ABAQUS section to the Connection Region Options dialog box, which is accessed via the Region Options button in the Connection Region dialog box. This option is used to specify the type of Analytical Rigid Surface, CYLINDRICAL, SEGMENTS, or REVOLUTION, to write to the ABAQUS input file.
- Added Connect, Connector Set command to create sets of Connectors (Contact or Glued).

Geometry

- Added Modify, Reflect menu which can be used to reflect Points, Curves, Surfaces, or Solids without creating a copy. When reflecting a geometric entity, all geometric entities needed by the entity will also be reflected.
- Added Geometry, Curve, Centerline command, which attempts to create a curve which represents the centerline of a geometric solid.
- Added Modify, Update Other, Align Surface Parameters command.
- Enhanced several commands which perform Boolean and other operations on solid geometry to work properly with geometric entities that contain boundary surfaces and/or combined curves.
- Enhanced Geometry, Surfaces, Ruled Surface command to Geometry, Surfaces, Ruled / Between Curves, which adds options to control surface tangents on one or both sides of the newly created surface.
- Enhanced commands for geometric entities on the Modify, Move By menu, the Modify, Rotate By menu, and the Modify, Align menu by adding an option to also move/rotate/align mesh associated with selected geometric entities. In addition, added Always Create Parent CSys option, which can be used to force creation of a “parent” coordinate system to be used as the reference coordinate system for any coordinate systems which need to be created to maintain the proper definition of finite element entities, such as boundary conditions or regions.
- Enhanced the commands on the Geometry, Copy menu, the commands Geometry, Rotate menu, and the commands on the Geometry, Reflect menu by adding an option to automatically select associated mesh, along with additional options.

GUI - Toolbars and Icons

View and View - Simple Toolbars

- Updated Mesh Size item on the View Style icon menu on the View and View - Simple toolbars to be Mesh Size/Locations, as it now toggles visibility of both mesh size on curves and mesh points.

Select Toolbar

- Added By Size command to Selector Actions icon menu on Select toolbar. Only available when Curve, Surface, or Solid is the active entity type in the Selector Entity icon menu. Curves can be selecting by Length, Surfaces by Area, or Solids by Volume.
- Added Connector Set command to context-sensitive menu when Connector is the active entity type in the Selector Entity icon menu of the Select toolbar, which allows creation of a Connector Set containing any number of Connectors.

GUI - Dockable Panes

Mesh Point Editor - New for 12.0!

- Added the Tools, Mesh Point Editor command, which displays the Mesh Point Editor dockable pane. The Mesh Point Editor provides you with a tool to interactively create, edit, and/or manage mesh points ("hard points") using an intuitive table control. Much like the Data Table, each Mesh Point appears as a single row separated into a number of columns when it enters the Mesh Point Editor. Also, once in the Mesh Point Editor, information about the mesh Points can be sorted, filtered, and evaluated to help you understand each Mesh Point to a greater degree. Unlike the Data Table, the Mesh Point Editor allows you to modify certain aspects of each Mesh Point in the model, including individual field modification in certain columns.

Model Info Tree

- Added Connector Set command to context-sensitive menu for Connectors, which allows creation of a Connector Set containing any number of Connectors.
- Added Text entities to the Selection List when they are the active entity in the Select toolbar. Also, implemented context-sensitive menu for text items in the Selection List.

Meshing Toolbox - General

- Added ability to use the "Esc" key to deactivate the Select icon for all tools which feature entity selection.
- Improved performance when automatically updating rigid elements as geometry is being remeshed via the Meshing Toolbox. For example, as test model with approximately 1.4 Million Nodes and 1.1 Million Elements, including 800 Rigid elements, demonstrates an 8X performance improvement for each individual mesh size change.

Meshing Toolbox - Geometry Editing Tool

- Added Slice to the Operation section of the Geometry Editing tool, which is used to divide surfaces, including boundary surfaces, using a slice plane defined after the surfaces have been selected.
- Added Midpoint and Curve/Location options to the Break Method section when Operation is set to Curve Break in the Geometry Editing tool. Midpoint breaks the selected curve(s) at the midpoint of each curve, while Curve/Location breaks the selected curve(s) using the closest distance between the specified location and each curve.
- Added Include Internal Loops and Split Solid options when Operation is set to Pad in the Geometry Editing tool. The Split Solid option attempts to subdivide solid geometry using solids generated from the "pad" or "pad with washer" pattern.

- Added Include Non-Circular Loops, Include Non-Circular Loops, and Split Solid options when Operation is set to Washer in the Geometry Editing tool. The Split Solid option attempts to subdivide solid geometry using solids generated from the “washer” pattern.
- Added Along Vector option to Project/Move To section when Operation is set to Project/Move Point in the Geometry Editing tool. This option moves/projects the selected point(s) along a vector specified after the points have been selected. This can be useful when trying to modify the overall shape of solids and surfaces.

PostProcessing Toolbox - Contour Tool

- Added Dynamic Control button to the Criteria section, which accesses the View, Advanced Post, Dynamic Criteria command. See Output and Post-Processing for more information.

Charting Pane

- Added “2..Vector Combination vs. Set” option to the Type drop-down in the Chart Data Series dialog box.

Data Surface Editor

- Added Criteria option to Output Map Data Conversion Options section of Define Options for Variation dialog box for use with the Output Map Data Surface.
- Added Preview icon button to Define Variation Between Coordinates Data Surface dialog box, which graphically displays lines between 2, 4 or 8 locations. This can be helpful when trying to determine if any “twisting” has occurred during selection of coordinates, especially when using a 4-point or 8-point data surface. In addition, added checking which alerts the user if FEMAP believes the data surface is “twisted”.

Data Table

- Added Transform button to Results to Add to Data Table dialog box, which is accessed via the Add Output Columns icon. Clicking this button displays the standard Results Transformation dialog box, which is used to select transformation options for Nodal Vector Output being added when nodes are already in the Data Table or Element Output on Line Elements, Shell Elements, and/or Solid Elements already in the Data Table.
- Added Filter Selected Entities button to Data Table Filter dialog box, which is accessed via the Filter Rows icon. Clicking this button displays the standard entity selection dialog box for the type of entity which is currently in the Data Table and the rows representing the selected entities are removed from the Data Table.

Interfaces - FEMAP Neutral

- Updated Neutral Read and Write for v12.0 changes
- Updated File, Import, Femap Neutral command to better handle cases where geometric entities have inconsistent geometry scale factors.

Interfaces - Nastran

- Added support for Optimization Variables for Materials (DVMREL1 entries) and Elements (DVEREL1 and associated GROUP entries), along with adding support for many more for Properties (DVPREL1 entities) for Design Optimization (SOL 200). See Optimization section.
- Added support for more types of Optimization Limits (DRESP1, DCONSTR, and DCONADD entries) for Design Optimization (SOL 200). See Optimization section.
- Added check when reading Coordinate Systems, Nodes and Elements to issue error and skip the entities if the ID is greater than 99,999,999.
- Updated process when importing or attaching to .op2 files created by response spectrum analysis to always skip principal stress and total vector calculations.
- Updated process when importing or attaching to .op2 files to first sort output sets by subcase, then by time step.

Interfaces - NX Nastran

- Added support for SOL 401 (Multi-Step Structural Solution) that supports a combination of static (linear or nonlinear), modal (real eigenvalue), and bolt pre-load subcases. Specifically, this includes adding the Multi-Step Control Options dialog box, along with the Solution and Convergence Options, Contact/Bolt Preload Control Options, and Creep Options dialog boxes to the Analysis Set Manager when Analysis Type is set to “27..Multi-Step Structural” to write the NLCNTL entry for the Master Case and/or subcases. In addition, added Analysis Type drop-down to the Master Requests and Conditions dialog box, which is used to select the type of analysis for the Master Case, and writes the appropriate ANALYSIS = entry to Case Control.
- Added support for SOL 402 (Multi-Step Nonlinear Kinematic Solution) that supports a combination of subcase types (static linear, static nonlinear, nonlinear dynamic, bolt pre-load, modal, Fourier, buckling) and large rotation kinematics. Specifically, this includes adding the Multi-Step Control Options dialog box, along with the Solution and Convergence Options dialog box to the Analysis Set Manager when Analysis Type is set to “28..Multi-Step Nonlinear Kinematic” to write the NLCNTL2 entry for the Master Case and/or subcases. In addition, this includes adding the Multi-Step Global Control Options dialog box to optionally write the NLCNTLG entry to the Master Case.
- Added Subcase (ID) Time Steps dialog box to the Analysis Set Manager when Analysis Type is set to “27..Multi-Step Structural” (SOL 401) or “28..Multi-Step Nonlinear Kinematic” (SOL 402) to write TSTEP1 to the Master Case (ID = 0) or subcase.
- Added Analysis Type drop-down to the Analysis Case dialog box in the Analysis Set Manager when Analysis Type set to “27..Multi-Step Structural” (SOL 401) or “28..Multi-Step Nonlinear Kinematic” (SOL 402). This is used to specify the type of analysis for a subcase, which writes the appropriate ANALYSIS = entries to the Case Control section. In addition, added the Step Control section to the Analysis Case dialog box, which is used to select Sequentially Dependent or Not Sequentially Dependent, and writes the SEQDEP entry to the appropriate subcase in Case Control.
- Added MATNL and corresponding drop-down to NASTRAN Bulk Data Options dialog box, which is only available in the Analysis Set Manager when Analysis Type is set to “27..Multi-Step Structural” (SOL 401) or “28..Multi-Step Nonlinear Kinematic” (SOL 402). Writes PARAM,MATNL with the selected option.

- Added ability to create Connector Set for support of Multi-Step Structural (SOL 401) and Multi-Step Nonlinear Kinematic (SOL 402). Connector Sets can be selected in the Contact Sets and/or Glue Sets sections of the Boundary Conditions dialog box in Analysis Set Manager when Analysis Type is set to “27..Multi-Step Structural” (SOL 401) or “28..Multi-Step Nonlinear Kinematic” (SOL 402) for the Master Case. For subcases, only the Contact Sets section is available. See Connections (Regions, Properties, and Connectors) section.
- Added tabs in Connection Property dialog box for support of Multi-Step Structural (SOL 401) and Multi-Step Nonlinear Kinematic (SOL 402). See Connections (Regions, Properties, and Connectors) section.
- Added read/write support for Cohesive Elements (CHEXCZ, CPENTCZ), along with the Cohesive Property (PSOLCZ) and Cohesive Material (MATCZ). See Elements, Materials, and Properties sections.
- Added support for Topology Optimization in Design Optimization (SOL 200), including ability to define Topology Optimization regions (DETVEL1 and associated GROUP entries) and specify Manufacturing Constraints (DMNCON entries). See Optimization section.
- Added ability to Read / Write output requests for Elastic Strain (ELSTRN) and Thermal Strain (THSTRN), as well as renaming the Strain item in the NASTRAN Output Requests dialog box to Total Strain.
- Added support for reading ply-by-ply laminate results created by random response analysis, from the .op2 file. This includes ply-by-ply Stresses including von Mises Stress for PSDF (OESPSD1C), Cumulative Root Mean Square output (OESXNO1C), and Positive Crossing (OESCRM1C) output sets, along with ply-by-ply Strains for PSDF (OSTPSD1C) and Cumulative Root Mean Square (OSTCRM1C) output sets.
- Added support for Strength Ratio output in new format generated by NX Nastran 12 for certain conditions.
- Updated the Rigid Element Thermal Expansion option in Translator Options section of NASTRAN Bulk Data Options dialog box in the Analysis Set Manager to be Rigid Element Method with a corresponding drop-down. When Analysis Type is set to “27..Multi-Step Structural” (SOL 401) or “28..Multi-Step Nonlinear Kinematic” (SOL 402), select from “0..AUTO”, “2..LINEAR”, or “3..STIFF”. When Analysis Type is set to other supported analysis types, select from “1..LAGRAN” or “2..LINEAR”. Writes the RIGID = entry to the Master Case.

