ATA ENGINEERING, INC.

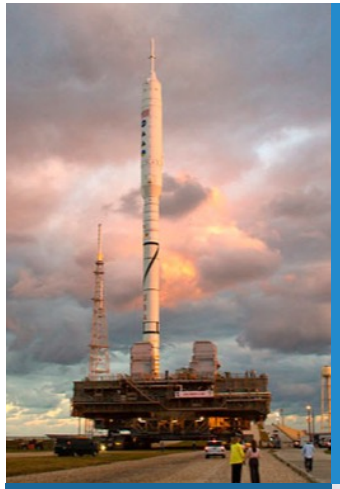
What's New in Femap 2301

March 30, 2023

Andy Haines, Siemens Digital Industries Software

What We Do

ATA Engineering's **high-value engineering services** help solve our customers' toughest product design challenges



Aerospace

Spacecraft

Hypersonics & Composites

Defense

Industrial & Mining Equipment

Themed Entertainment

Who We Are

We are an **employee-owned** small business with a **full-time staff of nearly 200**, more than 160 of whom are degreed engineers

Our Expertise

- Design: From initial concept development to detailed structural design
- Analysis: Comprehensive structural, fluid, acoustic, and thermal analysis services
- Test: Industry-leading structural test services for extreme loading environments

ATA is a Value-Added Reseller for Siemens Digital Industries Software

ATA offers training, free resources, and hotline support for a variety of Siemens products.

Hotline Support

Nastran Training Material

Software Implementation

Siemens Preferred Training Provider

Software Integration

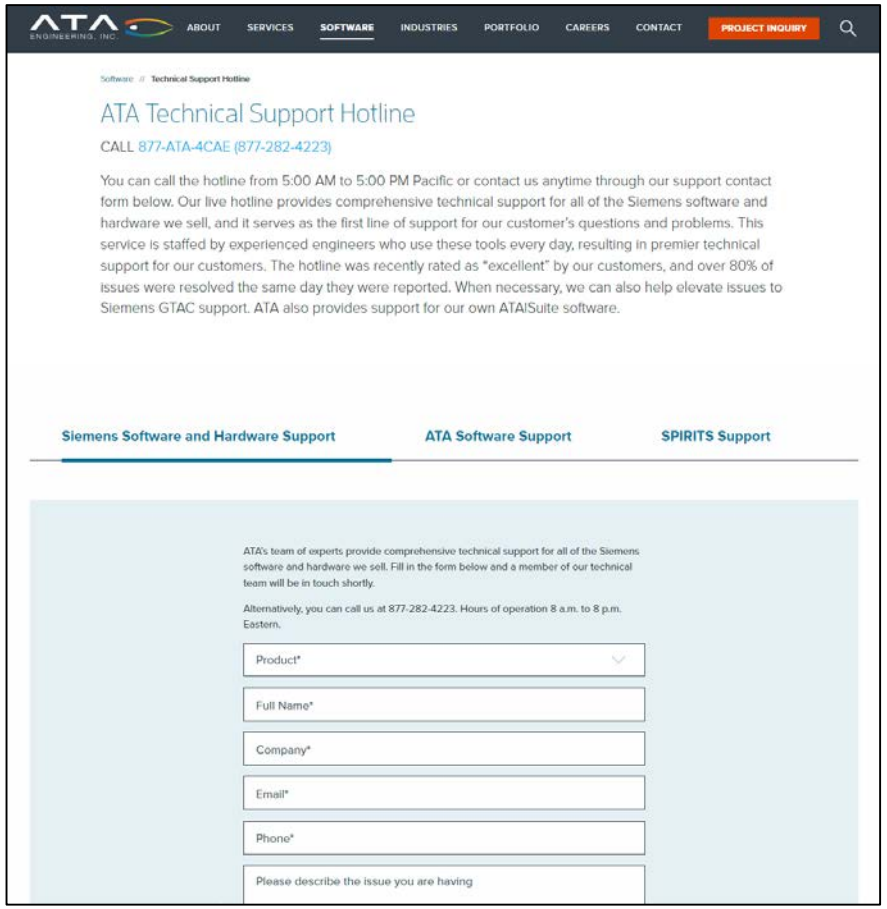
Expert Partner
Digital Industries Software

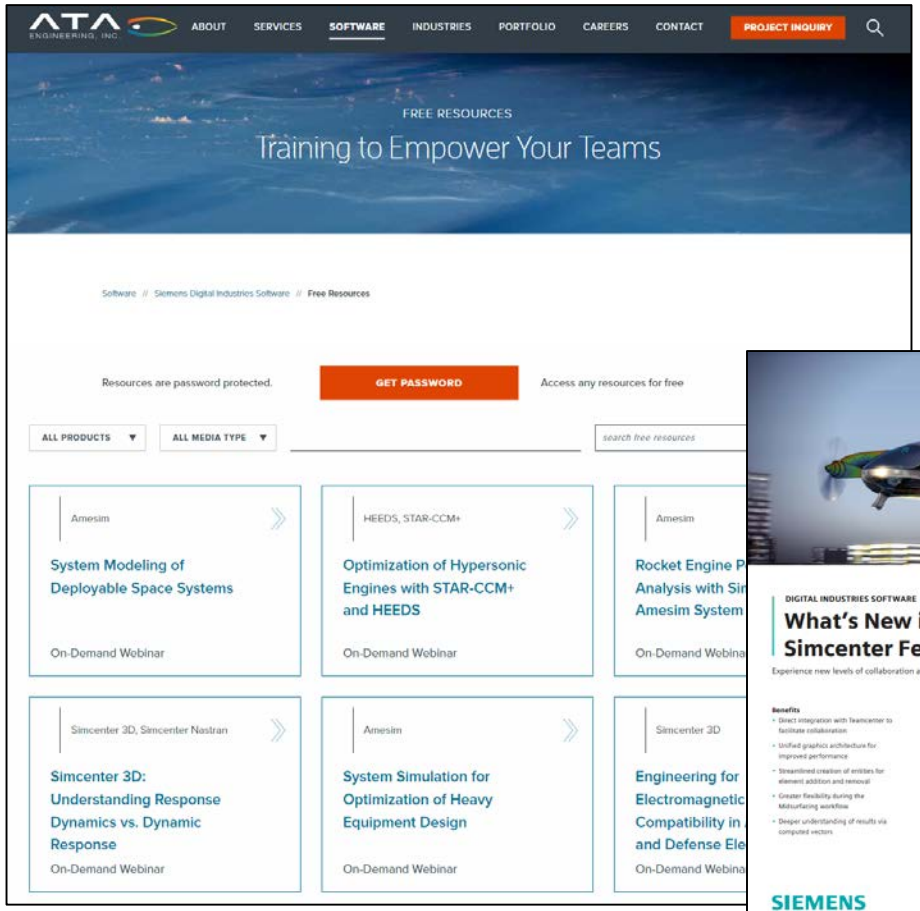
SIEMENS

ATA Engineering is recognized as a Certified Expert Partner with validated expertise in Femap, STAR-CCM+, and Simcenter 3D.

Visit Our Website for Product Information and Free Resources

https://www.ata-e.com/software/



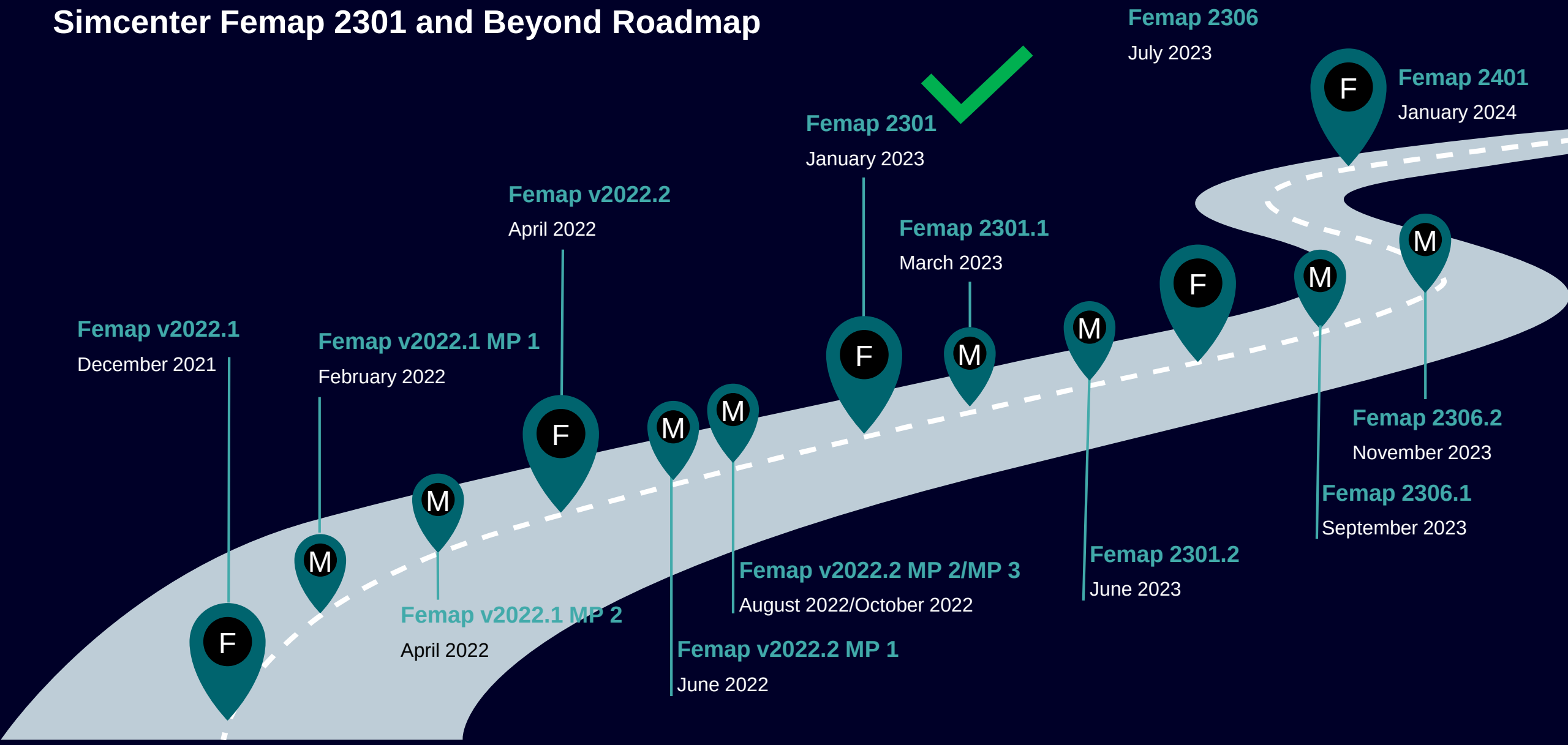


What's New in Femap 2301

March 30, 2023

Andy Haines, Siemens Digital Industries Software

Simcenter Femap 2301 and Beyond Roadmap



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

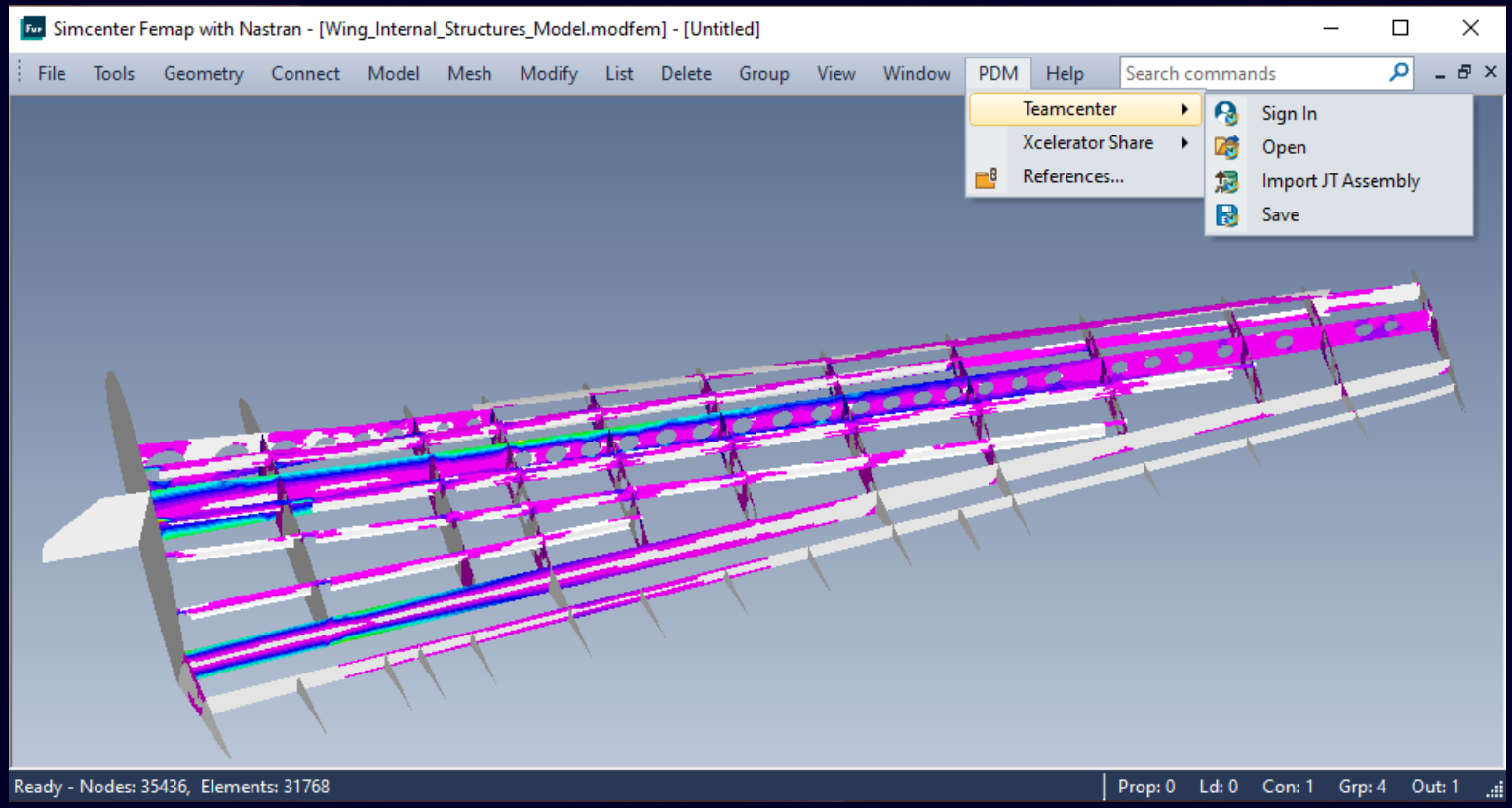
SIEMENS

Simcenter Femap 2301
Teamcenter Integration

Allows Femap files to be managed from within the Teamcenter Environment

Interaction with Teamcenter is accomplished through the **PDM -> Teamcenter** menu or Teamcenter toolbar

New PDM menu also contains **Xcelerator Share** menu (formally **File -> Share** menu) and **References** command (formally **File -> References** command)



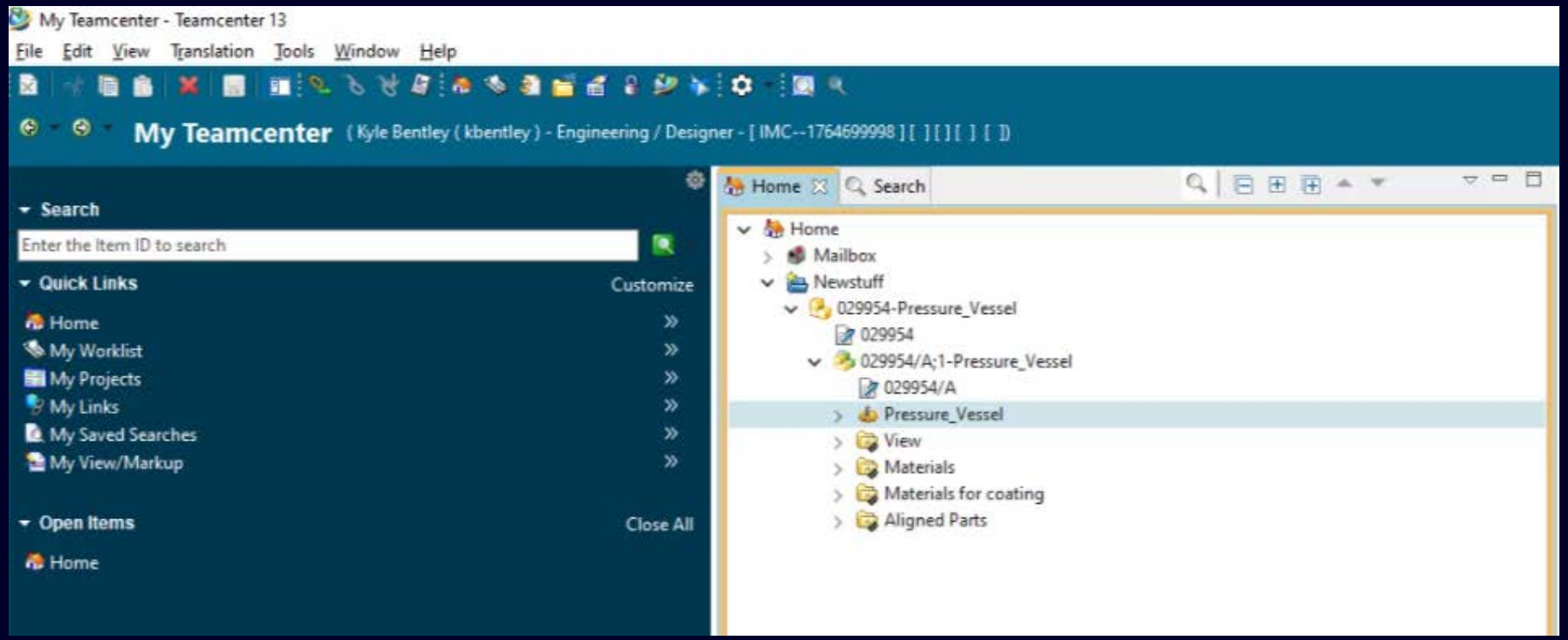
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Teamcenter Integration

Femap requires the Teamcenter Rich Application Client (RAC) for implementation

- “Items” need to be created in RAC to use with Femap



Note: Interaction with Teamcenter Active Workspace planned for future version

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Teamcenter Integration

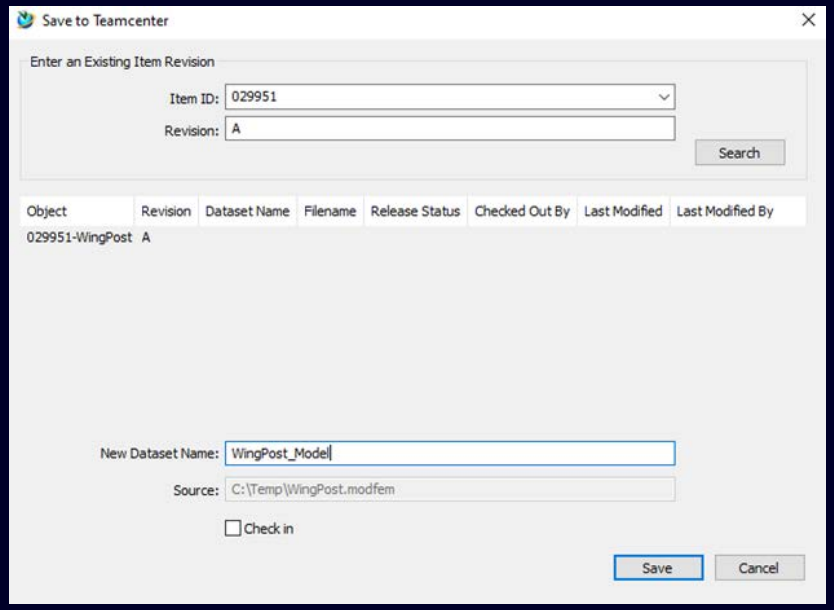
Sign-in to connect to user's Teamcenter Environment

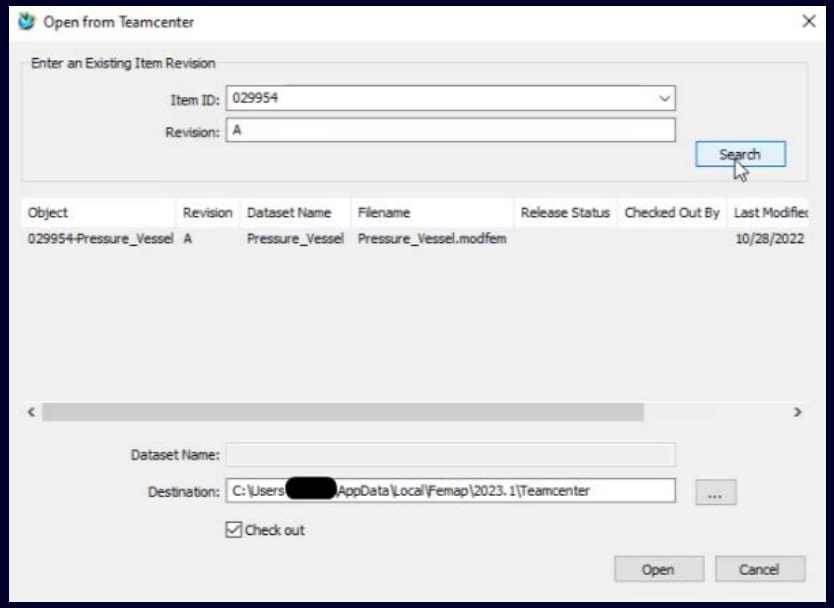
- Allows file check-in and check-out
- Revisions and revision history documentation

Create a dataset that hosts the Femap .modfem file directly in Teamcenter

Save and Open Femap models to/from the Teamcenter database

- **PDM -> Teamcenter -> Save** allows user to check-in file
- **PDM -> Teamcenter -> Open** allows user to check-out file





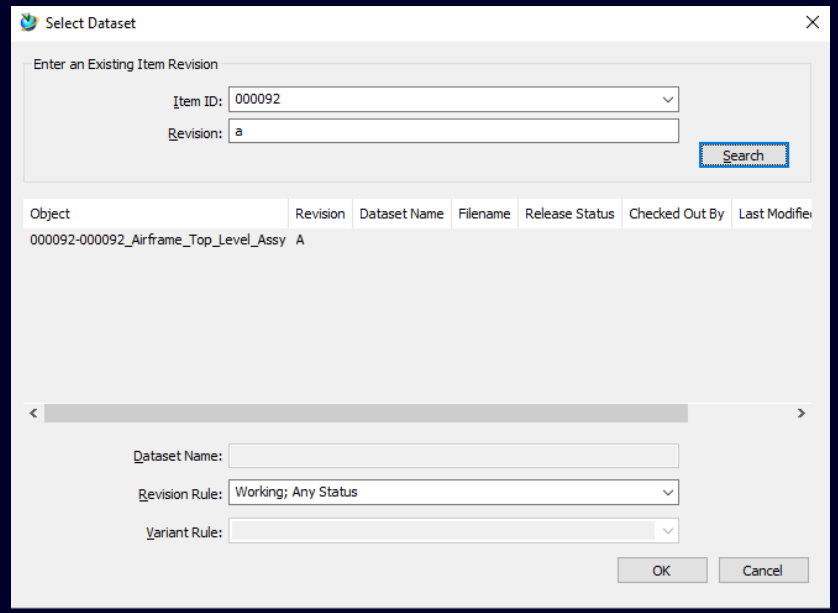
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Teamcenter Integration

The **PDM -> Teamcenter -> Import JT Assembly** command imports Assembly Item IDs from a Teamcenter Dataset into Femap

- Femap uses a generated JT model on each item revision to import the geometry
- Femap looks at each assembly in Teamcenter using the BVR (BOM View Revision) to find the associated part IDs and sub-assembly IDs
- The JT files can be generated/saved from multiple CAD systems within Teamcenter
  - Note: JT files can be manually imported one-by-one into Teamcenter if they are stored on the disk



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

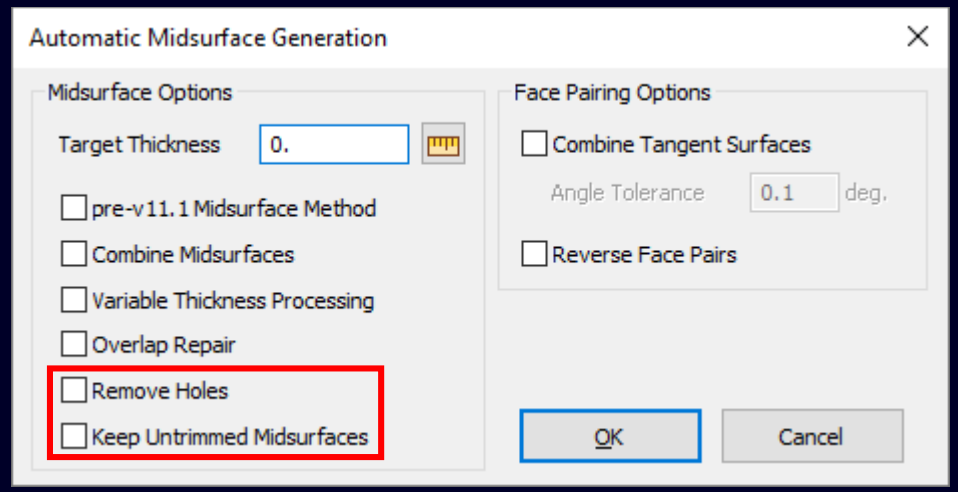
SIEMENS

Simcenter Femap 2301
Geometry – Midsurfacing Enhancements

Two new options have been added to the **Geometry -> Midsurface -> Automatic** command, both designed to allow more flexibility during the midsurfacing workflow

Remove Holes – When used, any internal holes in the resulting midsurface geometry will be deleted

Keep Untrimmed Midsurfaces – When used, any midsurface which is found but not trimmed properly is kept instead of being deleted

