



What's New in Femap 2401

March 14, 2024

Andy Haines, Siemens Digital Industries Software

San Diego • Los Angeles • Berkeley • Albuquerque • Denver • Huntsville • Washington D.C.

 [ata-engineering](https://www.linkedin.com/company/ata-engineering)  [@ATAEngineering](https://twitter.com/ATAEngineering)  (858) 480-2000  www.ata-e.com



Aerospace



Spacecraft



High-Speed
Systems



Defense



Industrial &
Mining
Equipment



Themed
Entertainment



What We Do

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

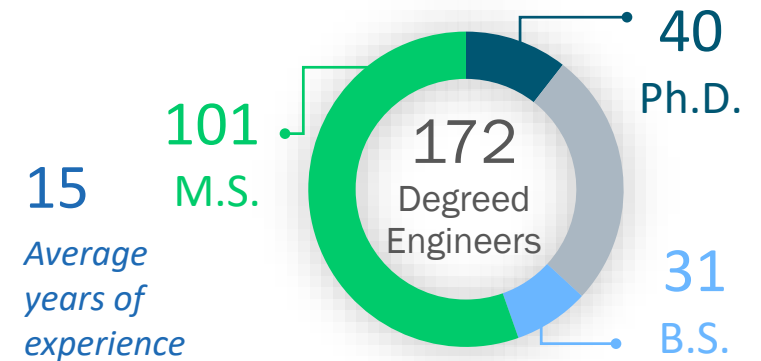


Who We Are

We are an **employee-owned** small business with a **full-time staff of over 210**

Subject-matter 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



Our Siemens Software Services

ATA is a partner and value-added reseller for Siemens Digital Industries Software

Simcenter

**Simcenter 3D | Femap | Nastran
STAR-CCM+ | HEEDS | Amesim
Test & Measurement**

Teamcenter

- Hotline Support
- Software Integration and Implementation
- Training Material

NX CAD & CAM



Expert
Partner

SIEMENS

Digital Industries Software

Visit Our Website for Product Information and Free Resources

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

ATA Technical Support Hotline
CALL 877-ATA-4CAE (877-282-4223)

You can call the hotline from 5:00 AM to 5:00 PM Pacific or contact us anytime through our support contact form below. Our live hotline provides comprehensive technical support for all of the Siemens software and hardware we sell, and it serves as the first line of support for our customer's questions and problems. This service is staffed by experienced engineers who use these tools every day, resulting in premier technical support for our customers. The hotline was recently rated as "excellent" by our customers, and over 80% of issues were resolved the same day they were reported. When necessary, we can also help elevate issues to Siemens GTAC support. ATA also provides support for our own ATASuite software.

Siemens Software and Hardware Support ATA Software Support SPIRITS Support

ATA's team of experts provide comprehensive technical support for all of the Siemens software and hardware we sell. Fill in the form below and a member of our technical team will be in touch shortly.

Alternatively, you can call us at 877-282-4223. Hours of operation 8 a.m. to 8 p.m. Eastern.

Product*
Full Name*
Company*
Email*
Phone*
Please describe the issue you are having

FREE RESOURCES
Training to Empower Your Teams

Resources are password protected. [GET PASSWORD](#) Access any resources for free

ALL PRODUCTS ALL MEDIA TYPE search free resources

- Amesim: System Modeling of Deployable Space Systems (On-Demand Webinar)
- HEEDS, STAR-CCM+: Optimization of Hypersonic Engines with STAR-CCM+ and HEEDS (On-Demand Webinar)
- Amesim: Rocket Engine P... Analysis with Sim... Amesim System (On-Demand Webinar)
- Simcenter 3D, Simcenter Nastran: Simcenter 3D: Understanding Response Dynamics vs. Dynamic Response (On-Demand Webinar)
- Amesim: System Simulation for Optimization of Heavy Equipment Design (On-Demand Webinar)
- Simcenter 3D: Engineering for Electromagnetic Compatibility in and Defense Ele... (On-Demand Webinar)

DIGITAL INDUSTRIES SOFTWARE
What's New in Simcenter Femap 2301
Experience new levels of collaboration and results exploration

Benefits

- Direct integration with Teamcenter to facilitate collaboration
- Unified graphics architecture for improved performance
- Streamlined creation of entities for element addition and removal
- Greater flexibility during the meshing workflow
- Deeper understanding of results via computed vectors

Summary

Simcenter Femap 2301 software is an advanced finite element modeling (FEM) software application used at a pre- and post processor for engineering simulation and analysis. Advanced workflows in Simcenter Femap enable analysts to model components, assemblies, and systems, and digitally analyze a model's response under real world conditions.

Simcenter Femap 2301 introduces key features and updates to enhance your productivity and collaboration, streamlining your design processes for geometry, meshing, analysis, and postprocessing. With the Command Hooks, you can save time by quickly and easily searching for current and new commands within Simcenter Femap. Additionally, the Applications Programming Interface (API) has been expanded to provide programmatic access to more functionality for the purpose of automation and customization.

SIEMENS siemens.com/simcenter

What's New in Femap 2401

March 14, 2024

Andy Haines, Siemens Digital Industries Software

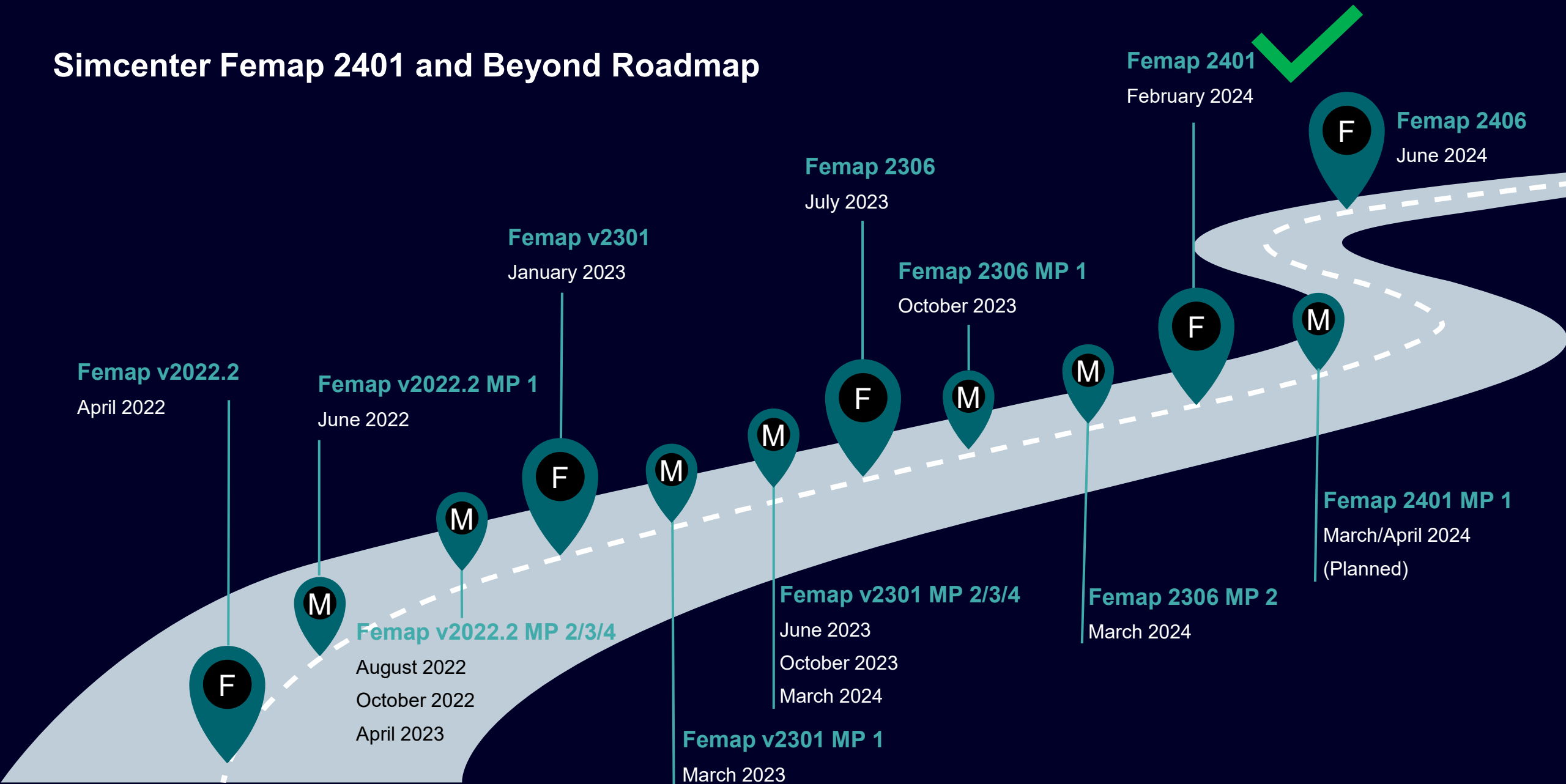


Simcenter
Simulation & Test

What's New in 2401

Simcenter Femap

Simcenter Femap 2401 and Beyond Roadmap





What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

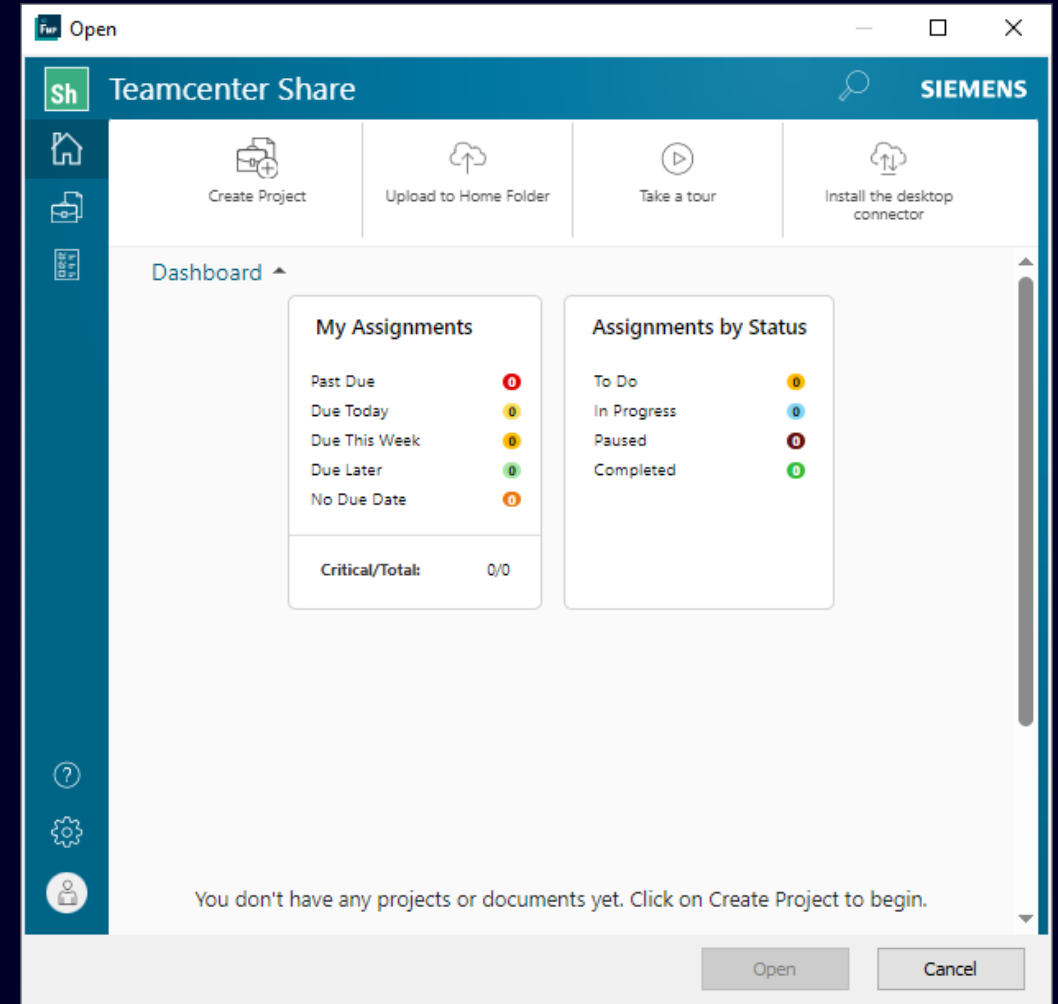
Miscellaneous and API

Simcenter Femap 2401 Teamcenter Share

Siemens Xcelerator Share is now Teamcenter Share

To reflect this in the User Interface, the names of the following have changed:

- *PDM -> Siemens Xcelerator Share* menu is now *PDM -> Teamcenter Share*
- *Tools -> Toolbars -> Siemens Xcelerator Share* is now *Tools -> Toolbars -> Teamcenter Share*
- *Siemens Xcelerator Share* toolbar is now called *Teamcenter Share*





What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

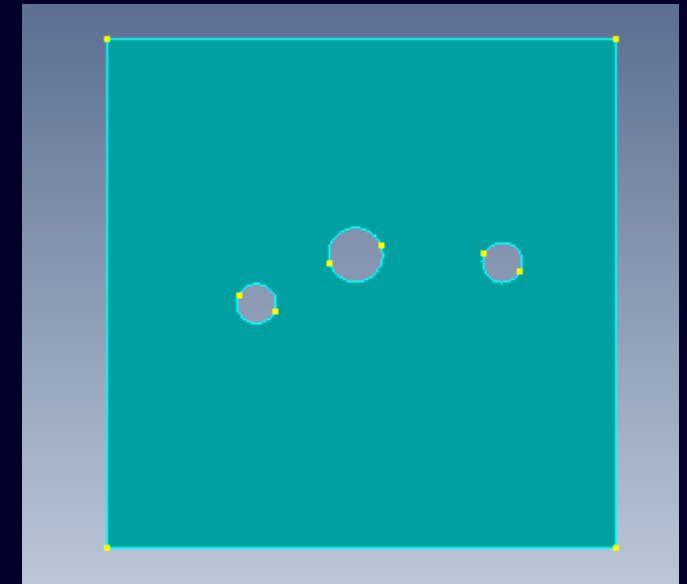
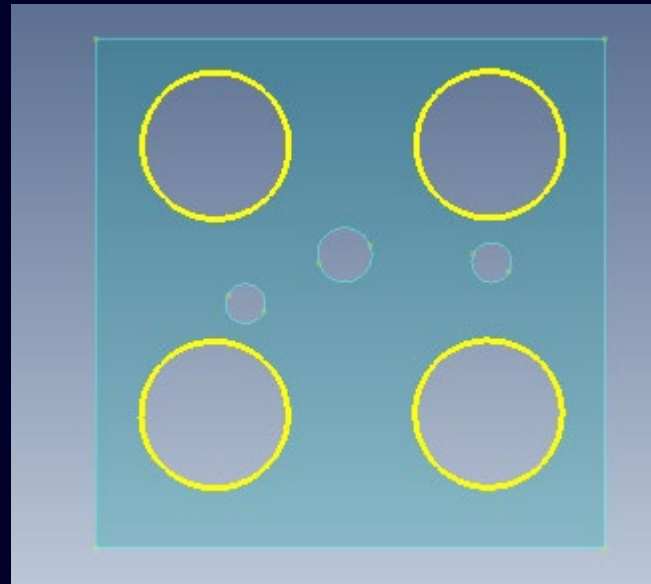
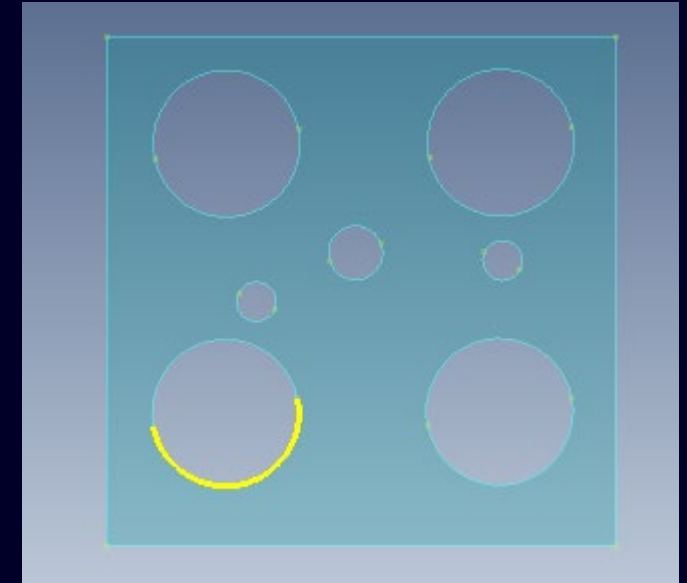
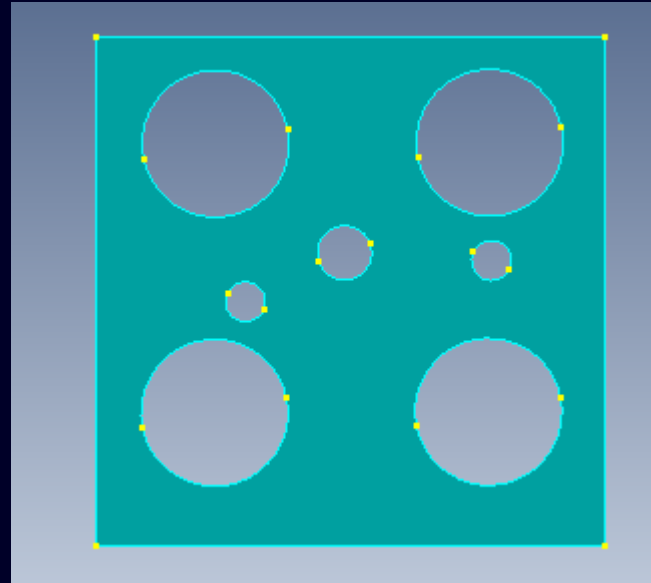
Miscellaneous and API

Simcenter Femap 2401 Geometry

“Add Matching Holes” has been added to the *Pick*^ menu of the Standard Entity Selection dialog box for curves

Simply select a curve of any hole and then use this option to select all other holes in the model which have the same radius as the selected curve

Helpful when using the *Geometry -> Surface -> Remove Hole* command, along with any tool in the *Meshing Toolbox* used for defeaturing geometry which contains holes