Interfaces - MSC Nastran

- Added Support for output from Fastener Elements (CFAST).
- Added support for SRCOMPS parameter.
- Added support for MSC Nastran Strength Ratio output.
- Added Database read / write support for MSC BCTABLE items: HCT, HCV, HNC, BNC, EMISS. These can only be accessed via the API.
- Updated the Rigid Element Thermal Expansion option in Translator Options section of NASTRAN Bulk Data Options dialog box in the Analysis Set Manager to be Rigid Element Method with a corresponding drop-down. When Analysis Type is set to a supported analysis type, select from “1..LAGRAN”, “2..LINEAR”, or “4..LGELIM”. Writes the RIGID = entry to the Master Case.

Interfaces - ANSYS

Added an entirely new ANSYS interface for version 12, which includes support for modern element types, along with other modern inputs and options. The new translator, and all user interface components throughout FEMAP which were updated to support the new translator, are available when the Enable Legacy Ansys Interface option is DISABLED in the General Solver Options section on the Interfaces tab of the File, Preferences command.

New features include:

- Analysis Set Manager - New dialog boxes to specify options for different types of analysis in any subcase. These include the Ansys Analysis Case, ANSYS Load Step, ANSYS Time Step Options, ANSYS Modal Analysis Options, ANSYS Buckling Analysis Options, ANSYS Transient Dynamics Options, ANSYS Harmonic Analysis Options, ANSYS Nonlinear Options, ANSYS Output Requests. In addition, the type of analysis being performed by each "case" can be selected using the Analysis Type drop-down in the Ansys Analysis Case dialog box.
- Connection Property - ANSYS tab has been updated to offer more options than were previously available. See Connections (Regions, Properties, and Connectors).
- Elements and Properties - 40 of the 44 element/property types in FEMAP are mapped to modern ANSYS element types.
- Element Formulations - An Ansys tab has been added to the Element Formulation dialog box, which is used to specify element type (Ansys Option section) and up to six KEYOPT values for each element type (Ansys Keyopt section).
- Materials - 30 Material types are mapped to modern ANSYS material models (constant or tabular values).
- Loads and Boundary Conditions - High-level of support for different boundary and loading conditions on nodes, elemental, and geometric entities (constant of tabular values)
- API - All options in the new ANSYS interface are available to be set using the Application Programing Interface (API).

Interfaces - ABAQUS

- Added support for analytical rigid surfaces of type CYLINDER, SEGMENTS, or REVOLUTION. See Connections (Regions, Properties, and Connectors).
- Added Support For ABAQUS 2018 ODB results files.
- Added reading Pore Pressure "POR" results from the ODB results file.
- Added export of CBUSH elements as ABAQUS CONN3D2 elements, which reference *CONNECTOR BEHAVIOR and *CONNECTOR ELASTICITY entries. In addition, added Write CBUSH as Matrix option to ABAQUS Model Options dialog box, which when enabled, writes out CBUSH elements *MATRIX INPUT and *MATRIX ASSEMBLE entries.

Interfaces - LS-DYNA

- Added support for *CONTROL_TIMESTEP entry for both Explicit Transient Dynamics and Implicit Transient Dynamics. For both types of analysis, this is specified by clicking the Advanced button in the Solver Options section of the LS-DYNA Analysis Control dialog box in the Analysis Set Manager, which opens the LS-Dyna Solver Options - *CONTROL dialog box.

Interfaces - Geometry

- Added support for Parasolid 31.0, Solid Edge 2019, NX 12.0, Pro/Engineer CREO 5, CATIA V5-6R2018, ACIS 2018 1.0, SolidWorks 2018, and JT 9.0
- Updated the File, Export, Geometry command by adding the Topology Optimization option, which requires output from a NX Nastran Topology Optimization analysis to export a STL file of the optimized shape. To dynamically specify elements to use for the faceted representation, click the Options button, then use the Topology Optimization STL File Output dialog box, which is very similar to the one used by the View, Advanced Post, Dynamic Criteria command. See Output and Post-Processing for more information.

Element - Solid Cohesive - New for 12.0!

- Added Solid Cohesive Element Type, which are currently only used by the NX Nastran Multi-Step Nonlinear Solution Sequences (SOL 401 and SOL 402). Solid Cohesive elements can be created manually or by using the Mesh, Editing, Cohesive Meshing command. In addition, while they are "solid" elements, the nodes on the top and bottom faces may be coincident (i.e.,"flat").

Element - Formulations

- Added Ansys tab to the Element Formulation dialog box, which is used to specify element type (Ansys Option section) and up to six KEYOPT values for each element type (Ansys Keyopt section) for the new ANSYS translator. If the Enable Legacy Ansys Interface option is enabled in the General Solver Options section of the Interfaces tab of the File, Preferences command, the Legacy Ansys Options section will be available instead.

Materials

- Added NX Nastran Cohesive (MATCZ Sol 401, 402) to Other Types. Used in conjunction with Cohesive Elements to model adhesion between different types of elements when using NX Nastran Multi-Step Nonlinear Solution Sequences (SOL 401 and SOL 402), as well as consider damage during progressive ply failure in models with laminate elements.

Properties

- Added the Solid Cohesive Property Type. Used for modeling cohesion in NX Nastran Multi-Step Nonlinear Solutions (SOL 401 and SOL 402), along with damage during progressive ply failure in models with laminate elements.
- Added Edit Layup icon button to Define Property - LAMINATE PLATE Element Type and Define Property - LAMINATE SOLID Element Type dialog boxes, which can be used to edit the layup currently selected in the Layup drop-down.

Layups

- Enhanced "Compute" capability to always calculate equivalent properties using multiple approaches (i.e., both including and excluding "Membrane/Bending Coupling") and added "FOR INFORMATION PURPOSES ONLY" section, which explicitly states the calculated values "Membrane/Bending Coupling Excluded" for those values.

Loads and Boundary Conditions

- Updated Model, Load, Map Output From Model command by adding Criteria option to Data Conversion section, which is designed to be used between models which have an identical or very similar mesh. When using the Criteria option, Conversion Type and Unmapped Values are not available.
- Updated Model, Load, From Freebody command, when using the Multi-Model option, by adding the Re-center Reference Node option to the Toggle Rigid Element Nodes dialog box. This dialog box may appear near the end of the command and is used to add or remove nodes from any of the rigid or interpolation elements automatically created by previous steps in the command.

Optimization

- Added Model, Optimization, Variables and Topology Regions command which opens the Optimization Variable and Topology Region Manager, which is used to create, edit, copy, renumber, delete selected, or delete all Optimization Variables (relationships) and/or Topology Regions.
- Added Model, Optimization, Limits command which opens the Optimization Limit Manager, which is used to create, edit, copy, renumber, delete selected, or delete all Optimization Limits (responses).
- Added Model, Optimization, Manufacturing Constraints command which opens the Manufacturing Constraint Manager, which is used to create, edit, copy, renumber, delete selected, or delete all Manufacturing Constraints specified for Topology Optimization.

Meshing

- Added Modify, Reflect menu which can be used to reflect Nodes or Elements without creating a copy.
- Added the Mesh, Editing, Cohesive Meshing command to create a layer of Cohesive Elements.
- Updated the Mesh, Edge Member command by dividing it into two commands which are now on the Mesh, Edge/Skin Elements submenu. Mesh, Edge/Skin Elements, Line Elements on Edges creates line elements on selected edges. Mesh, Edge/Skin Elements, Planar Elements on Faces creates Shell Elements on the selected faces of planar or Solid Elements.
- Updated various dialog boxes used for meshing to change the Node Param, Elem Param, Formulation, and Material Orientation buttons to icon buttons and added “tooltips” for those and for the Property icon button.
- Enhanced Mesh Points to include assigning Mesh Points on Points, Curves, and Solids, as well as Surfaces. Mesh points also now have a unique symbol that can be controlled with View Options and may be created or edited using the Mesh Point Editor dockable pane.
- Enhanced various meshing commands to automatically delete any extraneous planar element on surfaces after meshing surfaces which have a Matched - Link to Surface mesh approach assigned.
- Enhanced commands for Nodes and Elements on the Modify, Move By menu, the Modify, Rotate By menu, and the Modify, Align menu by adding an option to also move/rotate/align geometry associated with selected Node and Elements. In addition, added Always Create Parent CSys option, which can be used to force creation of a “parent” coordinate system to be used as the reference

coordinate system for any coordinate systems which need to be created to maintain the proper definition of entities, such as boundary conditions or regions.

- Enhanced commands for Coordinate Systems on the Modify, Move By menu, the Modify, Rotate By menu, and the Modify, Align menu by adding an option to also move/rotate/align geometry and mesh associated with selected Coordinate Systems. In addition, added Always Create Parent CSys option, which can be used to force creation of a "parent" coordinate system to be used as the reference coordinate system for any coordinate systems which need to be created to maintain the proper definition of entities.
- Enhanced the commands on the Mesh, Copy menu, the commands Mesh, Rotate menu, and the commands on the Mesh, Reflect menu by adding an option to automatically select associated geometry, along with additional options.

Output and Post-Processing

- Added View, Advanced Post, Dynamic Criteria command. This functionality can also be accessed by using the Dynamic Control button when Style is set to Criteria in the Contour tool of the PostProcessing Toolbox.
- Added ability to transform line element output into a selected Coordinate System. A typical "use case" for this type of transformation is to view results from Nastran Spring/Damper elements (CBUSH) in a single coordinate system, even though the elements may be oriented in different coordinate systems.
- Added "7..Visible Min/Max" option to the Level Modes section for the Contour/Criteria Levels option in View, Options. This option considers ALL visibility options currently specified for the current view to automatically determine the maximum and minimum values from the Contour Output Vector currently displayed, then interpolates between them to determine all intermediate values. These visibility options include visibility check boxes for Property, Material, Element Type, and/or Element Shape; visibility of individual elements; visible Groups; visible Layers; and/or elements being drawn/erased by the Draw/Erase toolbar.
- Added warning when attempting to transform data in a Output Set which was created using either the RSS Combination or Envelope methods of the Model, Output, Process command. The warning will also be issued for an Output Set created by the RSS Combination or Envelope methods available for a Results Set Processing Table in the Function/Table Editor or Results Set Processing Data Surface in the Data Surface Editor, when the output is stored in the database.
- Updated Post Titles to show End A and End B output vectors when displaying a contour on line element.

