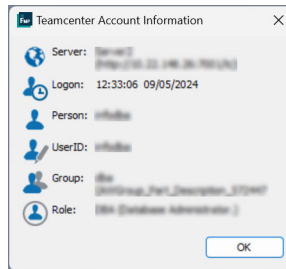


1. What's New - version 2406

Changes and enhancements made to Femap for version 2406 are listed here and in the Femap readme.

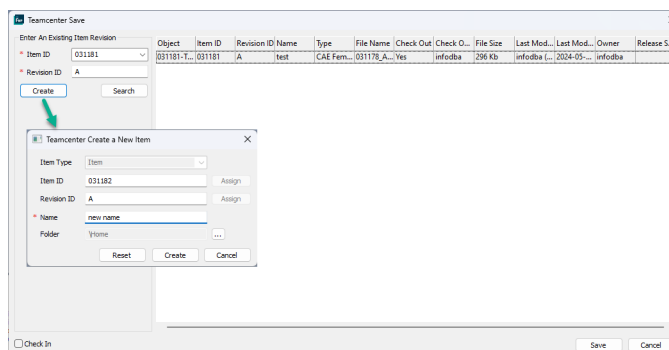
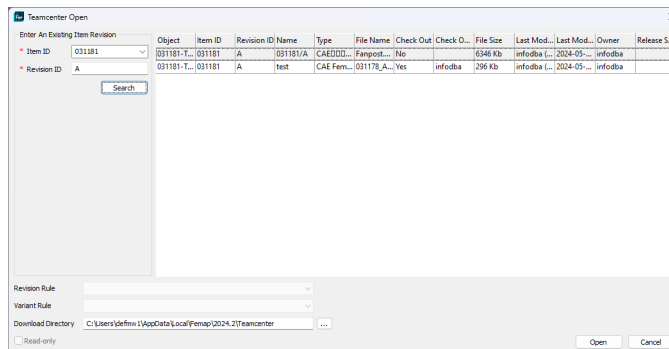
Teamcenter Integration

View Account Information



- The Account Information dialog box now posts successful login, and details about the current session.

Open/Save Files from/to Teamcenter



- Migrated the supported dataset from 'CAE Analysis' to 'CAE Femap'.

Note:

Any files saved to the 'CAE Analysis' dataset are now supported only for the Teamcenter Open operation.

- Added support for opening of mod and neu files.
- Created a new user interface for selecting data from Teamcenter operations.
 - Added ability to reorder columns or toggle visibility of columns.
 - Retain changes done in column orders and visibility for any subsequent usage of the dialog box.
 - For modfem files opened from the 'CAE Femap' dialog box, the **Read-only** check box lets you open the file as read-only. By default, **Read-only** is off and the data checked-out post file open.
 - Auto-fill the current **Item-ID** and its **Revision-ID** upon opening the Teamcenter Save dialog box.
 - Added retention of last 5 most recently used items for the Teamcenter search during Teamcenter Open, Save and Import Shattered JT Assembly commands.
 - For the Teamcenter Save operation, added a warning for empty dataset names in the Teamcenter Save dialog box. If you enter an Item Revision with no 'CAE Femap' dataset, an empty row is created with empty name. The warning icon and a tool-tip appear for the empty dataset name. Upon clicking of the **Save** button, a new dataset is created under the Item-Revision object of type 'CAE Femap' and the file is uploaded in the dataset.

New Item Revision Creation

- Added a **Create** button in the Teamcenter Save dialog box. Opens the Teamcenter Create a New Item dialog box containing options to define/assign **Item ID**, **Revision ID**, **Name** and **Folder**.
- The **Folder** option Browse button opens the Select Folder dialog box which lists the folders available in Teamcenter.
- Newly created **Item-ID** and **Revision-ID** gets auto-filled in the Teamcenter Save dialog box **Search** operation input

Clear Dataset Items When Searching

- New dataset items are appended to the table. When you press the Search button, the current matches are cleared from the table.

Check Out/Check In from/to Teamcenter

- **Check-out** and **Check-in** buttons added to the Teamcenter toolbar.



Teamcenter Check Out



Teamcenter Check In

- Buttons are activated exclusively for files opened from Teamcenter.
- **Check Out** provides an exclusive access to the data opened from Teamcenter.
- **Check In** is activated when the active model is checked out to the logged in user.
 - Check In saves the current model opened from Teamcenter and uploads it to Teamcenter.
 - After a successful upload, the data is checked-in and the exclusive lock is released.

Layup Builder Enhancements

The following enhancements have been made to Layup Builder in Femap version 2406.

Change names of HDF5 related commands, options, and settings

Throughout the Layup Builder user interface, the term "Fibersim" is replaced with "Composites HDF5".

Save Prompt Upon Exiting Layup Builder

A prompt now presents you with an option Save when you exit the Layup Builder pane.