What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

Simcenter Femap 2401

Laminates – Layup Builder

The *Layup Builder*, a new Dockable Pane, can be used to interactively create any number of Layups using an intuitive table control

- Accessed by the *Tools -> Layup Builder* command

Layup Builder

ID = 1
Title = 7 to 15 Ply Layup Stack

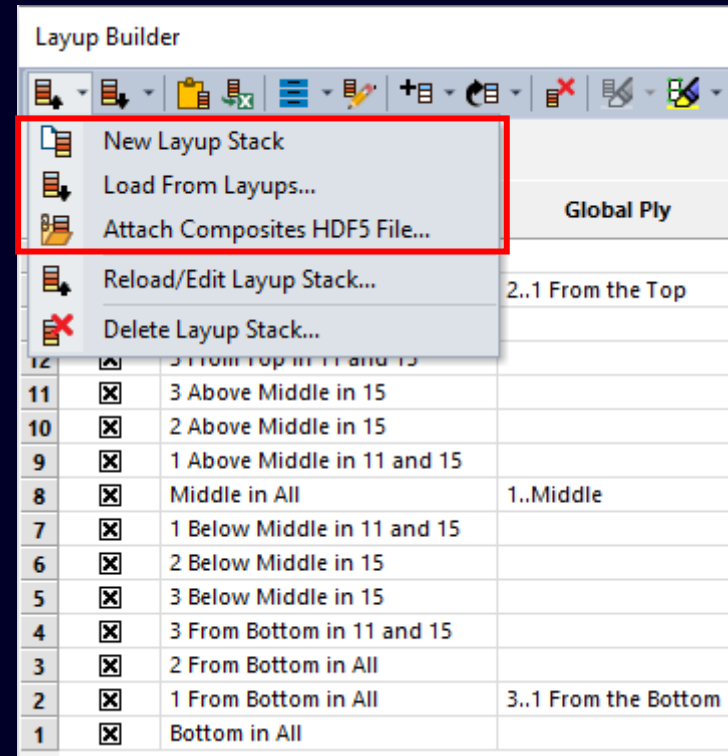
	On/Off	Title	Global Ply	Material	Thickness	Orientation			Entity Selection
						Angle	Direction		
15	<input checked="" type="checkbox"/>	Top in All		2..Glass/EP, Jones p. 70	0.1	0.			480 Elements
14	<input checked="" type="checkbox"/>	1 From Top in All	2..1 From the Top	2..Glass/EP, Jones p. 70	0.1	45.			480 Elements
13	<input checked="" type="checkbox"/>	2 From Top in All		2..Glass/EP, Jones p. 70	0.1	-45.			480 Elements
12	<input checked="" type="checkbox"/>	3 From Top in 11 and 15		2..Glass/EP, Jones p. 70	0.1	0.			324 Elements
11	<input checked="" type="checkbox"/>	3 Above Middle in 15		2..Glass/EP, Jones p. 70	0.1	90.			156 Elements
10	<input checked="" type="checkbox"/>	2 Above Middle in 15		2..Glass/EP, Jones p. 70	0.1	-45.			156 Elements
9	<input checked="" type="checkbox"/>	1 Above Middle in 11 and 15		2..Glass/EP, Jones p. 70	0.1	45.			324 Elements
8	<input checked="" type="checkbox"/>	Middle in All	1..Middle	2..Glass/EP, Jones p. 70	0.1	90.			480 Elements
7	<input checked="" type="checkbox"/>	1 Below Middle in 11 and 15		2..Glass/EP, Jones p. 70	0.1	45.			324 Elements
6	<input checked="" type="checkbox"/>	2 Below Middle in 15		2..Glass/EP, Jones p. 70	0.1	-45.			156 Elements
5	<input checked="" type="checkbox"/>	3 Below Middle in 15		2..Glass/EP, Jones p. 70	0.1	90.			156 Elements
4	<input checked="" type="checkbox"/>	3 From Bottom in 11 and 15		2..Glass/EP, Jones p. 70	0.1	0.			324 Elements
3	<input checked="" type="checkbox"/>	2 From Bottom in All		2..Glass/EP, Jones p. 70	0.1	-45.			480 Elements
2	<input checked="" type="checkbox"/>	1 From Bottom in All	3..1 From the Bottom	2..Glass/EP, Jones p. 70	0.1	45.			480 Elements
1	<input checked="" type="checkbox"/>	Bottom in All		2..Glass/EP, Jones p. 70	0.1	0.	<1,0,0.>		480 Elements

Simcenter Femap 2401

Laminates – Layup Builder

Unlike the *Model -> Laminates -> Layup* command used to create a new Layup in Femap, the *Layup Builder* offers multiple options to create “Layup Stacks” which can then be used to create multiple Layups and Properties at once based on selection of Elements, Surfaces, and/or Groups:

- **New Layup Stack** – Plys are entered into the table control using similar workflow to Layup Manager
- **Load from Layups** – Table control is populated using information from an existing Layup in the model
- **Attach Composites HDF5 File** – Table control is populated using information from an external file created by a different application, which currently can only come from Fibersim, another Siemens DI Software product



Simcenter Femap 2401

Laminates – Layup Builder

When using *Attach Composites HDF5 File*, there can be as many as 3 steps during attach process:

1. Select HDF5 (*.h5) file
2. If units are detected in HDF5 file, and certain units do not match setting for “Geometry Scale Factor” on *Geometry/Model* tab in *File -> Preferences*, user is prompted to scale HDF5 data
3. A preview of the Ply locations is shown in the graphics window and user is prompted to examine locations before continuing

Layup Builder

ID = Not Saved
Title =
Composites HDF5 File = D:\2401 Stuff\Layup_Builder\Fibersim_fairing\LAM001.h5

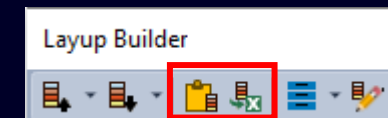
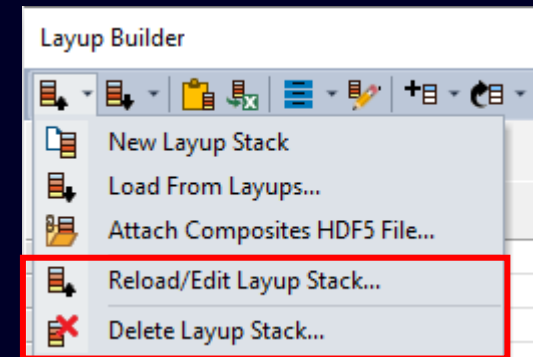
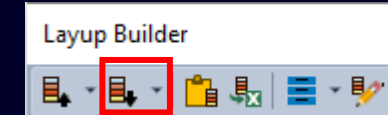
	On/Off	Composites HDF5		Title	Global Ply	Material	Thickness	Orientation			Entity Selection
		Component	Layer					Angle	Direction		
10	<input checked="" type="checkbox"/>	P10	100			3..fabrics/PPG_PL_3K	± 0.1905	± 90.	< ± >		
9	<input checked="" type="checkbox"/>	P9	90			3..fabrics/PPG_PL_3K	± 0.1905	± 45.	< ± >		
8	<input checked="" type="checkbox"/>	P8	80			3..fabrics/PPG_PL_3K	± 0.1905	± -45.	< ± >		
7	<input checked="" type="checkbox"/>	P7	70			3..fabrics/PPG_PL_3K	± 0.1905	± -45.	< ± >		
6	<input checked="" type="checkbox"/>	P6	60			3..fabrics/PPG_PL_3K	± 0.1905	± 45.	< ± >		
5	<input checked="" type="checkbox"/>	P5	50			3..fabrics/PPG_PL_3K	± 0.1905	± -45.	< ± >		
4	<input checked="" type="checkbox"/>	P4	40			3..fabrics/PPG_PL_3K	± 0.1905	± 0.	< ± >		
3	<input checked="" type="checkbox"/>	P3	30			3..fabrics/PPG_PL_3K	± 0.1905	± 45.	< ± >		
2	<input checked="" type="checkbox"/>	P2	20			3..fabrics/PPG_PL_3K	± 0.1905	± -45.	< ± >		
1	<input checked="" type="checkbox"/>	P1	10			3..fabrics/PPG_PL_3K	± 0.1905	± 45.	< ± >		

Simcenter Femap 2401

Laminates – Layup Builder

Once the table control has been populated, various Layup Builder Tools are available to update and investigate the Layup Stack

- **Save Table** – Saves the populated table to the database so it can be reloaded then edited
- **Reload/Edit Layup Stack** – Reloads a saved Layup Stack into the Layup Builder
- **Delete Layup Stack** – Deletes any number of saved Layup Stacks from the database
- **Copy to Clipboard** – Copies contents of Layup Builder to clipboard
- **Send to Excel** – Sends contents of Layup Builder to Excel

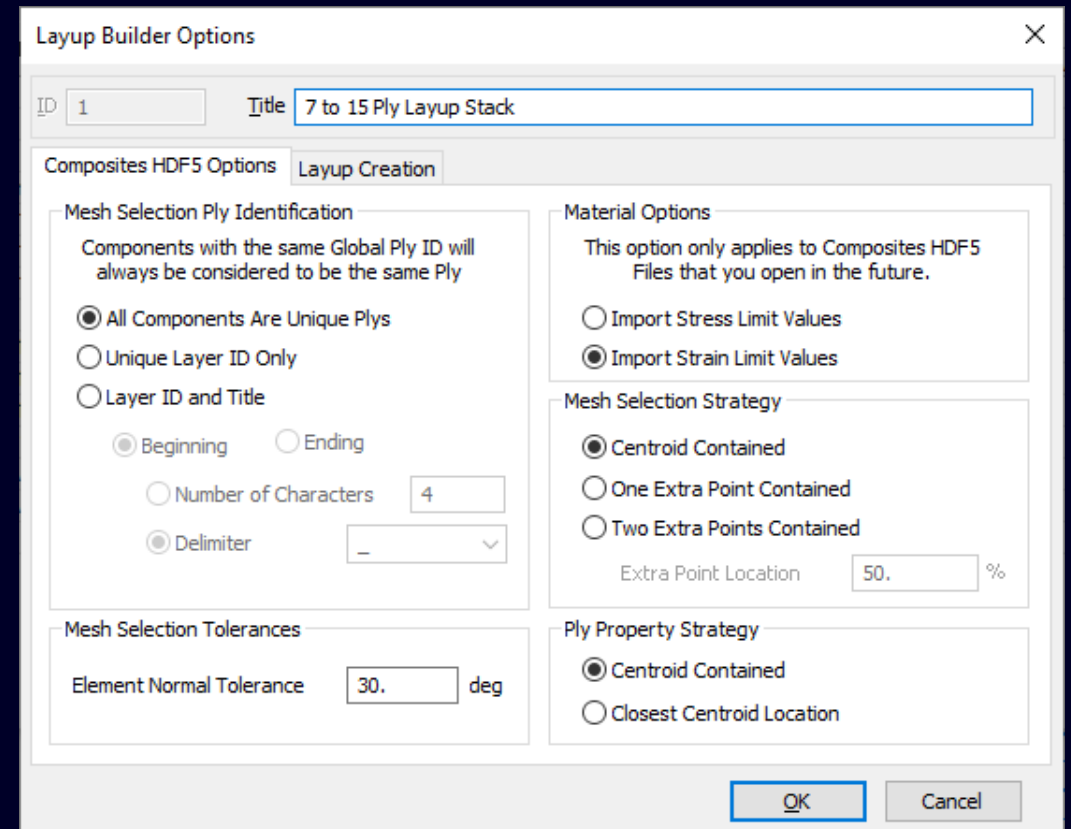
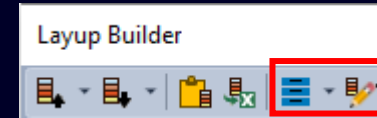


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Tools – All

- **Show/Hide Columns** – Allows user to select which columns are currently being displayed in the table control and toggle the *AutoFit to Window Width* option
- **Edit Layup Builder Options** – Opens *Layup Builder Options* dialog box which contains two tabs:
 - **Composites HDF5 Options** – Specifies options only used during and/or after the population of the table control using *Attach Composites HDF5 File*
 - **Layup Creation** – Specifies options used during creation of Layups as well as others used for viewing and/or showing items in the graphics window

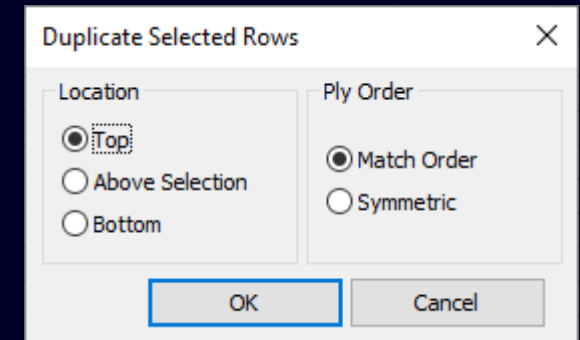
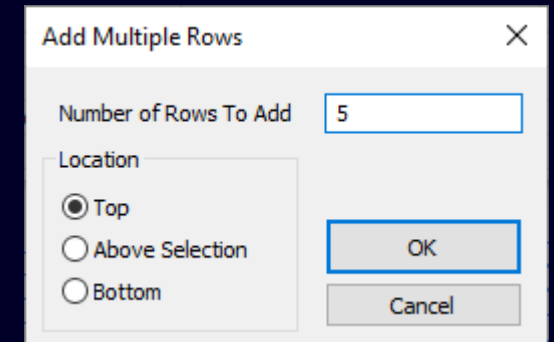
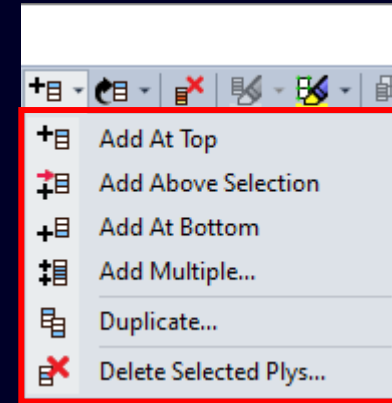


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Tools – All

- **Add at Top/Add at Bottom/Add Above Selection** – Depending on selected command, adds new row to the top, bottom, or above the selected row in the current Layup Stack
- **Add Multiple** – Opens *Add Multiple Rows* dialog box allowing user to specify *Number of Rows to Add* at the top, bottom, or above the selected row in the current Layup Stack
- **Duplicate** – Opens *Duplicate Selected Rows* dialog box allowing user to duplicate selected row(s) by placing them above, below, or above the selected row using either matching order or symmetric order
- **Delete Selected Plys** – Deletes selected row(s) from current Layup Stack

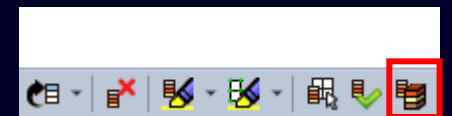
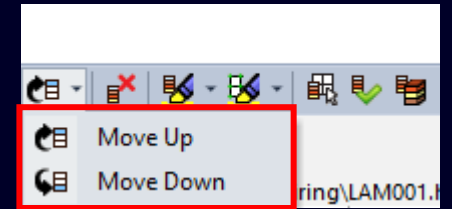


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Tools – All

- **Move Up/Move Down** – Moves selected row(s) up or down one row at a time
- **Clear All** – Clears all contents of the table control
- **Check Ply Info** – Creates the *Errors* Column which indicates if each row has all required info to properly create a ply in any number of Layups
 - To further diagnose why a particular row is not valid, hover the cursor over the field in the Errors column for that row and a “Tooltip” with error information will be displayed
- **Build Layups** – Once everything has been specified, this button is used to automatically create all required Layups



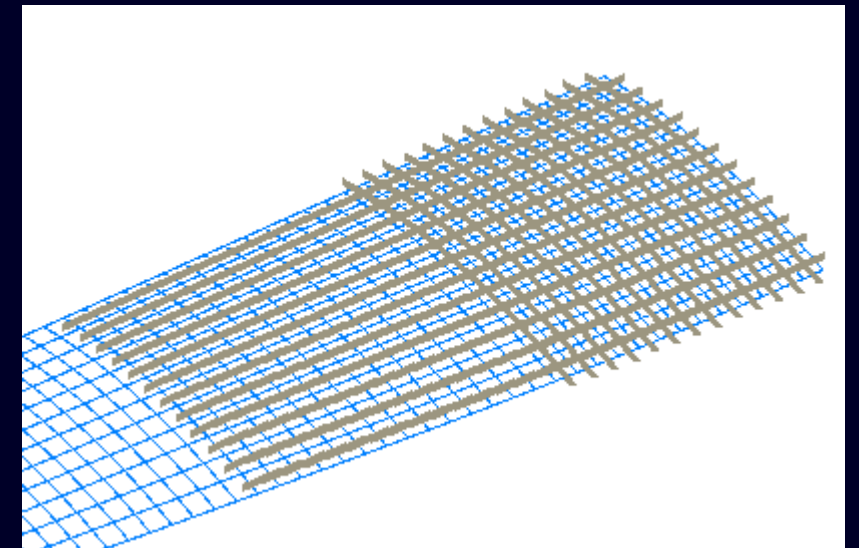
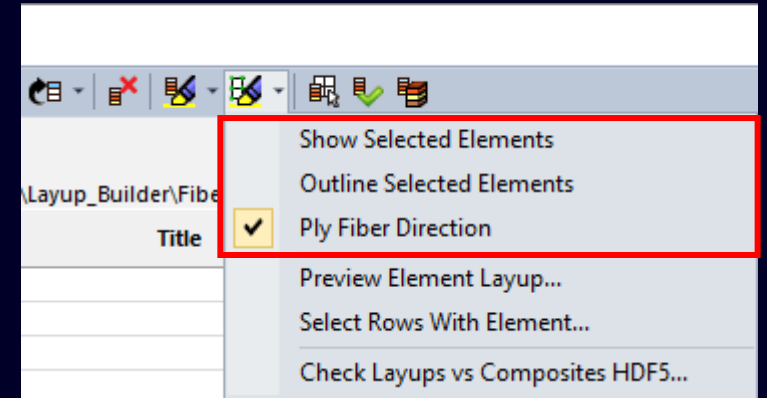
Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Tools – All