Groups and Layers

- Added the Group, Operations, Generate From Criteria command, which is only available when a Criteria plot is being displayed in the graphics window (i.e., Contour Style is set to Criteria in the View, Select command or Criteria is selected from the Style drop-down in the Contour Tool of the PostProcessing Toolbox). Creates a group containing all elements which fulfill the criteria options currently set for the active view, which may be different than all visible elements, especially if there are multiple element types in the model.

- Added the Group, Mesh Point submenu, which allows you to define, edit, and delete the rules that will be used to select mesh points. You can select mesh points into your group based on their ID, Color, Layer, associated with geometric entities (on Point, on Curve, on Surface, in Solid), or any combination of these methods.
- Improved performance of the "Add Related Entities" capability of Groups, which can be done using various methods throughout FEMAP. Previously in models that had large numbers of geometry-based contact, along with large numbers of solid elements, this command could take a long time to complete. This also improved performance of other commands and tools which use this capability, such as the Entity Locator found in the Meshing Toolbox.

Listing

- Added List, Geometry, Mesh Point command to list details about mesh points to the Messaged pane and any other destination(s) specified by the List, Destination command.
- Updated List, Output, Results to Data Table command by adding Show Summary Table option to the Send Results to Data Table dialog box, which adds a summary table to the bottom of the Data Table and contains Max and Min Values, their corresponding IDs, Sum of the Values, and the Average Value for each column.
- Updated List, Output, Results to Data Table command by adding Transform section to the Send Results to Data Table dialog box. This section can be used to send transformed output data to the Data Table using the options specified for the active view (Active View option) or specified via the Transform… button (Custom... option).
- Updated List, Output, Results Ranking to Data Table command by adding Transform button to the Rank Output to Data Table dialog box. This section can be used to send ranked output data to the Data Table, which has been transformed using the options specified via the Transform… button.
- Updated List, Output, Contoured Results to Data Table command by adding a header to the to of that Data Table, which describes the selected output sets/output vectors in the Data Table, including transformations.

Tools

Report Generator

- Added the Tools, Report Generator command. This tool queries the user's machine to make sure Microsoft Word is installed, and if so, will automatically link to Microsoft Word and create a report using options specified by the user. Options to specify include which entities to include and which formats to use when creating pictures in the report.

Model Merge

- Added Copy in Current Model option to the Merge/Extract section. This option allows entities from a model to be merged into the same model. Typically, this would be used in conjunction with one of the Orientation/Transform options to create duplicate entities in a different location.
- Added Keep Loads and Constraints in Original Sets option to Options section. When enabled, which is the default when using the Copy in Current Model option, load sets and constraint sets will not be renumbered. Instead, any load set or constraint set which exists in both the From Model and

To Model will be combined into a single load set or single constraint set in the To Model, using the original ID.

- Added Orientation/Transform section, which replaces the Transform Merged Model option and corresponding drop-downs to select coordinate systems.
- Updated File, Merge command to skip rebuilding solid faceting of previously existing solids.
- Updated File, Merge to better handle cases where geometry entities have inconsistent geometry scale factors.

User Interface - General

- Added “X” icon on the tab for each view which can be used to quickly close a view or an entire model, but only when a single view is currently visible for the model.
- Added Previous On icon button to select Output Sets and/or Output Vectors which where last selected via the Select Output Set(s) or Select Results dialog boxes found in various commands throughout FEMAP.
- Updated View Axes to display an Orientation Cube when clicked, which can be used to quickly rotate models to “standard views” and other orientations which are the average of two or three different “standard views”.
- Updated Contour Legend to be an interactive screen entity.
- Updated View Legend to be an interactive screen entity.
- Updated Post Titles to be an interactive screen entity.
- Added “By Size” option in the “Pick^” menu of the standard entity selection dialog box to select Curves by Length, Surfaces by Area, and Solids by Volume using the desired Size Specification option.
- Added “Add Connected Tangent Curves” option in the “Pick^” menu of standard entity selection dialog box and is only available when selecting curves. Using the Add Connected Tangent Curves command allows you to quickly add “connected tangent curves” to the selection list by first selecting any number of curves. This is a helpful picking tool when using the Geometry, Curve - From Surface, Offset Curves/Washer command.
- Added ability to use “ALT+S” and “ALT+E” in Standard Entity Selection dialog boxes to choose “Select All” and “Reset”, respectively.
- Updated “Property Data” to be “Element / Property Data” in the Contour Model Data dialog to reflect the fact that some of the quantities are specified as Element Data.

Preferences

Graphics

- Added Best Possible option to the Graphics Options section. This option, when enabled, attempts to use information from the computer running FEMAP (installed RAM, Graphics Card, version of Open GL, etc) to determine what should be the “best possible” settings for several items in the Graphics Options section. When enabled, the Performance Graphics, Max VBO MB, Min VBO B, and Memory Optimization options, along with the setting for the Vertex Arrays drop-down, cannot be specified by the user. When disabled, these items can specified individually

- Updated Mesh Size in the Include In Dynamic Rotation section to be Mesh Locations, as both Mesh Size on curves and Mesh Points are now not included in dynamic rotation unless this option is enabled.

Geometry/Model

- Added Surface Meshing in Memory to Meshing and Properties section. This preference determines whether additional memory will be allocated by the FEMAP boundary mesher. If this option is selected, FEMAP will allocate new memory to create the mesh. If it is not selected, FEMAP will utilize the memory allocated in the database to perform the mesh. By allocating new memory, the FEMAP mesher can run significantly faster than if it is limited to the database memory. Therefore, this option should almost always be turned on. The only reason to turn this option off is if the available memory on the current machine is low enough that allocation of new memory is extremely limited.

Interfaces

- Added Enable Legacy Ansys Interface option to the General Solver Options section. When this option enabled, the Analysis Set Manager and other dialog boxes will use the "legacy" version of the ANSYS analysis interface and translator (i.e., essentially what was in FEMAP versions prior to version 12).
- Added Enterprise License Server option to the Nastran Options section, which can be used to specify a license server if using an enterprise version of NX Nastran. Simply enter the "@" symbol followed by the name of the license server (i.e., @Nastran_Server_Name).

API

New and updated API Objects and Attributes

- Added CopyTool (feCopyTool) object to the API. Also, added Repetitions, CopyInSameLocation, AlwaysCreateParentCSys, UsePattern, and ReflectionTrapWidth attributes to the feCopyTool Object.
- Added MoveTool (feMoveTool) object to the API. Also, added AlwaysCreateParentCSys and ReflectionTrapWidth attributes to the feMoveTool Object.
- Added Discrete Value Set (feDiscreteValueSet) object to the API. Also, added IntegerType and RangeType attributes to the feDiscreteValueSet Object.
- Added Mesh Point (feMeshPt) object to the API. Also, added DefinitionID, PointID, layer, color, OnGeometryType, OnGeometryID, NodeOnGeometry, DistanceToGeometry, Locked, LocationOnGeometry, and vLocationOnGeometry to feMeshPtObject.
- Added Mesh Point Definition (feMeshPointDefinition) object to the API. Also, added title attributes to the feMeshPointDefinition Object.
- Added Optimization Manufacturing Constraints (feOptMC) object to the API. Also, added title, ManConType, Vec1, vVec1, Vec2, vVec2, Vec3, vVec3, Vec4, vVec4, nField1, dField1, and dField2 attributes to the feOptMC Object.
- Added Optimization Relationship (Variable) (feOptRel) object to the API. Also, added title, TypeField, NameField, RelationshipType, TopologyActive, Bounds, vBounds, RelationshipID, and BoundType attributes to the feOptRel Object.

- Added Optimization Response (Limits) (feOptResp) object to the API. Also, added title, rtype, ptype, atta, attb, ResponseCategory, VectorID, attbFieldType, attiDataType, and attaBlank attributes to the feOptRel Object.
- Added SubcaseAnalysisType attribute to feAnalysisCase Object.
- Added NasModeOn, NasModeEigrSet, NasModeMethod, NasModeSolutionType, NasModeEstRoots, NasModeDesiredRoots, NasModeNormOpt, NasModeNormNode, NasModeNormDOF, NasModeMassForm, NasModeXYOn, NasModeXYRefNode, NasModeFreqRange, vNasModeFreqRange, NasModesSkipEigr, NasModeImagFreqRange, vNasModeImagFreqRange, NasModesComplexConverge, NasModesComplexRegionWidth, NasModeDampOverall, NasModeXYRequest, and vNasModeXYRequest attributes for Nastran Modal Analysis subcases to feAnalysisCase Object.
- Added NasMsnlCntOn, NasMsnlCntSkipNLCNTL, vNasMsnlCntConv_flags, vNasMsnlCntConv_value, NasMsnlCntMaxbis, NasMsnlCntMaxdiv,NasMsnlCntMaxiter, NasMsnlCntMaxqn, NasMsnlCntEpsbolt, NasMsnlCntZerbolt, NasMsnlCntItrbolt, NasMsnlCntMisfblt, NasMsnlCntMsglvlb, NasMsnlCntLvar, NasMsnlCntMsglvl, NasMsnlCntSolver, NasMsnlCntThrmst, NasMsnlCntTvar, NasMsnlCntFollowk, NasMsnlCntKupdate, NasMsnlCntSpink, NasMsnlCntStfoptn, NasMsnlCntStressk, NasMsnlCntTstepk, NasMsnlCntCntmdiv, NasMsnlCntFsymtol, NasMsnlCntKsym, NasMsnlCntKsymtol, NasMsnlCntMsglvlc, NasMsnlCntUsolver, NasMsnlCntCrcerat, NasMsnlCntCrcinc, NasMsnlCntCreep, NasMsnlCntCricoff, NasMsnlCntCrinfac, NasMsnlCntCrmfmn, NasMsnlCntCrmfmx, NasMsnlCntCrteabs, NasMsnlCntdCrteco, NasMsnlCntCrterel, NasMsnlCntPlastic, NasMsnlCntAutotim, NasMsnlCntDtinit, NasMsnlCntDtmax, NasMsnlCntDtmin, NasMsnlCntDtsbcdt, NasMsnlCntEqmfmin, NasMsnlCntEqmfmx, NasMsnlCntTsccr, NasMsnlCntTsceq, NasMsnlCntTscumat, NasMsnlCntUmfmin, and NasMsnlCntUmfmx attributes for NX Nastran SOL 401 subcases to the feAnalysisCase Object.
- Added NasMsnlkCnt2on, NasMsnlkCnt2SkipNLCNTL2, NasMsnlkCnt2DISLIM, NasMsnlkCnt2ROTLIM, NasMsnlkCnt2DEFLIM, NasMsnlkCnt2LVAR, NasMsnlkCnt2TVAR, NasMsnlkCnt2DIPR, NasMsnlkCnt2NORM, NasMsnlkCnt2CREEP,NasMsnlkCnt2PLASTIC, NasMsnlkCnt2STAB, NasMsnlkCnt2MADI, NasMsnlkCnt2ERCD, NasMsnlkCnt2PRED, NasMsnlkCnt2IMPL, NasMsnlkCnt2BETA, NasMsnlkCnt2GAMA, NasMsnlkCnt2ALFA, NasMsnlkCnt2TETA, NasMsnlkCnt2PRCO, NasMsnlkCnt2HPRCO, NasMsnlkCnt2ILNS, NasMsnlkCnt2PRLN, NasMsnlkCnt2AMIN, NasMsnlkCnt2AMAX, NasMsnlkCnt2ITMA, NasMsnlkCnt2PRCR, NasMsnlkCnt2REFP, NasMsnlkCnt2PRCQ, NasMsnlkCnt2REFU, NasMsnlkCnt2PRCE, NasMsnlkCnt2REFE, NasMsnlkCnt2IT1K, NasMsnlkCnt2IT2K, NasMsnlkCnt2IT3K, NasMsnlkCnt2PLAS, NasMsnlkCnt2CIBL, NasMsnlkCnt2DTI0, NasMsnlkCnt2HMIN, NasMsnlkCnt2HMAX, NasMsnlkCnt2RUP, NasMsnlkCnt2RDOW, NasMsnlkCnt2RSUB, NasMsnlkCnt2RELC, NasMsnlkCnt2DCON, NasMsnlkCnt2PRCS, NasMsnlkCnt2IMPG, NasMsnlkCnt2IMPR, and NasMsnlkCnt2IMPV attributes for NX Nastran SOL 402 subcases to the feAnalysisCase Object.
- Added NasMsnlTSOn, NasMsnlTSSkipTSTEP, NasMsnlTSNumIncrements, NasMsnlTSEndTime, NasMsnlTSSkipFactor, and NasMsnlTSOutputFreq attributes for NX Nastran SOL 401 and SOL 402 subcases to the feAnalysisCase Object.