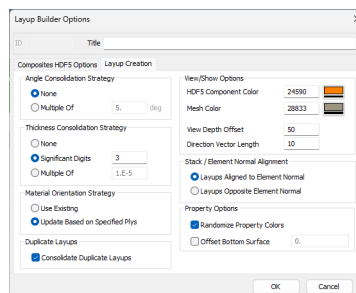
Update Layup Builder When Deleting Material

If you are working in the Layup Builder and deleting a material from the Femap user interface, the Layup Builder pane is updated to reflect the change. Rows using that material are no longer listed.

Bi-direction Composites HDF5 Data Exchange

Layup Builder now supports bi-direction Composites HDF5 data exchange. Users commonly go from Femap FEA to Fibersim as a starting point.

New Options — The Layup Builder Options dialog box is enhanced with two new options located in the newly added Property Option section of the **Layup Creation** tab.



Randomized Property Colors

A new option is available to randomize new property colors when generating layups. This is turned on in the **Layup Creation** tab of the Layup Builder Options dialog box.

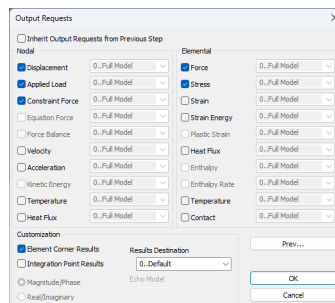
Set Offset on Laminates

A new option is available to set an offset on laminates. This is turned on, and the offset specified in the **Layup Creation** tab of the Layup Builder Options dialog box.

Import of Abaqus Output Requests

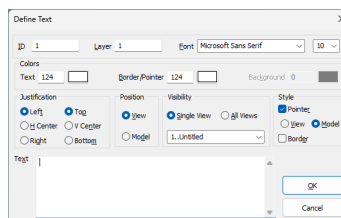
The import of Abaqus output requests is the final piece of functionality added to Femap's Abaqus *STEP import functionality. Femap version 2401 focused on importing other data contained in a *STEP. Femap version 2406 is focused on importing Output Requests for Abaqus. Additionally, improvements are added to the Abaqus Output Request dialog box and to the export of the output requests. These improvements ensure the propagation, re-specification, and suppression of output requests are done in a consistent manner.

A new check-box **Inherit Output Requests from Previous Step** is added to the Output Requests dialog box in an Abaqus Solver analysis set. When on, you can propagate the results from the previous step. When off, you can re-specify the output requests specifically for the current step. Also, support is added to the import of output requests specified in the Abaqus input file. This includes improved warning messages, and the display of relevant information in the warning messages during the export of the output requests.



Define Text in 3D Model Space

The **Tools** → **Text** command is enhanced to create the Text object in View space, but also have the pointer be attached to a point in 3D model space. Two new radio buttons are added in the **Style** section of the Define Text dialog box. They control the **Pointer** behavior. If you choose the **Position** of the text to be in **View**, you also have the option to put the **Pointer** in **View** or **Model**. If the text is in **Model** position, the pointer will only be allowed to be placed in Model space.



Support for Rigid Elements RROD and RBAR

In Simcenter Nastran, for SOL 401, an existing element architecture with *small strains* is available that improves performance. When you run your analysis with *large strains* turned off (PARAM, LGSTRN, 0), the software now uses the improved approach by default. The small strain architecture now supports RROD and RBAR rigid elements.

In Femap, RROD and RBAR rigid elements can now be created with the **Model** → **Element** command. The RROD and RBAR rigid elements are defined on two new tabs added to the Define Rigid Element dialog box. The **RROD** and **RBAR** tabs contain connectivity and degrees of freedom options for both.

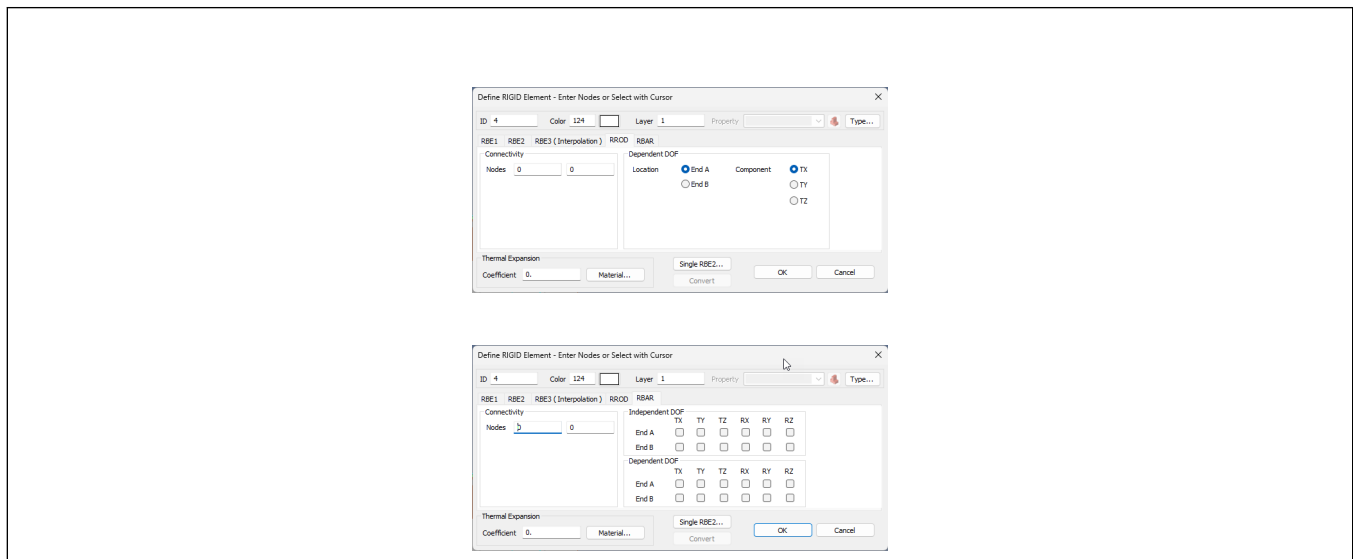
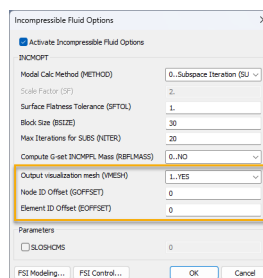