- **Show/Preview Mesh Info** – The top section offers options to specify how items will be highlighted in the graphics window:
- **Show Selected Elements** – Highlights all elements which are selected in rows currently selected in the table control
- **Outline Selected Elements** – Highlights outline of all elements which are selected in rows currently selected in the table control
- **Ply Fiber Direction** – Displays Fiber Direction on all elements which are selected in row(s) currently selected in the table control

*Note: Highlight/Display color is specified on Layup Creation tab of Layup Builder Options dialog box

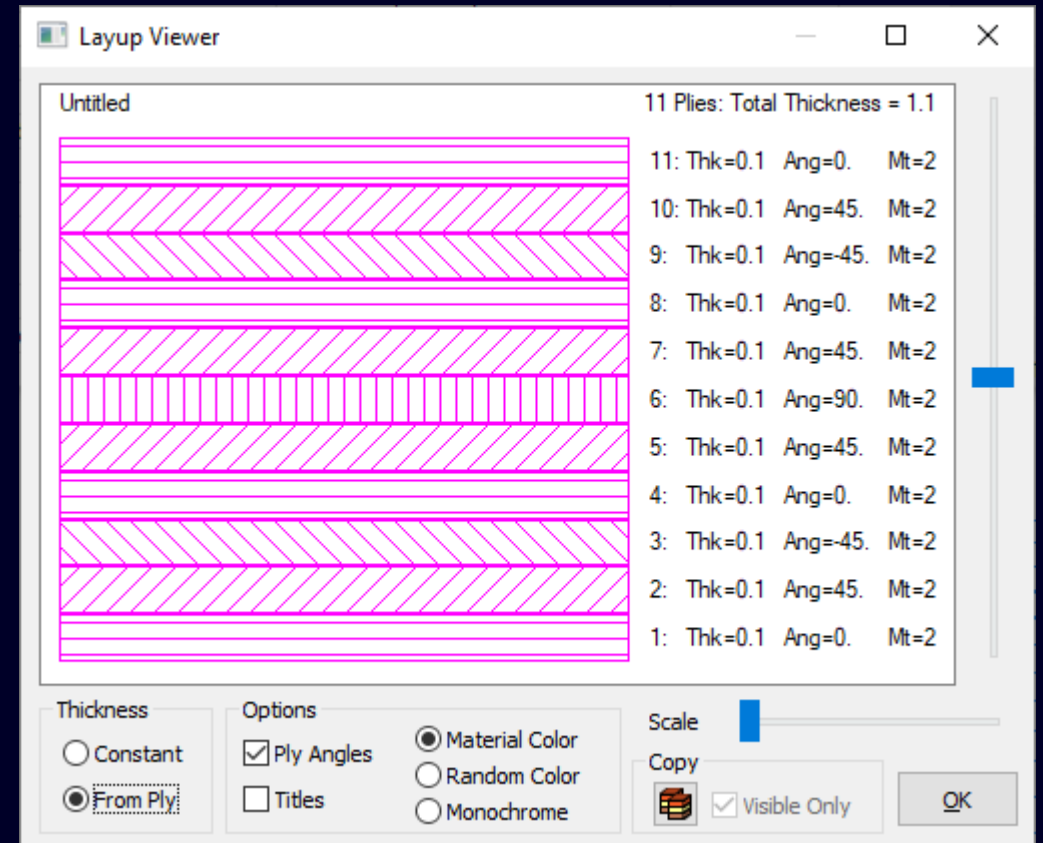
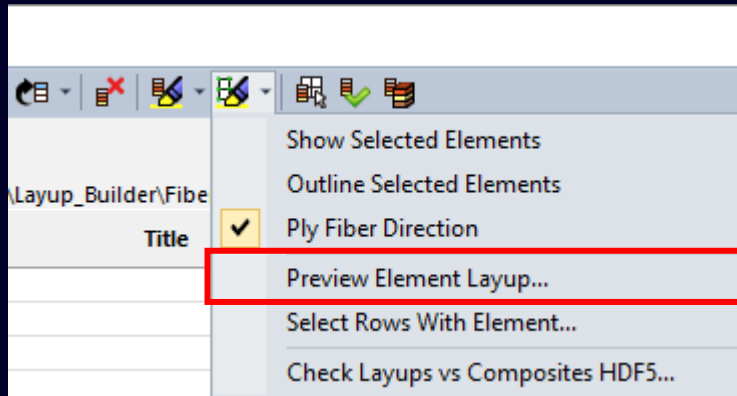


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Tools – All

- **Show/Preview Mesh Info** – The bottom section offers additional tools to further understand what has been defined in Layup Stack
- **Preview Element Layup** – Prompts user to select an element, then displays Layup to be potentially created for that element in Layup Viewer dialog box

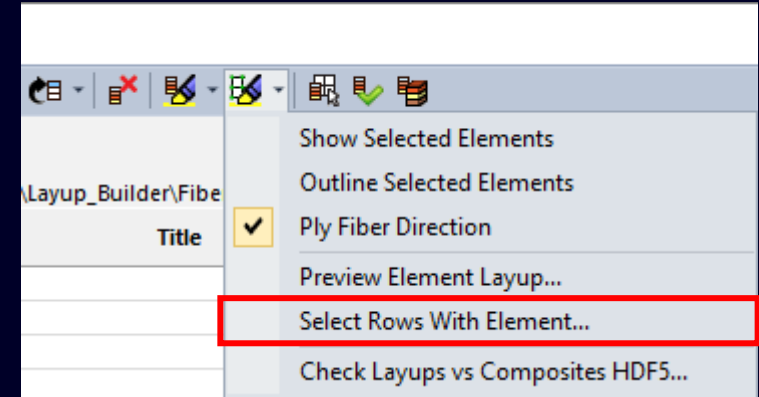


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Tools – All

- **Show/Preview Mesh Info** – The bottom section offers additional tools to further understand what has been defined in Layup Stack
- **Select Rows With Element** – Prompts user to select an element, then highlights which rows have that element currently selected in the table control



Layup Builder									
ID = 1 Title = 7 to 15 Ply Layup Stack									
	On/Off	Title	Global Ply	Material	Thickness	Orientation			Entity Selection
						Angle	Direction		
15	<input checked="" type="checkbox"/>	Top in All		2..Glass/EP, Jones p. 70	0.1	0.			480 Elements
14	<input checked="" type="checkbox"/>	1 From Top in All	2..1 From the Top	2..Glass/EP, Jones p. 70	0.1	45.			480 Elements
13	<input checked="" type="checkbox"/>	2 From Top in All		2..Glass/EP, Jones p. 70	0.1	-45.			480 Elements
12	<input checked="" type="checkbox"/>	3 From Top in 11 and 15		2..Glass/EP, Jones p. 70	0.1	0.			324 Elements
11	<input checked="" type="checkbox"/>	3 Above Middle in 15		2..Glass/EP, Jones p. 70	0.1	90.			156 Elements
10	<input checked="" type="checkbox"/>	2 Above Middle in 15		2..Glass/EP, Jones p. 70	0.1	-45.			156 Elements
9	<input checked="" type="checkbox"/>	1 Above Middle in 11 and 15		2..Glass/EP, Jones p. 70	0.1	45.			324 Elements
8	<input checked="" type="checkbox"/>	Middle in All	1..Middle	2..Glass/EP, Jones p. 70	0.1	90.			480 Elements
7	<input checked="" type="checkbox"/>	1 Below Middle in 11 and 15		2..Glass/EP, Jones p. 70	0.1	45.			324 Elements
6	<input checked="" type="checkbox"/>	2 Below Middle in 15		2..Glass/EP, Jones p. 70	0.1	-45.			156 Elements
5	<input checked="" type="checkbox"/>	3 Below Middle in 15		2..Glass/EP, Jones p. 70	0.1	90.			156 Elements
4	<input checked="" type="checkbox"/>	3 From Bottom in 11 and 15		2..Glass/EP, Jones p. 70	0.1	0.			324 Elements
3	<input checked="" type="checkbox"/>	2 From Bottom in All		2..Glass/EP, Jones p. 70	0.1	-45.			480 Elements
2	<input checked="" type="checkbox"/>	1 From Bottom in All	3..1 From the Bottom	2..Glass/EP, Jones p. 70	0.1	45.			480 Elements
1	<input checked="" type="checkbox"/>	Bottom in All		2..Glass/EP, Jones p. 70	0.1	0.	<1,0,0.>		480 Elements

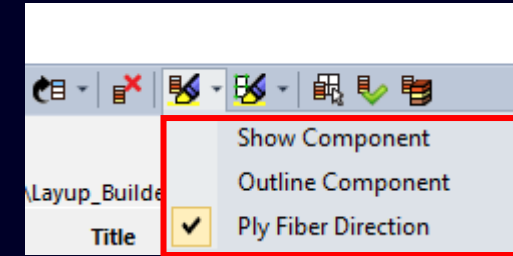
Simcenter Femap 2401

Laminates – Layup Builder

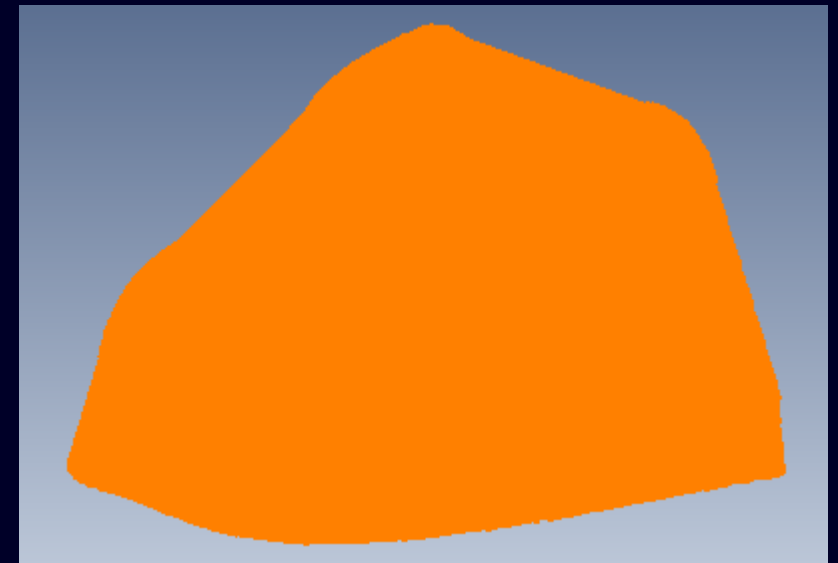
Layup Builder Tools – Composites HDF5 Only

- **Show Composites HDF5 Info** – Provides 3 options to display data from HDF5 file in the graphics window:
 - **Show Component** – Displays all “triangles” which are found in the HDF5 file as a “Filled Color Surface”
 - **Outline Component** – Displays overall outline of each component based on the “triangles” found in the HDF5 file
 - **Ply Fiber Direction** – Displays Fiber Direction using information from the HDF5 File

*Note: Display color is specified on *Layup Creation* tab of *Layup Builder Options* dialog box



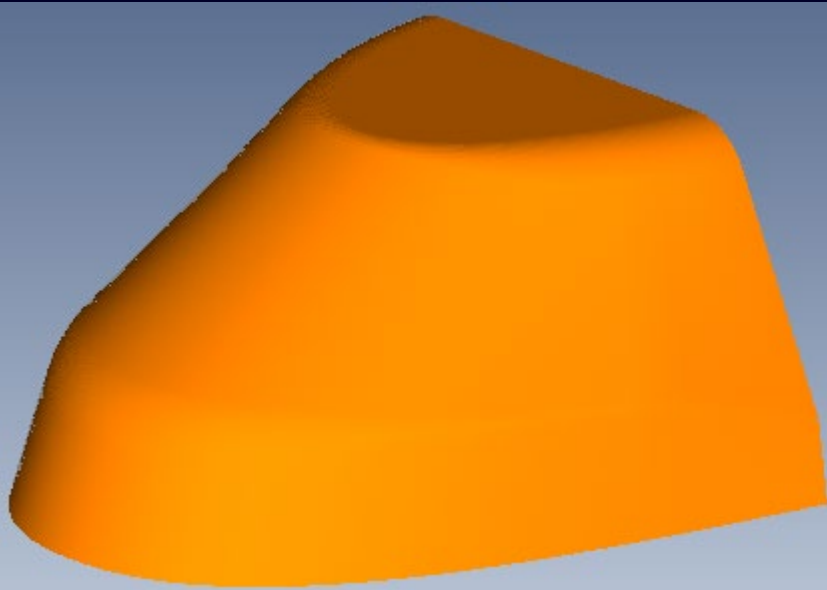
Ply Fiber Direction



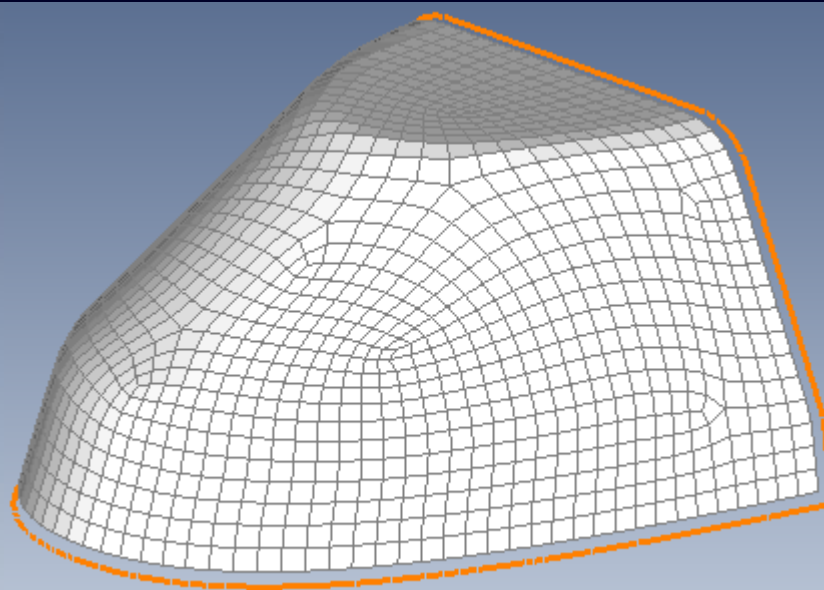
Simcenter Femap 2401

Laminates – Layup Builder

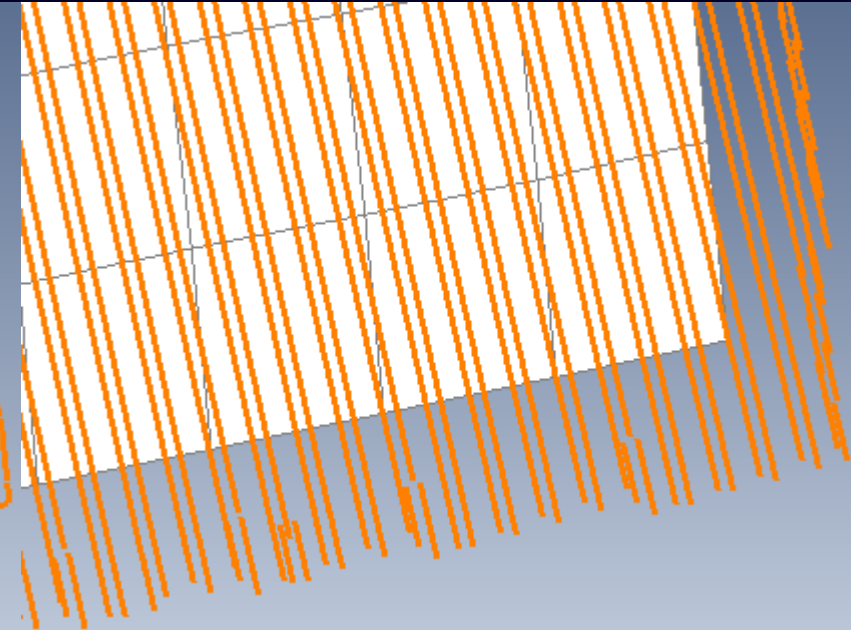
Show Component



Outline Component



Ply Fiber Direction
(Zoomed In)

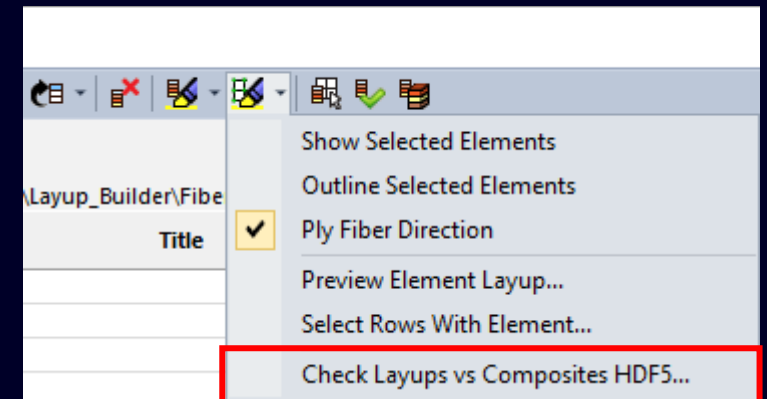
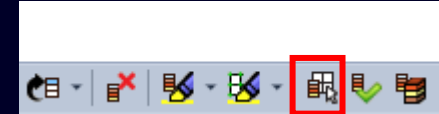


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Tools – Composites HDF5 Only

- **Select Mesh from Composites HDF5 Ply Info** – Uses “triangles” and other information from HDF5 file to automatically determine which elements should be selected by each row
 - Automatic selection of elements can be adjusted via *Composites HDF5 Options* tab of the *Layup Builder Options* dialog box
- **Show/Preview Mesh Info** – The bottom section offers additional tools to further understand what has been defined in Layup Stack
- **Check Layups vs Composites HDF5** – Lists information about the selected element(s) to the Messages window (“worst case”) and Data Table (individual elements) which compares what is specified for each element with the data from the attached HDF5 file
- This command is meant to perform “spot checking” only, as choosing too many elements will result in a large amount of data



Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns

Depending on the method used to populate the table control, only certain columns are available

- For *New Layup Stack* and *Load from Layups* the available columns are:

On/Off	Title	Global Ply	Material	Thickness	Orientation			Failure Theory	Entity Selection	Errors
					Angle	Direction				
<input checked="" type="checkbox"/>				0.	0.			From Property		

- For *Attach Composites HDF5 File* the available columns are:

On/Off	Composites HDF5		Title	Global Ply	Material	Thickness	Orientation			Failure Theory	Entity Selection	Errors
	Component	Layer					Angle	Direction				
<input checked="" type="checkbox"/>	P1	10			2..fabrics/PPG_PL_3K	± 0.1905	± 45.	< ± >		From Property		

Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns

The following Columns do not have a Context-Sensitive menu

- **On/Off** – Used by Build Layups tool to know if a row should be considered (box with “x”) or not (“empty” box) when creating new Layups and Properties
- **Title** – Allows user to enter a descriptive title for any row, but title is only used in *Layup Builder* and does not provide any ability to automatically title Layups or Properties

All other Columns have a Context-Sensitive menu used for various purposes

- **Composite HDF5 (Component and Layer)** – Always automatically populated by attached HDF5, but can become editable using context-sensitive menu
- **Global Ply** – Used to select a Global Ply for a row via a drop-down control
- **Material** – Used to select a Material for a row via drop-down control or can be automatically populated by attached HDF5
- **Thickness** – Used to enter a Thickness value directly in cell of a row or can be automatically populated by attached HDF5

On/Off	Title
<input checked="" type="checkbox"/>	Top in All
<input checked="" type="checkbox"/>	1 From Top in All
<input checked="" type="checkbox"/>	2 From Top in All
<input checked="" type="checkbox"/>	3 From Top in 11 and 15
<input checked="" type="checkbox"/>	3 Above Middle in 15
<input checked="" type="checkbox"/>	2 Above Middle in 15
<input checked="" type="checkbox"/>	1 Above Middle in 11 and 15
<input checked="" type="checkbox"/>	Middle in All
<input checked="" type="checkbox"/>	1 Below Middle in 11 and 15
<input checked="" type="checkbox"/>	2 Below Middle in 15
<input checked="" type="checkbox"/>	3 Below Middle in 15
<input checked="" type="checkbox"/>	3 From Bottom in 11 and 15
<input checked="" type="checkbox"/>	2 From Bottom in All
<input checked="" type="checkbox"/>	1 From Bottom in All
<input checked="" type="checkbox"/>	Bottom in All

Composites HDF5	
Component	Layer
P10	100
P9	90
P8	80
P7	70
P6	60
P5	50
P4	40
P3	30
P2	20
P1	10

Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns

- **Orientation** – This section contains 3 columns used for ply orientation purposes:
 - **Angle** – Used to specify orientation angle of ply to be created by a particular row with regard to the “0° direction”
 - Typically angle values in cells can be edited directly
 - The “0° direction” can be defined:
 - By each element itself based on node order (default)
 - Within Femap by using Modify -> Update Elements -> Material Orientation command
 - By specifying a Fiber Direction in the Direction Column in the Layup Editor
 - **Direction** – Contains information about Material Orientation/Fiber Direction of each row
 - If not specified, element orientation or Material Orientation used as “0° direction”
 - If specified, vector or other values specified used as “0° direction”
 - **Use Ply to Set Material Orientation (Symbol)** – Indicates if the Use Ply to Set Material Orientation option is turned on or off for a particular row

Orientation		
Angle	Direction	
0.		
45.		
-45.		
0.		
90.		
-45.		
45.		
90.		
45.		
-45.		
90.		
0.		
-45.		
45.		
0.	<1.,0.,0.>	

Orientation		
Angle	Direction	
± 90.	< ± >	
± 45.	< ± >	
± -45.	< ± >	
± -45.	< ± >	
± 45.	< ± >	
± -45.	< ± >	
± 0.	< ± >	
± 45.	< ± >	
± -45.	< ± >	
± 45.	< ± >	

Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns

- **Entity Selection** – Used to select which elements will be considered for each row when automatically creating Layups and Properties
 - 3 manual methods for element selection exist via Context-Sensitive menu:
 - Elements selected by ID
 - Elements associated to selected Surfaces
 - Elements which are referenced by a single selected Group
 - When using a Layup Stack populated by *Attach Composites HDF5 File*, the user can have elements automatically selected for each row by using the Select Mesh From Composites HDF5 Ply Info tool
- *Note: If the mesh in the model or any of the values in the Layup Stack have changed since the initial use of the Select Mesh From Composites HDF5 Ply Info tool, it should be used again to ensure that all the proper elements are assigned to each row

Entity Selection
480 Elements
40 Surfaces
480 Elements
324 Elements
48 Elements ; 4 Surfaces ; 101..Construct 1
144 Elements
324 Elements
480 Elements
324 Elements
12 Surfaces
156 Elements ; 101..Construct 1
324 Elements
480 Elements
480 Elements
480 Elements

Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns – Context Sensitive menus

- **Composites HDF5 (Component and Layer) – HDF5 Only**
 - **Auto Create Title** – Automatically creates a Title based on text and values in Component and Layer columns
 - **Auto Create Global Ply** – Automatically creates a Global Ply with information from row (“Title” =“Composites HDF5 Layer (Layer ID)”)
 - **Allow Edit of Composites HDF5 Columns** – When on, allows user to manually edit text and values in the Component and Layer columns
- **Global Ply** – Individual cells can typically be modified via drop-down
 - **Auto Create Global Ply** – Automatically creates a Global Ply based on information in each selected row (“Title” varies based on fields)
 - **Clear Selected** – Clears selected Global Ply from selected rows
 - **Global Ply Manager** – Opens Global Ply Manager dialog box

Composites HDF5		Title	
Component	Layer		
P10			
P9			
P8			
P7			
P6			

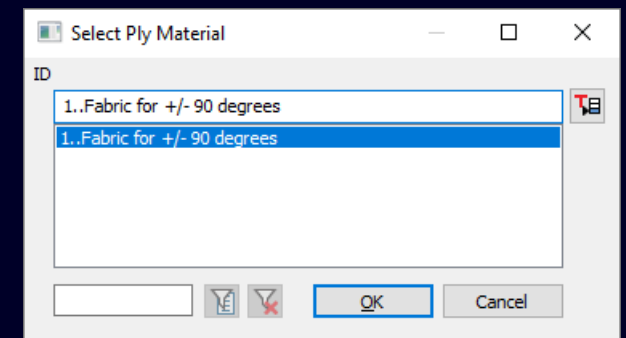
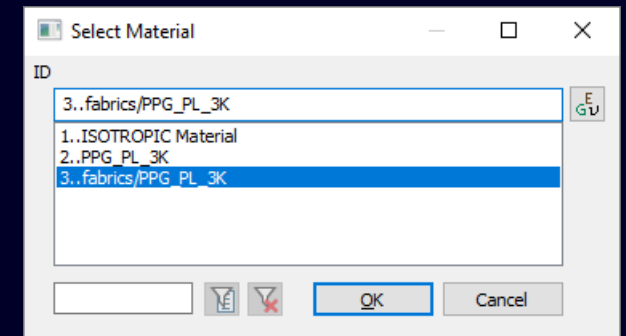
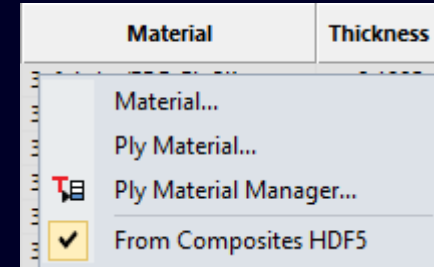
Global Ply	Mat

Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns – Context Sensitive menus

- Material – Individual cells can typically be modified via drop-down
 - **Material** – Opens *Select Material* dialog box which allows selection of an existing Material from a list which can be filtered as well as the ability to create a new Material via Define Material dialog box
 - **Ply Material** – Opens *Select Ply Material* dialog box which allows selection of an existing Ply Material from a list which can be filtered as well as the ability to create a new Ply Material via Ply Material Manager dialog box
 - **Ply Material Manager** – Opens the *Ply Material Manager* dialog box
- *Note: Following option only available when the table control has been populated Attach Composites HDF5 File command
- **From Composites HDF5** – When on, Material specified by information in attached HDF5 file, while when off, Material specified via drop-down or other commands

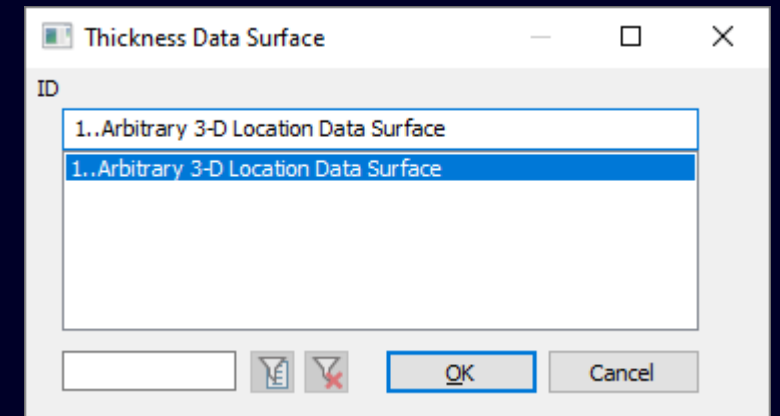
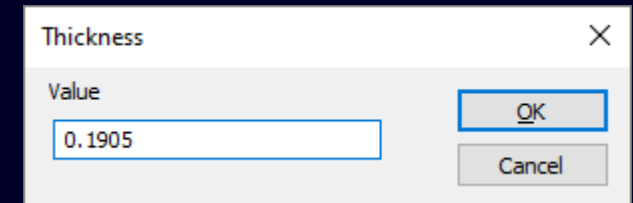
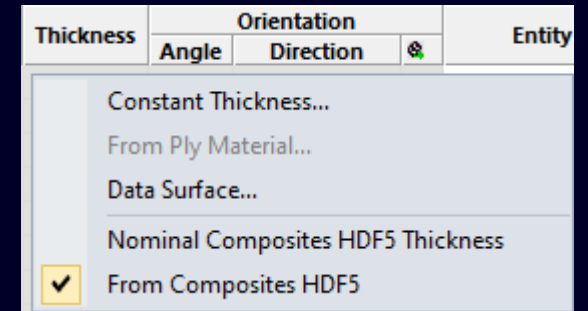


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns – Context Sensitive menus

- **Thickness** – Individual cells can typically be modified directly in cell
 - **Constant Thickness** – Opens *Thickness* dialog box which allows user to enter thickness used for all selected cells
 - **From Ply Material** – Only used when a Ply Material has been selected in the Material cell for a row
 - **Data Surface** – If certain types of Data Surfaces exist in the model, one can be selected using the *Thickness Data Surface* dialog box
- *Note: Following options only available when the table control has been populated Attach Composites HDF5 File command
- **Nominal Composites HDF5 Thickness** – Converts Thickness data from attached HDF5 file to a Thickness which can be modified
 - **From Composites HDF5** – When on, Thickness specified by information in attached HDF5 file, while when off, Thickness is specified by other commands

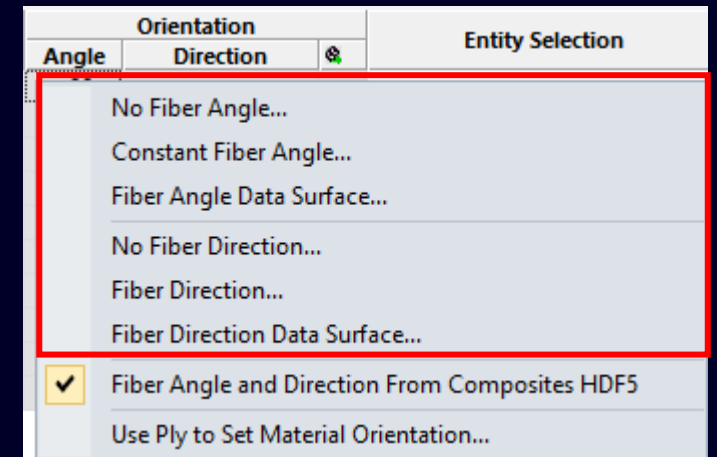


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns – Context Sensitive menus

- **Orientation (Angle, Direction, and Use Ply to Set Material Orientation)**
 - **No Fiber Angle** – Clears value of selected cells in Angle column
 - **Constant Fiber Angle** – Sets value of selected cells in Angle column to a single specified value
 - **Fiber Angle Data Surface** - Sets value of selected cells in Angle column to reference a scalar-based Data Surface
- **No Fiber Direction** – Clears vector and/or other values of selected cells in the Direction column
- **Fiber Direction** – Specifies vector values of selected cells in the Direction column using Standard Vector Definition dialog box
- **Fiber Direction Data Surface** – Specifies vector values of selected cells in the Direction column using a vector-based Date Surface

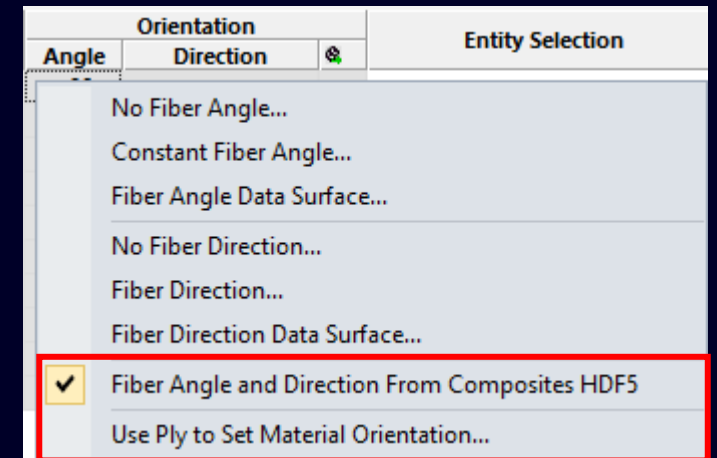


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns – Context Sensitive menus

- **Orientation (Angle, Direction, and Use Ply to Set Material Orientation)**
 - **Fiber Angle and Direction From Composites HDF5** – When on, values for both the Angle and Direction cells in a particular row are specified by information in attached HDF5 file
 - **Use Ply to Set Material Orientation** – When on, uses the vector or other values specified in the Direction column to set the Material Orientation of elements selected in Entity Selection Column
 - If user wants to set the same Material Orientation/Fiber Direction for all elements currently selected in Layup Builder, this option and Direction only need to be set in “Row 1”

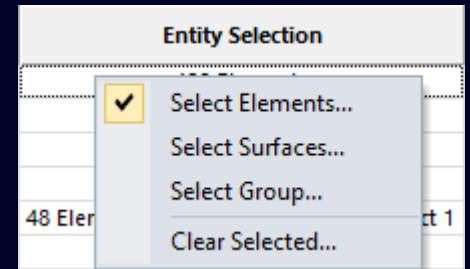


Simcenter Femap 2401

Laminates – Layup Builder

Layup Builder Columns – Context Sensitive menus

- **Entity Selection** – Values cannot be edited directly within cell
 - **Select Elements** – Use Standard Entity Selection dialog box for elements to select any number of elements for selected row(s)
 - **Select Surfaces** – Use Standard Entity Selection dialog box for surfaces to select elements associated any number of surfaces for selected row(s)
 - **Select Group** – Use Select Group dialog box to select a single group which references elements for selected row(s)
 - **Clear Selected** – Clears all selected entity types from selected row(s)
- **Errors** – While the cells in the Errors column do not actually have a Context-Sensitive menu, if a cell is **Red**, place the cursor over the cell and a Tooltip will appear explaining why a particular row is invalid



Simcenter Femap 2401

Laminates – Add Plys to Mesh

Model -> Laminates -> Add Plys to Mesh command

- **Allows user to add plys to any number of selected laminate shell elements by automatically creating all Layups and corresponding Properties needed to update the elements based on the settings for the *Where to Add*, *Ply Type*, *Ply Definition*, and *Nastran SOL 401/402 Property* sections in the *Add New Ply to Mesh* dialog box**

Add New Ply to Mesh

Title: Add 1 Ply above "Middle" Global Ply

Where to Add

Top Bottom Above Global Ply Below Global Ply 1..Middle

Ply Type

Ply Number of Plys: 1

Ply Material

Layup

Layup - Symmetric

Layup - Anti

Layup - AntiSymmetric

Ply Definition

Material: 2..Glass/EP, Jones p. 70

Thickness: 0.01

Angle: 0

Nastran SOL 401/402 Property

Ply Failure Theory

OK Cancel

Simcenter Femap 2401

Laminates – Add Plys to Mesh

Model -> Laminates -> Add Plys to Mesh command

- **Title** – Enter a descriptive title, if desired
- **Where to Add** – Choose *Top*, *Bottom*, *Above Global Ply*, or *Below a Global Ply*
- **Ply Type** – Choose *Ply*, *Ply Material*, *Layup*, *Layup – Symmetric*, *Layup – Anti*, or *Layup – AntiSymmetric*
- **Ply Definition** – This section changes depending on option set in *Ply Type*:
 - **Ply** – Specify *Material*, *Thickness*, and *Angle*
 - **Ply Material** – Specify *Ply Material* and *Angle*
 - **Layup (all options)** – Specify *Layup* and “Base” *Angle* for Layup
- **Nastran SOL 401/402 Property** – Specify *Ply Failure Theory* for SOL 401/402 when *Ply Type* is set to *Ply*

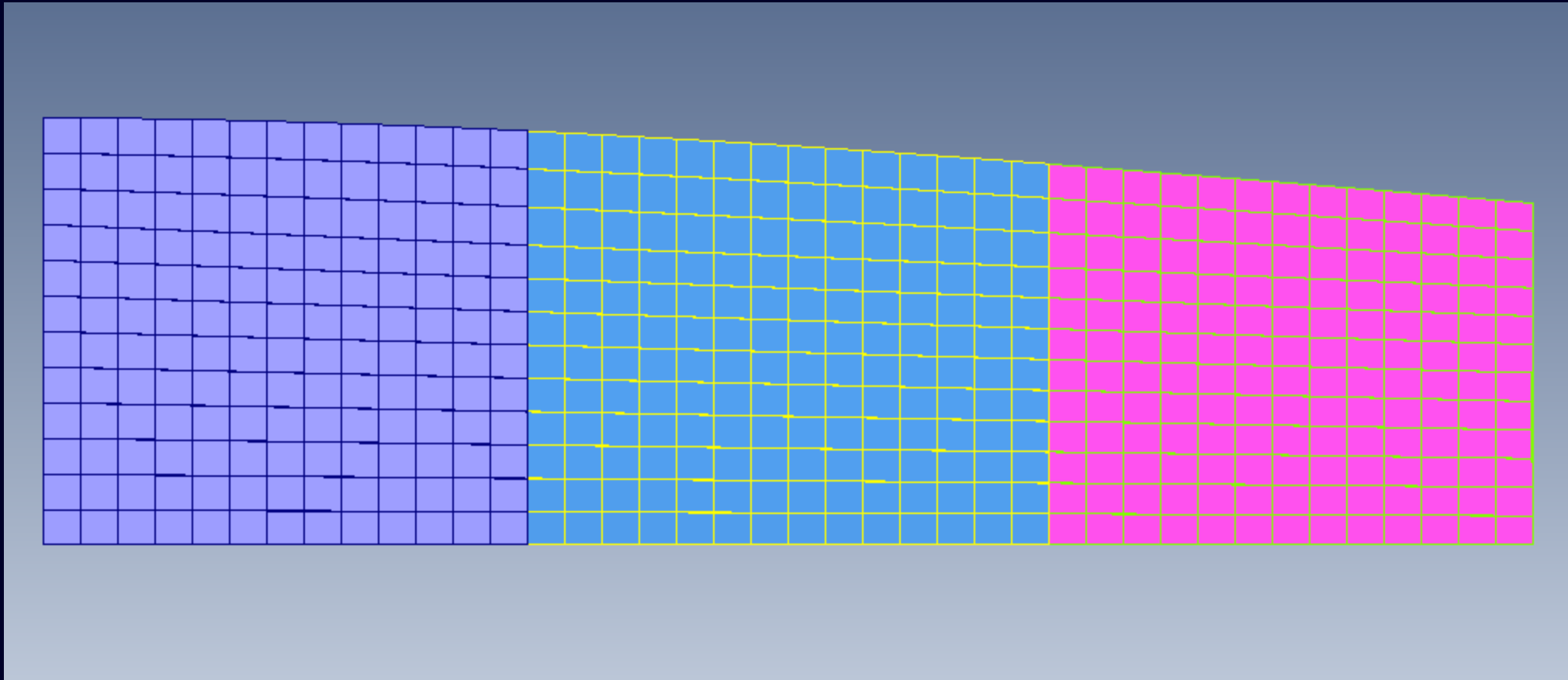
The screenshot shows the 'Add New Ply to Mesh' dialog box. The 'Title' field contains 'Add Ply Material to Bottom'. Under 'Where to Add', the 'Bottom' radio button is selected. The 'Ply Type' section has 'Ply Material' selected, and the 'Number of Plys' is set to 1. The 'Ply Definition' section shows '1..0.01 Thick Ply Material' selected for 'Ply Material', with 'Thickness' and 'Angle' (0) fields. The 'Nastran SOL 401/402 Property' section has 'Ply Failure Theory' selected. 'OK' and 'Cancel' buttons are at the bottom.