- Added AnsLdstepOn, AnsKbc, AnsPstres, AnsEqslv, AnsNewConstraint, AnsNewLoad,AnsNewContact,AnsTimestepOn, AnsTime, AnsAutots, AnsNsubs, AnsNsbstep, AnsNsbmx, AnsNsbmn, AnsDtime, AnsDtmin, AnsDtmax, AnsPlslimit, AnsPlslimitvalue, AnsCrplimit, AnsCrplimitvalue, AnsDpplimit, AnsDpplimitvalue, AnsDsplimit, AnsDsplimitvalue, AnsBisect, AnsBisectfactor, AnsPredictor,AnsMidtol, AnsTolerb, AnsResfq, AnsOutputOn, AnsOutresAll,AnsOutprAll, AnsOutresAllFreq,AnsOutprAllFreq, AnsOutresAllNth, AnsOutprAllNth, AnsOutresBasic, AnsOutprBasic, AnsOutresBasicFreq, AnsOutprBasicFreq, AnsOutresBasicNth, AnsOutprBasicNth, AnsOutresRsol, AnsOutprRsol, AnsOutresRsolFreq, AnsOutprRsolFreq, AnsOutresRsolNth, AnsOutprRsolNth, AnsOutresNsol, AnsOutprNsol, AnsOutresNsolFreq, AnsOutprNsolFreq,AnsOutresNsolNth,AnsOutprNsolNth, AnsOutresVel,AnsOutprVel, AnsOutresVelFreq,AnsOutprVelFreq, AnsOutresVelNth, AnsOutprVelNth,AnsOutresAcc, AnsOutprAcc, AnsOutresAccFreq, AnsOutprAccFreq, AnsOutresAccNth, AnsOutprAccNth, AnsOutresEsol, AnsOutprEsol, AnsOutresEsolFreq, AnsOutprEsolFreq, AnsOutresEsolNth, AnsOutprEsolNth, AnsModalOptOn, AnsModalMethod, AnsModalNmode, AnsModalFreqb, AnsModalFreqe, AnsModalNrmkey,AnsModalStrmck,AnsModalRangefact, AnsModalBlocksize, AnsModalRobustlev, AnsModalCompute, AnsModalReusekey, AnsModalSymmeth, AnsBuckleOptOn, AnsBuckleMethod,AnsBuckleNmode, AnsBuckleShift, AnsBuckleLdmulte,AnsBuckleRangekey,AnsBuckleStrmck, AnsHarmonicOptOn, AnsHarmonicMethod, AnsHarmonicMinmode, AnsHarmonicMaxmode, AnsHarmonicFreqb, AnsHarmonicFreqe, AnsHarmonicLogopt,AnsHarmonicReimky,AnsHarmonicClust, AnsTransOptOn, AnsTransMethod, AnsTransMinmode, AnsTransMaxmode, AnsTransTintopt, AnsTransDmpsfreq, AnsTransLumpm, AnsTransAlphad, AnsTransBetad, AnsTransDmpstr, AnsTransDmprat, AnsTransGamma, AnsTransGammavalue, AnsTransAlpha, AnsTransDelta, AnsTransAlphaf, AnsTransAlpham, AnsNlOptOn, AnsNropt, AnsNeqit, AnsCnvtolU, AnsUtoler, AnsCnvtolRot, AnsRottoler, AnsCnvtolF, AnsFtoler, AnsCnvtolM, AnsMtoler, AnsCnvtolDvol, AnsDvoltoler, AnsCnvtolHdsp, AnsHdsptoler, AnsNlgeom, AnsArclen, AnsLnsrch, and AnsPred attributes for ANSYS cases to the feAnalysisCase Object.
- Added AbaModCbushAsMatrix attribute for ABAQUS Master Case to the feAnalysisMgr Object.
- Added NasMsnlCntOn, NasMsnlCntSkipNLCNTL, vNasMsnlCntConv_flags, vNasMsnlCntConv_value, NasMsnlCntMaxbis, NasMsnlCntMaxdiv,NasMsnlCntMaxiter, NasMsnlCntMaxqn, NasMsnlCntEpsbolt, NasMsnlCntZerbolt, NasMsnlCntItrbolt, NasMsnlCntMisfblt, NasMsnlCntMsglvlb, NasMsnlCntLvar, NasMsnlCntMsglvl, NasMsnlCntSolver, NasMsnlCntThrmst, NasMsnlCntTvar, NasMsnlCntFollowk, NasMsnlCntKupdate, NasMsnlCntSpink, NasMsnlCntStfoptn, NasMsnlCntStressk, NasMsnlCntTstepk, NasMsnlCntCntmdiv, NasMsnlCntFsymtol, NasMsnlCntKsym, NasMsnlCntKsymtol, NasMsnlCntMsglvlc, NasMsnlCntUsolver, NasMsnlCntCrcerat, NasMsnlCntCrcinc, NasMsnlCntCreep, NasMsnlCntCricoff, NasMsnlCntCrinfac, NasMsnlCntCrmfmn, NasMsnlCntCrmfmx, NasMsnlCntCrteabs, NasMsnlCntdCrteco, NasMsnlCntCrterel, NasMsnlCntPlastic, NasMsnlCntAutotim, NasMsnlCntDtinit, NasMsnlCntDtmax, NasMsnlCntDtmin, NasMsnlCntDtsbcdt, NasMsnlCntEqmfmin, NasMsnlCntEqmfmx, NasMsnlCntTsccr, NasMsnlCntTsceq, NasMsnlCntTscumat, NasMsnlCntUmfmin, and NasMsnlCntUmfmx attributes for NX Nastran SOL 401 Master Case to the feAnalysisMgr Object.