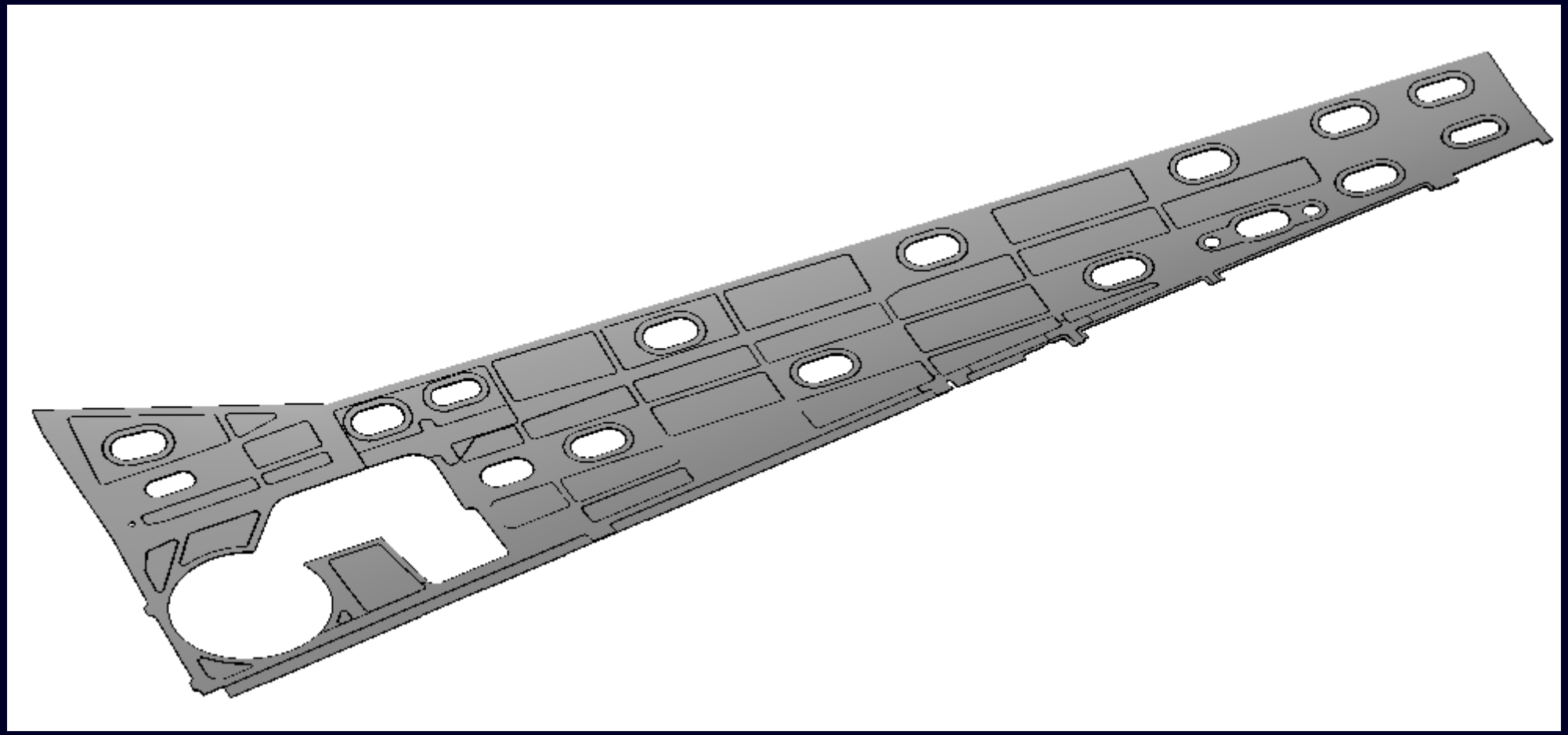


© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Geometry – Midsurfacing Enhancements

Original Geometry

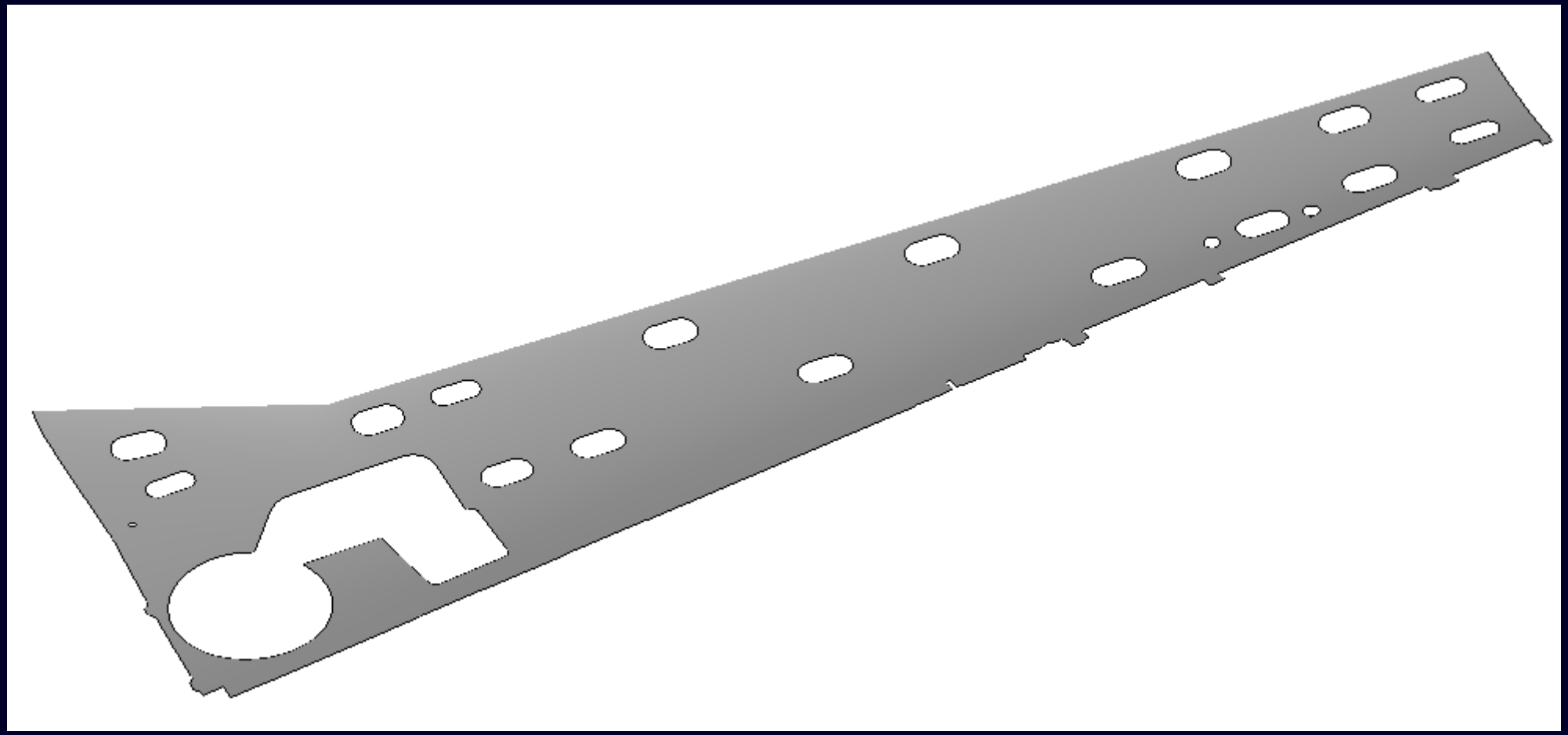


© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Geometry – Midsurfacing Enhancements

Remove Holes option Turned Off

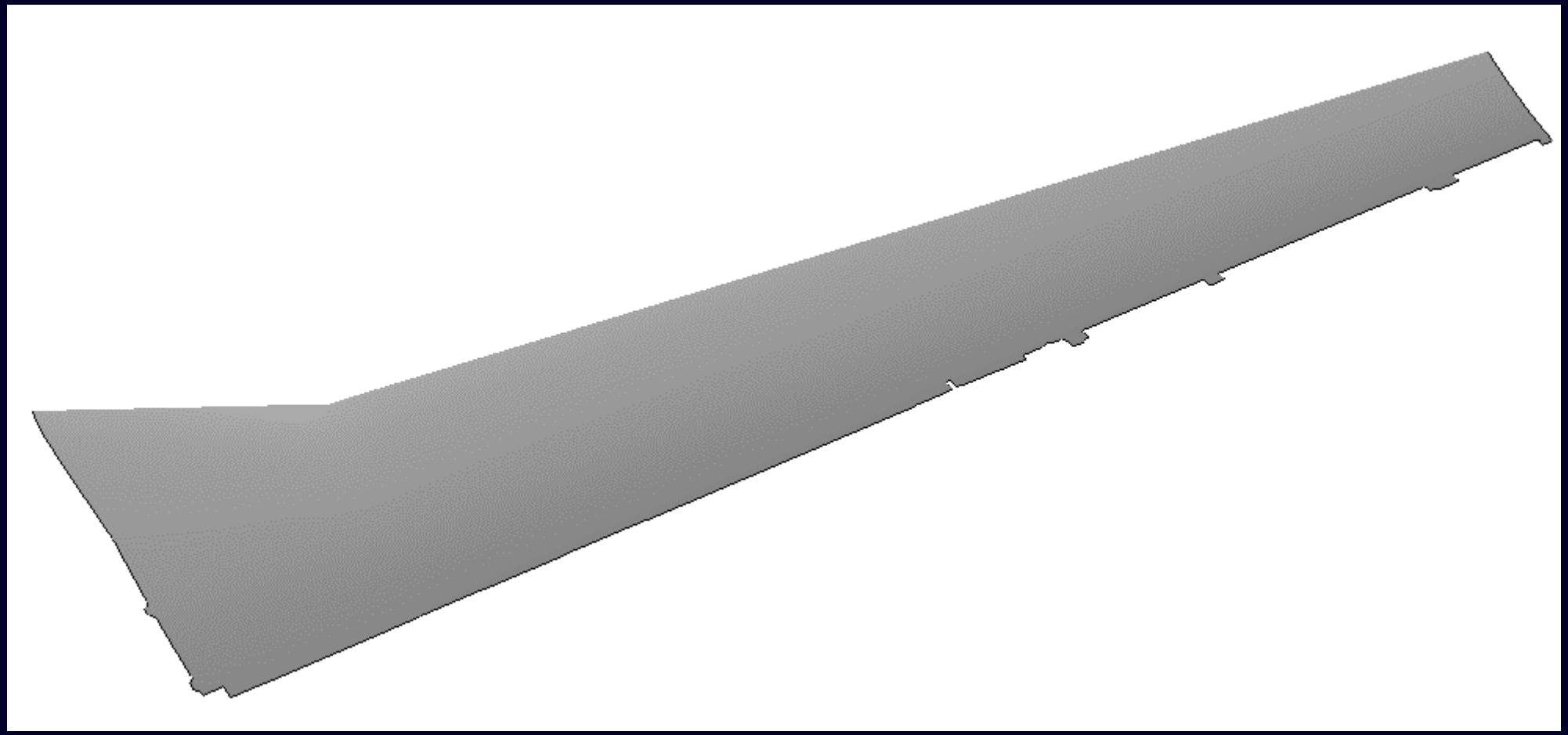


© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Geometry – Midsurfacing Enhancements

Remove Holes option Turned On



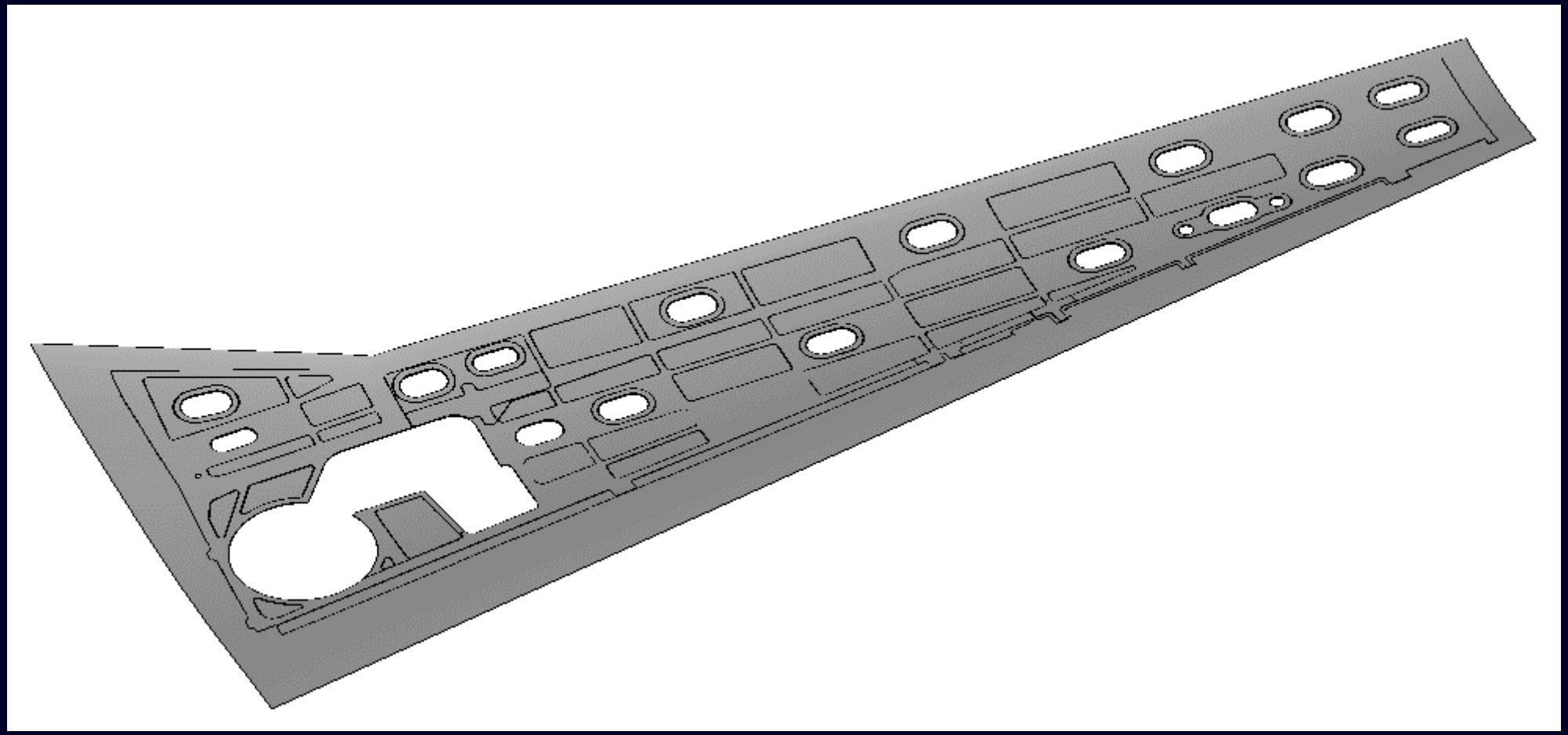
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301

Geometry – Midsurfacing Enhancements

Keep Untrimmed Midsurfaces option Turned On
(both Original Geometry and Midsurfaces shown for clarity)



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Geometry – Miscellaneous

Added support for reading Points in the Catia V4 direct geometry interface

Updated various tools in the *Meshing Toolbox* which can be used to move/project "Femap points" which also modify curves, surfaces, and solids created with Femap's "Standard" Geometry engine (i.e., non-Parasolid geometry)

- Especially helpful when updating wireframe models which contain "Femap Curves" meshed with line elements, as nodes are moved with curves

Improved the **Group -> Curve -> From Node** and **Group -> Surface -> From Node** commands to work with non-Parasolid FEMAP geometry

- Previously these commands only worked with Parasolid geometry
- This improvement was also made to associated selection methods in standard entity selection dialog, CurvesAsSet and SurfacesAsSet methods on API Point object, and Surfaces and SurfacesAs Set methods on API Curve object

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

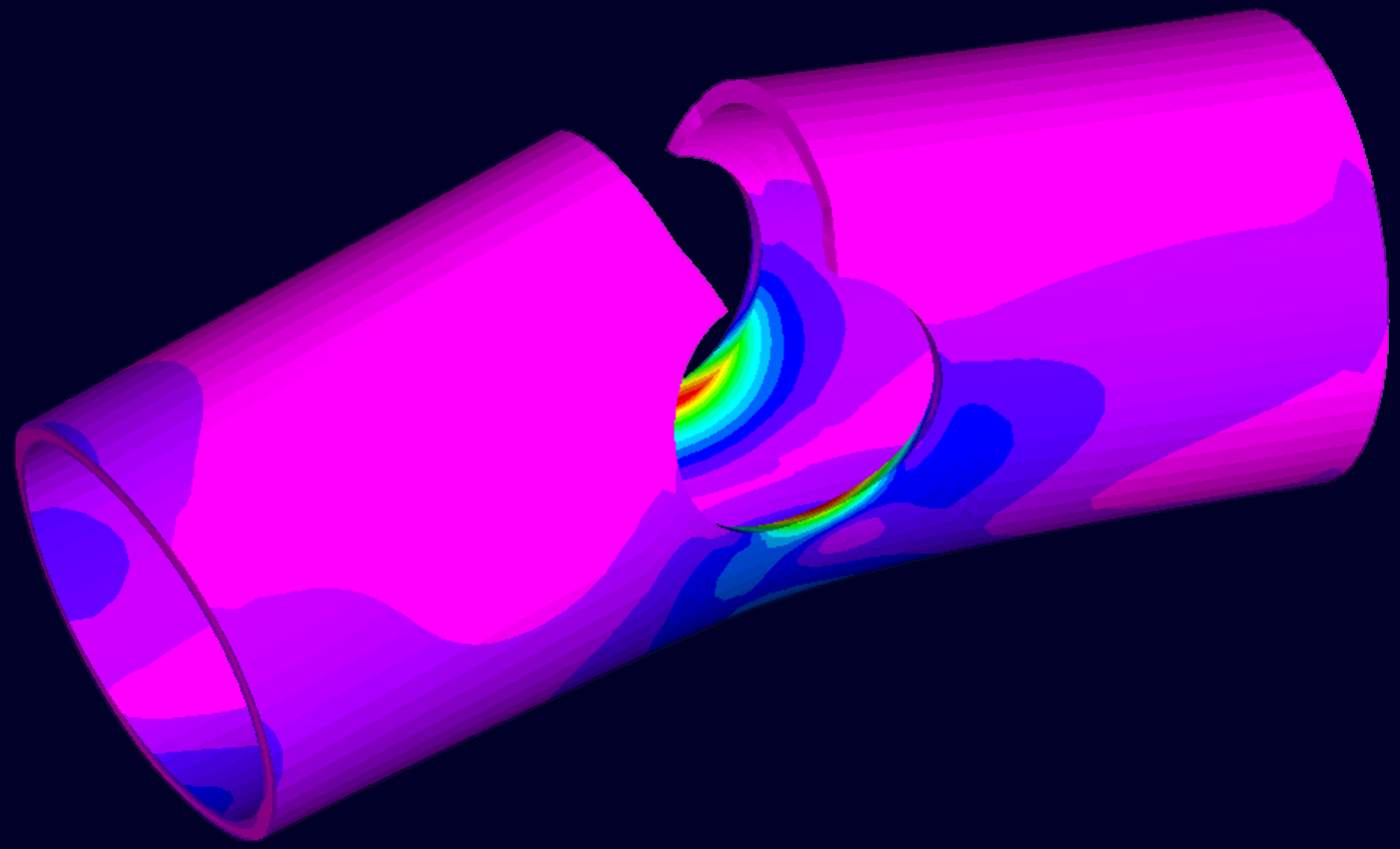
Simcenter Femap 2301
Preprocessing – Element Add | Remove

Simulation Entities used to perform element addition and removal when using Simcenter Nastran SOL 401

Accessed by **Model -> Simulation Entities -> Element Add | Remove** command or via Model Info tree

Both Element Add | Remove Entities and Sets referencing multiple Entities can be created

Creates ELAR, ELAR2, or ELARADD entries in Nastran input file



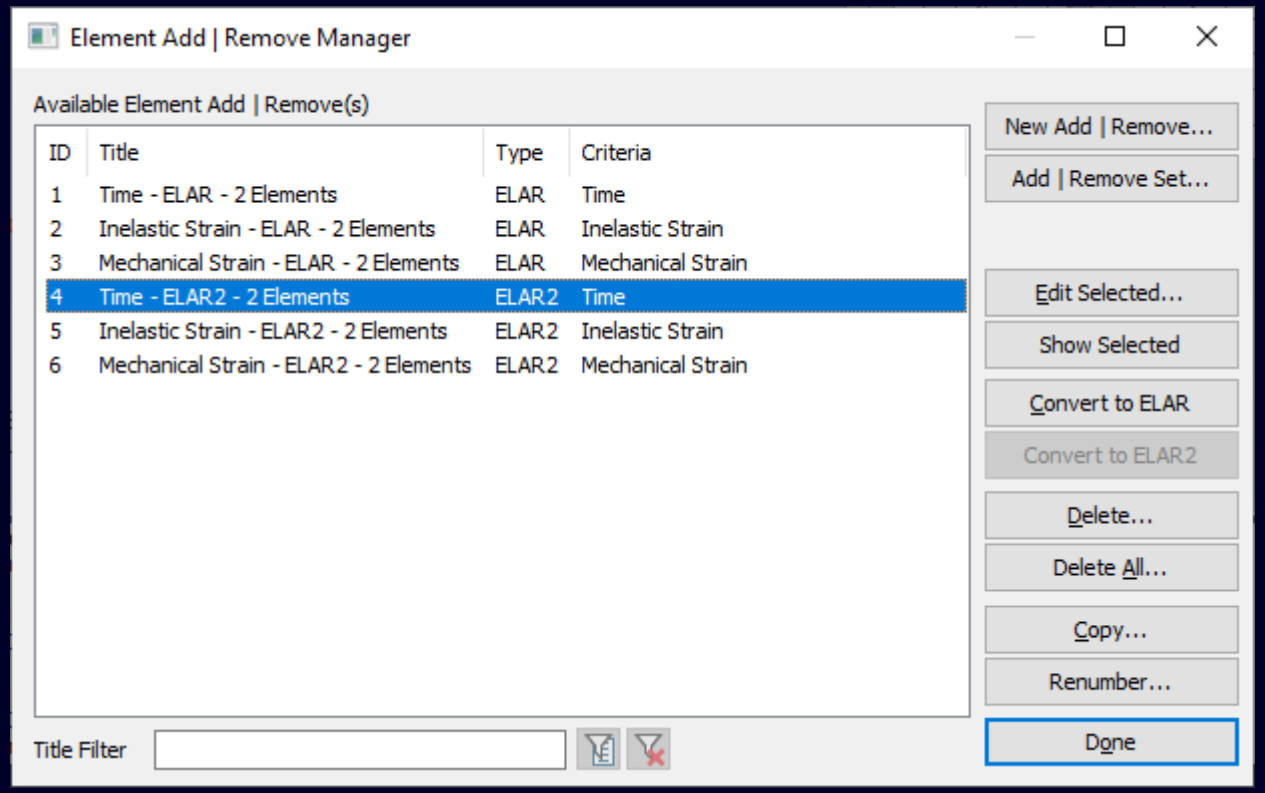
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Preprocessing – Element Add | Remove

Element Add | Remove Manager

- *New Add* | *Remove* – Creates ELAR or ELAR2 entity
- *Add* | *Remove Set* – Creates ELARADD from existing entities
- *Edit Selected*
- *Show Selected*
- *Convert to ELAR* – Creates multiple ELARs from ELAR2
- *Convert to ELAR2* – Coverts ELAR(s) into ELAR2(s)
- *Delete* and *Delete All*
- *Copy*
- *Renumber*



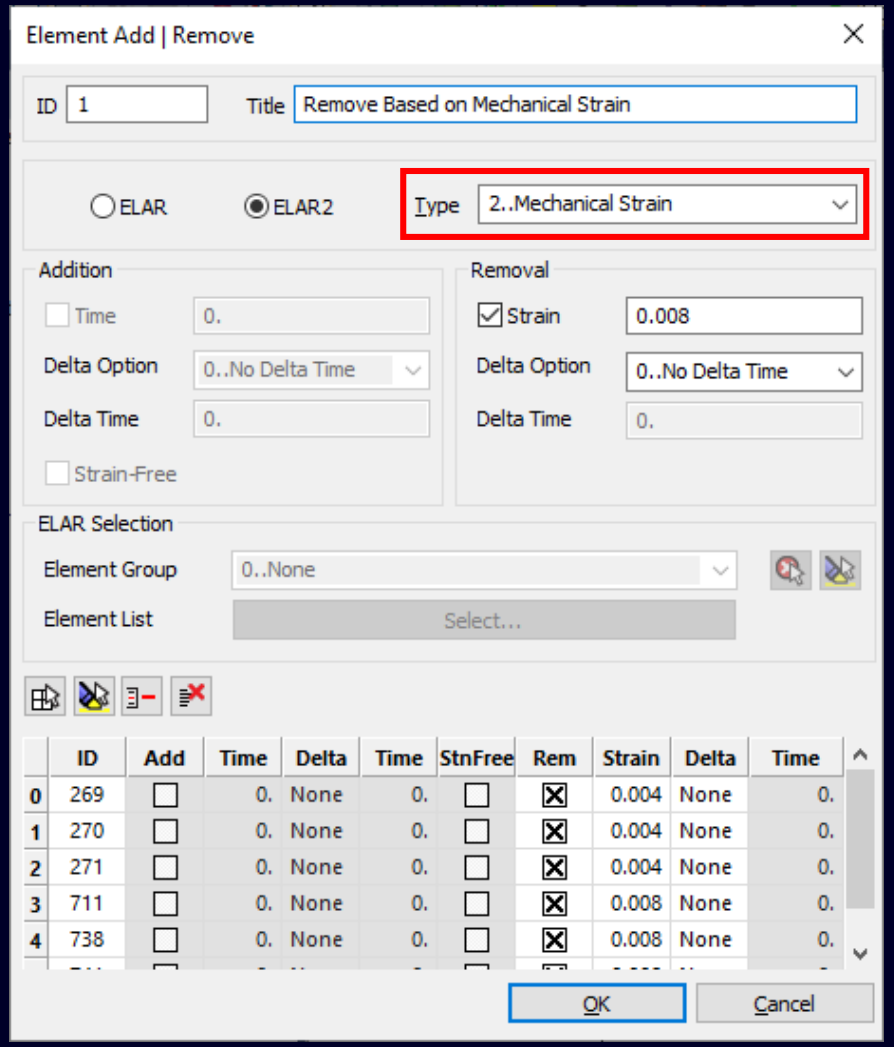
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Preprocessing – Element Add | Remove