The screenshot shows the 'Add New Ply to Mesh' dialog box. The 'Title' field is empty. Under 'Where to Add', the 'Above Global Ply' radio button is selected, and a dropdown menu shows '2..1 From the Top'. The 'Ply Type' section has 'Layup' selected, and the 'Number of Plys' is set to 1. The 'Ply Definition' section shows '1..7 Plies' selected for 'Layup', with 'Thickness' and 'Angle' (90) fields. The 'Nastran SOL 401/402 Property' section has 'Ply Failure Theory' selected. 'OK' and 'Cancel' buttons are at the bottom.

Simcenter Femap 2401

Laminates – Add Plys to Mesh

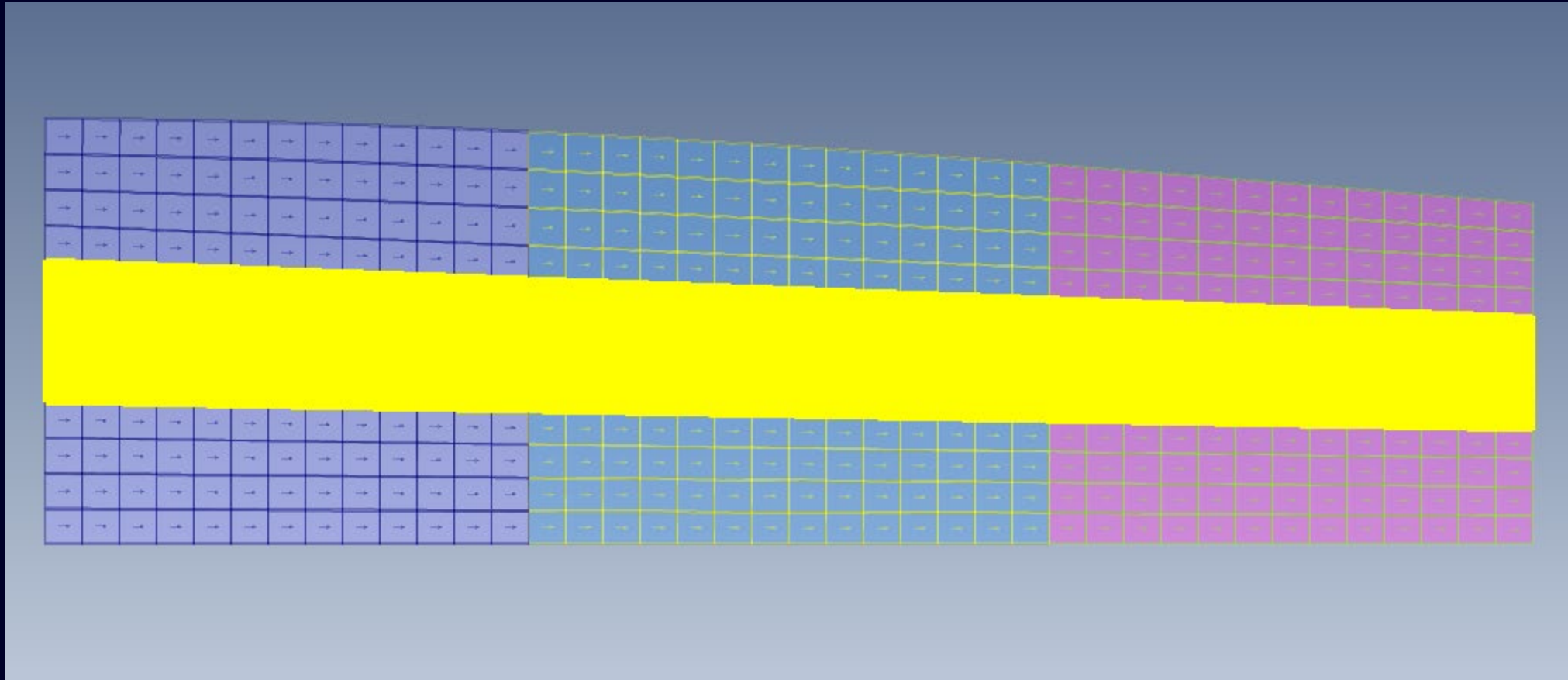
Model -> Laminates -> Add Plys to Mesh command – Original Model has 3 Layups/Properties



Simcenter Femap 2401

Laminates – Add Plys to Mesh

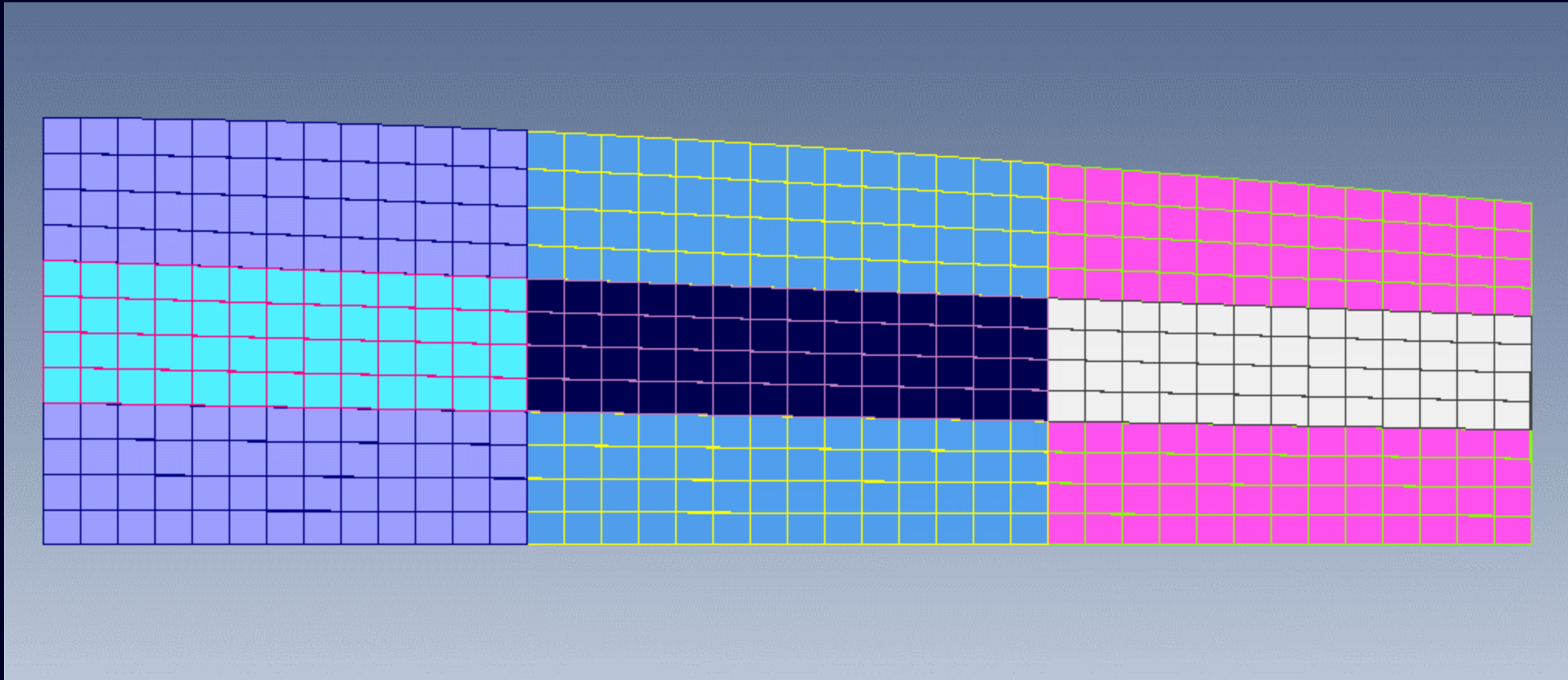
Model -> Laminates -> Add Plys to Mesh command – Elements where ply will be added



Simcenter Femap 2401

Laminates – Add Plys to Mesh

Model -> Laminates -> Add Plys to Mesh command – Updated Model has 6 Layups/Properties



Simcenter Femap 2401

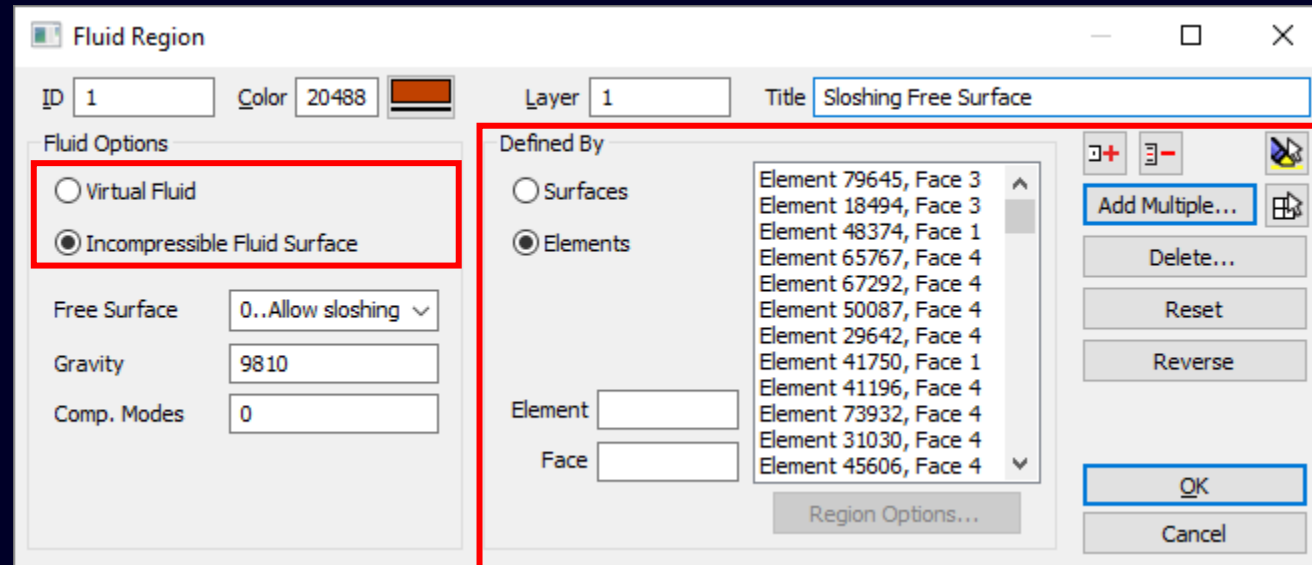
Fluid Region – Incompressible Fluid Surface

The **Connect -> Fluid Region** command displays **Fluid Region** dialog box

The **Fluid Options** section contains two options:

- **Virtual Fluid** - Used to specify options and entities written to MFLUID entry
- **Incompressible Fluid Surface** – Used to specify options and entities written to INCOMPFL entry

Defined By – Regardless of option selected in **Fluid Options**, used to select either faces of elements themselves or determine element faces based on association to selected geometric surfaces



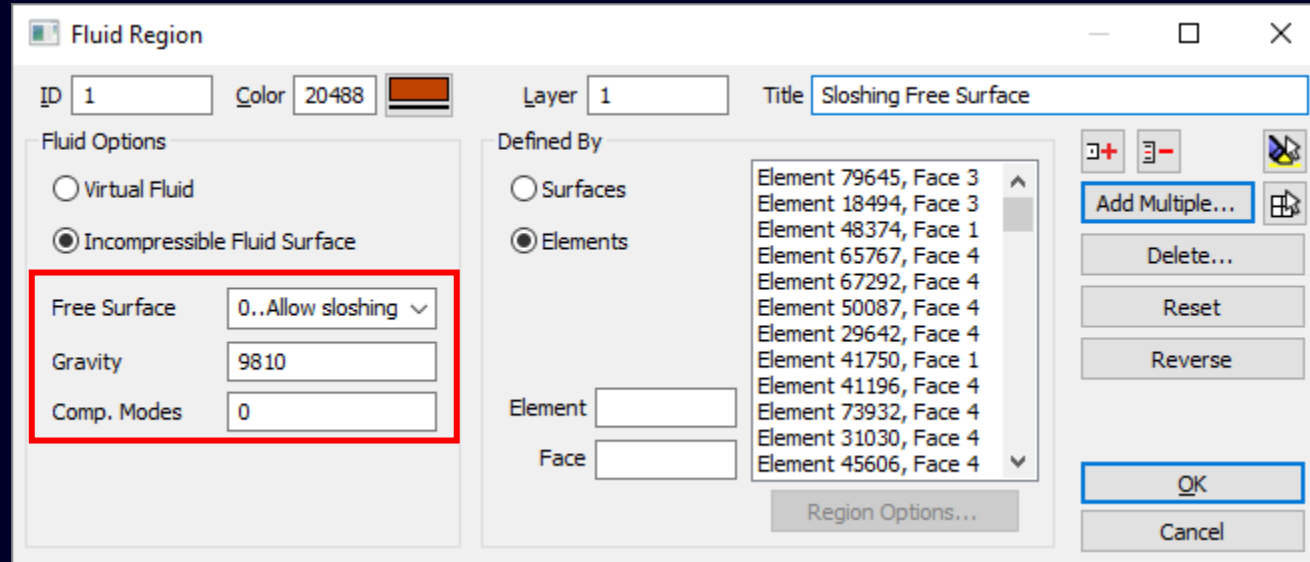
Simcenter Femap 2401

Fluid Region – Incompressible Fluid Surface

The Connect -> Fluid Region command displays Fluid Region dialog box

The Fluid Options section also contains additional options for Incompressible Fluid Surface

- **Free Surface** – Specifies whether fluid volume contains free surface or is completely enclosed by structure
 - If fluid volume contains a free surface, specify whether free surface is sloshing or non-sloshing
- **Gravity** – When free surface is allowed to slosh, sets the acceleration due to gravity
- **Comp. Modes** – Number of modes associated with sloshing surface after Component Mode Synthesis (CMS) reduction



Simcenter Femap 2401

Fluid Region – Incompressible Fluid Surface

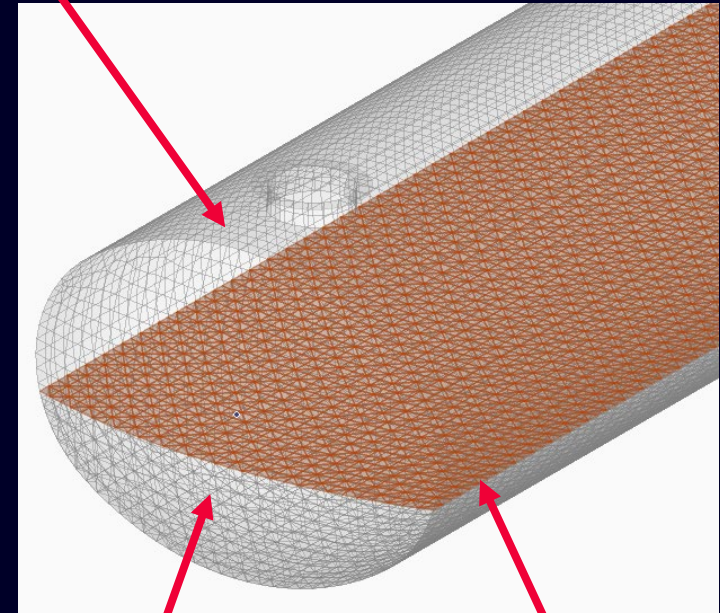
Defining the fluid region for the defined fluid mass approach works differently than defining the fluid region for the virtual fluid mass (MFLUID) approach

- **Defined Fluid Mass** – Specify the free surface for the incompressible fluid
 - If the fluid volume is fully enclosed, then it does not contain a free surface and the region should be empty

*Note: When modeling the structure, the best practice is to use plate elements and avoid using 3D elements

- **Virtual Fluid Mass** – Specify the fluid-structure interface, which are “wetted” plate elements that define the structural portion of the fluid boundary