- Added NasMsNLKGlobalOn, NasMsNLKGlobalRESO, NasMsNLKGlobalSTRMEAS, NasMsNLKGlobalIREF, NasMsNLKGlobalINLY NasMsnlkCnt2on, NasMsnlkCnt2SkipNLCNTL2, NasMsnlkCnt2DISLIM, NasMsnlkCnt2ROTLIM, NasMsnlkCnt2DEFLIM, NasMsnlkCnt2LVAR, NasMsnlkCnt2TVAR, NasMsnlkCnt2DIPR, NasMsnlkCnt2NORM, NasMsnlkCnt2CREEP,NasMsnlkCnt2PLASTIC, NasMsnlkCnt2STAB, NasMsnlkCnt2MADI, NasMsnlkCnt2ERCD, NasMsnlkCnt2PRED, NasMsnlkCnt2IMPL, NasMsnlkCnt2BETA, NasMsnlkCnt2GAMA, NasMsnlkCnt2ALFA, NasMsnlkCnt2TETA, NasMsnlkCnt2PRCO, NasMsnlkCnt2HPRCO, NasMsnlkCnt2ILNS, NasMsnlkCnt2PRLN, NasMsnlkCnt2AMIN, NasMsnlkCnt2AMAX, NasMsnlkCnt2ITMA, NasMsnlkCnt2PRCR, NasMsnlkCnt2REFP, NasMsnlkCnt2PRCQ, NasMsnlkCnt2REFU, NasMsnlkCnt2PRCE, NasMsnlkCnt2REFE, NasMsnlkCnt2IT1K, NasMsnlkCnt2IT2K, NasMsnlkCnt2IT3K, NasMsnlkCnt2PLAS, NasMsnlkCnt2CIBL, NasMsnlkCnt2DTI0, NasMsnlkCnt2HMIN, NasMsnlkCnt2HMAX, NasMsnlkCnt2RUP, NasMsnlkCnt2RDOW, NasMsnlkCnt2RSUB, NasMsnlkCnt2RELC, NasMsnlkCnt2DCON, NasMsnlkCnt2PRCS, NasMsnlkCnt2IMPG, NasMsnlkCnt2IMPR, and NasMsnlkCnt2IMPV attributes for NX Nastran SOL 402 Master Case to the feAnalysisMgr Object.
- Added NasMsnlTSOn, NasMsnlTSSkipTSTEP, NasMsnlTSNumIncrements, NasMsnlTSEndTime, NasMsnlTSSkipFactor, and NasMsnlTSOutputFreq attributes for NX Nastran SOL 401 and SOL 402 Master Case to the feAnalysisMgr Object.
- Added NasOptimCycles,NasOptimOn, NasOptimIsTopology, NasBulkMatnl, NasOptimMinDesobj, NasOptimDresp, NasOptimGoal, NasOptimIntervalVal, NasOptimEcho, NasOptimInterval, NasOptimMCDelayVal, NasOptimMCDelay, and NasOptimIntervalVal attributes for NX Nastran and MSC Nastran SOL 200 Master Case to the feAnalysisMgr Object.
- Added AnsLdstepOn, AnsKbc, AnsPstres, AnsEqslv, AnsNewConstraint, AnsNewLoad,AnsNewContact,AnsTimestepOn, AnsTime, AnsAutots, AnsNsubs, AnsNsbstep, AnsNsbmx, AnsNsbmn, AnsDtime, AnsDtmin, AnsDtmax, AnsPlslimit, AnsPlslimitvalue, AnsCrplimit, AnsCrplimitvalue, AnsDpplimit, AnsDpplimitvalue, AnsDsplimit, AnsDsplimitvalue, AnsBisect, AnsBisectfactor, AnsPredictor,AnsMidtol, AnsTolerb, AnsResfq, AnsOutputOn, AnsOutresAll,AnsOutprAll, AnsOutresAllFreq,AnsOutprAllFreq, AnsOutresAllNth, AnsOutprAllNth, AnsOutresBasic, AnsOutprBasic, AnsOutresBasicFreq, AnsOutprBasicFreq, AnsOutresBasicNth, AnsOutprBasicNth, AnsOutresRsol, AnsOutprRsol, AnsOutresRsolFreq, AnsOutprRsolFreq, AnsOutresRsolNth, AnsOutprRsolNth, AnsOutresNsol, AnsOutprNsol, AnsOutresNsolFreq, AnsOutprNsolFreq,AnsOutresNsolNth,AnsOutprNsolNth, AnsOutresVel,AnsOutprVel, AnsOutresVelFreq,AnsOutprVelFreq, AnsOutresVelNth, AnsOutprVelNth,AnsOutresAcc, AnsOutprAcc, AnsOutresAccFreq, AnsOutprAccFreq, AnsOutresAccNth, AnsOutprAccNth, AnsOutresEsol, AnsOutprEsol, AnsOutresEsolFreq, AnsOutprEsolFreq, AnsOutresEsolNth, AnsOutprEsolNth, AnsModalOptOn, AnsModalMethod, AnsModalNmode, AnsModalFreqb, AnsModalFreqe, AnsModalNrmkey,AnsModalStrmck,AnsModalRangefact, AnsModalBlocksize, AnsModalRobustlev, AnsModalCompute, AnsModalReusekey, AnsModalSymmeth, AnsBuckleOptOn, AnsBuckleMethod,AnsBuckleNmode, AnsBuckleShift, AnsBuckleLdmulte,AnsBuckleRangekey,AnsBuckleStrmck, AnsHarmonicOptOn, AnsHarmonicMethod, AnsHarmonicMinmode, AnsHarmonicMaxmode, AnsHarmonicFreqb, AnsHarmonicFreqe, AnsHarmonicLogopt,AnsHarmonicReimky,AnsHarmonicClust, AnsTransOptOn, AnsTransMethod, AnsTransMinmode, AnsTransMaxmode, AnsTransTintopt,

AnsTransDmpsfreq, AnsTransLumpm, AnsTransAlphad, AnsTransBetad, AnsTransDmpstr, AnsTransDmprat, AnsTransGamma, AnsTransGammavalue, AnsTransAlpha, AnsTransDelta, AnsTransAlphaf, AnsTransAlpham, AnsNlOptOn, AnsNropt, AnsNeqit, AnsCnvtolU, AnsUtoler, AnsCnvtolRot, AnsRottoler, AnsCnvtolF, AnsFtoler, AnsCnvtolM, AnsMtoler, AnsCnvtolDvol, AnsDvoltoler, AnsCnvtolHdsp, AnsHdsptoler, AnsNlgeom, AnsArclen, AnsLnsrch, and AnsPred attributes for ANSYS to the feAnalysisMgr Object.

- Added CombinationType and AbsoluteCombination attributes to feChart Object.
- Added IsCombination attribute to feConnect Object.
- Added attrTopology, MeshPointID, and vMeshPointID attributes to feCurve Object.
- Added InPlanePropExcMemBend, vInPlanePropExcMemBend, BendingPropExcMemBend, and vBendingPropExcMemBend attributes to feLayup Object.
- Added UseCriteria attribute to feMapOutput Object.
- Added attrTopology, attrMidsideOnGeometry and attrMaxMidsideAngle attributes to feSolid Object.
- Added FeatureLineAngle, ViewLegendJustification, PostTitlesJustification, ContourLegendHorizontal, ContourLegendReversed, ViewLegendLocation, vViewLegendLocation, PostTitlesLocation, vPostTitlesLocation, ContourLegendLocation, vContourLegendLocation, ContourLegendWidthPct, DiscreteValueOn, DiscreteValueSetID, ModelDataContourOn, ModelDataContourGroup, ModelDataContourLabels, ViewLegendLockJustification, PostTitlesLockJustification, ContourLegendTopLeftLabel, and ContourLegendLockOrientation, attributes to the to feView Object. Also, updated AxisLocation and vAxisLocation.
- Updated label attribute on the feAeroSurf Object.
- Updated formulation and vformulation attributes on the feElem Object.

New and Updated API Methods

- Added GetOptLimits and PutOptLimits for NX Nastran and MSC Nastran SOL 200 subcases to feAnalysisCase Object.
- Added KBC, PSTRES, EQSLV, TIME, AUTOTS, NSUBST, DELTIM, CUTCONTROL, MIDTOL, OUTRES, OUTPR, MODOPT, LANBOPTION, PCGOPT, SUBOPT, SNOPTION, QRDOPT, BUCOPT, HROPT, HARFRQ, HROUT, TRNOUT, LUMPM, ALPHAD, BETAD, DMPSTR, DMPRAT, TINTP, NROPT, NEQIT, CNVTOL, NLGEOM, ARCLEN, LNSRCH, and PRED for ANSYS Cases to feAnalysisCase Object.
- Added GetOptVars, PutOptVars, GetOptLimits, PutOptLimits, GetOptMCs, and PutOptMCs for NX Nastran and MSC Nastran SOL 200 Master Case to feAnalysisMgr Object.
- Added KBC, PSTRES, EQSLV, TIME, AUTOTS, NSUBST, DELTIM, CUTCONTROL, MIDTOL, OUTRES, OUTPR, MODOPT, LANBOPTION, PCGOPT, SUBOPT, SNOPTION, QRDOPT, BUCOPT, HROPT, HARFRQ, HROUT, TRNOUT, LUMPM, ALPHAD, BETAD, DMPSTR, DMPRAT, TINTP, NROPT, NEQIT, CNVTOL, NLGEOM, ARCLEN, LNSRCH, and PRED for ANSYS Master Case to feAnalysisMgr Object.
- Added ConnectionsByType, CombinationsByType, IsGlueConnector, and GetCombinations to feConnect Object.

- Added SetFriction, GetFriction, SetAnalyticSurfaceType, and GetAnalyticSurfaceType to feContact Object.
- Added Option Methods (Clear, SetNumbering, SetVectorPattern, SetCSysPattern, SetPropertyOption, IncludeOtherEntities, and AllOtherEntities) and Operation Methods (AlongVector, PointToPoint, BetweenCSys, BetweenVectors, BetweenPlanes, RotateAroundVector, RotatePointToPoint, Reflect, CreatePattern, and InPlace) to feCopyTool Object.
- Added MeshPointID, vMeshPointID, and InitMeshAttributes to feCurve Object.
- Added GetDataArray to feDataSurf Object.
- Added Get, Put, GetNumberLevels, GetLevelInteger, SetLevelInteger, AddLevelInteger, GetLevelFloat, SetLevelFloat, AddLevelFloat, Sort, GetTitle, SetTitle, CreateRangeInteger, CreateRangeFloat, and AutoCreateLevels to the feDiscreteValueSet object.
- Added GetAnsysFormulationArray, PutAnsysFormulationArray, GetAnsysKeyoptArray, PutAnsysKeyoptArray, HasProp, and HasMatl to t feElem Object.
- Added SetTotalLoad to feLoadDef Object.
- Added Put, Enable, IsEnabled, OnPoint, AutoDistanceToGeometry, UpdateMeshPoints, and UpdateMeshPointsOnGeometry to feMeshPt object.
- Added Get, Delete, ResetNextMeshPoint, NextMeshPoint, and CountMeshPoints to feMeshPointDefinition object.
- Added Option Methods (Clear and IncludeOtherEntities) and Operation Methods (AlongVector, PointToPoint, BetweenCSys, BetweenVectors, BetweenPlanes, RotateAroundVector, RotatePointToPoint, Reflect) to feMoveTool Object.
- Added SetAddm, GetAddm, SetCast, GetCast, SetCheckerboard, GetCheckerboard, SetCycSym, GetCycSym, SetExtrusion, GetExtrusion, SetMinSize, GetMinSize,SetMaxSize, GetMaxSize, SetPlSym, and GetPlSym to feOptMC object.
- Added SetupTopology, GetTopology, SetupShellT, GetShellT, SetupProp, GetProp, SetupMatl, GetMatl, and CalculateBounds to feOptRel object.
- Added Get, Put, SetEntities, GetEntities, SetupDisplacement, GetDisplacement, SetupSPCForce, GetSPCForce, SetupElem, GetElem, SetupESE, GetESE, SetupFreq, GetFreq, SetupEign, GetEign, SetupBuckle, GetBuckle, SetConstraints,and, and GetConstraints to feOptResp object.
- Added HasMatl to the feProp Object.
- Added SetLineTransform to the feResults Object.
- Added AddAllInRangeInSet and AddBySize to the feSet Object.
- Added InitMeshAttributes and Volume to the feSolid Object.
- Added InitMeshAttributes to the feSurface Object.
- Added FitVisible, SetModelDataContourForm, GetModelDataContourForm, SetModelDataContourType, GetModelDataContourType to the feView Object.
- Added GetFirstPathID, GetNextPathID to the feViewOrient Object.
- Updated SetDefaults on the feConnectionProp Object.

New and Updated Global Variables

- Added Pref_RenderBestAvailable, Pref_NXNastEntLicenseServer, Pref_FastHoleMeshing, and Pref_EnableLegacyAnsysInterface to set various preferences.

Removed Global Variables

- Removed Pref_NastranUseILP64, as this preference is no longer available.