Available criteria types for Element Add | Remove

- *Time* – Available for Addition or Removal
- *Inelastic Strain* – Available for Removal only
- *Mechanical Strain* – Available for Removal only



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

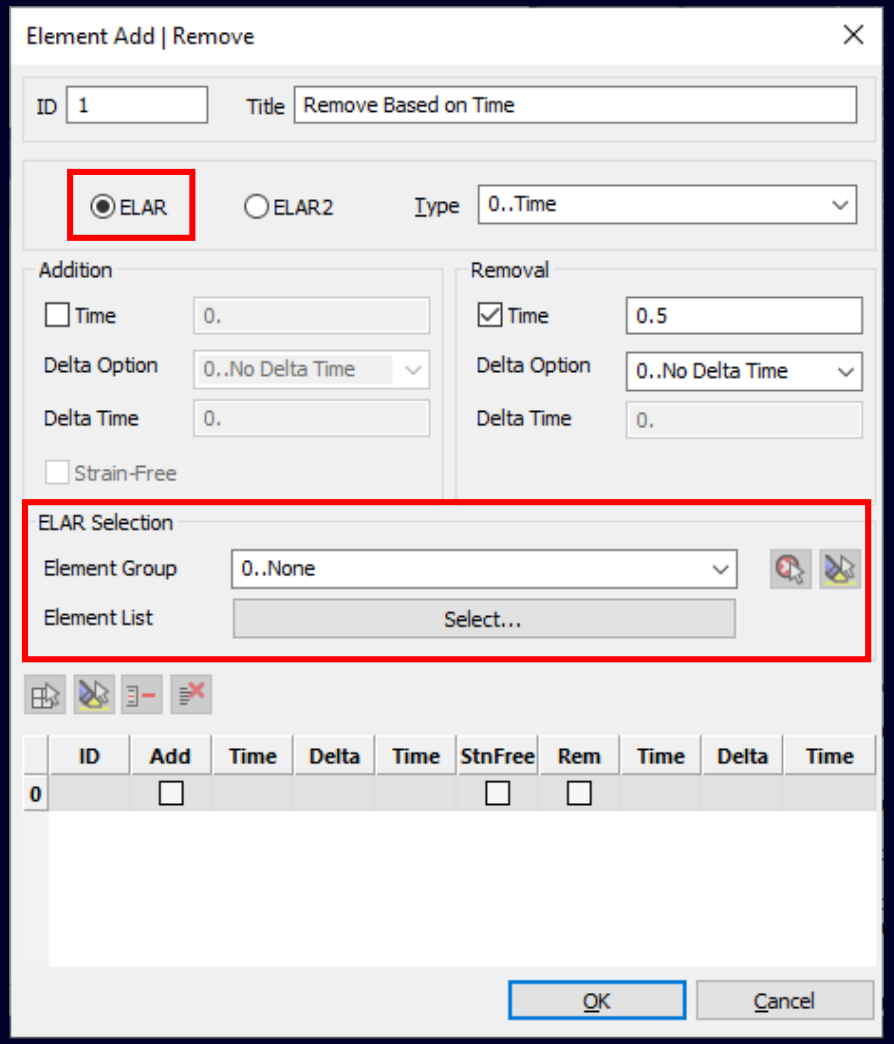
Simcenter Femap 2301
Preprocessing – Element Add | Remove

Available criteria types for Element Add | Remove

- *Time* – Available for Addition or Removal
- *Inelastic Strain* – Available for Removal only
- *Mechanical Strain* – Available for Removal only

Two options to create Element Add | Remove entity

- *ELAR* – Selected elements all use same value
  - Elements specified by choosing existing Group or by using Select button



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

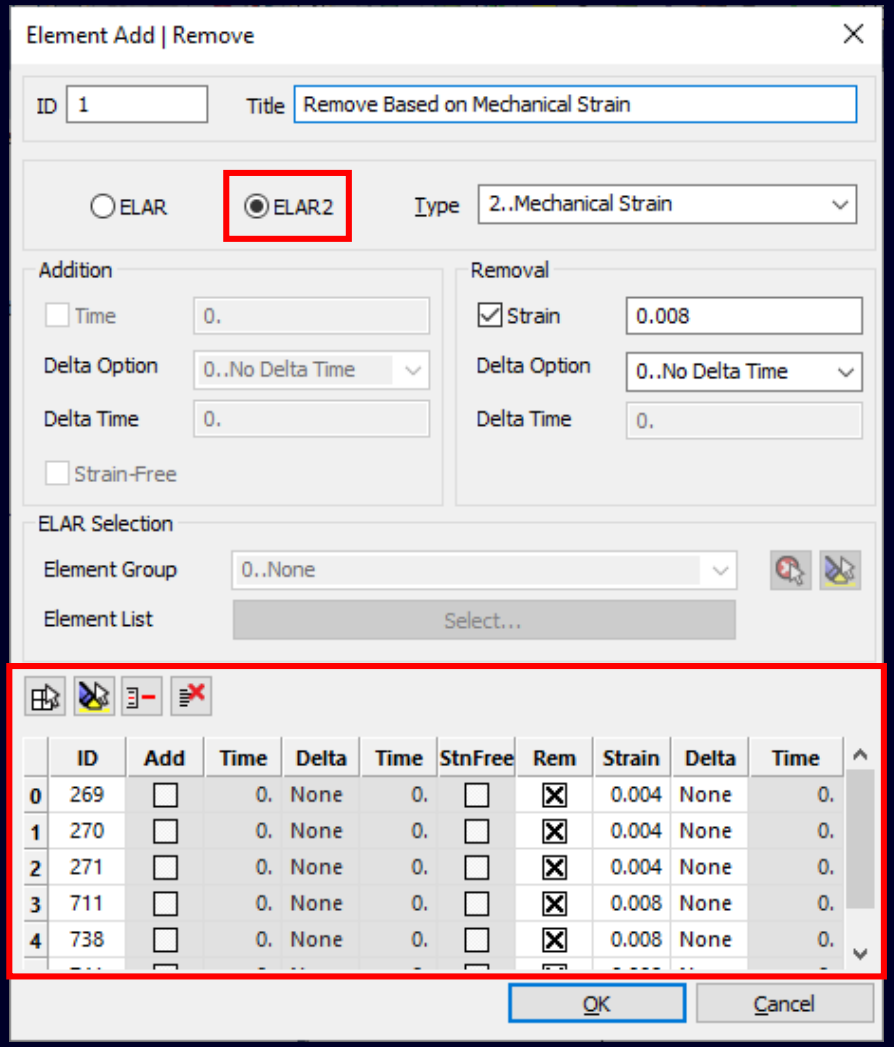
Simcenter Femap 2301
Preprocessing – Element Add | Remove

Available criteria types for Element Add | Remove

- *Time* – Available for Addition or Removal
- *Inelastic Strain* – Available for Removal only
- *Mechanical Strain* – Available for Removal only

Two options to create Element Add | Remove entity

- *ELAR* – Selected elements all use same value
  - Elements specified by choosing existing Group or by using Select button
- *ELAR2* – Individual value can be specified for each element
  - Elements selected using Select Elements icon above table, with other icons available to highlight, delete, or clear table
  - Values can be updated directly in the table



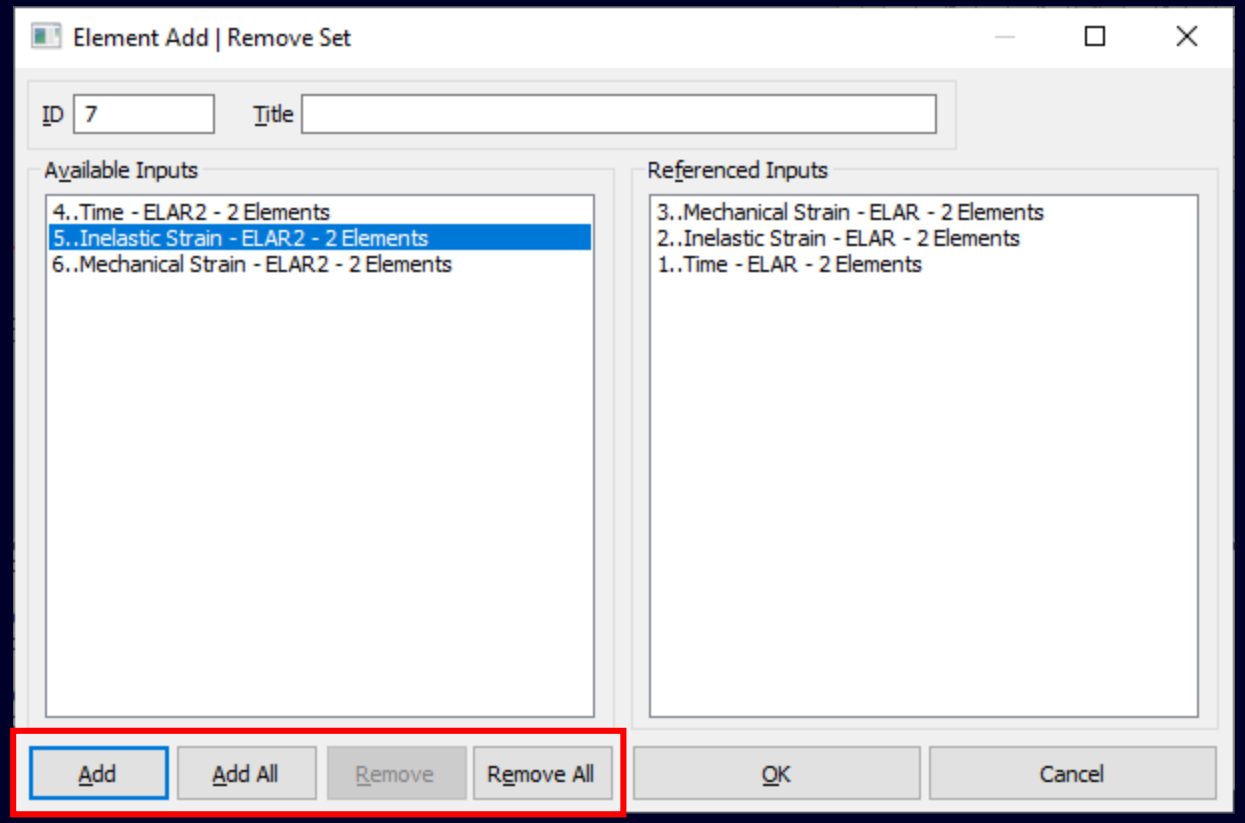
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Preprocessing – Element Add | Remove

Element Add | *Remove Set*

- Creates ELARADD
- Highlight any number of Element Add | Remove entities in *Available Inputs* and click *Add* to add them to *Referenced Inputs*
- *Remove* removes highlighted entities from *Referenced Inputs*
- *Add All* adds all *Available Inputs* to *Referenced Inputs*
- *Remove All* removes all items from *Referenced Inputs*



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

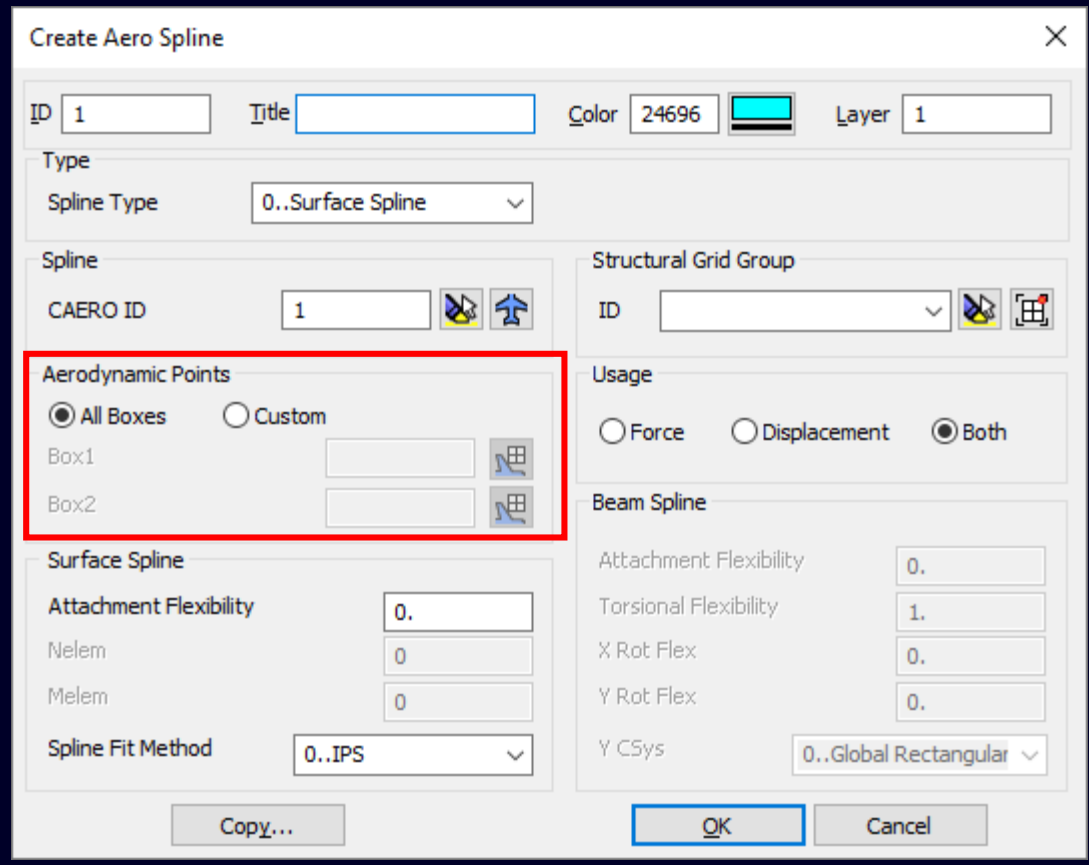
SIEMENS

Simcenter Femap 2301
Preprocessing – Aero Spline Enhancement

When choosing which Aero Boxes to include for an Aero Spline, there are now two options:

- *All Boxes* – Chooses all Aero Boxes on the Aero Panel/Body selected by *CAERO ID*
- *Custom* – Allows selection of a subset of Aero Boxes on the selected Aero Panel/Body

All Boxes allows user to update definition of an Aero Panel/Body and not have to update the Aero Spline when using all Aero Boxes on the Aero Panel/Body



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

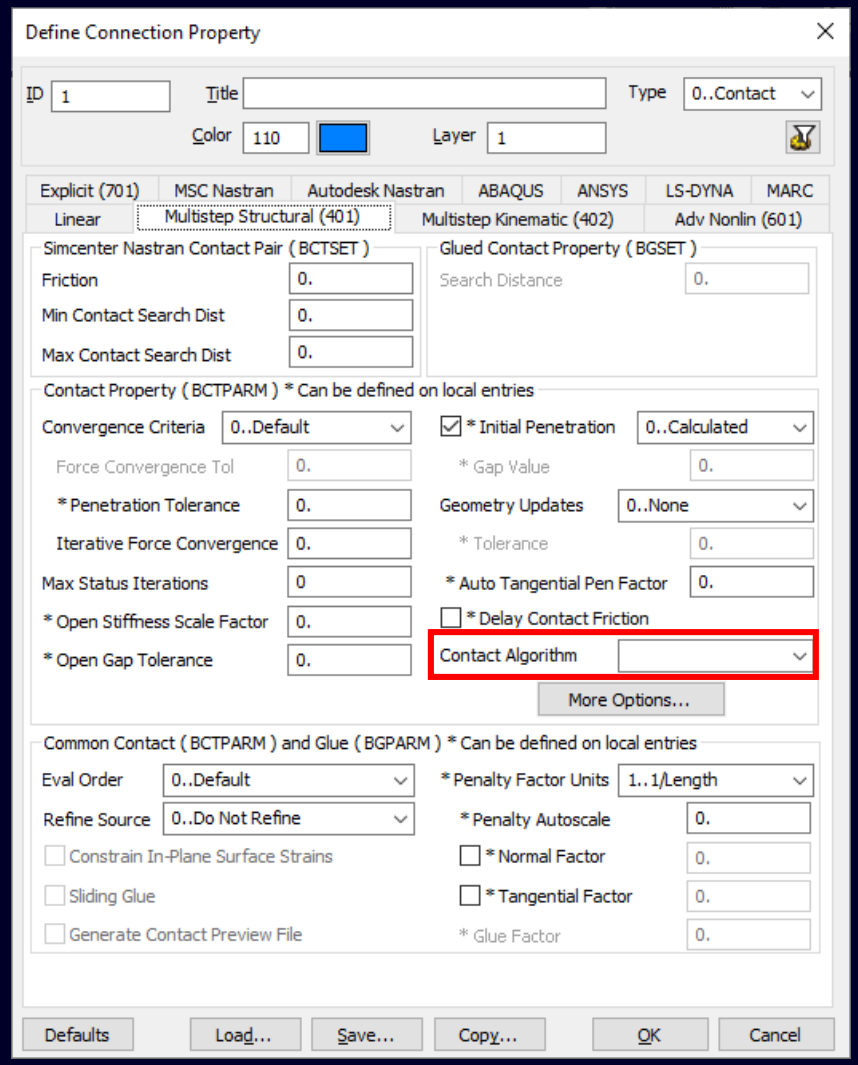
SIEMENS

Simcenter Femap 2301
Preprocessing – Connection Properties

Multistep Structural (401) tab
- *Contact Property (BCTPARM)* section
  - Added *Contact Algorithm* drop-down

These items have been removed as they are no longer used
- *Contact Property (BCTPARM)* section
  - **Closed Gap Tolerance (GAPTOL)*
  - **No Separation (NOSEP)*

- Dialog box accessed by *More Options…* button
  - *Stiffness Stabilization (KSTAB)*



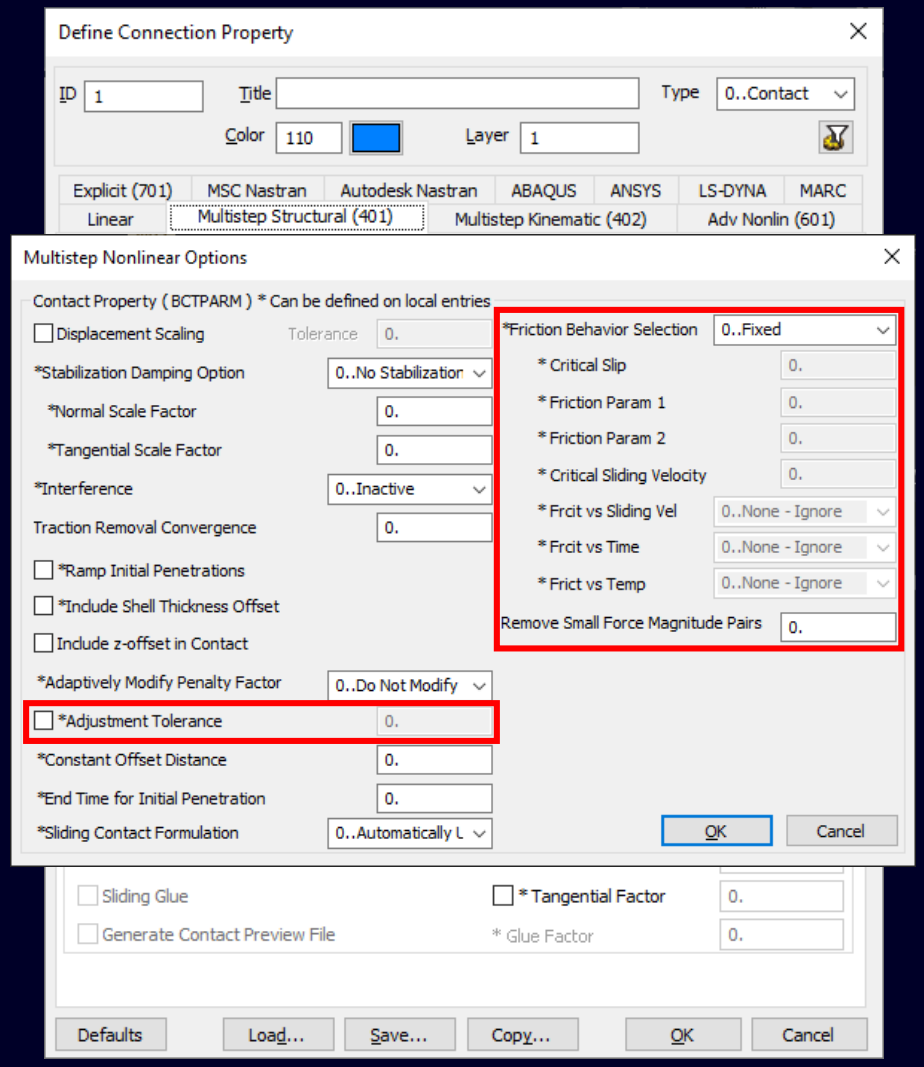
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Preprocessing – Connection Properties

MultiStep Structural (401) tab

- Dialog box accessed by *More Options…* button, Added:
  - **Friction Behavior Selection* drop-down
  - **Critical Slip* value
  - **Friction Param 1* value
  - **Friction Param 2* value
  - **Critical Sliding Velocity* value
  - **Frict vs Sliding* Velocity drop-down
  - **Frict vs Time* drop-down
  - **Frict vs Temp Velocity* drop-down
  - *Remove Small Force Magnitude Pairs* value
- Updated:
  - **Adjustment Tolerance (ADJUST)* - can now be toggled and also accepts negative values



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

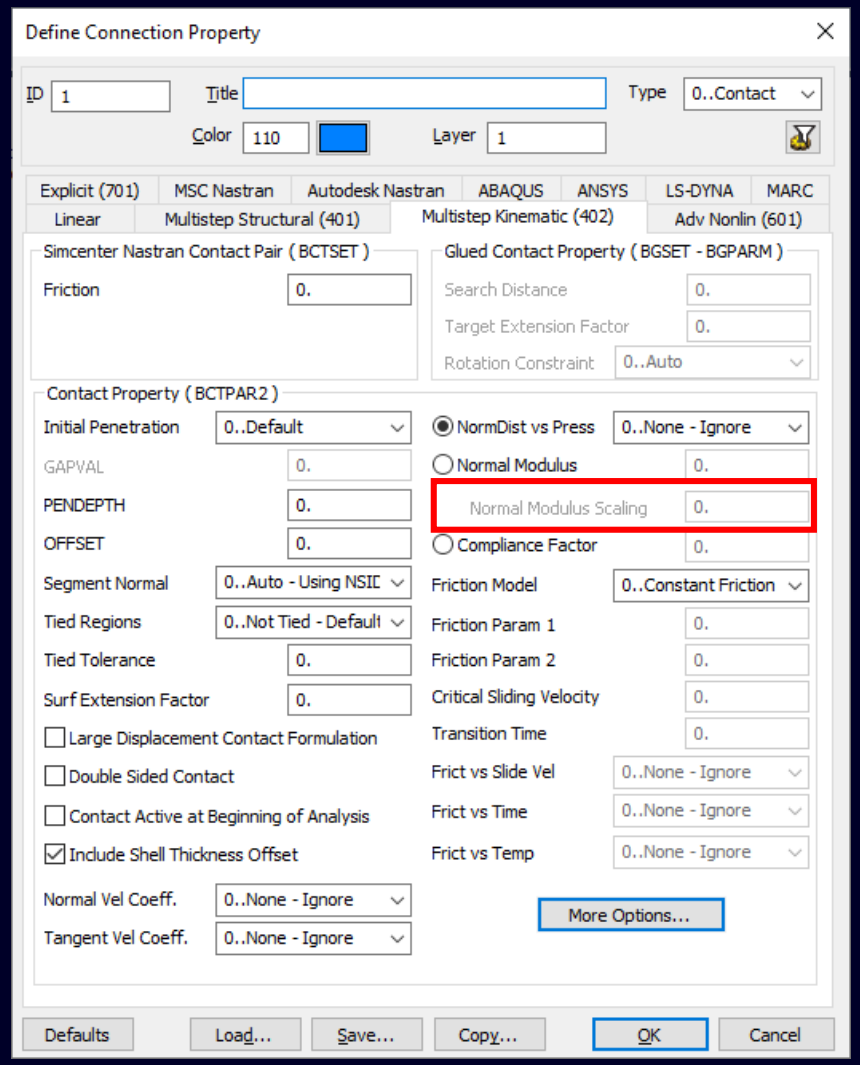
Simcenter Femap 2301
Preprocessing – Connection Properties

Multistep Kinematic (402) tab

- *Contact Property (BCTPAR2)* section
  - Added *Normal Modulus Scaling* value

- *More Options…* button added to accommodate new options

- Moved from Contact Property (BCTPAR2) section to dialog box accessed by *More Options…* button
  - *Regularization Models* drop-down and *Value* value (also added *Tangential Contact Stiffness Scale* value)
  - *Normal Reg. Type* drop-down and *Value* value



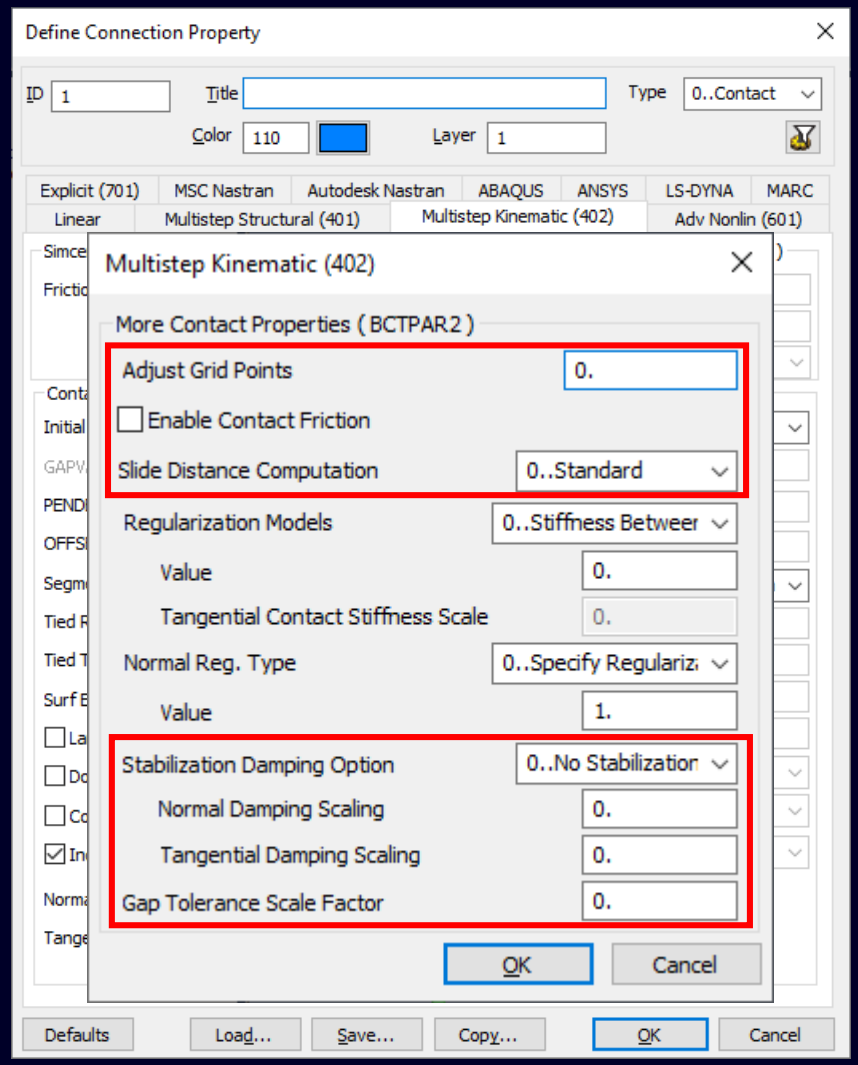
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Preprocessing – Connection Properties

MultiStep Kinematic (402) tab

- Dialog box accessed via *More Options…* button, Added:
  - *Adjust Grid Points* value
  - *Enable Contact Friction* Boolean
  - *Slide Distance Computation* drop-down
  - *Stabilization Damping Option* drop-down
  - *Normal Damping Scaling* value
  - *Tangential Damping Scaling* value
  - *Gap Tolerance Scale Factor* value



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Preprocessing – Groups

Updated **Group -> Operations -> Condense**

- Always updates group to use “ID rules” but now condensed group only contains IDs of existing entities

Added **Group -> Operations -> Reduce to Existing**

- Works like **Group -> Operations -> Condense** but it updates each rule type to contain its existing entities
- Does not condense the rules down to “ID rules” so user retains the same functionality of the Group

Updated **Group -> Operations -> Automatic Add**

- Now prevents adding entities that are being renumbered to new IDs
- Previously they were added because a new ID was being created even though it was not a new entity

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

Simcenter Femap 2301
Preprocessing – Miscellaneous

Updated how Constraint Equations are shown in the *Data Table*

- Previously, Nodes with multiple DOF did not show the Node ID with each DOF, which caused columns to be misaligned and mislabeled if user added multiple equations with differing numbers of DOF per node
- ID of the Constraint Equation within the Constraint Set is now shown as the ID instead of the selected node, as was previously done

Updated toolbar commands that show multiple different icons so that all instances of those commands are correctly updated if user places such a command on existing or custom toolbars

Added “8..Fluid” and “7..Other” as material types which can be selected in the standard entity selection dialog box when *Method^* is set to *Type* and are also now selected when using *Select All* when *Method^* is set to *Type*.