Structure (Plate Elements)



Fluid Free Surface Region

Fluid (Solid Elements)



What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

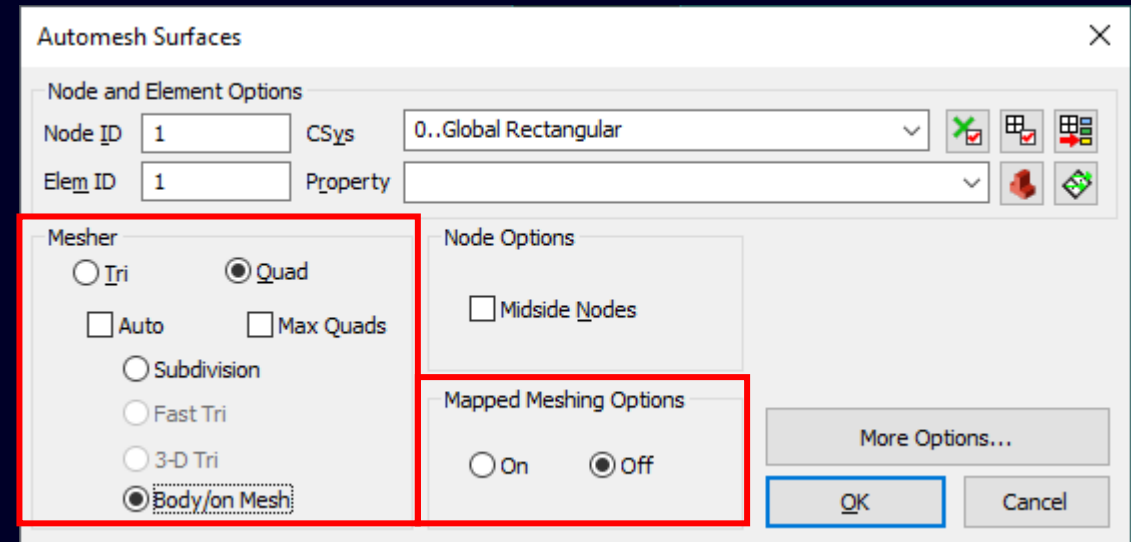
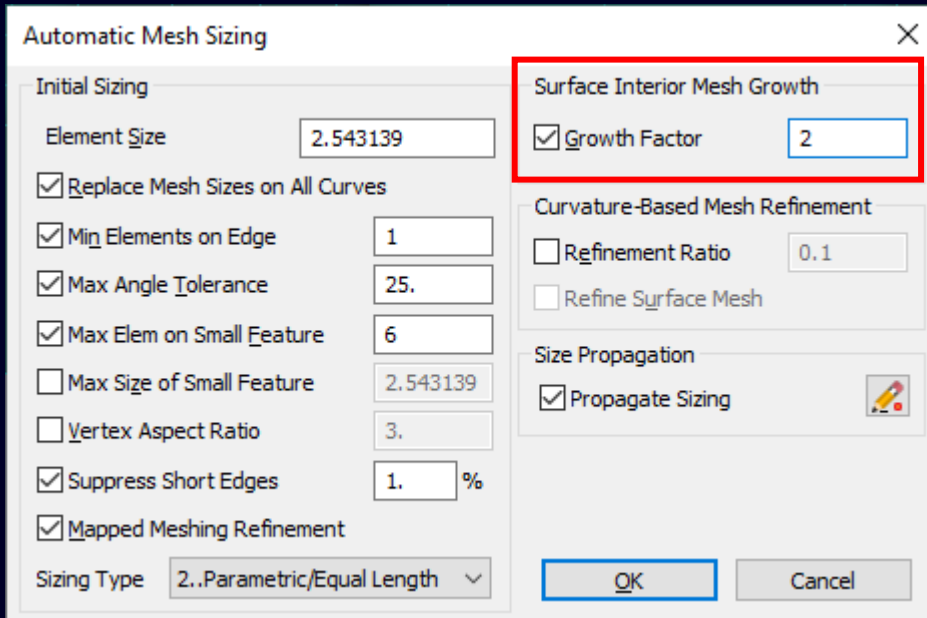
Postprocessing

Miscellaneous and API

Simcenter Femap 2401

Meshing – Body Mesher Enhancements

Updated Mesh -> Geometry -> Surface to attempt to use the Surface Interior Growth Factor specified via the Mesh -> Mesh Control -> Size on Surface or Mesh -> Mesh Control -> Size on Solid when Mesher is set to “Tri” or “Quad” and “Body/on Mesh” and Mapped Meshing Options is set to “Off”



Simcenter Femap 2401

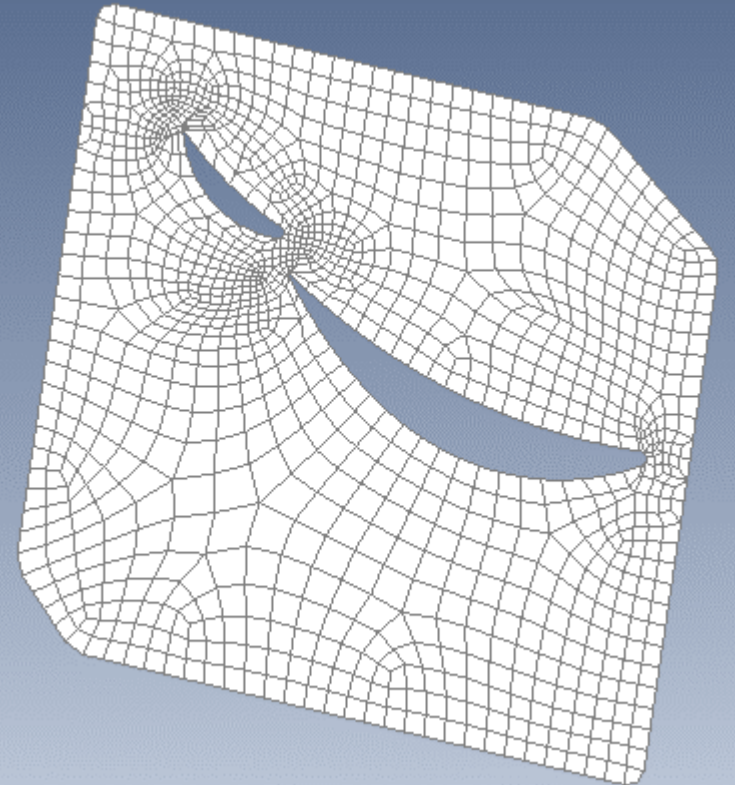
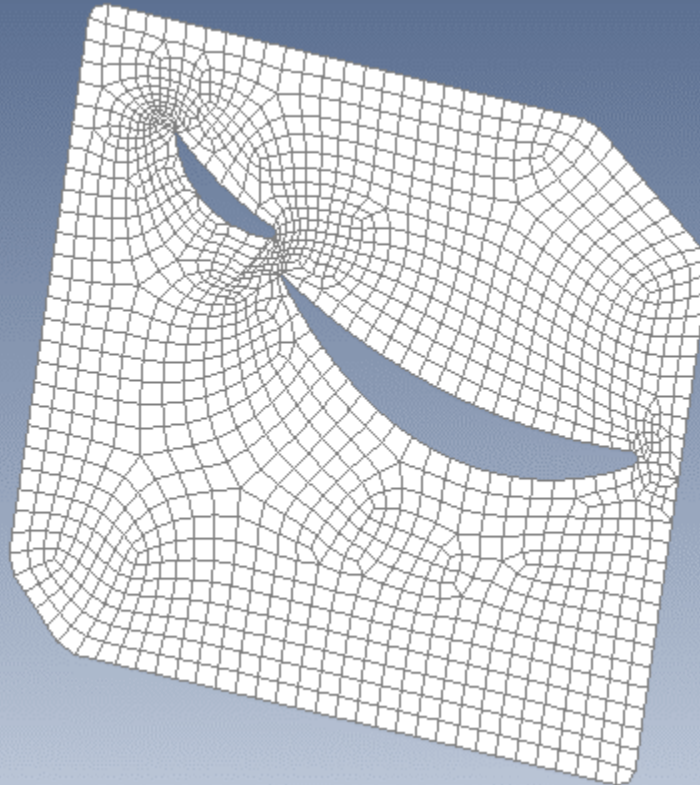
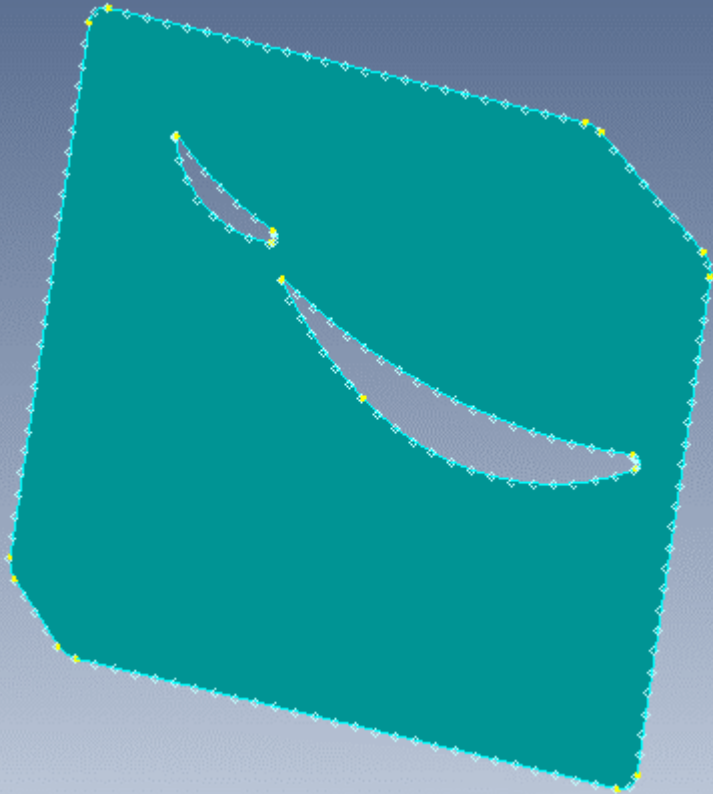
Meshing – Body Mesher Enhancements

Quad Mesh Example

Mesh Size = 6.0

No Growth Ratio

Growth Ratio = 2.0



Simcenter Femap 2401

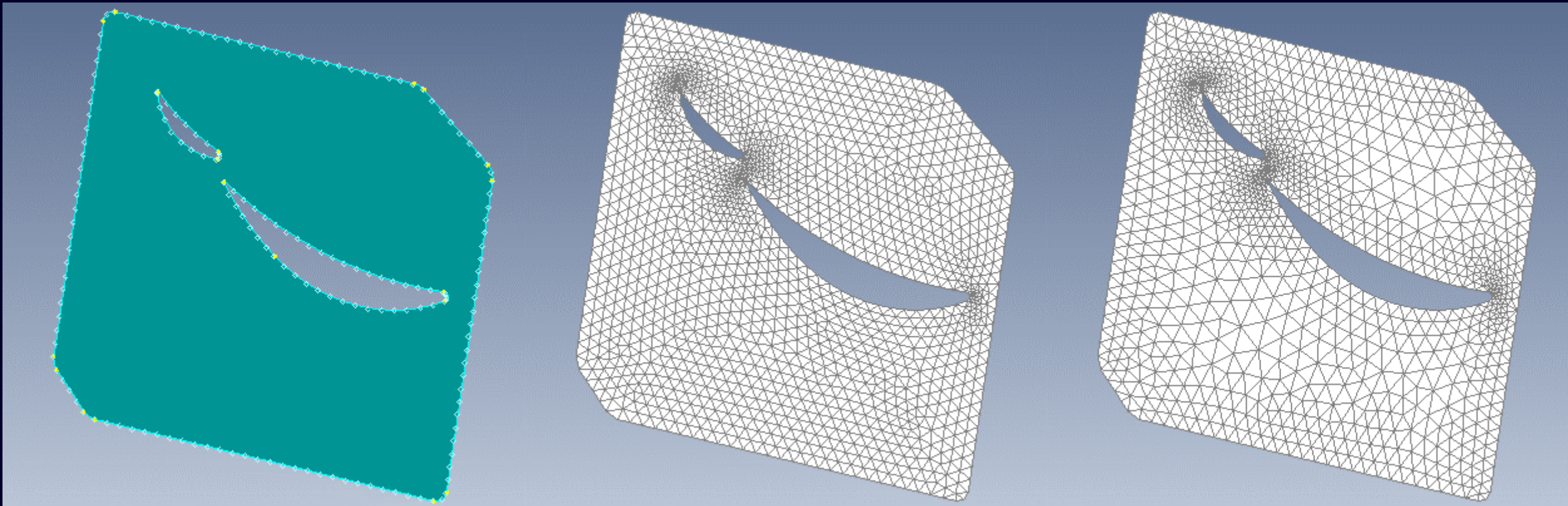
Meshing – Body Mesher Enhancements

Triangle Mesh Example

Mesh Size = 6.0

No Growth Ratio

Growth Ratio = 2.0





What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

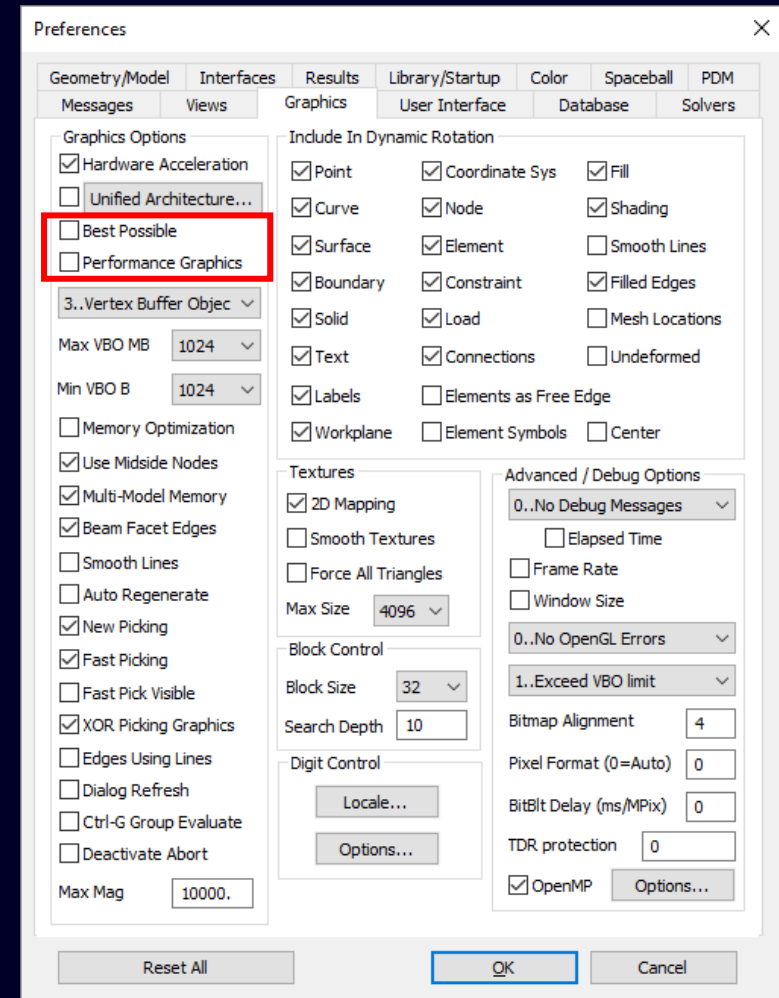
Simcenter Femap 2401

Performance Improvements – Unified Graphics Architecture

Femap currently contains three different “pipelines” for graphics: “original OpenGL”, “Performance Graphics”, and “Unified Graphics Architecture (UGA)”

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 the original two “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”



Simcenter Femap 2401

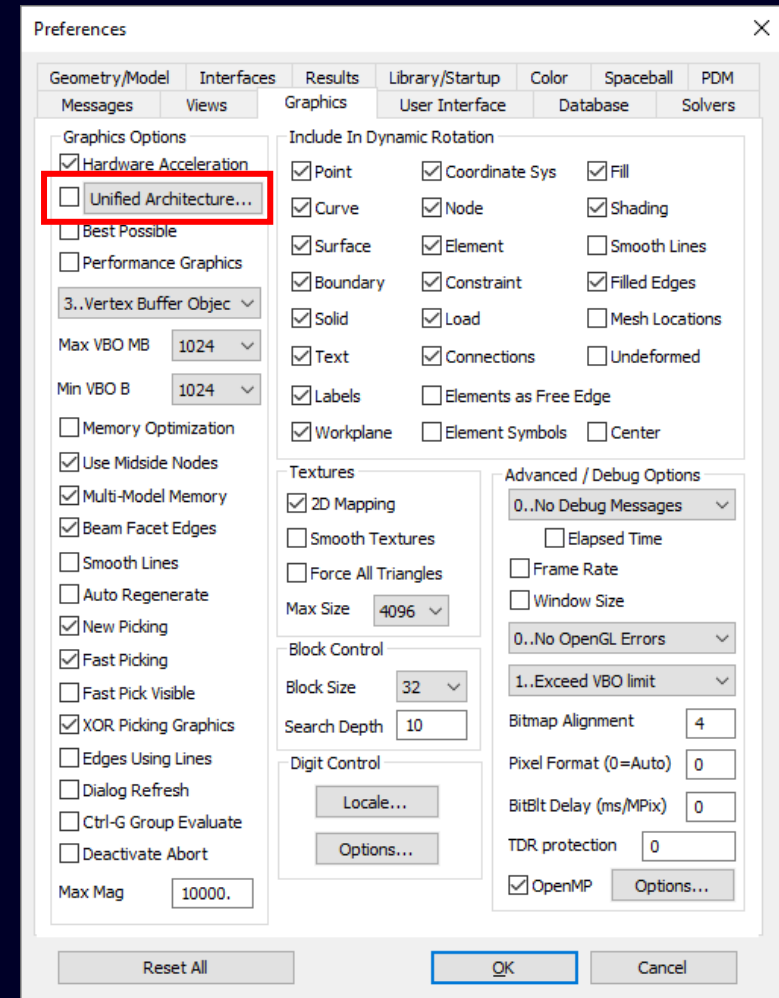
Performance Improvements – Unified Graphics Architecture Update

In Femap 2301, an effort to improve graphics performance across all aspects of Femap was started, which is still intended to span across multiple releases