The following functions have been added or updated:

- feFileWriteTopologyStl
- feSolidThicken
- feSurfaceMidAuto3
- feSurfaceOffset2
- feImprintLocationInSurface
- feResultsToDataTable2
- feMeshEdgeMembers2
- feMeshPoint2
- feMeshCurve2
- feModifyLineElementDirection
- feGroupFromCriteria
- feGetElementAdjacentFaces
- feSelectOutput2
- feSelectOutputSets2
- feOutputValueGetRGB
- feAppResetLocale
- feGetNodalAreaOrLength
- feGetNodalAreaOnSurface

New and Updated API Scripts (Custom Tools Menu):

- Added Custom Tools, Views, View Themes command (View Themes.exe), which runs as a dockable pane to provide easier creation and saving of backgrounds and entity colors.
- Added Custom Tools, Examples, PowerPoint - Group PostProcess Capture command (PowerPoint - Group PostProcess Capture.bas) that automatically creates a PowerPoint presentation, then create a slide containing a picture of each selected group. The script can also connect to an existing PowerPoint presentation and start creating new slides at a specific slide number or simply create imagines and save them to the specified directory (i.e., potentially not interact with PowerPoint).
- Updated existing Custom Tools, Data Table, Add Corner Thickness script. The script now works with the FEMAP Selector.
- Updated existing Custom Tools, Example, Using Text script. Added auto-repeat if turned on in FEMAP Preferences.
- Updated existing Custom Tools, PostProcessing, GROUP_POST_FNO script with option to export Full Model or individual Group(s).
- Updated existing Custom Tools, PostProcessing, List Output to Excel script to use Results Browsing Object for faster Output Vector listing.
- Updated existing Custom Tools, PostProcessing, Calculate Factor of Safety script for Solid Elements to prompt user for Limit Stress values if none are specified in material card.

- Updated existing Custom Tools, PostProcessing, Calculate Mid Plate Stresses_Results Browsing Object script to create additional vector for Mid-Plate Von Mises Stress.

Corrections

Views

- Corrected issue where bringing up the Performance Graphics Font Dialog from View Options Dialog and then moving the dialog will prevent the use of the mouse to select a font.
- Corrected issue where previous zoom values were not initialized correctly causing the magnification to be set to 0 (graphics system then uses 1e-4)
- Corrected issue that made picking difficult with Animate, Trace or Animate MultiSet active. The deformed model would be used to determine waht was picked and then the highlighting would be on the undeformed.
- Corrected issue where View, Rotate, Model and View, Zoom commands did not apply to all views correctly.
- Corrected issue where the values on the advanced post dynamic section cut dialog box did not update when scrolling the mousewheel while holding the Alt key down to dynamically move the cutting plane.
- Corrected issue when switching between views and the user clicks in the title bar of a non active view, the view is maximized.

Connection Properties, Regions, and Connectors

- Corrected issue when using the Mesh, Editing, Interactive or Mesh, Editing, Split commands, along with the Modify, Update Elements, Split Quad command which would cause regions to not include new elements which were created by spliting existing elements.

General

- Corrected issue which could cause application to exit unexpectedly when printing to graphics files.
- Corrected issue which allowed a model to be saved when an entity references a coordinate system that no longer exists. Now, FEMAP errors.

Geometry

- Corrected issue when attempting to use Modify, Edit, Point to edit solid points which never updated the solid.
- Corrected issue when projecting locations onto Arcs/Circles if the initial location being projected was at the center of the Arc/Circle. Previously it returned the center location which is not “on” the curve. This caused problems when breaking curves (PR# 8950203).
- Corrected issue when using Geometry, Curve - From Surface, Offset Curve/Washer command which could cause extremely short curves to be created, which could lead to an over-aggressive cleanup, resulting in distorted geometry.
- Corrected issue when using Geometry, Solid, Remove Face where removal of faces using the Parasolid VTK would not occur.

Graphics

- Corrected issue when picking zero length rigid elements, they are not visible by default. Changed default for rigid element symbols to be on.
- Corrected issue where preview arrows drawn directly into or out of the screen did not have a head or a tail.
- Corrected issue where CBUSH direction arrows were not drawn or not drawn correctly if zero length or were to ground
- Corrected issue with extremely poor performance with property based NSM regions. Large models showing 20 times improvement in performance (PR# 9089646)
- Corrected issue when using global transparency that caused the view legend, view axes, post titles and contour legend to be drawn transparent.
- Corrected issue where contour legend, view legend, post titles and view axes were drawn transparent when using Window, Show Entities command
- Corrected issue where origin color was not controlling the transparency of the origin.
- Corrected issue with incorrect colors for eliminated geometry entities.
- Corrected issue where points were not eliminated correctly in non Performance Graphics.
- Corrected issue where front picking of contact regions would possibly fail.
- Corrected issue where deleting output in a model with multiple views causes Loads and Boundary Conditions to disappear.
- Corrected issue where nodes were not drawn correctly when in non Performance Graphics free edge display and switching elements on and off.
- Corrected issue where non Performance Graphics only drew contour arrows on the free face solid elements.
- Corrected issue where beam cross section display did not obey blanking and erase/draw
- Corrected issue where beam cross section stress display did not work correctly for tapered beams.
- Corrected issue where criteria display of plate elements could not just draw criteria values and unfilled elements.
- Corrected issue where non Performance Graphics pressure loads did not display the phase value.
- Corrected issue where line contours disappear if you do List, Model, Node command, then rotate with middle mouse. This only happened in non-Performance Graphics with VBO on.
- Corrected issue with filled edges on cutting plane for Pyramid13 and Pyramid5.
- Corrected issue with labels not being transparent when transparency is on (PR# 7810217)
- Corrected issue with rotate/zoom about mouse not working if anything is transparent.
- Corrected issue where line elements override point elements making picking point elements difficult.
- Corrected issue where post titles disappear if the user does an element quality check from the meshing toolbox and kills the toolbox.
- Corrected issue where the no pick list was not being honored by fast pick visible.
- Corrected issue where picking connection regions by screen area was slow.
- Corrected several issues with free face and free edge displays when including whole model.
- Corrected issue when picking elements, if user rotates with middle mouse button and while holding it down, hits right mouse button, picking is broken until the model is rotated again.

- Corrected issue where aero splines were sometimes not drawing connected nodes immediately after reading from a Nastran deck.
- Corrected issue where streamlines were not depth buffered correctly which gave rise to confusing images with multiple streamlines
- Corrected issue which could cause a newly created coordinate system to not be drawn when when not using Performance Graphics

Performance Graphics

- Corrected issue with Performance Graphics drawing constraints in coordinate systems other than 0, 1 and 2 if the coordinate system IDs are not contiguous.
- Corrected issue with Performance Graphics with multiple views of the same model. If one view had beam sections or beam diagram and the other did not, switching between the views would give incorrect graphical display.
- Corrected issue where selecting Performance Graphics fonts only showed the list of fonts that were not hidden by the Operating System. All fonts are now listed for selection.
- Corrected issues with transparency that effected contour arrows, labels and symbols.
- Corrected issue where if you edited the property to change between CBUSH and Spring/Damper, the graphics did not change even with a ctrl-G.
- Corrected issue where constraints in cylindrical and spherical coordinate systems were not drawn correctly
- Corrected issue where undo did notwork with interactive mesh editing
- Corrected issue where moidifying material direction controlled by a material coordinate system was not updated graphically even after a ctrl-G
- Corrected issue where Window, Show Entities could cause FEMAP to exit unexpectedly when highlighting Connection Regions with specific options specified (PR# 9033536)
- Corrected issue where Dynamic Rotation About Cursor Location did not work with whole model (auto) transparency on.
- Corrected issue where rigids were not drawn in the undeformed model. Some undeformed color options were also not working correctly.
- Corrected issue where criteria did not work on RBE2 and RBE3.
- Corrected issue where symbols on line elements were contoured with first node values not the average value.
- Corrected issue where z arrowhead on coordinate systems were not drawn.
- Corrected issue where arrow head circles were drawn at the base of an arrow and not at the tip and this caused incorrect depth buffering.
- Corrected issue where clipping plane di not effect contour arrows.
- Corrected issue where color elements by material used the property color and not the lement color when the property had no material.
- Corrected issue where fast pick visible was not accounting for shrink.
- Corrected issue where 0 was drawn for a load with no function selected.

- Corrected issue when animating mixed Performance Graphics elements and non Performance Graphics elements (for example beam sections), nodes were not drawn correctly when stopping the animation to pick or rotate.
- Corrected issue where Performance Graphics pressure loads did not display the function ID.
- Corrected issue where Performance Graphics was not obeying black/white swap when printing.
- Corrected issue where a high ambient lighting can saturate the contour colors, resulting in an incorrect contour color.
- Corrected issue where nodes on elements in another layer are not drawn if the node layer is on and the element layer is off (PR# 7810484)
- Corrected issue where shells/plates were always included in the free face and they were not being controlled by the "all elements" switch in the Free Edge and Face option of View Options

GUI - General

Standard Entity Selection dialog box

- Corrected issues with the "Add Connected Fillets", "Add Tangent Surfaces", "Add Connected Elements", and "Add All Connected Elements" commands on the Pick^ menu in the standard entity selection dialog box. Previously, they worked correctly if you were selecting Surfaces or Elements, but did not work if you were selecting any other entity type but had switched to a Method that selected Surfaces or Elements.

GUI - Toolbars and Icons

Select Toolbar

- Corrected issue which caused FEMAP to become unresponsive if you used the Selector to choose Geometry-based loads and the Entity Editor or Data Table was active.

GUI - Dockable Panes

Model Info Tree

- Corrected issue that occurred any time you did an Undo/Redo across a command that internally called a File, Rebuild (like importing an analysis file). Previously the Model Info tree could be partially non-functional until it was reloaded manually. Now the tree is reloaded every time you do an Undo/Redo.
- Corrected issue where when updating the output set in the PostProcessing toolbox, the active output set in the Model Info tree was not updated as well

Meshing Toolbox - General

- Corrected issue when using the icon menu to toggle tools on/off in the Meshing Toolbox. Previously, "Geometry Tools", "Meshing Tools", and "Mesh Editing Tools" toggled on all "tools" in their section of the menu, while toggling off all of the other tools in the menu, however some tools were defined in the wrong section. Now, using any of these commands simply toggles on/off the tools in their section.

Meshing Toolbox - Feature Removal Tool

- Corrected issue when Feature Type set to Surfaces or Blends, where attempting to remove faces using the Parasolid VTK as a first step, would not occur.

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.
- Corrected issue where number of digits on freebody vectors were incorrectly controlled by digits for Contour / Criteria Style rather than digits for Freebody Vectors (IR# 8950588).

Charting Pane

- Corrected issue where updating a chart data series marker size via the dialog could result in excessively large marker sizes.
- Corrected issue which caused incorrect labeling for independent and dependent vectors for Vector vs Vector Chart Data Series dialog
- Corrected issue when using “vs. Position” where the relative position was being calculated before values were being transformed, which caused the coordinates to be presented in a different coordinate system. The new order of calculations performs the transformation before calculating the relative positioning to avoid a second translation of the coordinates
- Corrected issue where relative location ID was not correctly saved when the relative positioning option was enabled for “vs. Entity” Chart Data Series (PR# 9100245)
- Corrected issue with Chart Data Series dialog where output set may not be saved correctly for Vector vs Vector data series types
- Corrected issue in Chart Data Series dialog that would prevent the user from selecting vectors in the 9,000,000 - 10,000,000 ID range (PR# 8891224).
- Corrected issue where using the Delete All command in the Chart Data Series Manager, and there was no active chart being displayed, could cause FEMAP to exit unexpectedly.