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

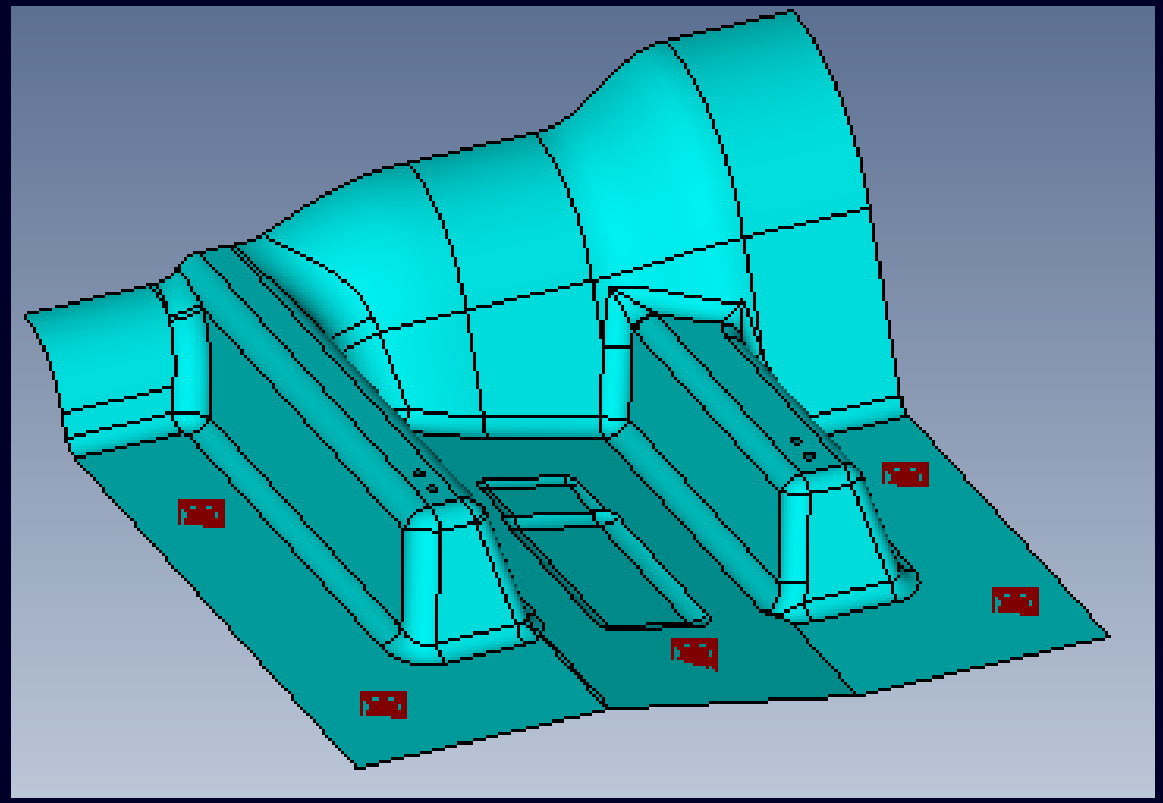
SIEMENS

Simcenter Femap 2301
Meshing – Body Mesher Enhancements

Mesh Points (hard points) created via the Mesh Point Editor are now recognized by Body Mesher both when using the **Mesh -> Bodies** command or when using the **Mesh -> Geometry -> Surface** command with Mesher set to *Body/on Mesh*

Midside Nodes tab added to dialog box of **Mesh -> Bodies** command to allow projection of midside nodes toward associated surfaces and curves, limited by user-defined distortion angle

Mesh Points specified on Geometry

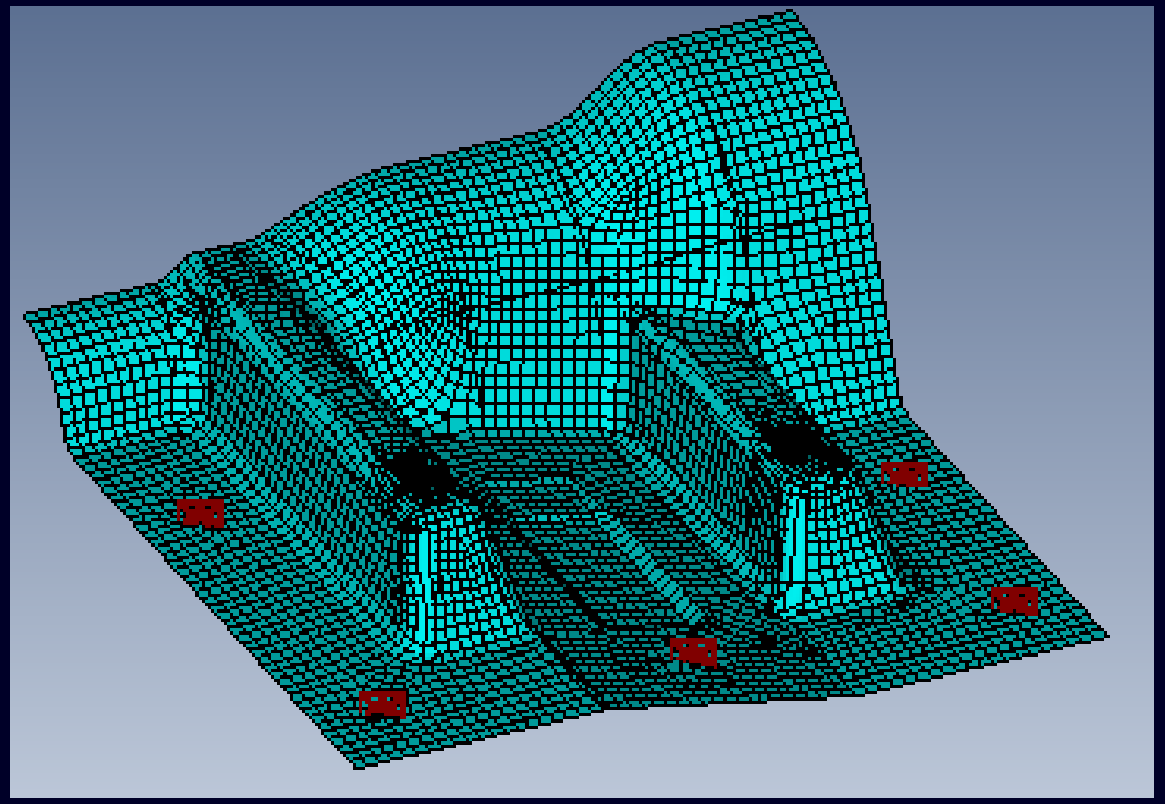


© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

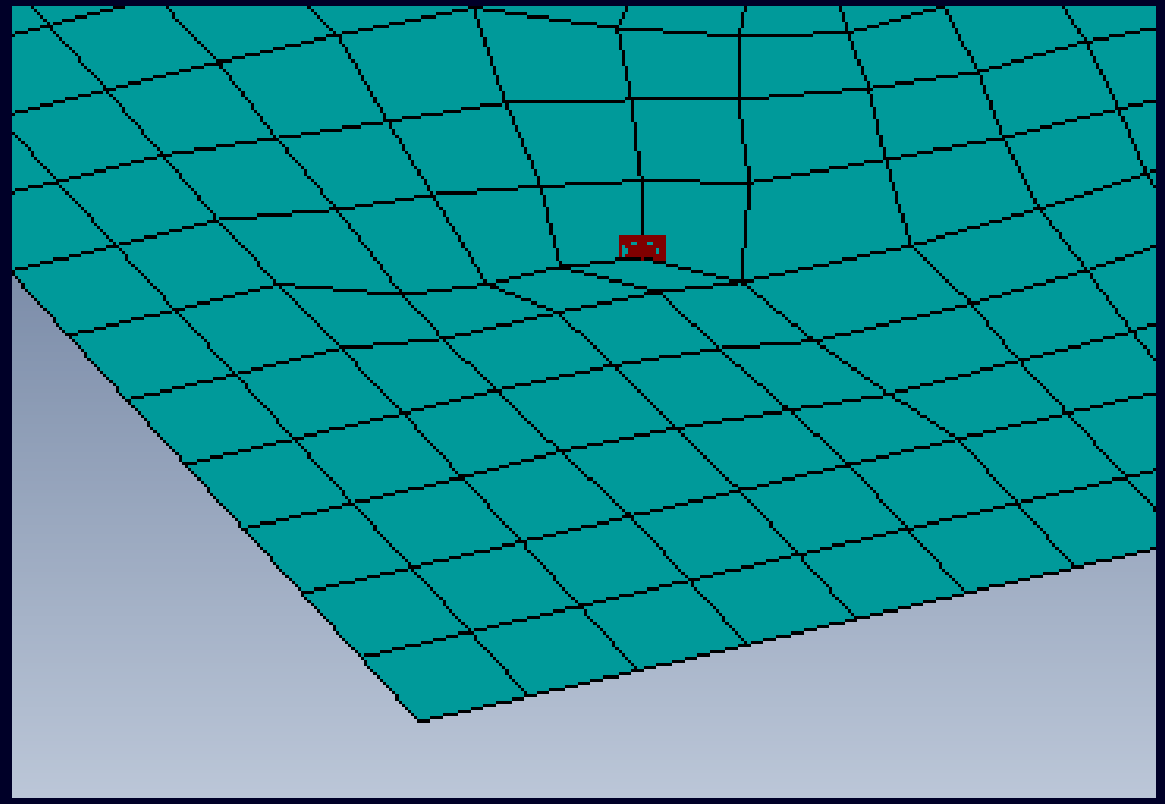
SIEMENS

Simcenter Femap 2301
Meshing – Body Mesher Enhancements

Mesh Points recognized by Body Mesher



Zoomed in on single mesh point



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

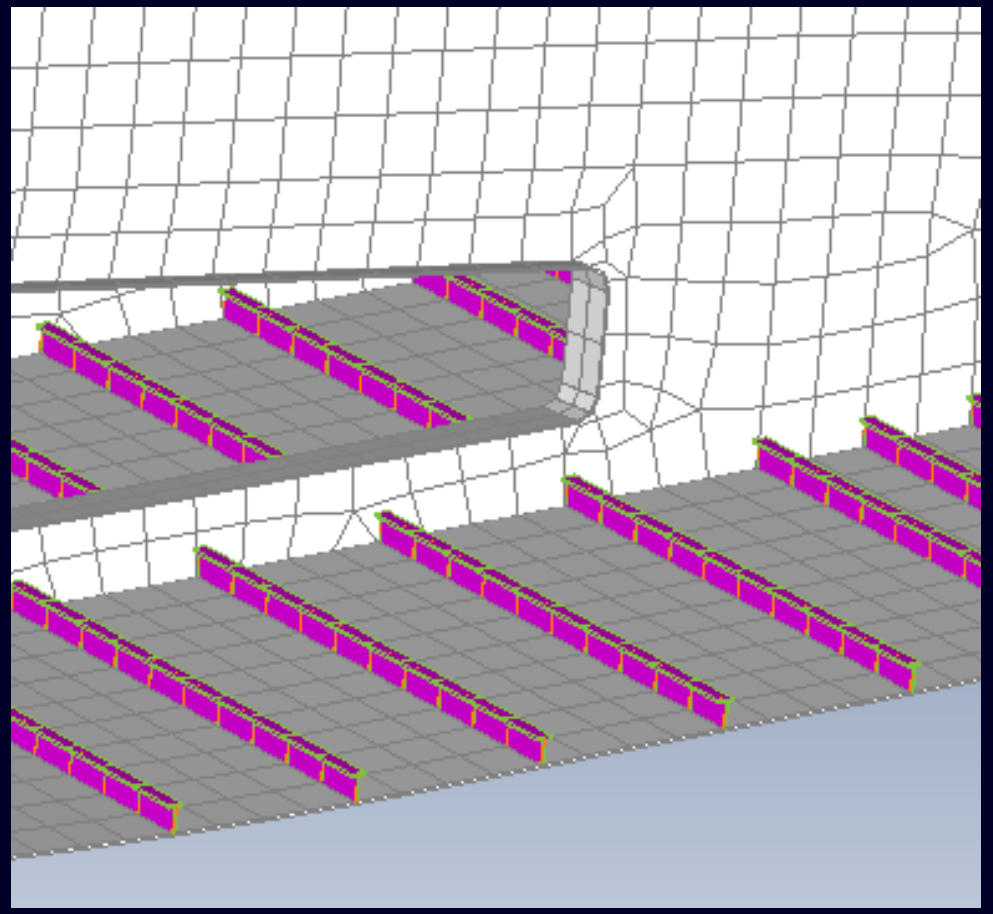
SIEMENS

Simcenter Femap 2301
Meshing – Mesh -> Mesh on Mesh Enhancements

Line elements connected to other element types can now be updated when using **Mesh -> Mesh on Mesh** command

- Allows mesh containing line element stiffeners to be completely refined or un-refined
- *Line Elements* tab added which controls if adjacent line elements should be treated together or individually during remeshing based on a user-defined *Orientation/Offset Deviation Limit Angle Along Edge* value

Original Model



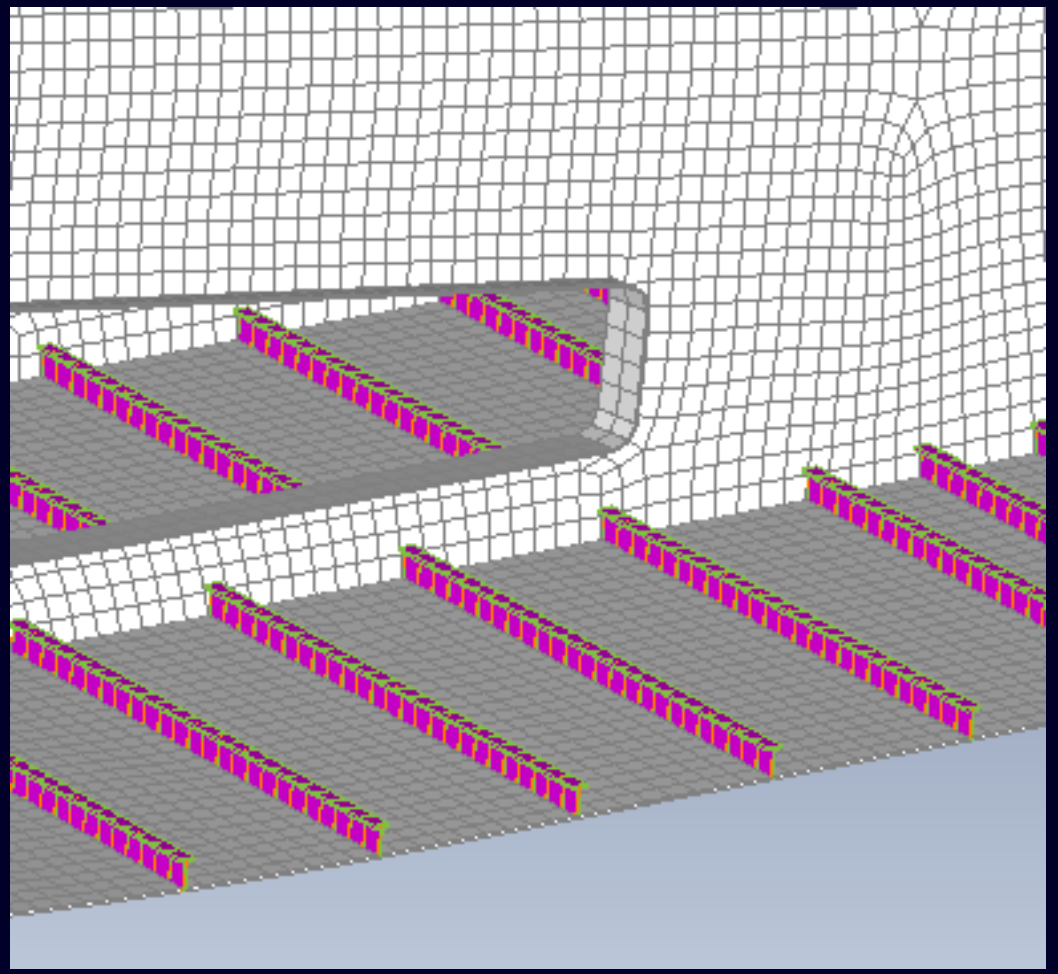
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

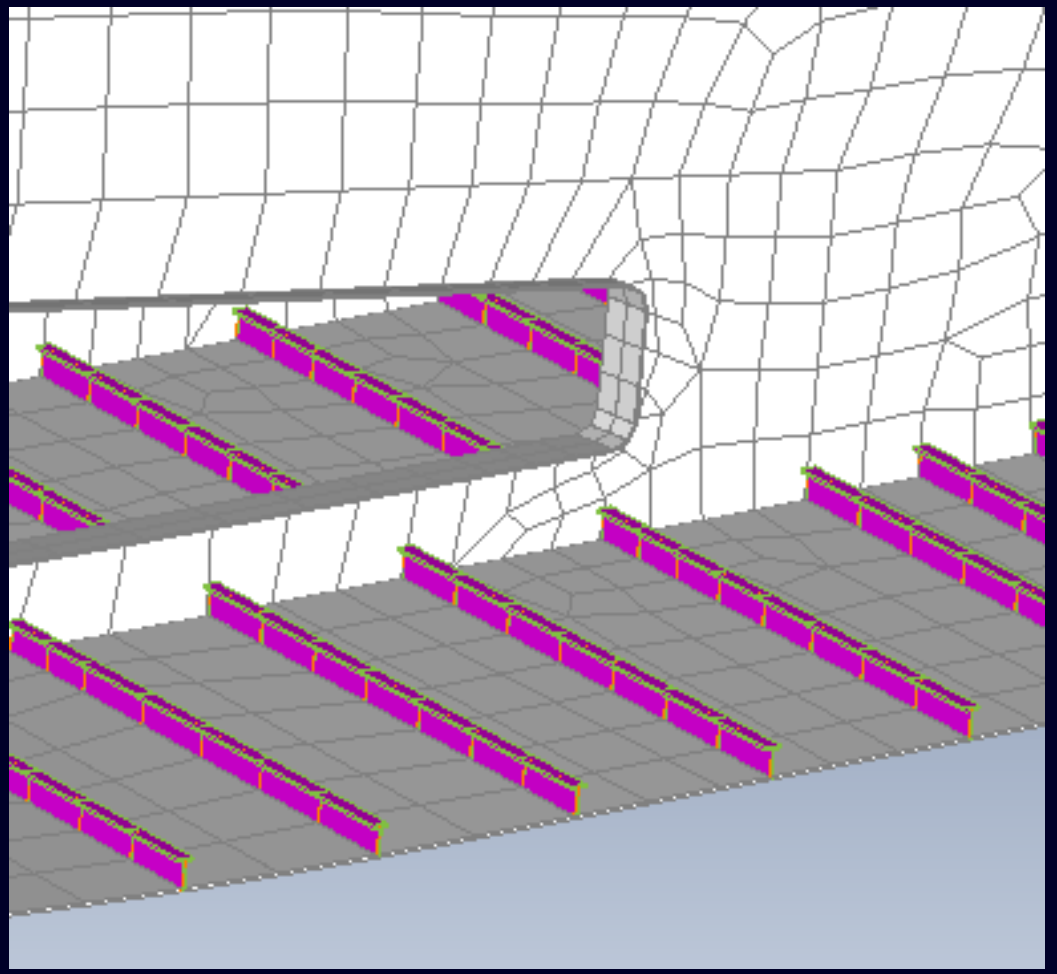
Simcenter Femap 2301

Meshing – Mesh -> Mesh on Mesh Enhancements

Refined Mesh



Un-refined Mesh



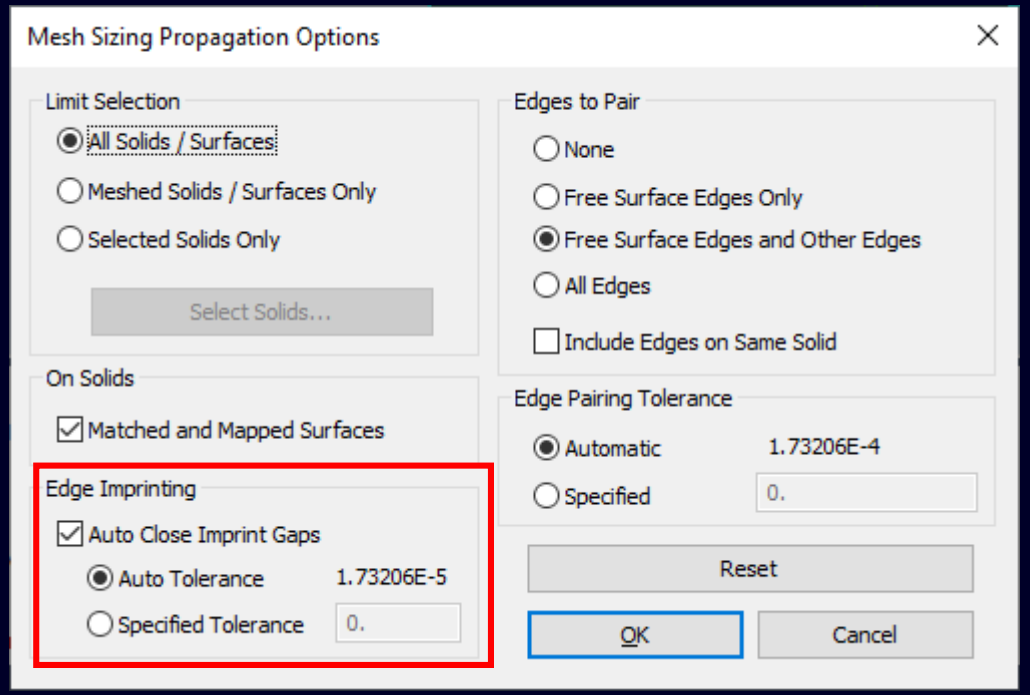
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Meshing – Mesh Sizing Propagation Options

Added *Edge Imprinting* section to *Mesh Sizing Propagation Options* dialog box accessed by various commands on **Mesh -> Mesh Sizing** menu or via icon in *Mesh Control Explorer* pane

- Added *Auto Close Imprint Gaps* option – when on, will extend projected imprints to closest edge when distance from end of imprinted curve to closest edge is within tolerance
- Two options for tolerance
  - *Auto Tolerance* – uses default “Merge Tolerance” (Length of “Model Box Diagonal”/10,000) divided by 10
  - Specified Tolerance – uses value supplied by user



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Meshing – Miscellaneous

Updated **Mesh -> Geometry Preparation** command

- Added confirmation question and warning message if *Suppress Internal Voids* option is selected and any void is much larger than the specified mesh size
- Intended to prevent accidental suppression of large void regions like the inside of hollow tubes with closed ends

Updated **Mesh -> HexMesh Bodies** command to better match specified mesh sizes in very specific case where *Size Surface if All Curves Sized* option is turned on, all curves on either a required surface are sized or *All Sized Curves* option is turned on, and curve sizing is close to *Target Element Size* value

Enhanced the **Mesh -> Editing -> Cohesive Meshing** command to properly support inserting Cohesive elements at a location where some portions of the mesh could be missing adjacent elements over part or all of the area.