This new Unified Graphics Architecture (UGA) uses a single “pipeline” and when fully implemented, UGA will provide a better experience for the user in quality, performance, and hardware support

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

Once UGA is completed, both the “Performance Graphics” and “original OpenGL” pipelines will be removed

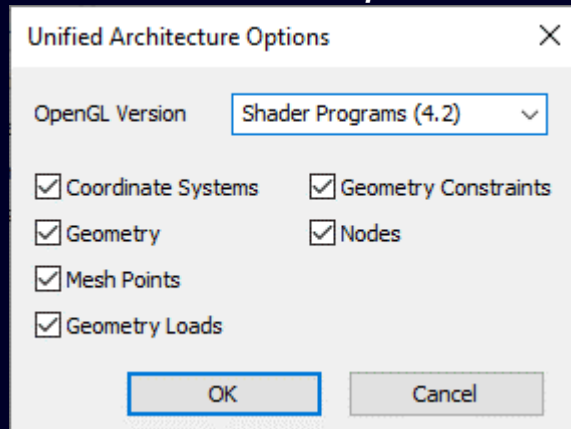


Simcenter Femap 2401

Performance Improvements – Unified Graphics Architecture Update

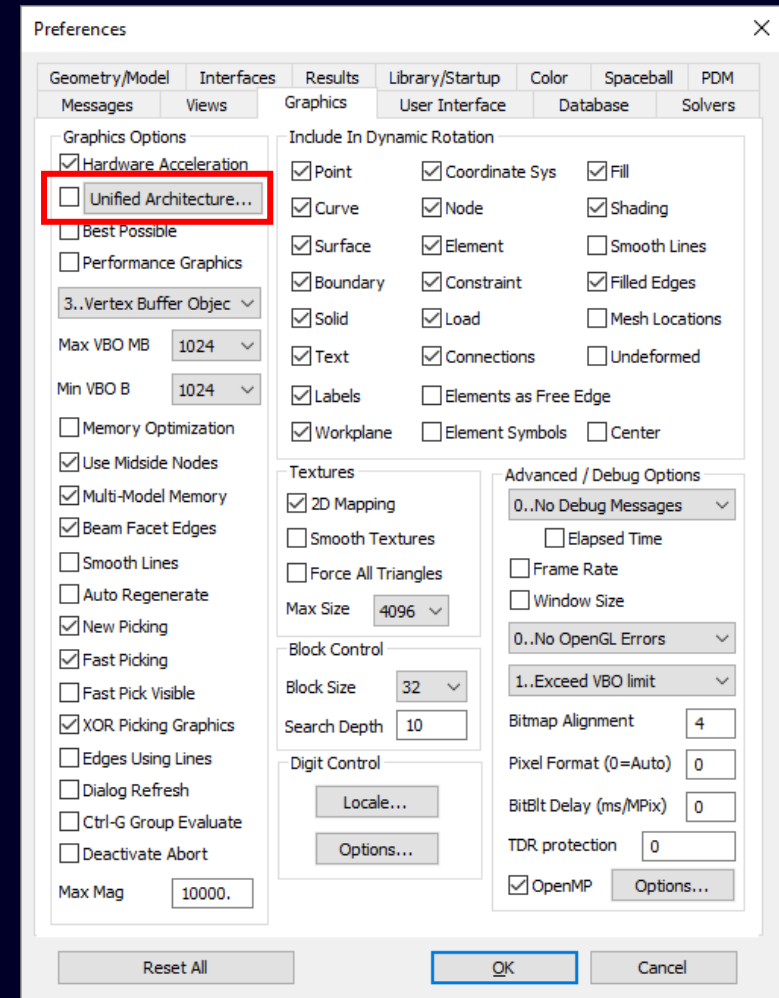
UGA is turned on by default but can be turned off using Unified Architecture option on the *Graphics* tab of *File -> Preferences*

For 2401, “Unified Architecture...” is now also a button which accesses the *Unified Architecture Options* dialog box:



This dialog box allows selection of *OpenGL version* as well as ability to use/not use UGA for supported entity types*

*Note: ONLY use if problem is encountered when using UGA

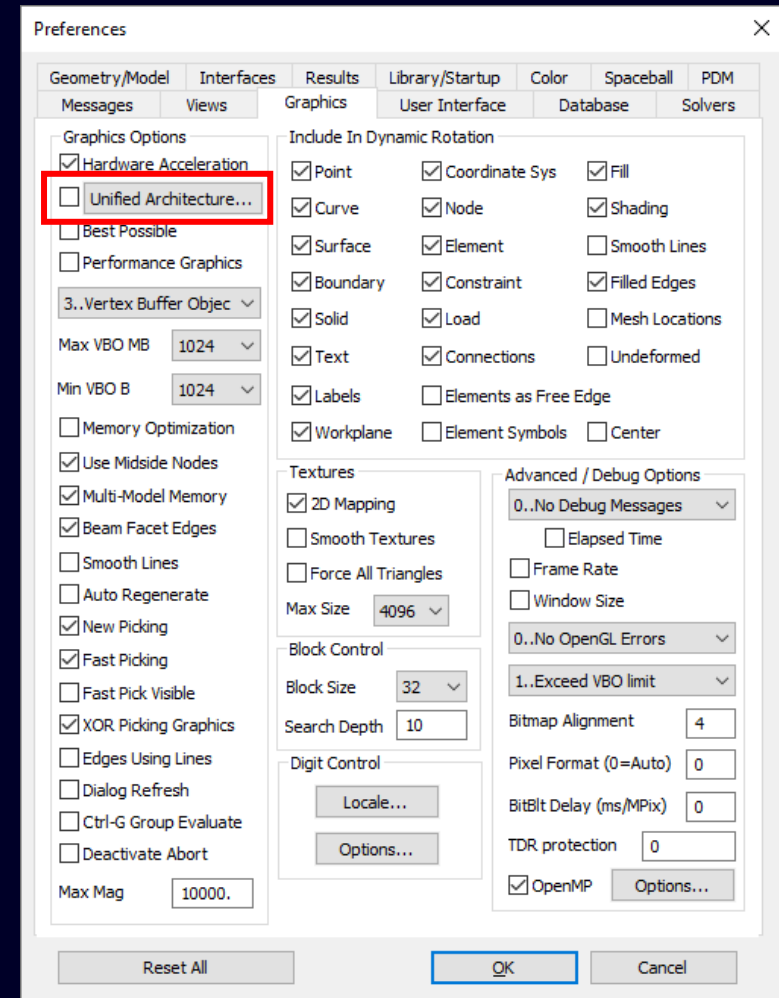


Simcenter Femap 2401

Performance Improvements – Unified Graphics Architecture Update

For Femap 2401, Nodes have been added to the UGA “pipeline”, which includes support for all display options for nodes which previously existed in the other two “pipelines”
These entity types were already supported in 2301 and 2306:

- Coordinate Systems
- Geometric Entities
 - Points
 - Mesh Points
 - Curves
 - Composite Curves
 - Surfaces
 - Boundary Surfaces
 - Solids
- Geometry-based Loads and Constraints





What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

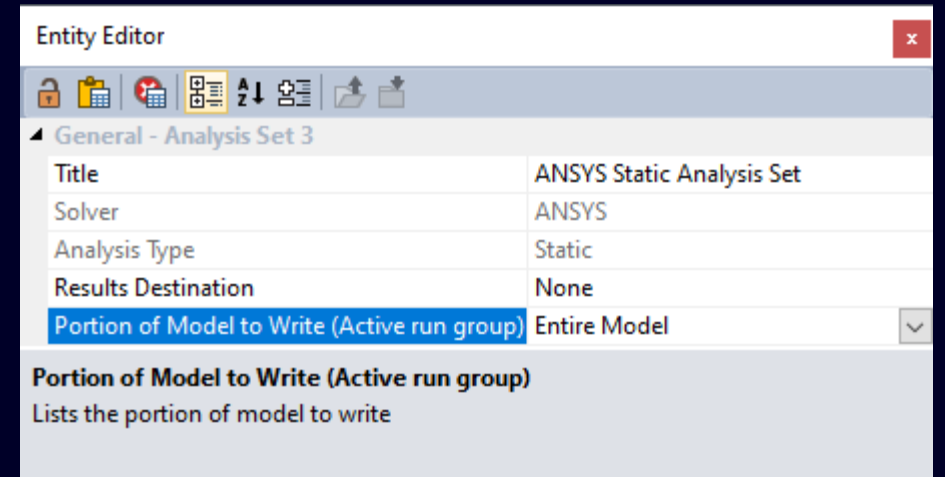
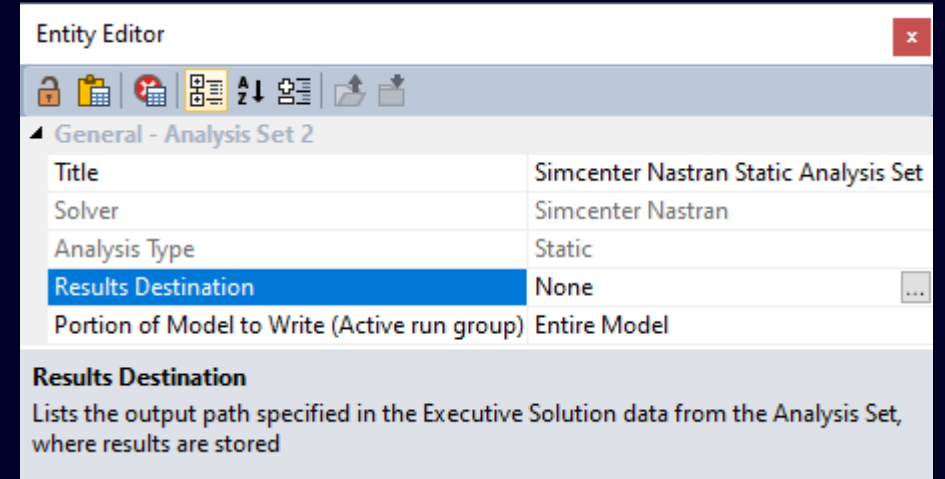
Simcenter Femap 2401

All Solvers – Entity Editor

In previous versions of Femap, the *Entity Editor* was not populated with any information when an Analysis Set was highlighted in the Analyses branch of the Model Info Tree

As an initial implementation for Femap 2401, the *Entity Editor* is now populated with a subset of information available for each Analysis Set:

- **Title** – Title of the Analysis Set (editable)
- **Solver** – Analysis Solver (read-only)
- **Analysis Type** – Analysis Type (read-only)
- **Results Destination** – If specified, displays full path to directory where output will be sent by the solver (editable)
- **Portion of Model to Write** – Displays ID and name of Group to be used to write a portion of the model (editable)

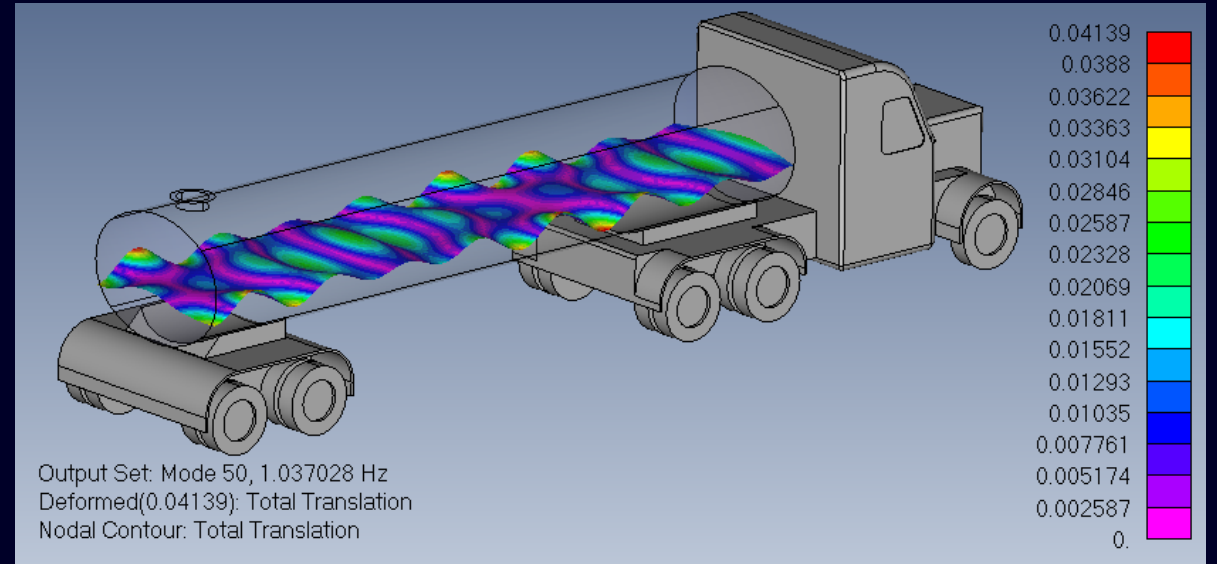
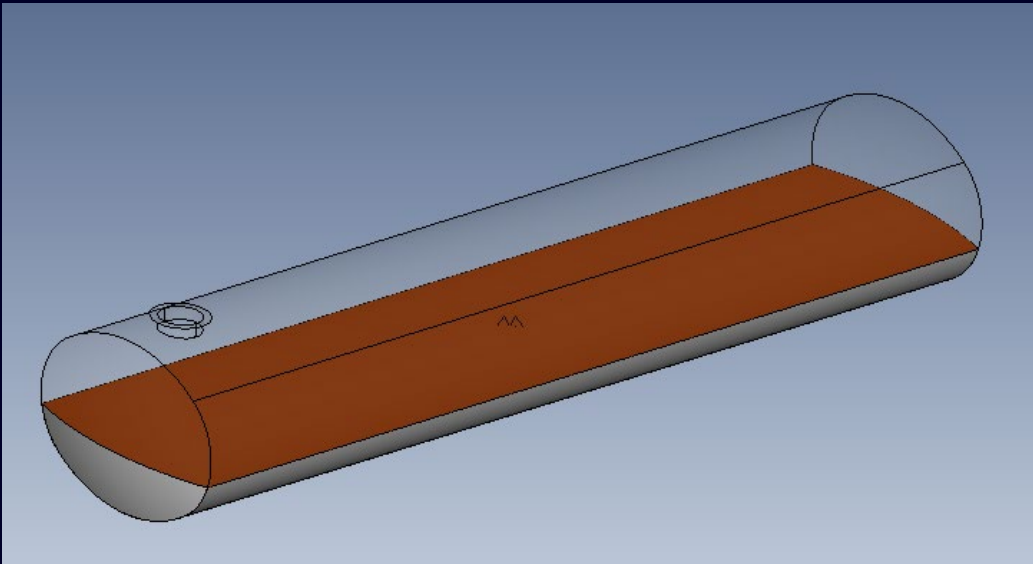


Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

Normal Modes (SOL 103) and Modal Frequency Response (SOL 111) analysis on models that contain incompressible fluids where fluid is defined with solid elements is possible with Simcenter Nastran

- The defined fluid mass approach is useful for complicated fluid volume shapes
- Previous versions of Femap included support for the virtual fluid mass method (MFLUID), which used boundary elements to model the fluid volume
- Defining an Incompressible Fluid Surface is done using Connect -> Fluid Region command

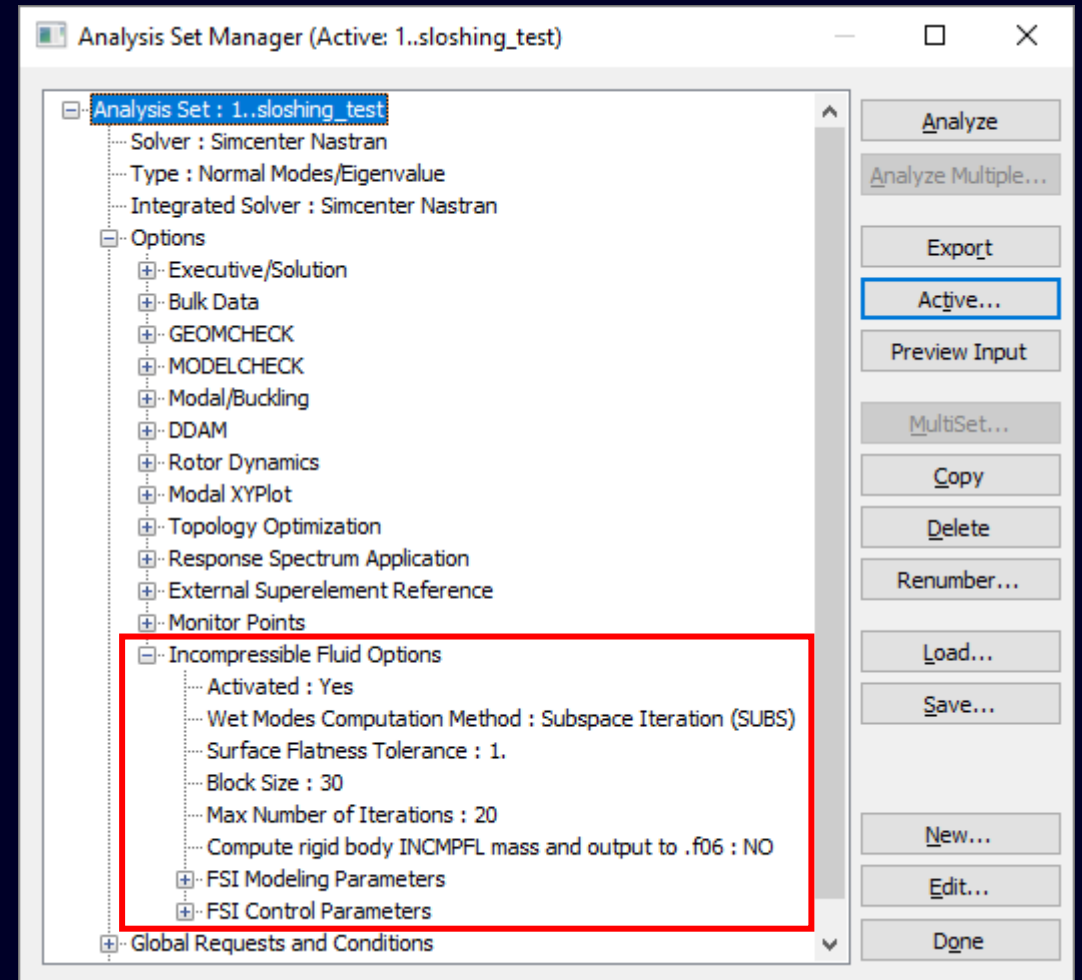


Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

Various options are available for configuring the analysis to optimize for efficiency. These are available in normal modes and modal frequency response analysis sets within the new “Incompressible Fluid Options” branch (Note: Not in “Next/Prev chain”)

- **Incompressible Fluid Options** – Specify options for modeling incompressible fluids with fluid elements
- **FSI Modeling Parameters** – Define the modeling parameters for the interface between the fluid and the structure (FSI = Fluid-Structure Interaction)
- **FSI Control Parameters** – Various control parameters for configuring the Fluid-Structure Interaction



Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

Incompressible Fluid Options

When *Activate Incompressible Fluid Options* is turned on, writes INCMOPT Bulk Data entry with specified values

- **Modal Calc Method** – Specifies the method for the modal calculation with the incompressible fluid
 - 0..Subspace Iteration (SUBS)
 - 1..Dry Modes plus Residual Vectors (DRYMODES)
 - 2..ARPACK (proposed for Simcenter Nastran 2406)
- **Scale Factor** – Only used for “1..Dry Modes Plus Residual Vectors (DRYMODES)” method, and is applied to the frequency range or number of modes for the modal calculation with the incompressible fluid to determine frequency range or number of modes for the modal calculation without the incompressible fluid

Incompressible Fluid Options

Activate Incompressible Fluid Options

INCMOPT

Modal Calc Method (METHOD) 0..Subspace Iteration (SU)

Scale Factor (SF) 2.