Figure 1-1. RROD and RBAR tabs

Bulk entry changes - Incompressible Fluid Options

A Simcenter Nastran version 2406 update to the INCMOPT bulk entry is incorporated into Femap's *Sloshing* analysis set options. The INCMOPT bulk entry specifies options for modeling incompressible fluids with fluid elements. Several options in Femap's Incompressible Fluid Options dialog box are added for outputting a visualization mesh for a sloshing surface.



Note:

The Incompressible Fluid Options dialog box can be accessed from the analysis set **Options** in the Analysis Set Manager.

The new options are related to the Simcenter Nastran VMESH bulk entry field.

Output visualization mesh (VMESH)

Outputs a visualization mesh for a sloshing surface. Options are: **0..No** (default) and **1..Yes**.

Note:

The software always outputs the mesh of PLOTTEL visualization elements it creates for a sloshing surface to the .op2 file. When **Output visualization mesh (VMESH)** is set to **1..Yes**, the software also outputs the mesh of PLOTTEL visualization elements to an ASCII file named *input_file_plotel.dat* and saves the file in the folder with the other files for the solution.

Node ID Offset (GOFFSET)

Sets a grid ID offset for the visualization mesh output. (Integer ≥ 0 ; Default = 0)

Element ID Offset (EOFFSET)

Sets an element ID offset for the visualization mesh output. (Integer ≥ 0 ; Default = 0)

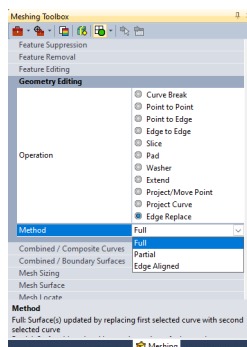
Read Aero Results from the op2 File

In previous versions of Simcenter Nastran, aero results from flutter solutions were only written to the *.f06 file making it difficult or confusing to setup Femap to read the flutter results for aero data. Femap version 2406 adds support for reading the aero results data from the *.op2 files.

Meshing Toolbox Enhancements

Geometry Editing: New Edge Replace Option

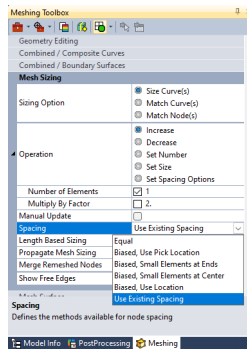
Edge Replace and **Methods** are added to the Geometry Editing command **Operation** options.



When editing geometry, the **Edge Replace** option extends Surface(s) connected to the first selected Curve to the second selected Curve via three **Methods**. **Full** replaces the first selected curve with the second selected curve. **Partial** updates the surface(s) by moving the points of the first selected Surface to the closest location on the second selected Curve. **Edge Aligned** replaces first selected edge to the second selected edge by linearly extending the adjacent edges of the first selected curve.

Mesh Sizing: New Spacing Option

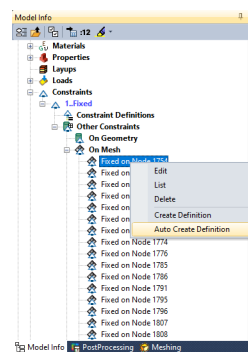
The option **Use Existing Spacing** is added to the Mesh Sizing command **Spacing** biasing options drop-down menu. When the option is used, the existing biasing option on the curve will be maintained.



Dockable Panes

Model Info Tree Constraints

- Added **Auto Create Definition** option to the Other Constraints context sensitive menus. The new option allows you to highlight any number of constraints and will automatically create new constraint definitions based on type and constraint. A new definition will be created for constraints of the same type which have different values and/or different settings, which differs from the Create Constraint command.



Nastran Modal Analysis Solver Defaults

For solutions that involve **real eigenvalue analysis with the Lanczos method** in Femap versions 2406 and higher (SOL 103 – Normal Modes, SOL 105 – Linear Buckling, SOL 111 – Modal Frequency, and SOL 112 – Modal Transient), the **MUMPS** solver is now the default. Reverting to the solver used in earlier

versions prior to version 2406 can be done by adding the `SYSTEM(810)=0` system cell to the analysis set.