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

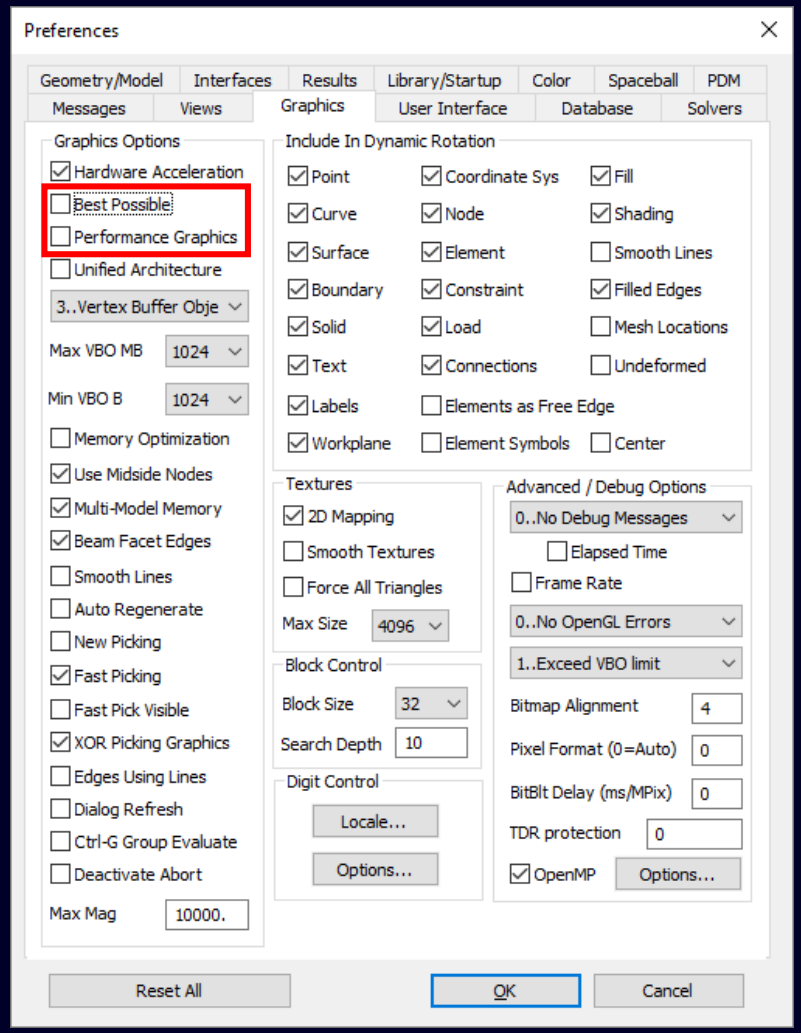
SIEMENS

Simcenter Femap 2301
Performance Improvements – Unified Graphics Architecture

Currently, Femap contains two different “pipelines” for graphics: “Performance Graphics” and “original OpenGL”

When the *Best Possible* or *Performance Graphics* option is turned on in **File -> Preferences**, everything that is supported by “Performance Graphics” is drawn using more modern OpenGL technology and everything which is not supported is drawn using “original OpenGL”

The drawback to having two different “pipelines” is that some entity types only appear “as expected” in the graphics window in “original OpenGL”, as line style, line width, fill style, and transparency are not supported by “Performance Graphics”



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

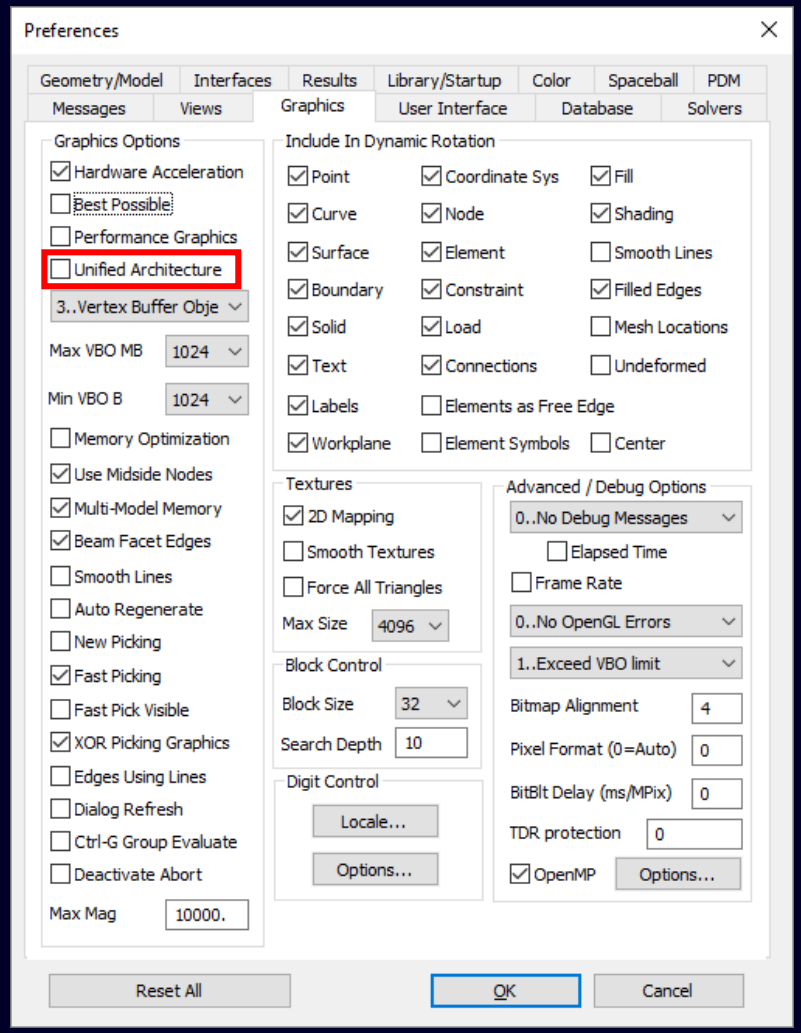
SIEMENS

Simcenter Femap 2301
Performance Improvements – Unified Graphics Architecture

In an ongoing effort to improve graphics performance across all aspects of Femap a new Unified Graphics Architecture (UGA), which uses a single “pipeline”, will be implemented over the next few releases

When fully implemented, UGA will provide a better experience for the user in quality, performance, and hardware support and both the “Performance Graphics” and “original OpenGL” pipelines will be removed

In addition, UGA will support line style, line width, fill style, and transparency along with being able to better support users using Intel graphics



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

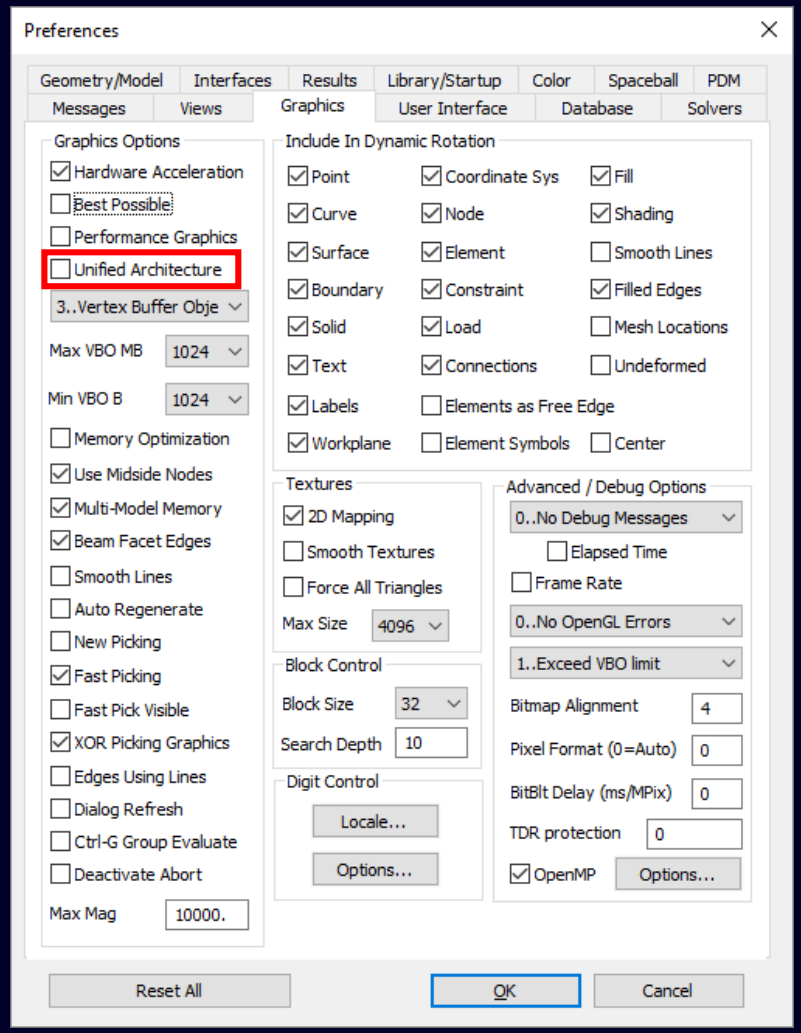
SIEMENS

Simcenter Femap 2301
Performance Improvements – Unified Graphics Architecture

UGA is turned on by default but can be turned off using *Unified Architecture* option on the *Graphics* tab of **File -> Preferences**

The following entity types are currently supported:

- Coordinate Systems
- Geometric Entities
  - Points
  - Mesh Points
  - Curves
  - Composite Curves
  - Surfaces
  - Boundary Surfaces
  - Solids
  - Volumes (may be removed in future version)
- Geometry-based Loads and Constraints



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Performance Improvements – Miscellaneous

Improved the performance of adding large numbers of Nodal, Elemental and Geometric Loads and Geometric Constraints to the *Data Table*

Improved performance adding and updating columns in Data Table API object

- One example of adding a Freebody with over 800,000 lines of data went from around 19 minutes to 34 seconds (33X improvement)

Improved the performance of loading certain facetted geometry.

- One model with complex surfaces showed an almost 50% speed improvement, in another about 27%.

Improved the performance of updating a mesh in the *Meshing Toolbox* when the model contained rigid elements and *Propagate Sizing* option is turned off

- Previously, unnecessary work was being done to find adjacent edges even though those would never be used

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

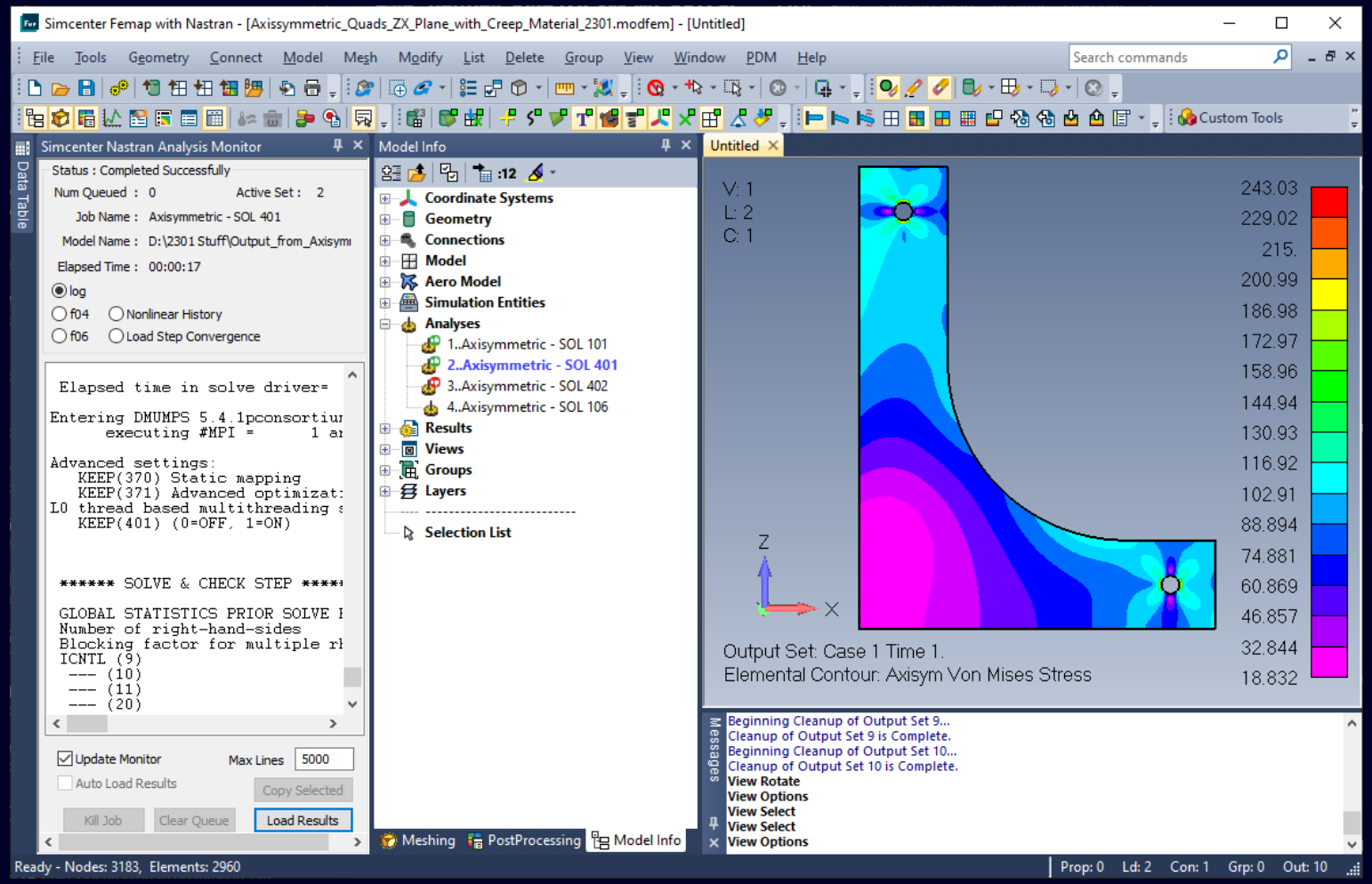
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301

Analysis and Solver Support – Analysis Monitoring Improvements

Several enhancements have been made to monitoring and recovering results from analyses which have been launched from within Femap



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

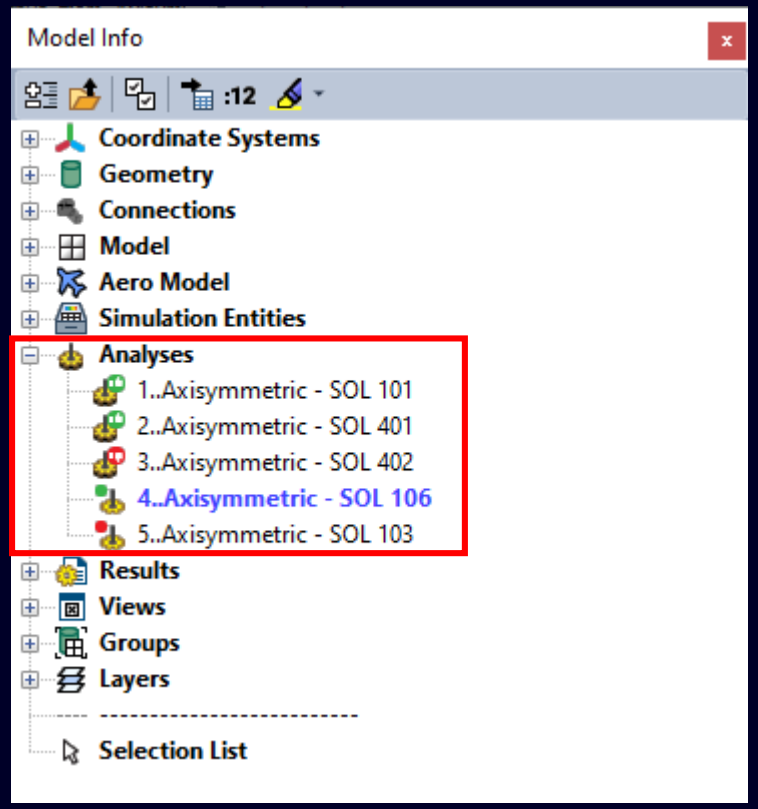
SIEMENS

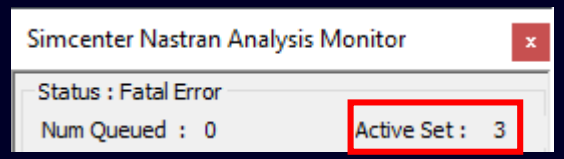
Simcenter Femap 2301
Analysis and Solver Support – Analysis Monitoring Improvements

Improved monitoring when analyzing multiple Analysis Sets at once

- Icon in *Model Info* tree to indicate if analysis was successful
  - Green “Square” – Analysis is running
  - Red “Square” – Analysis in analysis queue
  - Green “Thumbs Up” – Analysis completed successfully
  - Red “Thumbs Down” – Analysis did not complete successfully

- *Analysis Monitor* pane updates based on “active” Analysis Set in *Model Info* tree
  - “Active Set” shown in *Analysis Monitor* and “Load Results” loads results for active set





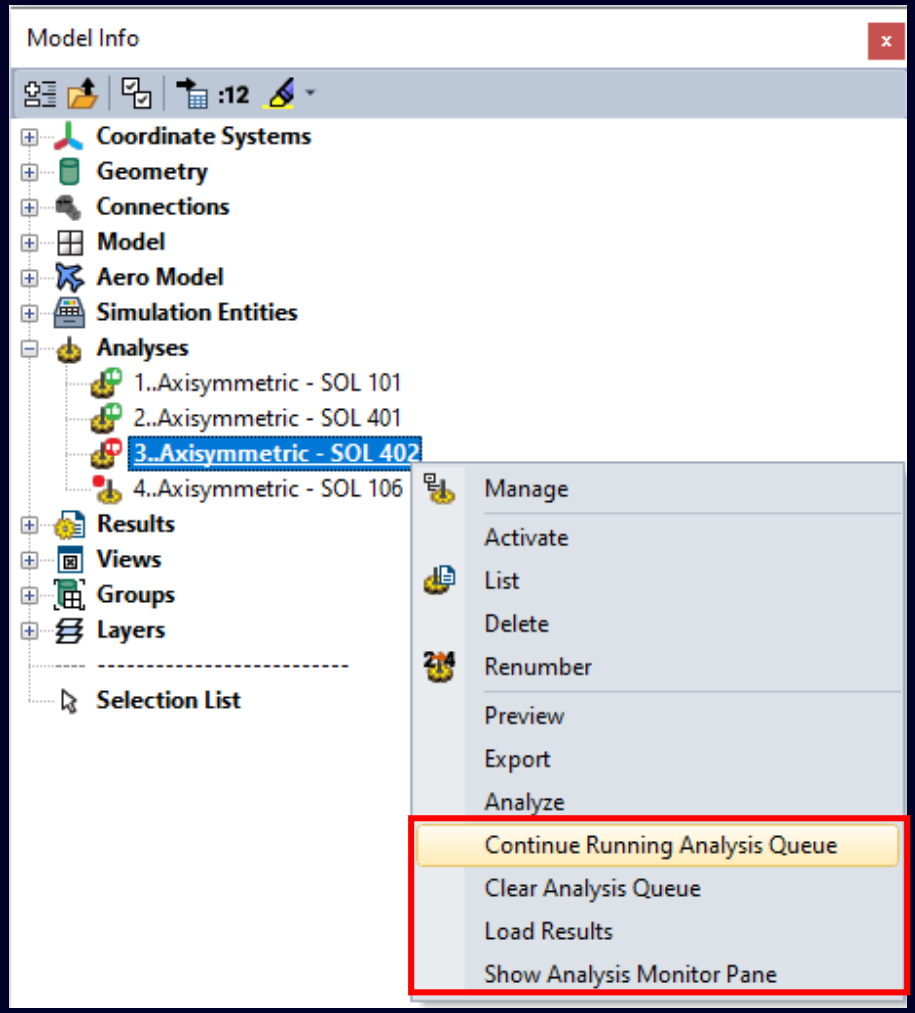
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Analysis and Solver Support – Analysis Monitoring Improvements

Context-Sensitive menu now offers:

- *Continue Running Analysis Queue* – offers ability to run next job in analysis queue if previous job failed
- *Clear Analysis Queue* – clears out the analysis queue (was available in previous versions, but has been enhanced to work with new features)
- *Load Results* – loads results from selected analysis set(s)
- *Show Analysis Monitor Pane* – opens the Analysis Monitor if it has been closed



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

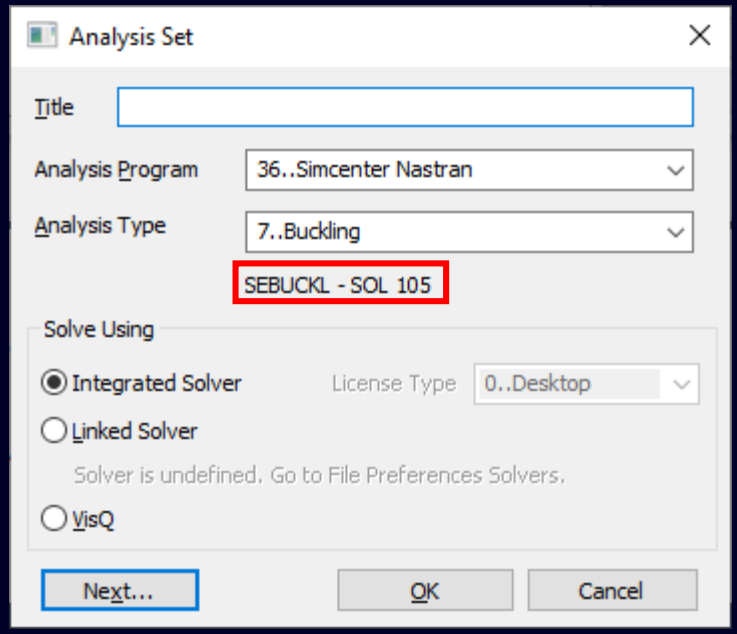
SIEMENS

Simcenter Femap 2301
Analysis and Solver Support – Analysis Set Manager Enhancement

In the *Analysis Set Manager*, a brief description of the analysis which is to be run by a specific Analysis Set is now displayed in the main *Analysis Set* dialog box.

These descriptions are based on a combination of the selected *Analysis Program* and *Analysis Type* and include information which is likely to be familiar to users familiar with solver nomenclature.

For example, the description for:
Analysis Program = “36..Simcenter Nastran” and *Analysis Type* = “7..Buckling” shows a description of “SEBUCKL – SOL 105”



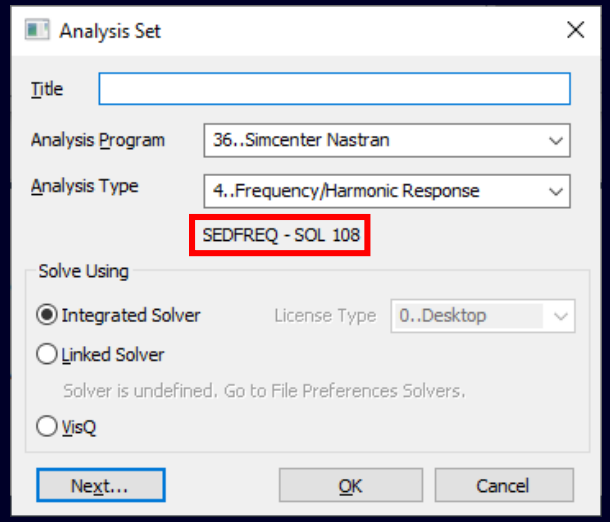
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

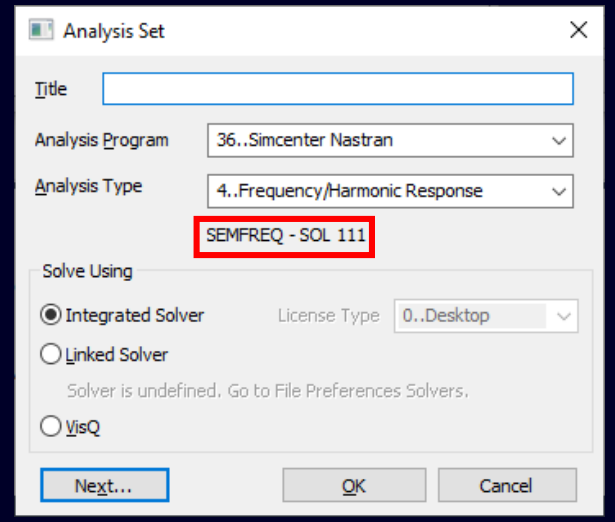
SIEMENS

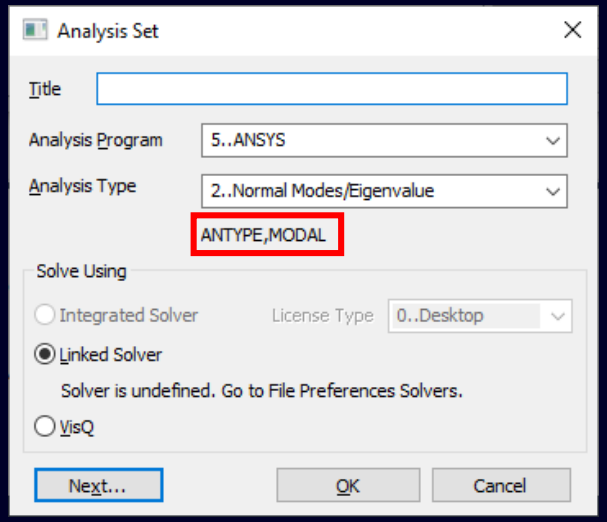
Simcenter Femap 2301
Analysis and Solver Support – Analysis Set Manager Enhancement

Additional Examples:
Analysis Program = “36..Simcenter Nastran” and *Analysis Type* = “4..Frequency/Harmonic Response” shows a description of “SEDFREQ – SOL 108” when set up for a Direct Frequency run, while it would show a description of “SEMFREQ – SOL 111” when setup for a Modal Frequency run

Analysis Program = “5..ANSYS” and *Analysis Type* = “2..Normal Modes/Eigenvalue” shows a description of “ANTYPE,MODAL”







© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

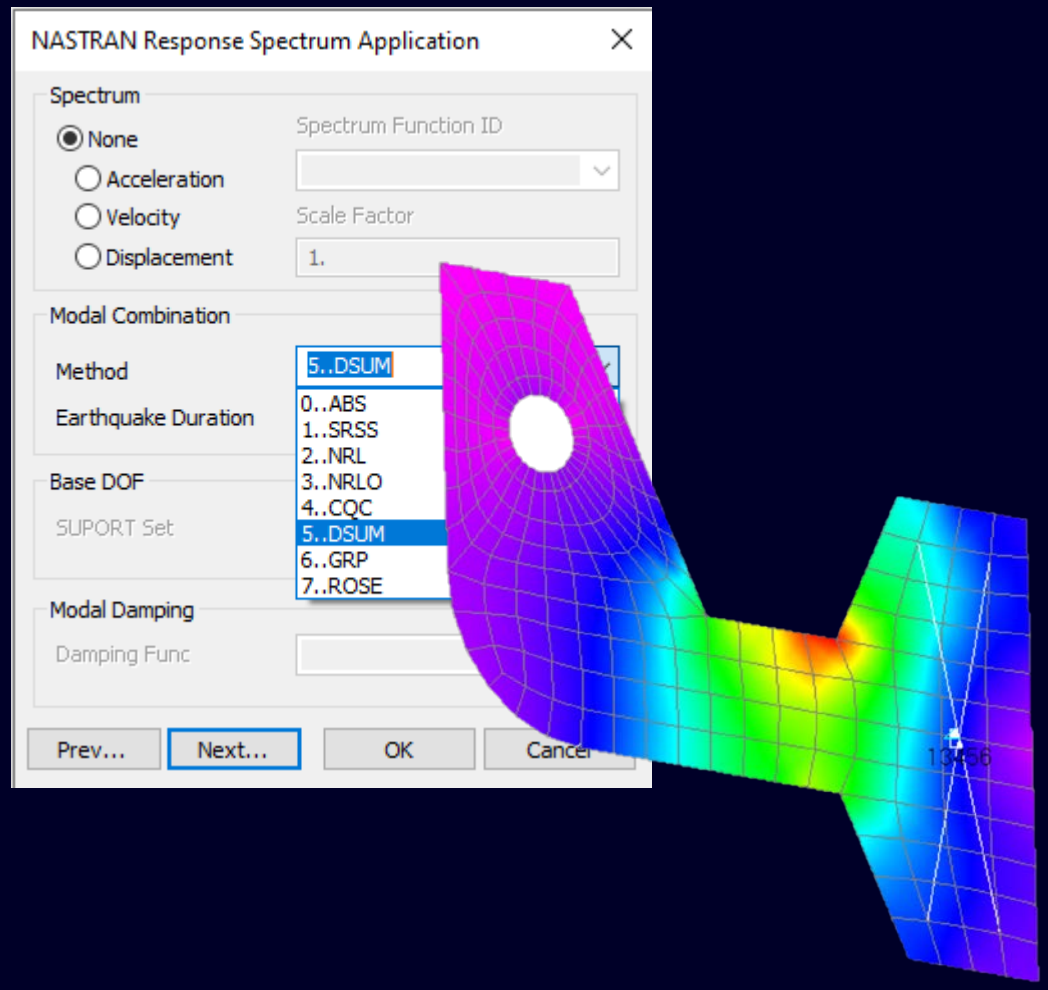
Simcenter Femap 2301
Analysis and Solver Support – Simcenter Nastran – Response Spectrum Application

Added support for Simcenter Nastran's simplified user interface for response spectrum application runs

- RSAPPLY case control command
- RSAPPLY bulk data entry
- RSPECTR bulk data entry
- RSPOPT bulk data entry

Includes support for the new modal combination methods

- *CQC*
- *DSUM* (FACTOR = *Earthquake Duration*)
- *GRP*
- *ROSE* (FACTOR = *Earthquake Duration*)



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

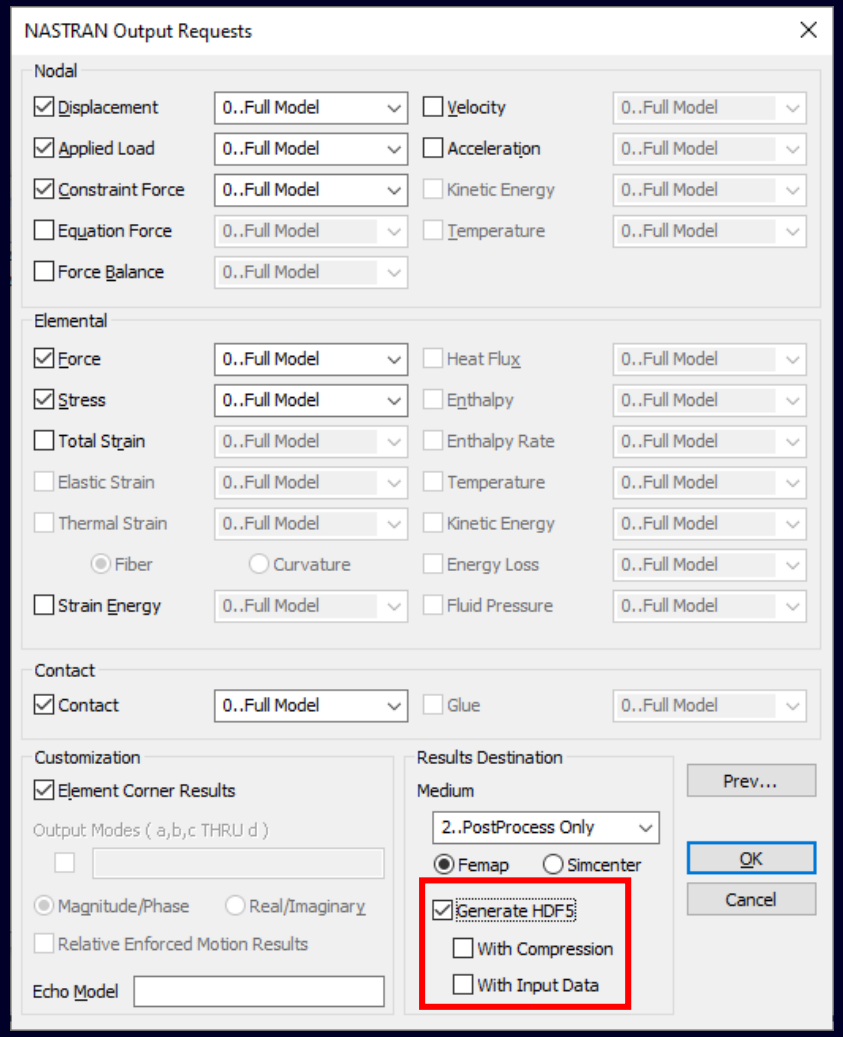
SIEMENS

Simcenter Femap 2301
Analysis and Solver Support – MSC Nastran – HDF5 Output Request

Output from MSC Nastran can now be requested in HDF5 format (NH5RDB) via the *NASTRAN Output Requests* dialog box

Depending on selected options, MDLPRM bulk data entry is written with appropriate value for HDF5 param

- *Generate HDF5* (no other options on) – writes 2
- *With Compression* turned on only – writes 3
- *With Input Data* turned on only – writes 0
- Both *With Compression* and *With Input Data* turned on – writes 1



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Analysis and Solver Support – Simcenter Nastran – SOL 401

Additions and Updates for Multi-Step Nonlinear (SOL 401) in Analysis Set Manager

- *Boundary Conditions*
  - Added *Element Add* | *Remove* drop-down to select Element Add | Remove entity or Element Add | Remove Set

- *Multi-Step Control Options*
  - Changed default for *Equilibrium Min Factor (EQMFMIN)* from to 0.2 to 0.476
  - Changed default for *Equilibrium Max Factor (EQMFMAX)* from to 5.0 to 1.9

- *Multi-Step Control Options -> Solution and Convergence*
  - Added *Always Output Results at Last Converged Step (LSTCONV)* Boolean
  - Updated *Diagnostic Level (MSGLVL)* from Boolean to drop-down to accommodate additional option

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Additions and Updates for Multi-Step Nonlinear (SOL 401) in Analysis Set Manager

- *Multi-Step Control Options -> Contact/Bolt Preload*
  - Added *Tangential Cont Stiff Options (KMODTN)* drop-down
  - Added *Modal Subcase Scale (KMODSCL)* value

- *Multi-Step Control Options -> Creep/Plasticity*
  - Added *Adjust Integration Factor (CRLIMR)* value
  - Added *Max Equivalent Plastic Strain (PLLIM)* value
  - Added *Max Plastic Strain Multiplying Factor (PLLIMF)* value

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

Additions and Updates for Multi-Step Kinematics (SOL 402) in Analysis Set Manager

- *Multi-Step Control Options*
  - Added *Storage Cycle for Grid Point Results (IA16)* value
  - Added *Storage Cycle for XY Plotting Results (IA19)* value
  - Added *Ramping Load Factor Interpolation (RFVAR)* drop-down
  - Added *Free Thermal Expansion (ITHE)* Boolean
  - Added *Laws of Excitation Storage (LL2)* value
  - Added *Max Equivalent Plastic Strain (PLLIM)* value
  - Added *Creep Strain Increment (CRICOFF)* value
  - Added *Adjust Integration Error (CRLIMR)* value
  - Update *Enable Inertial in Dynamics (INERTIA)* to drop-down to accommodate additional option
  - Changed default for *Minimum Decrease Ratio (EQMFMIN)* from 2.1 to 0.476

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

Additions and Updates for Multi-Step Kinematics (SOL 402) in Analysis Set Manager

- *Multi-Step Control Options -> Solution and Convergence Options*
  - Added *Stiffness Update (KUPDATE)* drop-down
  - Added *Max Time Step Reductions (MAXBIS)* value
  - Added *Modified Generalized Alpha Param (RHOINF)* value
  - Added *Tangential Contact Stiffness (KMODTN)* drop-down
  - Added *Diagnostic Output (MSGLVLC)* drop-down
  - Changed default for *Max Iter (ITMA)* from 10 to 20
  - Changed default for *Relative Disp Force (PRCQ)* from 1.0 to 0.1
  - Changed default for *Generalized Scheme Param (TETA)* from 0.80 to 0.55

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

Simcenter Femap 2301
Analysis and Solver Support – ABAQUS

Additions and Updates for ABAQUS

- Added support for SHELL TO SOLID CONNECTION for glued contact
- Added support for 13-node pyramid elements, which are degenerate hexahedral elements
- Added support for NASTRAN RBE3-like interpolation elements
- Updated Standard Beam Section to write out for Implicit Analysis, not just Explicit Analysis. Also corrected issues when reading standard sections from ABAQUS input file
- Updated reading of output files to read in Forces for Spring and DOF Spring elements, which are created using a STRESS request

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Additions and Updates for ANSYS

- Added the ability to read “meshed” cross-section for Beams from ANSYS input files and store them with the appropriate Beam Property so they can later be exported
- Added support to read in additional ANSYS APDL-style inputs which are often used in input files created by ANSYS Mechanical (formally ANSYS Workbench) and were not supported in previous versions

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

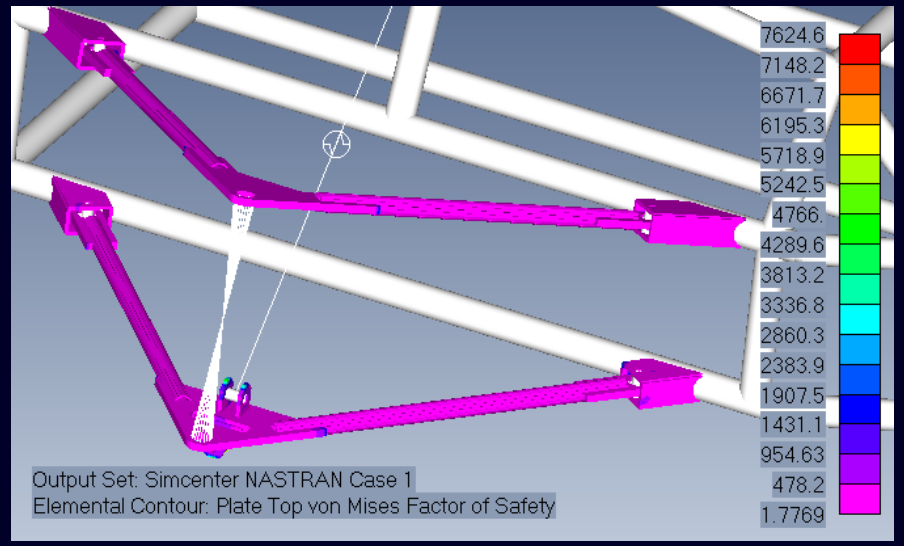
Miscellaneous and API

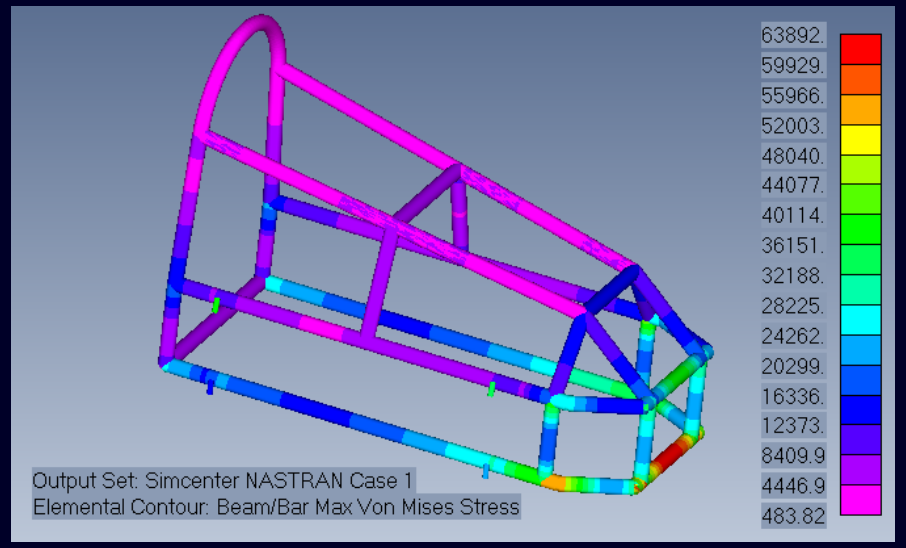
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Computed Vectors

The new Computed Vectors functionality captures additional key results data that may not be calculated by the solver via the **Model -> Output -> Computer Vectors** command





© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

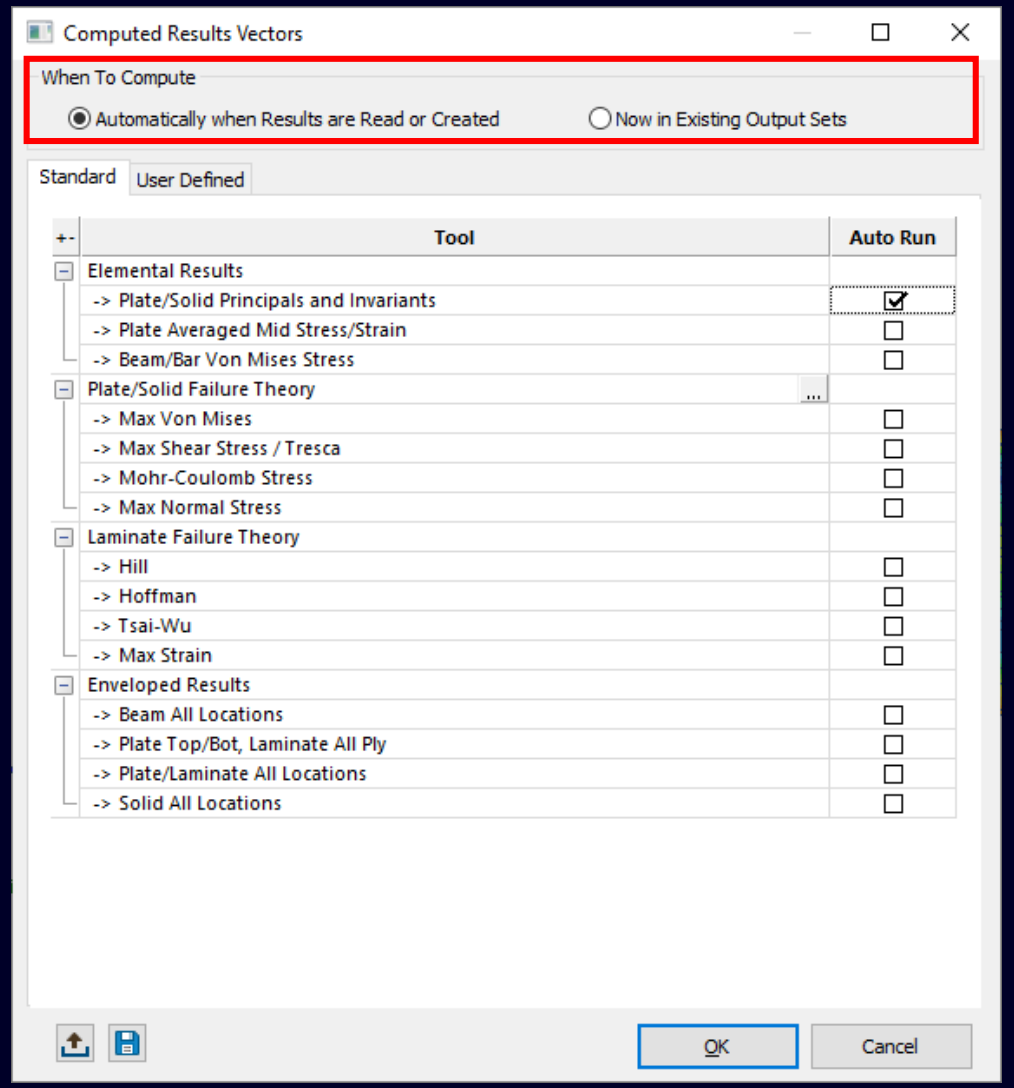
SIEMENS

Simcenter Femap 2301
Postprocessing – Computed Vectors

When To Compute (Two Options)

- *Automatically when Results are Read or Created*
  - If *Auto Run* is specified for a results quantity, appropriate output vectors are always created when results are read or otherwise created
  - Only vector with default set to *Auto Run* is *Plate/Solid Principals and Invariants*

- *Now in Existing Output Sets*
  - Check boxes in *Run Now* column, then click *Compute Now* button to compute vectors



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

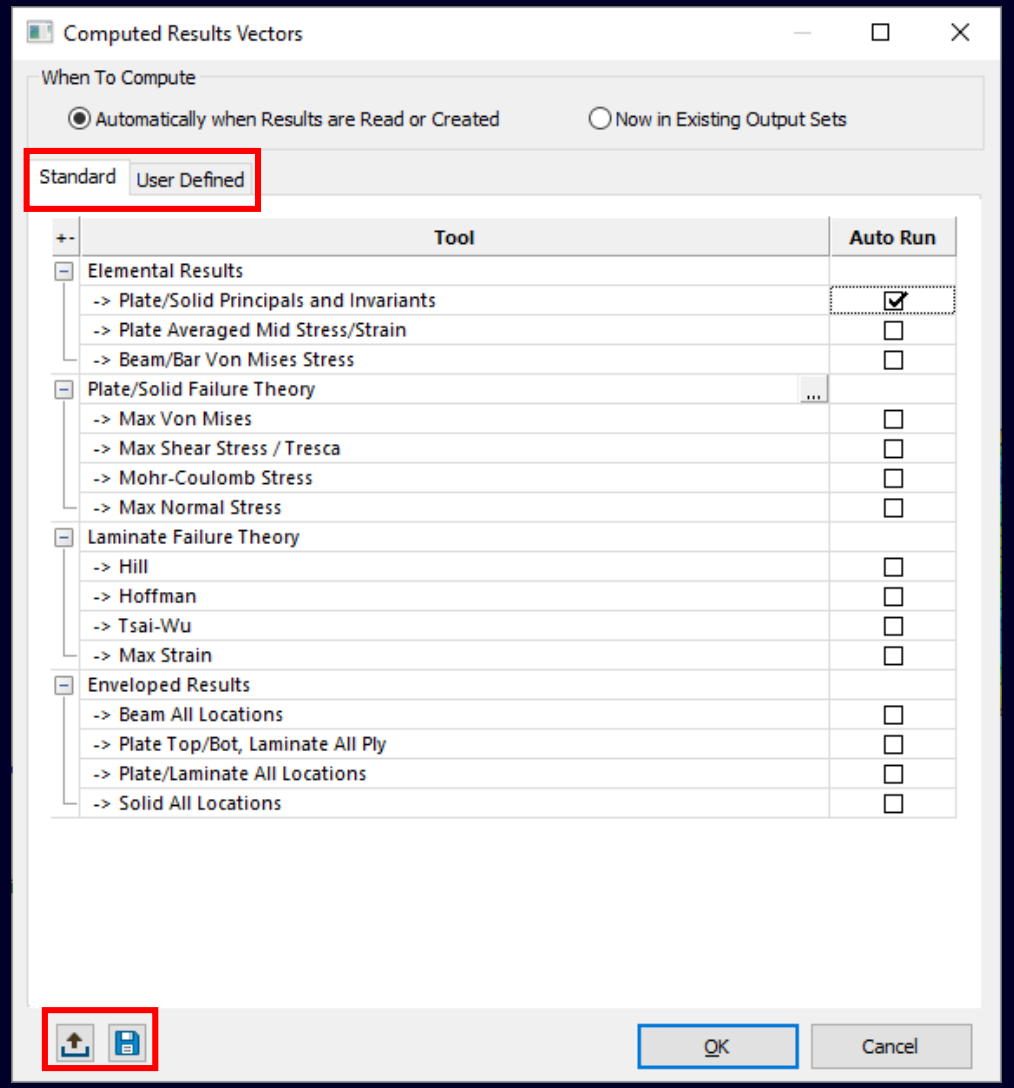
Simcenter Femap 2301
Postprocessing – Computed Vectors

Computed Vectors (Two Types)

- *Standard* – Select from list of pre-defined output types
- *User Defined* – Choose from Envelope, Combination, or API Script, then define

Library of vectors to compute can be saved

- Both *Standard* and *User Defined* can be saved to libraries
- Loaded via icon from within command
- Set on *Results* tab of **File -> Preferences** to select a library of vectors to always compute for all new models



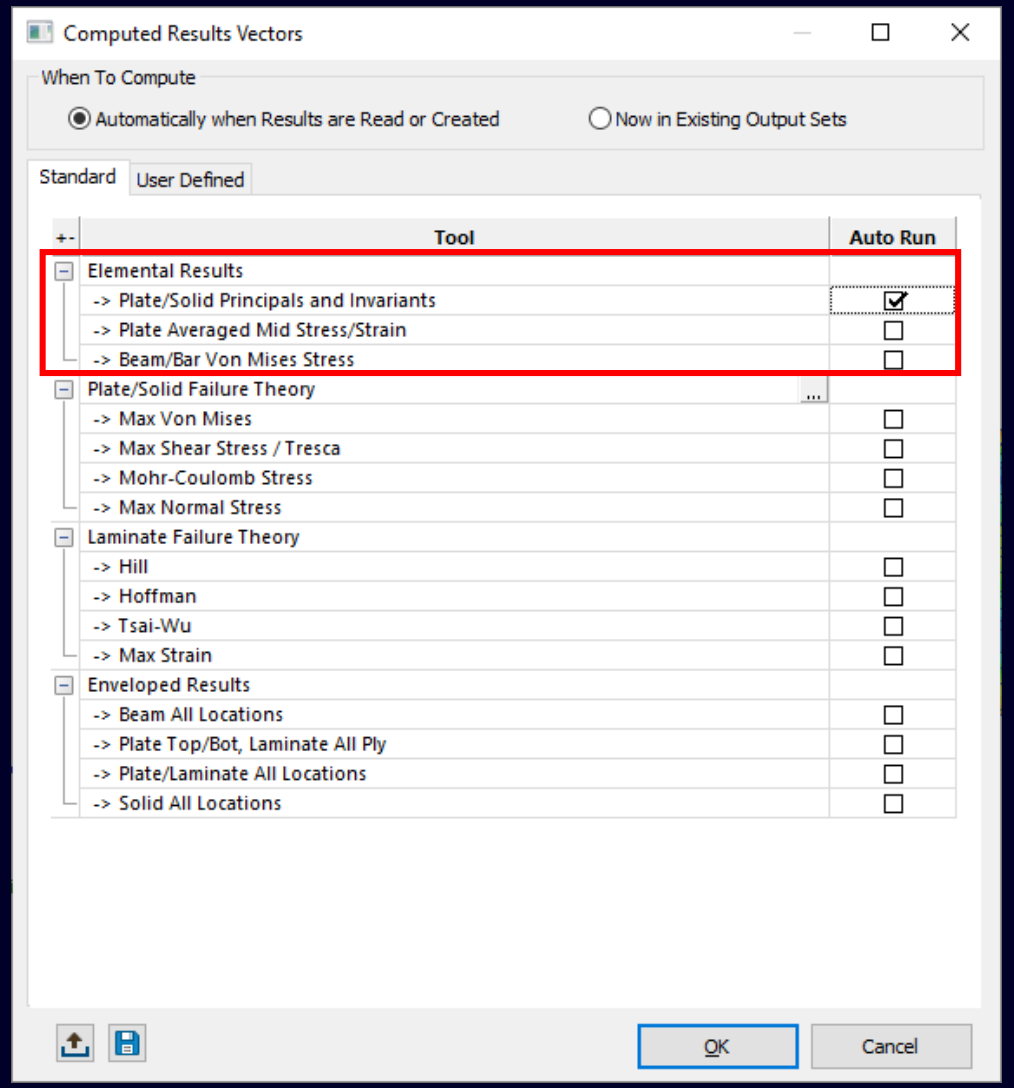
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Computed Vectors

Standard Computed Vectors – Results quantities that many users has asked for Femap to compute

- *Elemental Results*
  - *Plate/Solid Principals and Invariants* (*Identical to Compute Principal Stress/Strain* option on *Results* tab in **File -> Preferences** in previous versions)
  - *Plate Averaged Mid Stress/Strain* (Identical to *Compute Averaged Mid Stress/Strain* option on *Results* tab in **File -> Preferences** in previous versions)
  - *Beam/Bar Von Mises Stress*