Data Surface Editor

- Corrected issue where copying an Output Map Data Surface as text would not copy the ID row.
- Corrected issues where Tabular and Arbitrary 3D data surfaces could yield incorrect results when using Coordinate Systems other than Coordinate System 0.

Connection Editor

- Corrected issue in the Connection Editor that caused the connectors being displayed to be lost or corrupted if the entities were renumbered over top of existing entities. Previously if you closed the pane and reopened it it would reload properly but was initially incorrect. Now it is properly maintained.

Program File

- Corrected issue which occurred if you tried to record a program file but cancelled closing of other panes (like the API Programming pane). Previously, record would activate, but the panes were still open even though they could not be recorded. Now, record does not begin until all panes are closed.

Messages Window

- Corrected issue where duplicate message: “n Constraint Set(s) translated” could appear when writing Nastran deck (PR# 9133419)

Interfaces - FEMAP Neutral File

- Corrected issue which occurred when opening a model from a previous version and the user chose not to read the Views. Previously you would see a window for the model but no graphics because there was no View in the model. You could get around it by creating a View, but now that happens automatically.

Interfaces - Nastran

- Corrected issue which could cause time step 0.0 to be skipped when indexing op2 files.
- Corrected issue where PRGPST,YES card would be placed in Start Text in the Analysis Set, as well as enabling the flag in the NASTRAN Bulk Data Options, after importing a Nastran input file. This would cause the entry to be written twice to the Nastran input file during export.

Interfaces - NX Nastran

- Corrected issue where CBUSH Spring forces requests for Random (XYPRINT card) would not be written to dat file (PR# 8978258)
- Corrected potential crash when “auto” CBUSH Spring Force is requested.
- Corrected issue which caused Nastran analyses to fail if you had a line that contained nothing but spaces in the Bulk Data section and you were running with Desktop licensing.
- Corrected issue which caused a NSM entry in the Case Control section of an imported Nastran input file to be saved in to Start Text section of the Analysis Set, which resulting in two NSM entries being written to the Case Control section of the exported Nastran input file.
- Corrected issue which would cause an empty PBUSH entry (i.e., no fields defined) to not be “finished” with a new line when exporting a Nastran input file.
- Corrected issue in Nastran translator that could cause NLPARM = 0 to be written if the Use Load Set Options switch was used in the Analysis Set (PR# 8977113).
- Corrected issue where an invalid warning stating that Complex Modes were required for Rotor Dynamics would be issued (PR# 8839306)

Interfaces - ABAQUS

- Corrected issue where Femap was not reading pyramid results from ABAQUS ODB
- Corrected issue where in reading ABAQUS *.inp file, presence of a SYSTEM card with no xyz coordinates on succeeding lines (to define csys), will cause failure to read next header (keyword) in file, possibly resulting in incomplete or corrupt model.

- Corrected issue where ABAQUS Axisymmetric element DCAX would label corner temperature vectors incorrectly.
- Corrected issue where ABAQUS Membrane element would label corner temperature vector incorrectly.

Interfaces - MARC

- Corrected issue where Femap could exit unexpectedly when defining a MARC analysis set with no entries in the MARC Contact Table (PR# 9038253).

Interfaces - Comma-separated

- Corrected issues when reading Table 500/501 from an enhanced Comma-Separated results file if field were skipped (such as Tet Element Corner Results). Previously, the final field was not read. This problem did not occur if the Results file was attached, only when it was imported.

Loads and Boundary Conditions

- Corrected issue which resulted in Geometric Loads referencing a non-existing Coordinate System if the system they originally referenced was renumbered.
- Corrected issue when expanding nodal geometric loads when they were defined using the "Along Curve" or "Normal to Surface" directions and the load specified a coordinate system that was not aligned with the Global Rectangular coordinate system. In that case, previously, the components of the load were expanded as if they were in Global Rectangular, but they were then treated as if they were in the specified coordinate system.
- Corrected issue when expanding pressure loads on surfaces that were defined by invalid geometry, specifically, when the geometry had slope/normal discontinuities. Even though the real problem was the geometry irregularities, this correction will work around most geometry problems and result in pressures in the correct direction.
- Corrected issue with Load Sets set to NASTRAN Load Combination, which would occur if the underlying sets contained loads which were defined in a User Coordinate System and had one or more DOF/directions disabled. Previously, drawing, listing and sum forces was potentially impacted in these cases. Writing of the loads to NASTRAN however was not impacted by the problem and was correct.(PR# 9096500)

Meshing

- Corrected issue when meshing linked planar boundary surfaces, which in many cases would not have mesh properly in the past versions.
- Corrected issue which could occur when sizing solids for Hex Meshing which had combined curves and/or boundary surfaces.
- Corrected issue when sizing for Hex Meshing which would occur if adjacent edges in two solids were both defined by multiple curves. Previously the element count would be correct but the spacing of the nodes could be different resulting in free faces when you hex meshed.
- Corrected issue when inserting a mesh point on a spherical surface when it intersected the surface along an element edge.

- Corrected issue which occurred when trying to create a linked surface mesh between surfaces that had intermediate surfaces that were multi-surface boundaries.
- Corrected issue where mesh points in a surface mesh happened to be located along edges of elements that would have been created without the mesh points.
- Corrected issue when meshing very bad surfaces with triangular elements which caused edge intersections that could cause FEMAP to exit unexpectedly (PR# 9112369).
- Corrected issue which created incorrect elements when meshing surfaces which had previously been meshed with the Max Quads option and parabolic elements (PR# 9253333).
- Corrected issue when using the Mesh, Editing, Interactive or Mesh, Editing, Split commands, along with the Modify, Update Elements, Split Quad command which would cause regions to not include new elements which were created by splitting existing elements.

Elements - General

- Corrected issue where rigid, slideline and weld elements were not renumbered correctly by minimum node ID.

Elements - Spring/Damper

- Corrected issue where the location along the spring of the CBUSH element was not initialized correctly.

Properties

- Corrected issue with evaluation of neutral axis offset for PBEAML HAT1 section (PR# 9072562)
- Corrected issue where tube and curved tube property dialog did not check inner diameter was less than outer diameter.

Output and Post-Processing

- Corrected issue where hitting Apply or OK in the Contour Arrow Style of View Options will change the color of the first arrow component to black.
- Corrected issue where element, property and material ID were not highlighted in Model Data Contour dialog if they were previously selected.
- Corrected issue where beam diagram was used for certain invalid Model Data Contour displays.
- Corrected issue where Slide Line elements not processed by Model Data Contour
- Corrected issue where mouse wheel zoom for beam post in screen space was opposite to main view.
- Corrected issue which caused Result values to be incorrect if you created an “As Needed/ Temporary” (virtual) result set by processing other “As Needed/Temporary” (virtual) result sets. For example, a virtual envelope of virtual linear combinations.
- Corrected issue which prevented elemental contoured data values from appearing in tooltips when displaying a Section Cut or IsoSurface in the graphics window (PR# 9086026)

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.

Tools

- Corrected issue where Femap Skew check improperly reported failure for ideal penta element (PR# 8983591)
- Corrected issue which prevented Safe Merge from working when merging coincident nodes, but only occurred in very specific dimensional cases.
- Corrected issue with the Tools, Check, Planar command when trying the simple, degenerate case of simply checking 3 nodes (PR# 9104630)
- Corrected issue where Tools, Mass Properties, Mesh Properties command would create a mass element with zero value for My and Mz.

Model Merge

- Corrected issue which prevented Constraint Equations from being copied in File, Merge command.
- Corrected issue that prevented entities from being properly copied via the File, Mesh command if they had larger IDs than some other ID that was referenced to be copied but did not exist in the model.

Preferences

Interfaces

- Corrected issue with the “Improve Real Number Precision” preference when writing numbers between +/- 1.0E-3 and 1.0E-11 using wide field format. Previously, the precision was not improved to the maximum possible extent and in the lower ranges of these values could actually reduce precision compared to having the preference turned off.

API

- Corrected issues with a number of API calls to properly get surface and solid facets were broken.
- Corrected issue where calls to access beam section results was extremely slow. Looping over 10 output sets and for each looping over 3700 beam elements is almost 60 times faster.
- Corrected issue where call ro calculate beam section properties was not setting the reference point correctly.
- Corrected issue in Data Surfaces with Model to Model mapping using API (MapFromModelToSet and MapFromModelToSet2) when targetSetID was not equal to 0.
- Corrected issue where Skew was not being checked via call to CheckQuality.
- Corrected issue with SelectMultiID of the API Set object that caused pre-selected entities to not be checked if you were selecting Result Sets. Other data types worked previously
- Corrected issue where an add-in pane created via an API script could be closed via the system “X” while an API was actively communicating with FEMAP.

- Corrected issues which could cause nodes which are not associated to any elements being duplicated to also be duplicated when using various calls in the API that create duplicate elements.
- Corrected issue with return code from feFileAttachSave2 and cleared the undo stack for API operations on attached output to match the behavior of the User Interface.
- Corrected issue which caused Info_NextID to be overwritten when using the newer functionality for face selection.
- Corrected issues where return code FE_NOT_AVAILABLE was not being returned when using a set-based entity type which is not supported by a method (PR# 7930455)
- Corrected issue where certain AnalysisMgr properties could be updated via API without the changes being reflected in the Analysis Manager in the user interface.

FEMAP v11.4.2 New Features and Corrections

Updates and Enhancements

GUI - Toolbars and Icons

Panes Toolbar

- Added TMG Thermal/Flow icon, which opens the Thermal/Flow Model Info dockable pane.

GUI - Dockable Panes

Thermal/Flow Model Info - New for 11.4!

- For more information about using this pane, use the Help, TMG Thermal and Flow command to open a HTML version of the Femap Thermal/Flow User Guide.

Interfaces - NX Nastran

- Added support for NX Nastran 12.0

Interfaces - Geometry

- Added support for NX 12.0

Corrections

Interfaces - Nastran

- Corrected issue which could cause FEMAP to write NLPARM = 0 if the Use Load Set Options switch was used in the Analysis Set (PR# 8977113).
- Corrected issue which caused NASTRAN PBUSH Stress/Strain recovery coefficients to be written incorrectly. Previously the Stress Rotation Coefficient and the Strain Translation Coefficient were swapped.

GUI - Dockable Panes

Meshing Toolbox

- Corrected issue which caused Rigid Interpolation Elements (RBE3 elements) to have their Degrees-Of-Freedom and coefficient factors disappear when using any tool in the Meshing Toolbox which tried to automatically update the nodes of a Rigid Interpolation element. This would occur even if the number of nodes on the Rigid Interpolation did not change. Rigid Interpolation elements are now properly updated as long as the same Degrees-Of-Freedom and coefficient factors are specified for all of the Nodes to Average. If all of the Nodes to Average on a Rigid Interpolation element do not have the exact same Degrees-Of-Freedom and coefficient factors specified, the element is not automatically updated and a messaged is issued informing the user that the element must be updated manually.