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

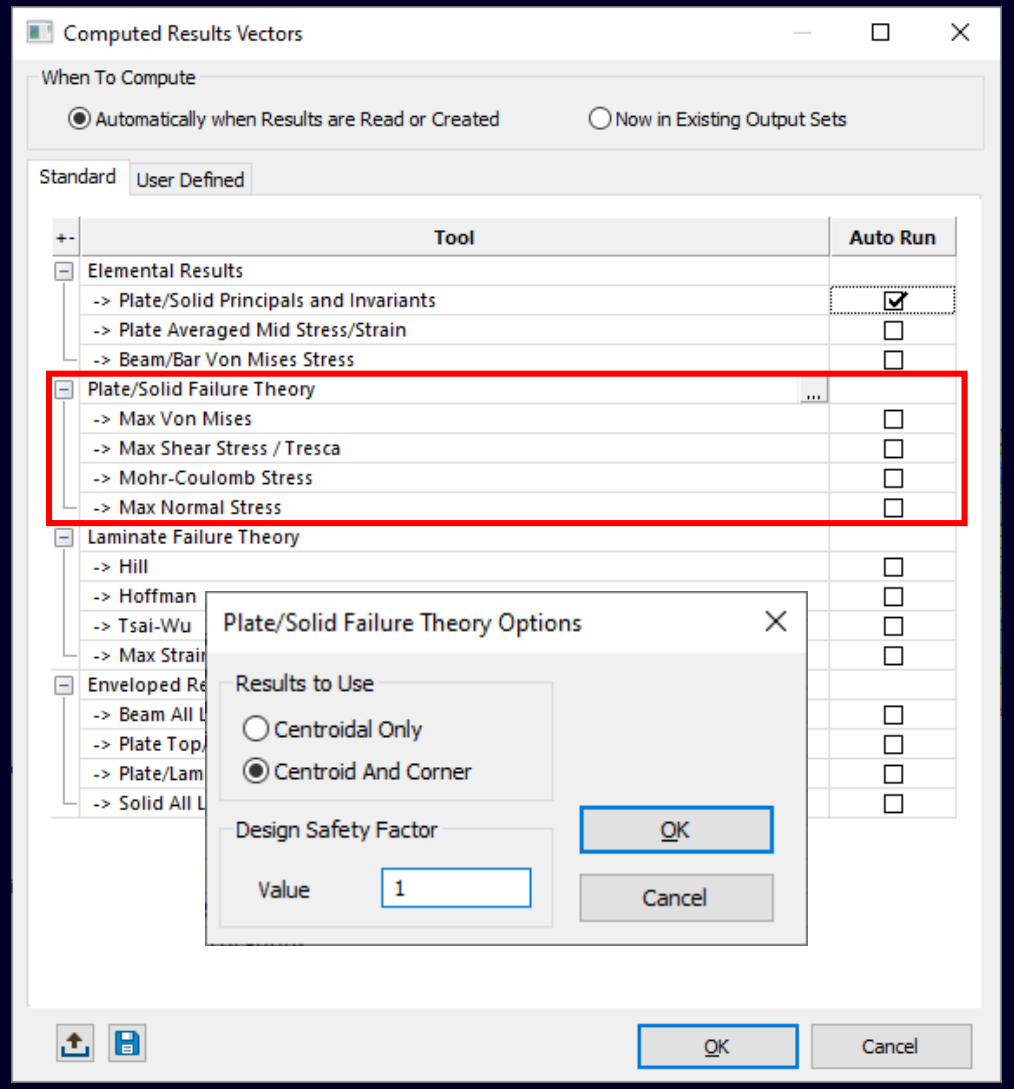
Simcenter Femap 2301
Postprocessing – Computed Vectors

Standard Vectors – Results quantities that many users has asked for Femap to compute

- *Plate/Solid Failure Theory*
  - *Max Von Mises*
  - *Max Shear Stress / Tresca*
  - *Mohr-Coulomb Stress*
  - *Max Normal Stress*

The *"…"* button opens the *Plate/Solid Failure Theory Options* dialog box

- *Results to Use* – Choose between *Centroid Only* or *Centroid and Corner*
- *Design Safety Factor – Value* used to compute Design Safety Factor for verification



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

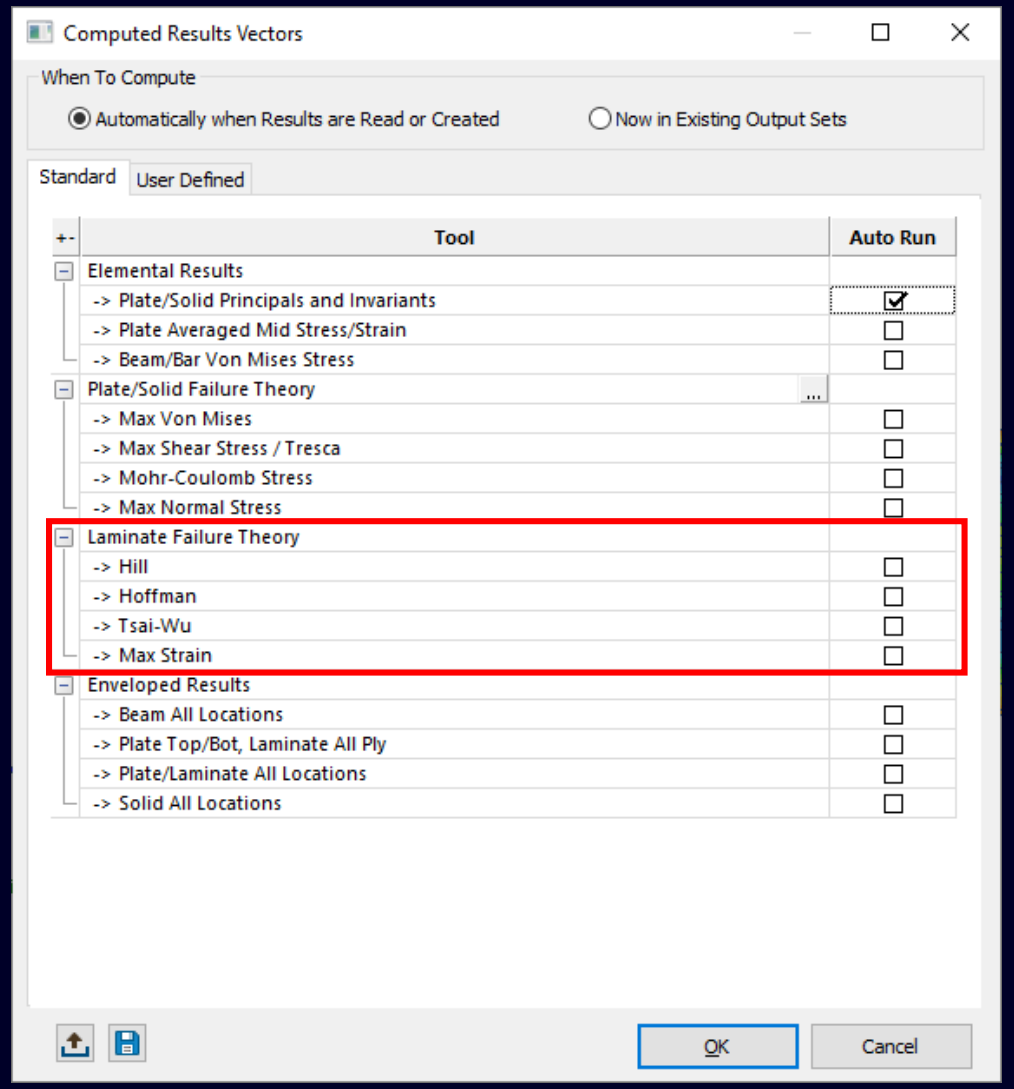
Simcenter Femap 2301
Postprocessing – Computed Vectors

Standard Vectors – Results quantities that many users has asked for Femap to compute

- *Laminate Failure Theory*
  - *Hill*
  - *Hoffman*
  - *Tsai-Wu*
  - *Max Strain*

Uses same Failure Theory calculations as Nastran solvers

Allows user to calculate different Failure Theory or recalculate using different material limits without re-running the analysis



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

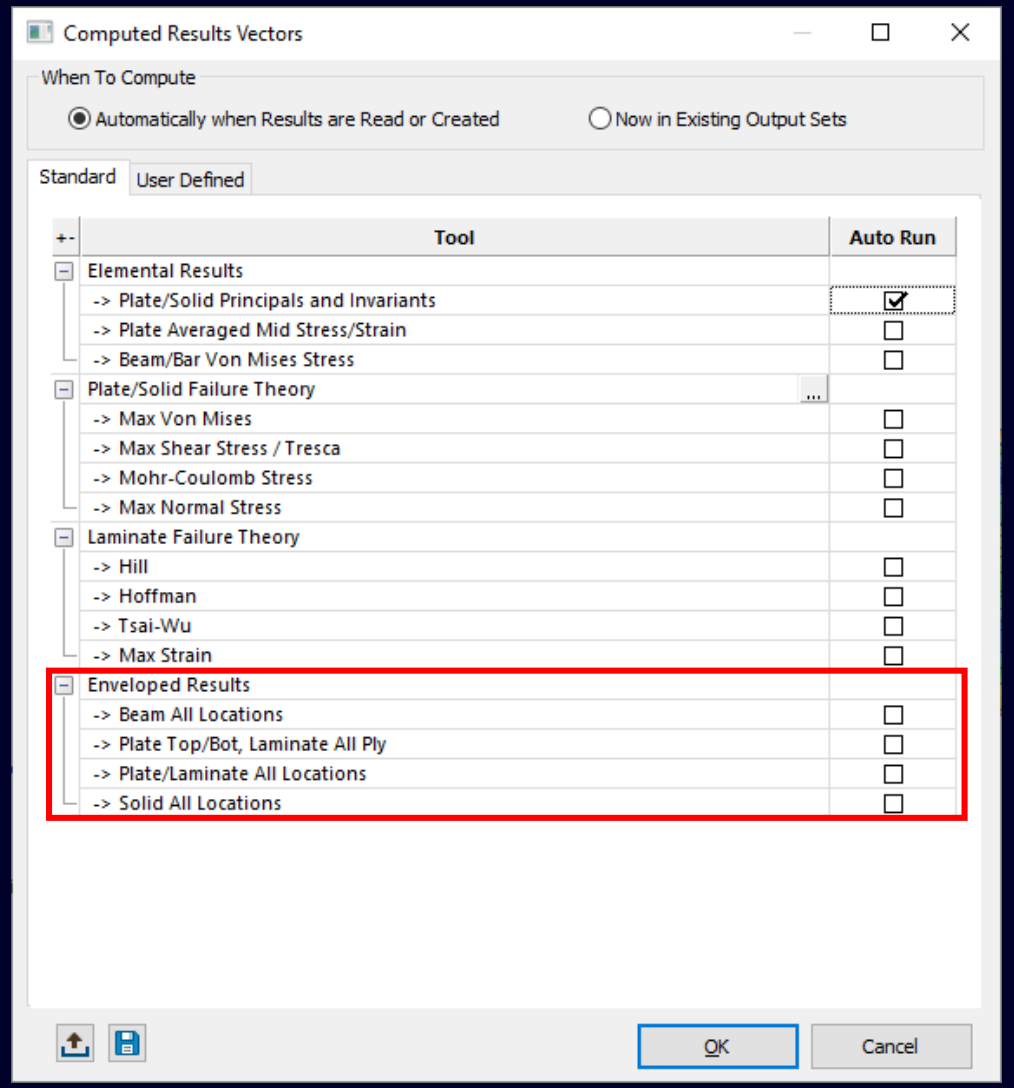
SIEMENS

Simcenter Femap 2301
Postprocessing – Computed Vectors

Standard Vectors – Results quantities that many users has asked for Femap to compute

- *Enveloped Results*
  - *Beam All Locations*
  - *Plate Top/Bottom, Laminate All Ply*
  - *Plate/Laminate All Locations*
  - *Solid All Locations*

These results quantities could always be calculated using **Model -> Output -> Process**, this just streamlines the creation of these output vectors



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

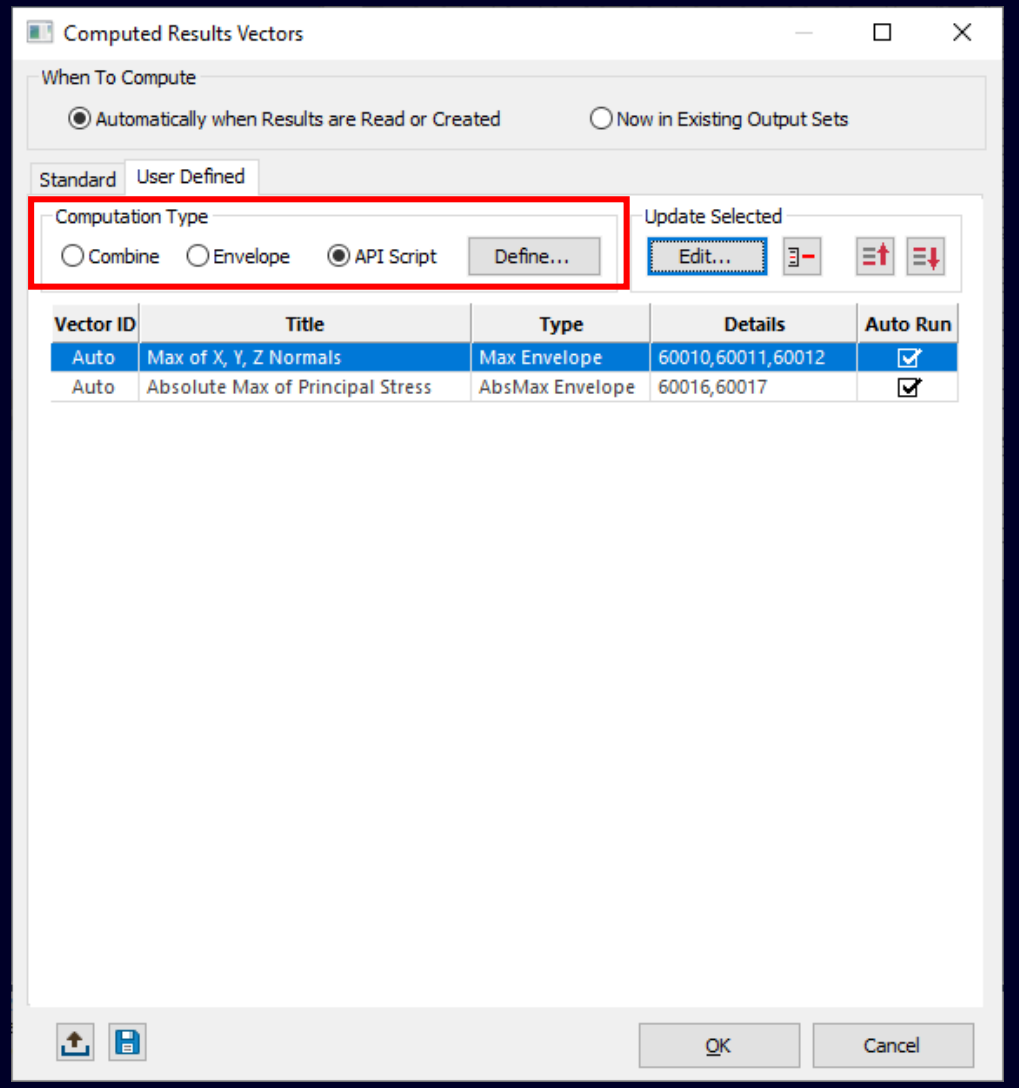
SIEMENS

Simcenter Femap 2301
Postprocessing – Computed Vectors

User Defined Computed Vectors – Results quantities which have been defined by the user for a particular model

- *Computation Type*
  - *Combine* – Creates Computed Vectors based on a *Linear Combination* or *RSS Combination* using any number of specified Output Vector IDs
  - *Envelope* – Creates a *Max Value*, *Min Value*, or *Max Absolute Value* Envelope using the specified Output Vector IDs
  - *API Script* – Creates Computed Vectors based on an API script

Click *Define* to define Computed Vector item…

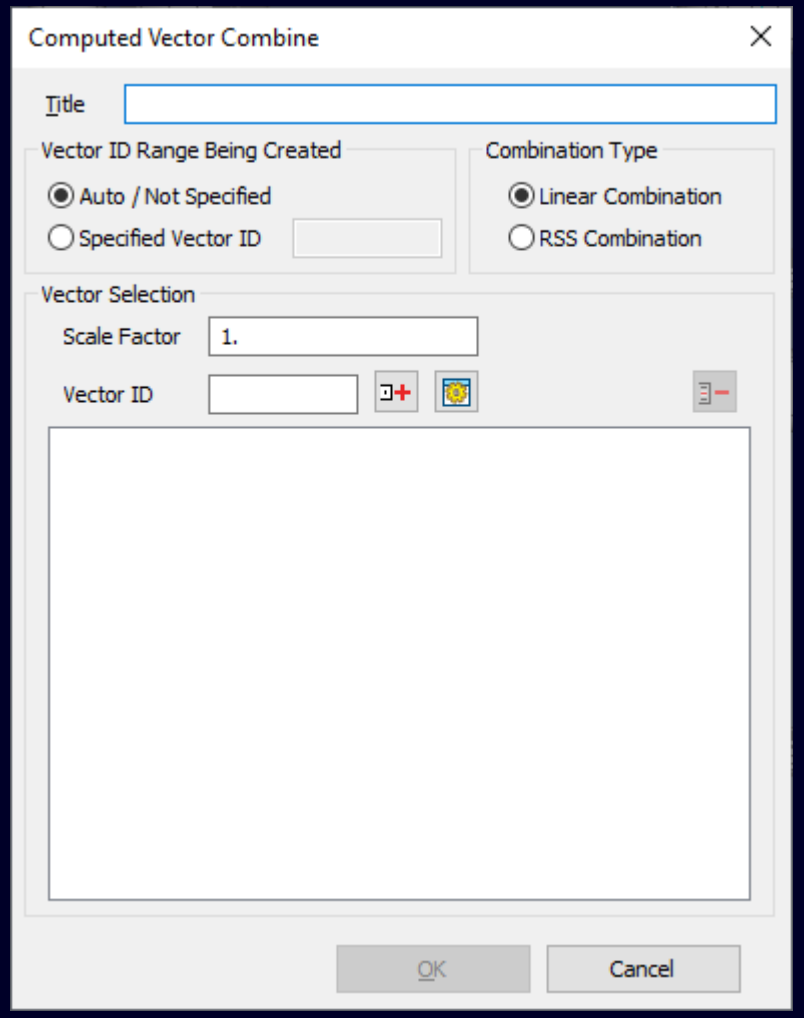


© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

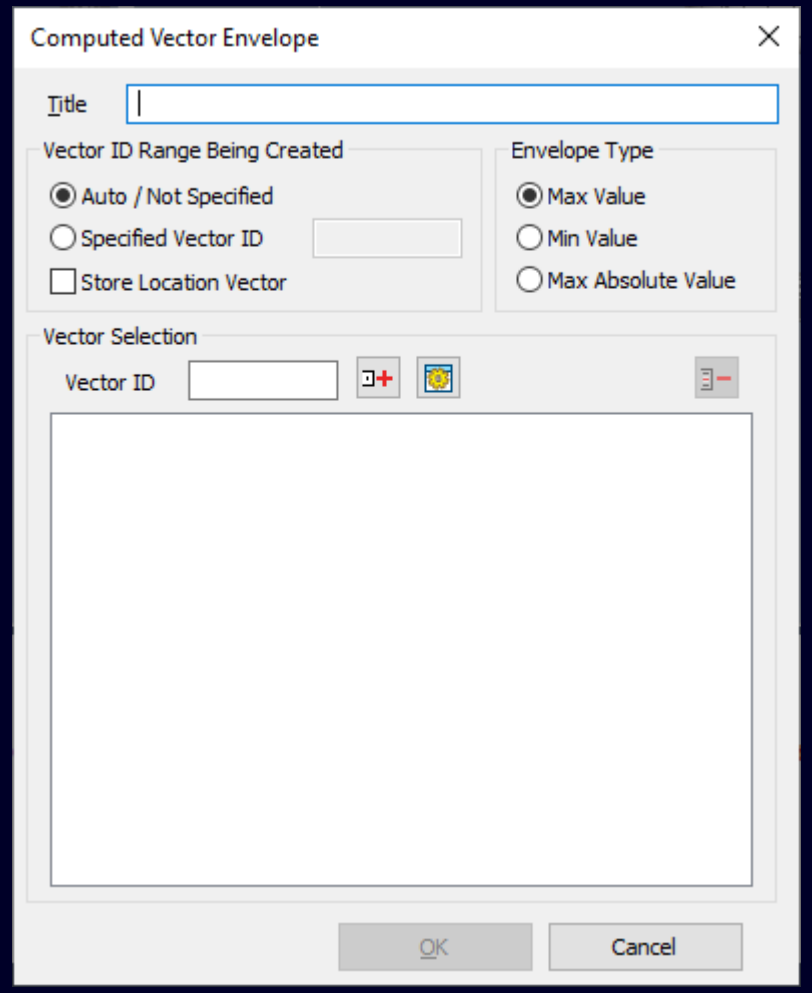
SIEMENS

Simcenter Femap 2301
Postprocessing – Computed Vectors

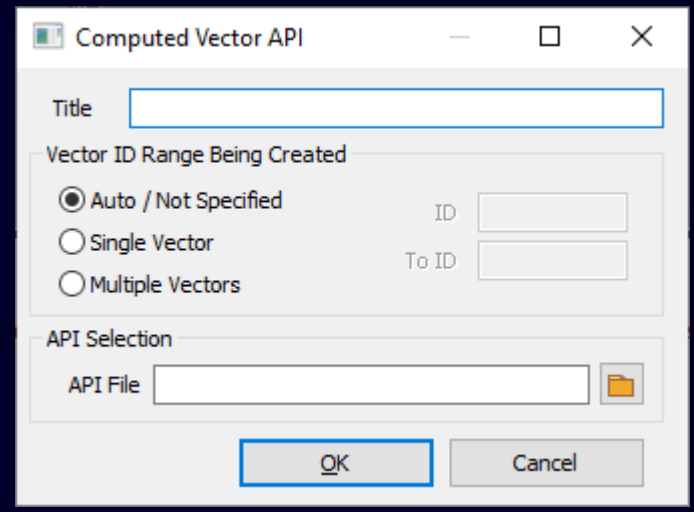
Combination



Envelope



API Script



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

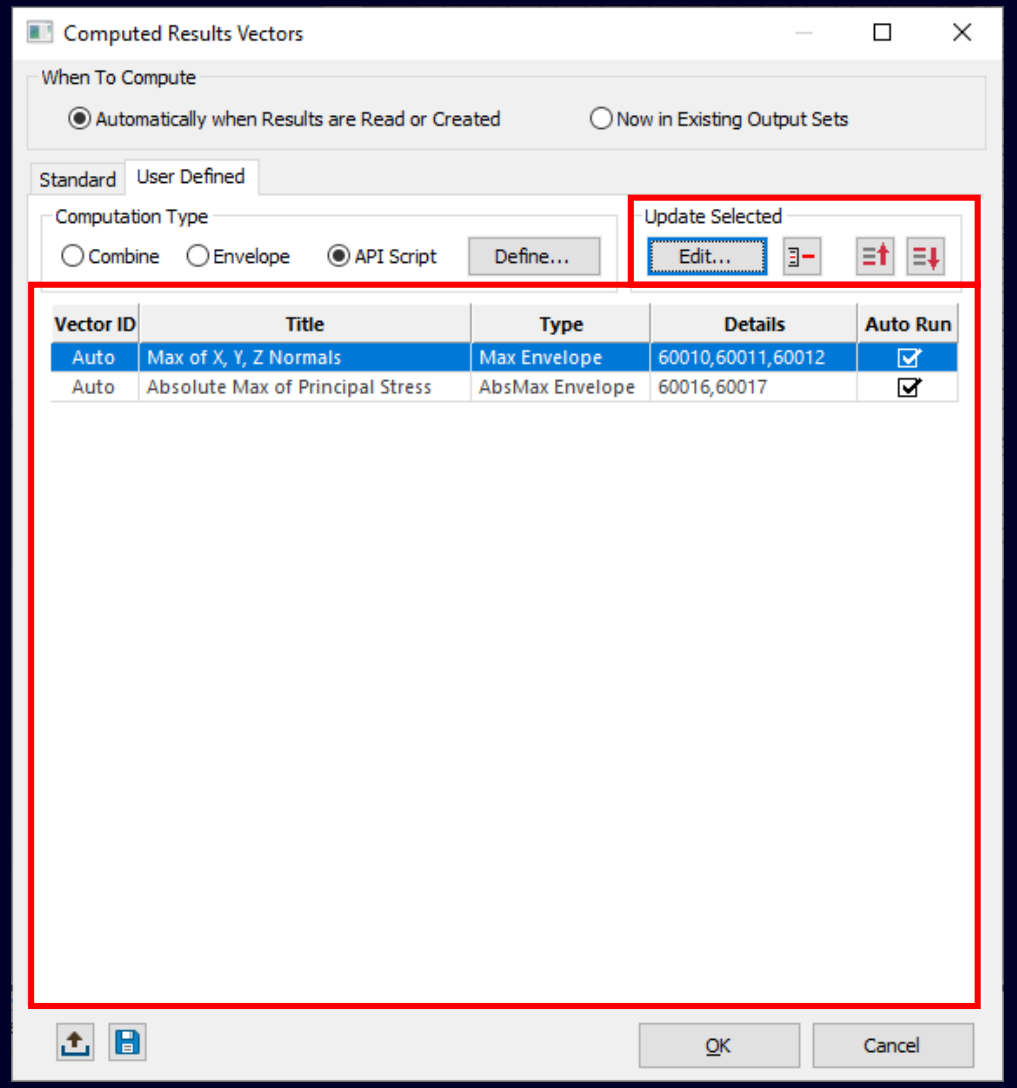
Simcenter Femap 2301
Postprocessing – Computed Vectors

Once defined, each *User Defined* Computed Vector item is added to the list

Much like *Standard* Computed Vectors, the *When to Compute* option is used to determine if the *Auto Run* or *Run Now* column is available

If needed, the icons in the *Update Selected* section can be used to:

- *Edit* – Edits selected item in the list
- *Delete* icon – Deletes selected item in the list
- *Move Up/Move Down* icons – Moves selected item up or down in list should Computed Vector item lower in the list rely on other item higher in list for vector IDs



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Data Conversion options

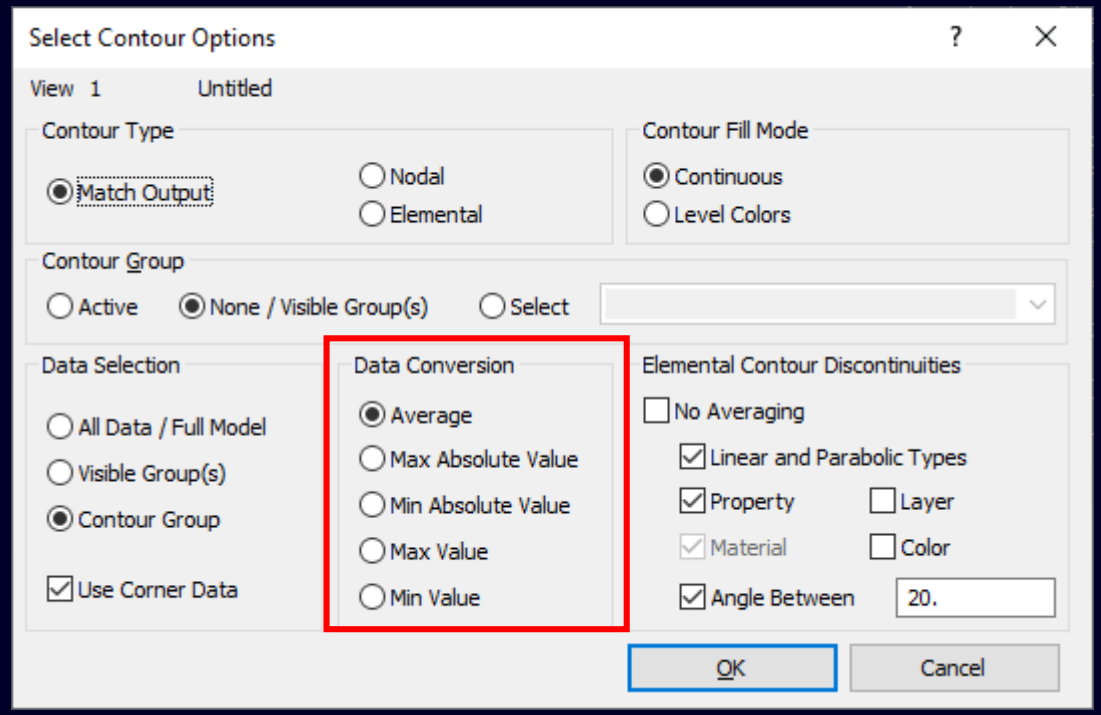
Updated options for “Data Conversion” throughout Femap

Previously, there were three options:

- *Average*
- *Max Value* (actually “Max Absolute Value”)
- *Min Value* (actually “Min Absolute Value”)

Now, available options are:

- *Average*
- *Max Absolute Value* (old “Max Value”)
- *Min Absolute Value* (old “Min Value”)
- *Max Value* (New!)
- *Min Value* (New!)



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Data Conversion options

These Data Conversion options are now all labeled consistently and available in all places they are used:

- *Contour Options* dialog box
- *Postprocessing Toolbox* for *Data Conversion*
- *View, Options* – “Post Titles” (displayed in Graphics Window)
- *Charting* pane dialog boxes and Legend in Chart Area
- **Model -> Output -> Process** – *Convert* tab
- *Output Map Data Surface* in *Data Surface Editor* pane
- **Model -> Load -> Map Output from Model** command
- *Entity Editor* and *Data Table* which contain info about a View
- API Enum zResultsConvert
- API Interpolate Class

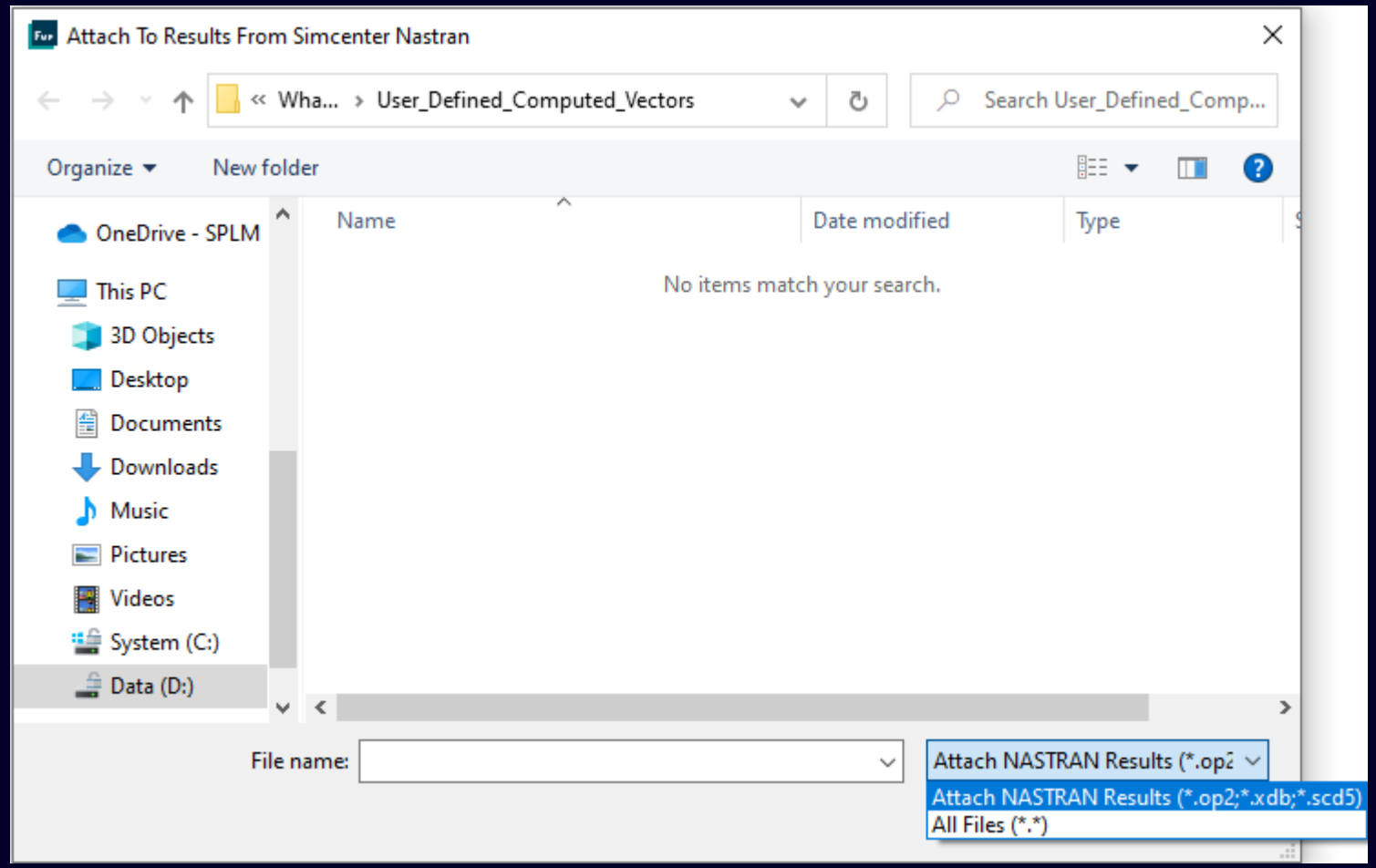
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Attach to Simcenter HDF5 File (*.scd5)

Support has been added to attach to HDF5 format results files (SCD5) generated by Simcenter Nastran using the **File -> Attach to Results** command

Note: There is no way to currently request the SDC5 file from Simcenter Nastran via the Femap user interface



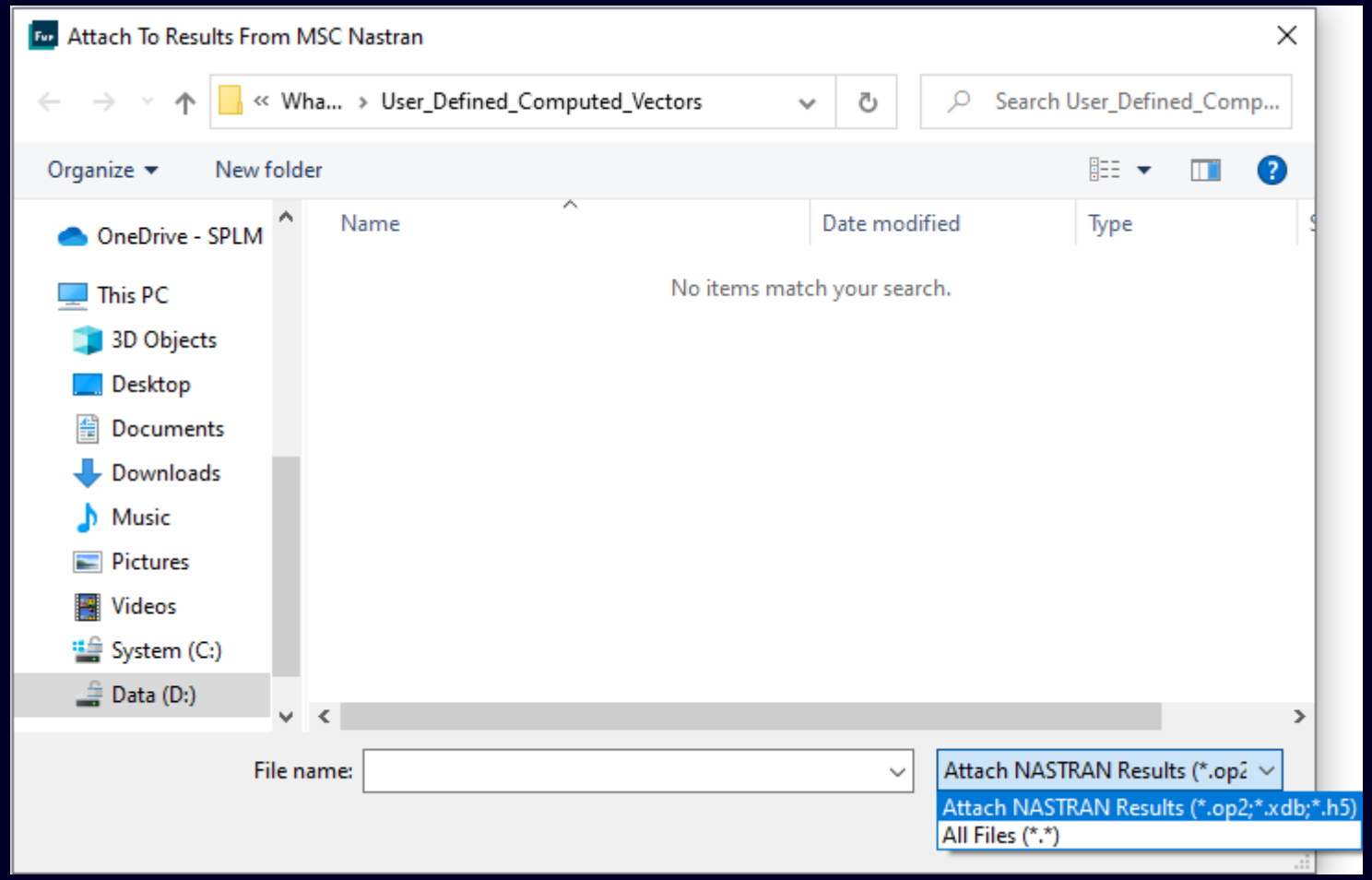
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Attach to MSC Nastran HDF5 File (*.H5)

Support has been added to attach to HDF5 format results files (NH5RDB) generated by MSC Nastran using the **File -> Attach to Results** command

Note: Results in this format can be requested within the Femap user interface via the *NASTRAN Output Requests* dialog box



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Attach to MSC Nastran HDF5 File (*.h5)

HDF5 Output Types currently supported by Femap (May be more by time of release)

NODAL

- NODAL\DISPLACEMENT
- NODAL\APPLIED\LOAD
- NODAL\SPC\FORCE
- NODAL\MPC\FORCE
- NODAL\ACCELERATION
- NODAL\VELOCITY

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Attach to MSC Nastran HDF5 File (*.h5)

ELEMENTAL STRESS

- ELEMENTAL\STRESS\ROD
- ELEMENTAL\STRESS\ROD\NL

- ELEMENTAL\STRESS\TRIA3
- ELEMENTAL\STRESS\TRIAR
- ELEMENTAL\STRESS\TRIA6
- ELEMENTAL\STRESS\QUADR
- ELEMENTAL\STRESS\QUAD4
- ELEMENTAL\STRESS\QUAD8
- ELEMENTAL\STRESS\QUAD\CN
- ELEMENTAL\STRESS\QUADR\NL
- ELEMENTAL\STRESS\QUAD4\NL
- ELEMENTAL\STRESS\TRIA3\NL
- ELEMENTAL\STRESS\TRIAR\NL

- ELEMENTAL\STRESS\TETRA
- ELEMENTAL\STRESS\HEXA
- ELEMENTAL\STRESS\PENTA
- ELEMENTAL\STRESS\PYRA

- ELEMENTAL\STRESS\TETRA\NL
- ELEMENTAL\STRESS\HEXA\NL
- ELEMENTAL\STRESS\PENTA\NL
- ELEMENTAL\STRESS\PYRA\NL

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Attach to MSC Nastran HDF5 File (*.h5)

ELEMENTAL FORCE

- ELEMENTAL\ELEMENT\FORCE\ROD
- ELEMENTAL\ELEMENT\FORCE\QUAD4
- ELEMENTAL\ELEMENT\FORCE\QUADR
- ELEMENTAL\ELEMENT\FORCE\TRIA3
- ELEMENTAL\ELEMENT\FORCE\QUADR\CT
- ELEMENTAL\ELEMENT\FORCE\TRIAR
- ELEMENTAL\ELEMENT\FORCE\TRIA6
- ELEMENTAL\ELEMENT\FORCE\QUAD8
- ELEMENTAL\ELEMENT\FORCE\QUAD4\CN

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
Postprocessing – Attach to MSC Nastran HDF5 File (*.h5)

ELEMENTAL STRAIN

- ELEMENTAL\STRAIN\ROD

- ELEMENTAL\STRAIN\QUAD\CN
- ELEMENTAL\STRAIN\QUAD4
- ELEMENTAL\STRAIN\QUAD8
- ELEMENTAL\STRAIN\QUADR
- ELEMENTAL\STRAIN\TRIA3
- ELEMENTAL\STRAIN\TRIA6
- ELEMENTAL\STRAIN\TRIAR

- ELEMENTAL\STRAIN\TETRA
- ELEMENTAL\STRAIN\HEXA
- ELEMENTAL\STRAIN\PENTA
- ELEMENTAL\STRAIN\PYRA

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

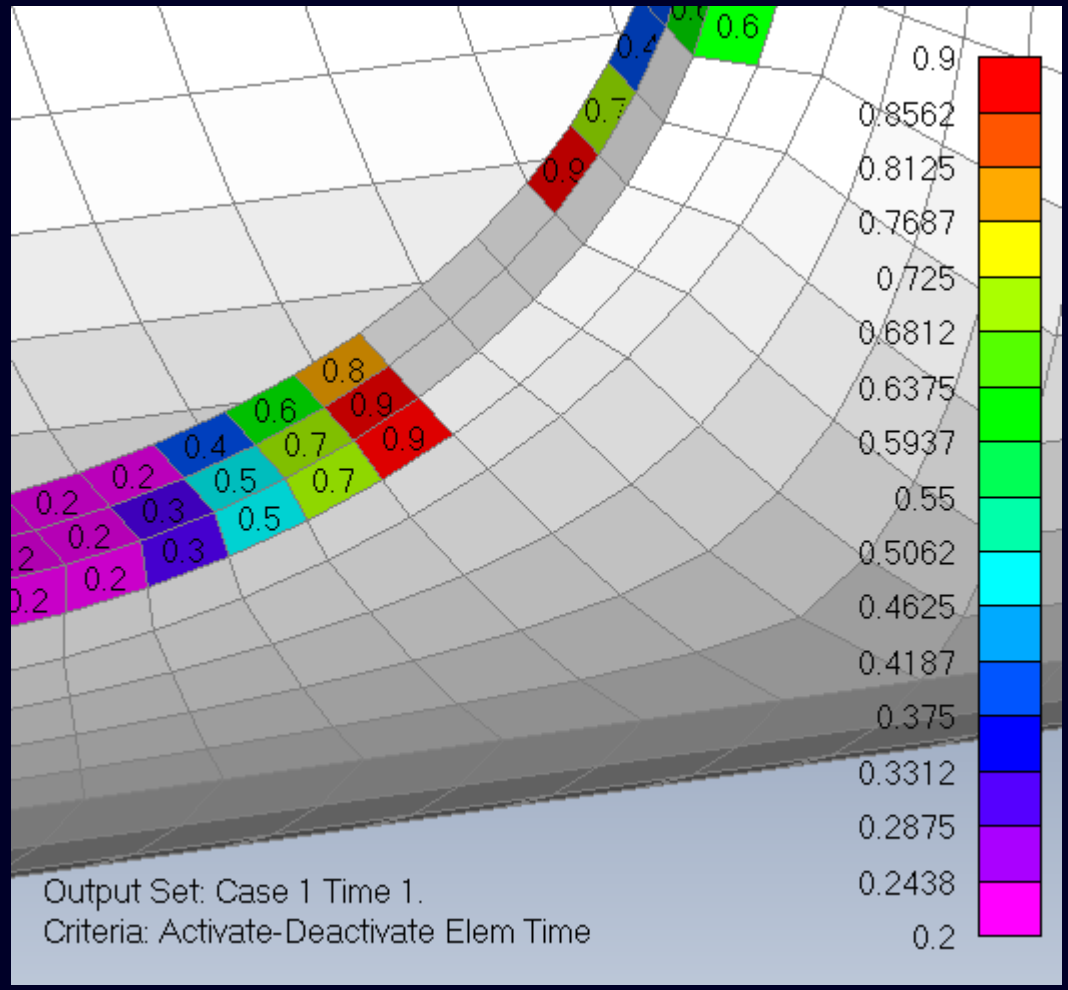
SIEMENS

Simcenter Femap 2301
Postprocessing – Support for results from Simcenter Nastran SOL 401 and SOL 402

Added support for results from Simcenter Nastran SOL 401 for Element Addition and Removal

- Pre-Born and Removed Elem Flag
- Activate-Deactivate Elem Time

Added support for various types of Strain results from Simcenter Nastran SOL 401 and SOL 402 for Axisymmetric, Plane Strain, and Plane Stress elements



© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

What’s new

Simcenter Femap 2301

Teamcenter Integration

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

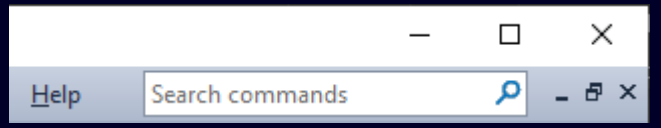
© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

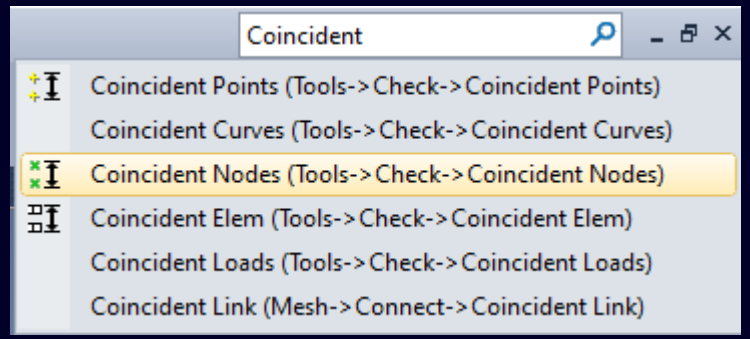
SIEMENS

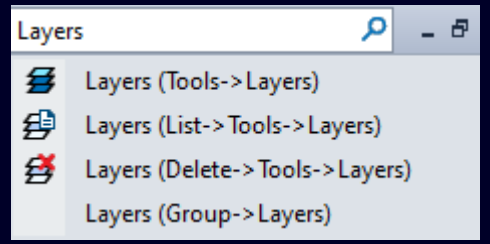
Simcenter Femap 2301
Miscellaneous – Command Finder

New for version 2301 is the *Command Finder* functionality!!!

- Accessed via the “Search Commands” field found in the menu bar to the right of the **Help** menu
- Type keyword(s) into field and available commands which contain the keyword will be listed after clicking the “magnifying glass” icon in the field or clicking *Enter*
- Navigate to the desired command in the list and the command will be invoked







© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
API – New functionality

Added new Computed Vectors Object (feComputedResultsVectors) along with adding required properties and methods to access the new Computed Vectors functionality programmatically

Added new Element Add | Remove Object (feElemAddRemove) along with adding required properties and methods to access the new Element Add | Remove functionality programmatically

Added feGroupReduceToExisting to provide programmatic access to the new **Group -> Operations -> Reduce to Existing** command

Added feMeshEditingSplit to access the **Mesh -> Editing -> Split** command programmatically

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap 2301
API – New functionality

Added SolidCleanupTool Object along with needed properties and methods to allow programmatic access to all options in **Geometry -> Solid -> Cleanup** command

Added feSurfaceMidAuto5 to allow programmatic access to all options in the **Geometry -> Midsurface -> Automatic** command including new *Remove Holes* and *Keep Untrimmed Surfaces* options

Added feCurveSplitPointToPoint2, feCurveSplitPointToEdge2, and feCurveSplitEdgeToEdge2, each of which now has an option for determining the surface to be split:

- 0 = Works the same as the previous API calls (i.e.,without the “2) and if a surface to split is not found, user is prompted to choose one
- 1 = Attempts to determine surface to split automatically and if one cannot be determined, returns FE_FAIL
- 2 = Option to enter ID of surface to split via 4th argument in call

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Simcenter Femap

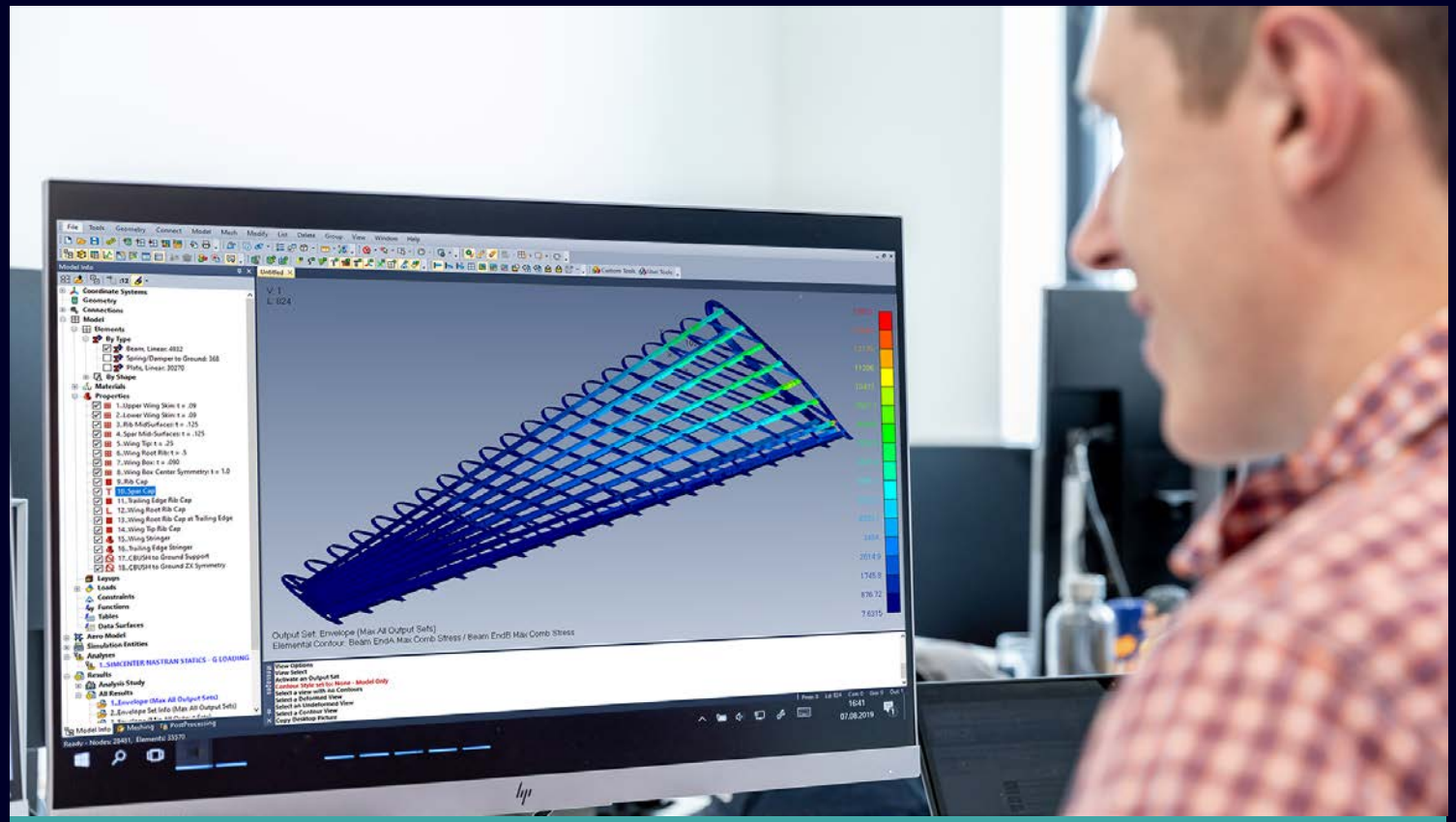
Simcenter Femap SaaS

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Try Simcenter Femap for free

30-day free trial



siemens.com/try-simcenter-femap

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Join the Simcenter Femap Community

Explore – Browse our different **blogs** dedicated to each solution, read our articles and hear about the latest news.

Share – Ask questions in our **forum**, get answers, connect with other users, and benefit from their experience.

Learn – Find the information you need in the **knowledge base** and improve your skills.

siemens.com/plm/community/simcenter

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

Q & A

Simcenter Femap 2301

© Siemens 2023 | Siemens Digital Industries Software | Where today meets tomorrow.

SIEMENS

Contact Us

13290 Evening Creek Drive
San Diego, CA 92128

(858) 480-2000

sales@ata-e.com

www.ata-e.com

@ATAEngineering

ata-engineering