Charting

- Corrected issue with the “2..Vector vs. Vector” Data Series type which would cause the Sort Data option to not be saved in the Chart Data Series dialog box, thus editing this type of Chart Data Series could change how the Data Series was displayed.

Output and Post-Processing

- Corrected issue when attached to a Nasrtan .op2 file that could cause nodal output which uses an output coordinate system to be incorrect. This could occur if another input file was imported into FEMAP between the time that the .op2 file was attached and when output data was actually retrieved from the .op2 file (PR# 8983429).
- Corrected issue where contour arrow plots of principal stresses and strains or principal stress/strain angle for an individual ply in a laminate were oriented with respect to the element coordinate system rather than the ply orientation (PR# 9015291).

API

- Corrected issue when using the RemoveMeshPoint method of the Surface Object, which would occur when the first defined mesh point was removed from the surface. This would zero out all counters and other API functions used downstream would then show there were no mesh points specified on the surface.
- Corrected issue which allowed an “add-in” dockable pane to be closed via the system “X” icon, even while an active api “conversation” was in progress.

FEMAP v11.4.1 New Features and Corrections

Updates and Enhancements

Interfaces - Geometry

- Added support for Parasolid 30.0, Solid Edge with Synchronous Technology 10, and Pro/Engineer Creo 4.

Interfaces - NX Nastran

- FEMAP with NX Nastran bundle now includes NX Nastran 11.0.2.

Output and Post-Processing

- Added ability to transform magnitude/phase complex output data.

API

New and Updated API Methods

- Added GetSavedSetType to the Set Object.
- Added CollectorSingleSymbol, CollectorAddSymbolEntityFaceNormalAutoLocations, CollectorAddSymbolEntityEdgeLocations, CollectorAddSymbolEntityEdgeAutoLocations, CollectorAddSymbolREAL8EntityFaceNormalAutoLocations, CollectorAddSymbolREAL8EntityEdgeLocations, and CollectorAddSymbolREAL8EntityEdgeAutoLocations to the User Defined Graphics Object.
- Updated SetTri3Orientation, SetTri6Orientation, SetQuad4Orientation, and SetQuad8Orientation for the Results Browsing Object, by allowing user to specify Material Direction for each argument. In addition, updated SetSolidOrientation to allow user to specify Element for each argument.

New and Updated Global Variables

- Added UndoGrayed global variable.
- Updated Info_OrientTria3StressOuput, Info_OrientTria3StrainOuput, Info_OrientTria3ForceOuput, Info_OrientTria6StressOuput, Info_OrientTria6StrainOuput, Info_OrientTria6ForceOuput, Info_OrientQuad4StressOuput, Info_OrientQuad4StrainOuput, Info_OrientQuad4ForceOuput, Info_OrientQuad8StressOuput, Info_OrientQuad8StrainOuput, and Info_OrientQuad8ForceOuput global variables, by allowing user to specify a value of 3 to specify Material Direction. In addition, Info_OrientSolidIsoOutput, Info_OrientSolidAnisoOutput, and Info_OrientSolidHyperOutput now allow user to specify a 3 for Element.
- Updated Pref_OrientTria3StressOuput, Pref_OrientTria3StrainOuput, Pref_OrientTria3ForceOuput, Pref_OrientTria6StressOuput, Pref_OrientTria6StrainOuput, Pref_OrientTria6ForceOuput, Pref_OrientQuad4StressOuput, Pref_OrientQuad4StrainOuput, Pref_OrientQuad4ForceOuput, Pref_OrientQuad8StressOuput, Pref_OrientQuad8StrainOuput, and Pref_OrientQuad8ForceOuput preference variables, by allowing user to specify a value of 3 to specify Material Direction. In addition, Pref_OrientSolidIsoOutput, Pref_OrientSolidAnisoOutput, and Pref_OrientSolidHyperOutput now allow user to specify a 3 for Element.

New and Updated Events

- Added FEVENT_UPDATEDSAVEDSET event, which indicates a “SavedSet” Set Object has been updated.
- Added FEVENT_LEAVINGMODEL event, which can be used in conjunction with FEVENT_SWITCHMODEL and indicates about to leave a model. lParam contains the identifier of the model being left.
- Added FEVENT_CLOSINGMODEL event, which can be used in conjunction with FEVENT_ENDMODEL and indicates closing a model. lParam contains the identifier of the model being closed.

The following functions have been added or updated:

- feAppUndoClear
- feChartPaneCopy
- feEntitySetVisibility2 (PR# 7993007)

Corrections

Analysis Manager

- Corrected issue when deleting FREQi entries by adding an error message that warns the user the FREQi entities may possibly be referenced by other Analysis Sets or used by the Model, Output, Forced Response command.

Interfaces - Nastran

- Corrected issue where any number of nodal loads, which have the same phase value and are in the same Load Set, could potentially produce a DPHASE entry for only a single node instead of all the appropriate nodes (PR# 8338460).
- Corrected issue which would cause CBAR element output to be imported into Beam element output vectors instead of Bar elements output vectors when both CBAR elements and at least one CBEAM elements were defined in the model (PR# 7847627).

Interfaces - MSC Nastran

- Corrected issue which would cause a Connector to be skipped when the Connector ID overlapped with a Connection Property ID (PR# 7948179).

Geometry

- Corrected issue where imported JT files would be read into layer 0 rather than the active layer
- Corrected issue which occurred when creating any planar boundary surface on a negative global plane, which would cause the “Holes appear outside outer boundary” message to be displayed.
- Corrected issue with the Geometry, Copy, Solid; Geometry, Scale, Solid; Geometry, Rotate, Solid; or Geometry, Reflect, Solid commands, where the color and layer of copied/scaled/rotated/reflected solid(s) would not match the original solid(s) when the Color and Layer option was enabled.

Graphics and Performance Graphics

- Corrected issue when Contour Style is set to Section Cut where scrolling the mouse wheel while holding down the ALT key would not redraw after each mouse wheel change
- Corrected issue where preview vectors would not be drawn for certain commands (Geometry, Solid, Primitive; Geometry, Curve - From Surface, Parametric Curve; and Mesh, Mesh Control, Mapped Divisions on Surface, among others) if the preview color had a transparency component
- Corrected issue with element edge coloring when Performance Graphics were not enabled. If element color was set to Property or Material, element edges would still be colored by Property or Material if the filled edge color was set to entity color (PR# 8966551)
- Corrected an issue which would occur if element display was turned off while displaying a free edge plot, the elements would still be displayed until the view was regenerated if Performance Graphics was not enabled.
- Corrected issue when Contour Style was set to Arrow where arrow plots of nodal values would only get drawn on free faces (no interior nodes) when Performance Graphics was enabled

GUI - Dockable Panes

Meshing Toolbox

- Corrected issue which caused rigid elements to not be properly updated when updating the mesh on boundary surfaces containing circles or internal loops.
- Corrected issue which caused rigid elements to not be properly updated when the rigid element was connected to more than two Parasolid bodies (surfaces or solids) and/or FEMAP boundary surfaces and the mesh was updated on any of the bodies/boundary surfaces.

Charting

- Corrected issue where logarithmic axes were not displayed correctly for the phase component of magnitude / phase plots in the charting pane (PR# 8985807).

Meshing

- Corrected an issue when meshing any curve-based boundary surface which also has an existing surface manually associated to the boundary surface, which caused the resulting mesh to follow only the boundary edges instead of following both the edges and underlying surface.

Output and Post-Processing

- Corrected issue in Chart Data Series dialog box that would prevent the user from selecting output vectors in the 9,000,000 to 9,999,999 range (PR# 8891224)
- Corrected issue where invalid results could be displayed in the Charting Pane or Data Table when data for multiple ABAQUS output sets was requested.
- Corrected issue with Model, Output, Expand Complex and on-the-fly expansion of complex results where the output was expanded using lagging phase angles rather than leading phase angles.
- Corrected issue where Freebody vector digit control was set to Contour Legend digits when Sum Data on Nodes option was not enabled
- Corrected issue which allowed laminate strength ratios to be linearly combined.

- Corrected issue with results transformation that would allow integer columns (i.e., Output Set IDs in Envelope sets) to be transformed.
- Corrected issue where output from shell laminate elements was being included when creating an envelope of only output from solid laminate elements using Model, Output, Process, thus creating solid laminate element output on both solid laminate and shell laminate elements (PR# 7819468)

API

- Corrected issue which did not allow the Enable method of Connection Region object to be used with NonStructural Mass regions.
- Corrected issue in feSolidExtractCenterlines where stress recovery points would not be created when extracting from circular cross-sections.
- Corrected issue where output orientation could not be set as Material Direction for shell elements or Element for solid elements for the Results Browsing object, Preferences, or directly on the main Application object (model).
- Corrected issue when FEMAP was started via API and spaceball was not initialized external APIs could not connect to the FEMAP COM server
- Corrected issue with Results Browsing Object where envelope columns could return results from unexpected columns if any transformation options were also set. The envelope columns previously also considered additional columns that were automatically added to calculate the transformation. Now only columns that were specifically added by the user are considered.

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.
- Updated titling of Groups created when Create Groups from INCLUDE files option is enabled and the absolute filename is longer than the allowable title length (79 characters). Previously the title was simply truncated to the filename with no path. Now, the full path is trimmed from the left on subdirectory boundaries to include as much of the path as will fit in the title.

Results

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

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
- feSolidAlignSplits
- feOutputGlobalPly

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).
- Corrected issue which could cause the current model orientation and post-processing options of a “non-active” view to not be honored when a “non-active” view became the “active view”.

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 interpreted 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 expanded 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.
- Corrected issue when using the Select method of the Set Object, which allowed selection of Group IDs using the Group drop-down in the standard entity selection dialog box. Now the Group drop-down is properly grayed, as it is in when selecting Group IDs using standard commands.

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.
- Corrected issue which caused the Contour Max/Min information to appear in the Postprocessing Legend for non-Contour displays if you had the Post Titles Legend Style option set to display them along with the Contour/Criteria Style Label Mode set to one of the Max/Min options. Now this information is only displayed for Contour displays (PR #7754829)

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.
- Corrected issue with the API methods feFileWriteFNO and feFileWriteFNO2. Previously, if you attempted to use them to write data from attached result files using a node/element group that only included IDs where there were no results in one or more of the selected Output Vectors, in some cases, the FNO that was created was corrupted.

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.
- Corrected issue when finding automatic connections between some pairs of toroidal surfaces.

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).

Listing

- Corrected issue in the List, Output, Contoured Results to Data Table command. If you used it to report data from a double-sided elemental contour, the Top/Bottom label in the Data Table was reversed

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, OuptutSet, 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 command 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 Entity 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.

FEMAP 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.