Surface Flatness Tolerance (SFTOL) 1.

Block Size (BSIZE) 30

Max Iterations for SUBS (NITER) 20

Compute G-set INCMPL Mass (RBFLMASS) 0..NO

Parameters

SLOSHCMS 0

FSI Modeling... FSI Control... OK Cancel

Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

Incompressible Fluid Options

- **Surface Flatness Tolerance** – Tolerance for determining whether the free surface is sufficiently flat to perform the modal analysis with the incompressible fluid
- **Block Size** – Only used for “0..Subspace Iteration (SUBS)” method, the number of eigenvectors calculated
- **Max Iterations for SUBS** – Only used for “0..Subspace Iteration (SUBS)”, maximum allowable number of iterations for convergence
- **Compute G-set INCOMPFL Mass** – Option to compute rigid body incompressible fluid mass and output it to the .f06 file

Incompressible Fluid Options

Activate Incompressible Fluid Options

INCMOPT

Modal Calc Method (METHOD) 0..Subspace Iteration (SU)

Scale Factor (SF) 2.

Surface Flatness Tolerance (SFTOL) 1.

Block Size (BSIZE) 30

Max Iterations for SUBS (NITER) 20

Compute G-set INCOMPFL Mass (RBFLMASS) 0..NO

Parameters

SLOSHCMS 0

FSI Modeling... FSI Control... OK Cancel

Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

Incompressible Fluid Options

Parameters – When on, writes PARAM,SLOSHCMS,“n”

- **SLOSHCMS** – Parameter for specifying whether the software performs Component Mode Synthesis (CMS) reduction on the sloshing surface and retains “n” modes
 - If both this field and *Comp. Modes* field on Fluid Region are specified, *Comp. Modes* value takes precedence for the incompressible fluid volume sloshing surface

FSI Modeling – Opens *FSI Modeling Parameters* dialog box used to specify advanced options such as coupling controls and search distances (writes ACMODL Bulk Data entry)

FSI Control – Opens *FSI Control Parameters* dialog box used to specify SKINOUT as well as if coupling data with pairing information is written (writes FLSTCNT Case Control entry)

Incompressible Fluid Options

Activate Incompressible Fluid Options

INCMOPT

Modal Calc Method (METHOD) 0..Subspace Iteration (SU)

Scale Factor (SF) 2.

Surface Flatness Tolerance (SFTOL) 1.

Block Size (BSIZE) 30

Max Iterations for SUBS (NITER) 20

Compute G-set INCMPL Mass (RBFLMASS) 0..NO

Parameters

SLOSHCMS 0

FSI Modeling... FSI Control... OK Cancel

Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

FSI Modeling Parameters – Accessed by *FSI Modeling* button

When *Activate Fluid-Structure Interface Modeling Parameters* is turned on, writes ACMODL Bulk Data entry with specified values

- **Type of Coupling (CTYPE)** – Determines which governing equation will be used by choosing either:
 - 0..Two Way Coupling (STRONG)
 - 1..Effect of Structure on Fluid Only (WEAK)
- **Search Units (SRCHUNIT)** – Controls how values specified for *Outward Norm Srch Dist (NORMAL)* and *Inward Normal Srch Dist (INTOL)* are used by choosing either:
 - 0..Relative – Values used as scale factors applied to maximum edge length of a fluid element free face (L_{max} in figure)
 - 1..Absolute – Values used as physical distances

FSI Modeling Parameters

Activate Fluid-Structure Interface Modeling Parameters

ACMODL

Type of Coupling (CTYPE) 0..Two-way Coupling (STF)

Search Units (SRCHUNIT) 0..Relative

Outward Norm Srch Dist (NORMAL) 0.5

Inward Normal Srch Dist (INTOL) 0.2

Angular Tolerance (OVL PANG) 60.

Coupling Method (AREAOP) 0..Recommended Method

OK Cancel

Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

FSI Modeling Parameters – Accessed by *FSI Modeling* button

- **Outward Norm Srch Dist (NORMAL)** – Value used to determine outward normal search distance (L_{outward} in figure)
 - If the fluid and structural element free faces are within search distance of one another and satisfy value specified for *Angle Tolerance (OVL PANG)*, they are coupled
- **Inward Norm Srch Dist (INTOL)** – Value used to determine the inward normal search distance (L_{inward} in figure)
 - If the fluid and structural element free faces are within search distance of one another, they are coupled
- **Angular Tolerance (OVL PANG)** – Angular tolerance in degrees used to decide whether a fluid free face and a structural face can be considered overlapping
 - If the angle between the normals of the fluid and structural faces exceeds the value, they are not coupled

Parameter	Value
Activate Fluid-Structure Interface Modeling Parameters	<input checked="" type="checkbox"/>
Type of Coupling (CTYPE)	0..Two-way Coupling (STF)
Search Units (SRCHUNIT)	0..Relative
Outward Norm Srch Dist (NORMAL)	0.5
Inward Normal Srch Dist (INTOL)	0.2
Angular Tolerance (OVL PANG)	60.
Coupling Method (AREAOP)	0..Recommended Method

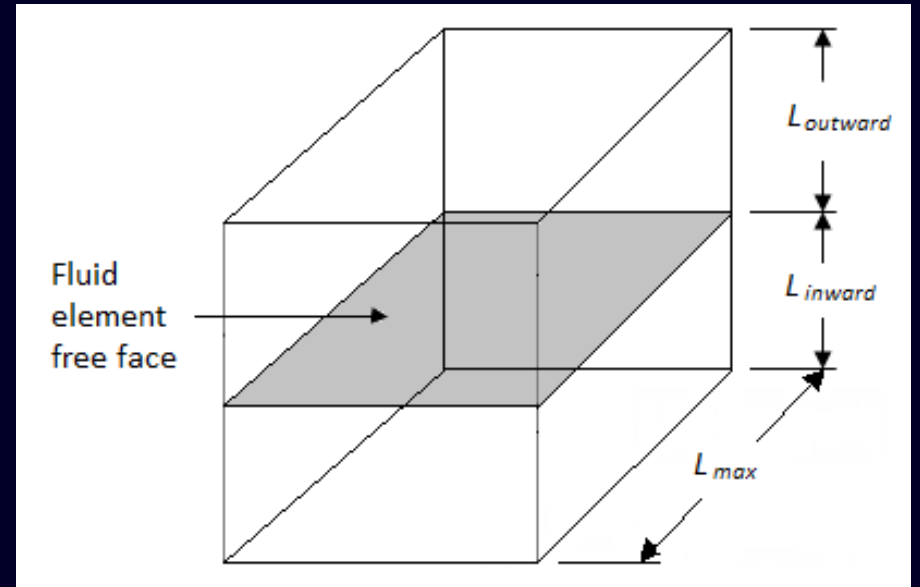
Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

FSI Modeling Parameters – Accessed by *FSI Modeling* button

Figure shows outward ($L_{outward}$) and inward (L_{inward}) normal search distances for a quadrilateral free face of a fluid element, as well as maximum edge length of a fluid element free face (L_{max}), which is only used when *Search Units (SRCHUNIT)* is set to “0..Relative”

- For each fluid element free face, the search for a structural free face with which to couple continues until a structural face is identified or search distance is reached
- **Coupling Method (AREAOP)** – Option to choose:
 - 0..Recommended Method
 - 1..RBE3 Method – Alternative fluid-structure coupling method, which applies an area correction and removes parallel disconnected faces from the coupling



Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

FSI Control Parameters – Accessed by *FSI Control* button

When *Activate Fluid-Structure Interface Control Parameters* is turned on, writes FLSTCNT Case Control entry with specified values

- **Symmetric Request (ACSYM)** – Specifies if fluid-structure analysis is symmetric or not by choosing either:
 - 0..NO – Solution is non-symmetric
 - 1..YES – Solution is symmetric
- **Particle Vel Output Type (ACOUT)** – Specifies the type of output created by a FORCE Case Control entry by choosing either:
 - 0..Peak value output (PEAK)
 - 1..Root-mean-square output (RMS)
- **Reference Pressure (PREFDB)** – Specifies value for reference pressure

FSI Control Parameters

Activate Fluid-Structure Interface Control Parameters

FLSTCNT

Symmetric Request (ACSYM)	1..YES
Particle Vel Output Type (ACOUT)	0..Peak value output
Reference Pressure (PREFDB)	1.
Coupled Solution (ASCOUN)	1..YES
Coupling Information (SKINOUT)	0..NONE
Coupling Matrix to PCH (AGGPCH)	0..NO
Matrix from SFE AKUSMOD (SFEF70)	0..NO

OK Cancel

Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

FSI Control Parameters – Accessed by *FSI Control* button

- **Couple Solution (ASCOUP)** – Specifies if a coupled fluid-structure analysis should be performed or not by choosing either:
 - 0..NO – Solution is not coupled
 - 1..YES – Solution is coupled
- **Coupling Information (SKINOUT)** – Specifies if coupling data with the pairing information is output, for debugging, by choosing from:
 - 0..NONE – Coupling data is not output
 - 1..PUNCH – Coupling data is written to *.dat and Punch (*.pch) files (solution proceeds, only coupled free faces are written)
 - 2..FREEFACE – Coupling data is written to *.dat and Punch (*.pch) files as well as to the coupling datablock in .op2 file (solution proceeds, both coupled and uncoupled faces are written)
 - 3..STOP – Works like “2..FREEFACE”, but solution stops immediately after pairing information is written

FSI Control Parameters

Activate Fluid-Structure Interface Control Parameters

FLSTCNT

Symmetric Request (ACSYM)	1..YES
Particle Vel Output Type (ACOUT)	0..Peak value output
Reference Pressure (PREFDB)	1.
Coupled Solution (ASCOUP)	1..YES
Coupling Information (SKINOUT)	0..NONE
Coupling Matrix to PCH (AGGPCH)	0..NO
Matrix from SFE AKUSMOD (SFEF70)	0..NO

OK Cancel

Simcenter Femap 2401

Simcenter Nastran – Dynamic Analysis with Defined Fluid Mass

FSI Control Parameters – Accessed by *FSI Control* button

- **Coupling Matrix to PCH (AGGPCH)** – Requests if output of the fluid-structure coupling matrix AGG will be written to the Punch (*.pch) file by choosing either:
 - 0..NO – Fluid-Structure coupling matrix output is not written
 - 1..YES – Fluid-Structure coupling matrix output is written
- **Matrix from SFE AKUSMOD (SFEF70)** – Requests import of a fluid-structure coupling matrix created by SFE AKUSMOD by choosing either:
 - 0..NO – Coupling matrix is not imported
 - 1..YES – Coupling matrix is imported

Parameter	Value
Activate Fluid-Structure Interface Control Parameters	<input checked="" type="checkbox"/>
FLSTCNT	
Symmetric Request (ACSYM)	1..YES
Particle Vel Output Type (ACOUT)	0..Peak value output
Reference Pressure (PREFDB)	1.
Coupled Solution (ASCOUP)	1..YES
Coupling Information (SKINOUT)	0..NONE
Coupling Matrix to PCH (AGGPCH)	0..NO
Matrix from SFE AKUSMOD (SFEF70)	0..NO

Simcenter Femap 2401

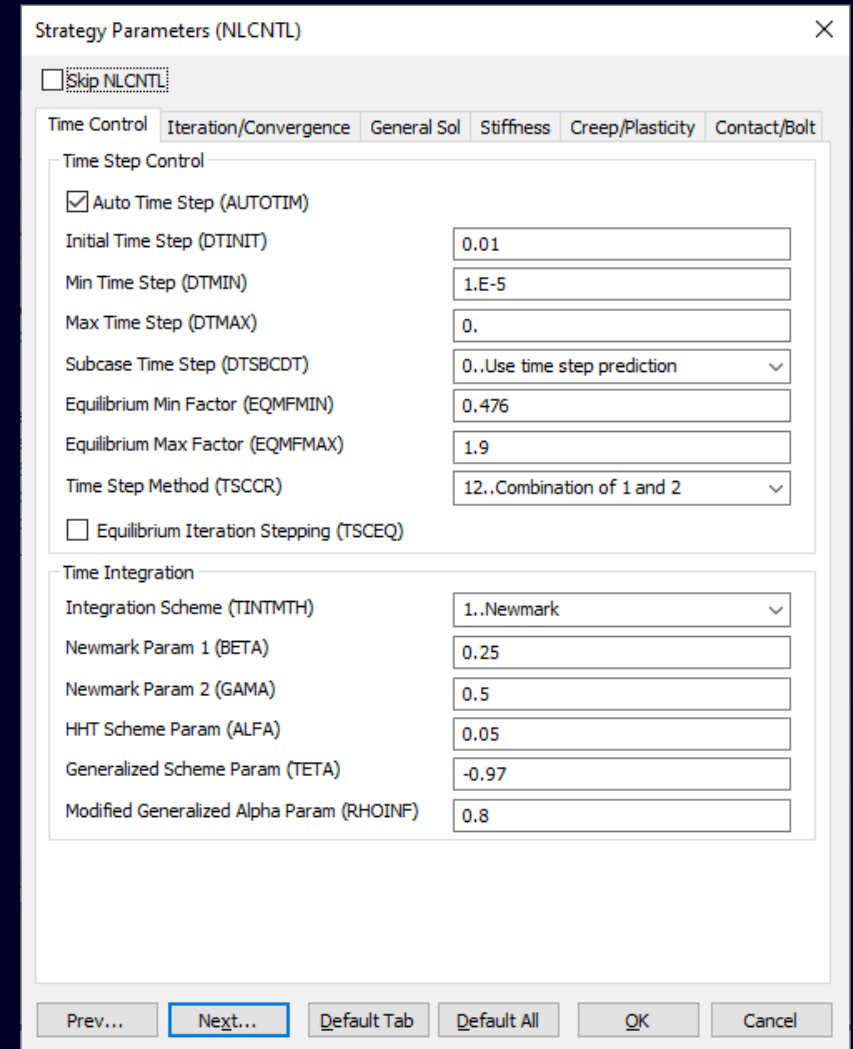
Simcenter Nastran – SOL 401

Updated name of “Control Options” in Analysis Set Manager tree structure to “Strategy Parameters” and added “Non-Default Values” branch below which only contains items with non-default values

Changed name of dialog box to “Strategy Parameters (NLCNTL)” and enhanced it to feature tabs (former “section” in dialog boxes):

- Time Control (Time Step Control and Time Integration)
- Iteration/Convergence (Convergence Parameters)
- General Sol (Solution Parameters and Time Step Control)
- Stiffness (Stiffness Parameters)
- Creep/Plasticity (Creep/Plasticity and Other Options)
- Contact/ Bolt (Contact, Bolt Preload, and Other Options)

Added *Default Tab* and *Default All* buttons



Simcenter Femap 2401

Simcenter Nastran – SOL 401 and SOL 402

Additions and Updates for Multi-Step Structural (SOL 401):

- *Strategy Parameters (NLCNTL) – Iteration/Convergence tab*
 - Added *Incremental Rotation Limit in Iteration (RTOLB)* value
- *Strategy Parameters (NLCNTL) – Stiffness tab*
 - Added *Include Stress Stiffening at First Iteration (STKFRST)* option
- *Strategy Parameters (NLCNTL) – Creep/Plasticity tab*
 - Changed default for *Adjust Integration Factor (CRLIMR)* from 0.0 to 1.0

Updated for Multi-Step Nonlinear Kinematic (SOL 402)

- *Strategy Parameters (NLCNTL) – Plasticity/Creep tab*
 - Changed default for *Creep Integration Factor (CRINFAC)* from 0.5 to 1.0

Time Control | **Iteration/Convergence** | General Sol | Stiffness | Creep/Plasticity | Contact/Bolt

Iteration and Convergence

<input type="checkbox"/> Displacement (EPSU)	0.01
<input type="checkbox"/> Load (EPSP)	0.01
<input checked="" type="checkbox"/> Work (EPSW)	1.E-6
Max Bisections (MAXBIS)	5
Max Divergences (MAXDIV)	3
Max Iteration/Step (MAXITER)	25
Max Quasi Newton Vectors (MAXQN)	10
Norm Criteria for Force (NORMP)	1..Norm 2 Criteria
Norm Criteria for Displacement (NORMU)	1..Norm 2 Criteria
Force Error Function Denominator (REFP)	-1.
Disp. Error Function Denominator (REFU)	-1.
Incremental Rotation Limit in Iteration (RTOLB)	20.

Time Control | Iteration/Convergence | General Sol | **Stiffness** | Creep/Plasticity | Contact/Bolt

Stiffness Parameters

Stiffness Update Strategy (KUPDATE)	0..Auto Stiffness Update
Quasi Newton-Raphson Iterations	2
Material Stiffness Matrix (STFOPTN)	3..Elastic/Tangent Stiffness Matrix
<input type="checkbox"/> Follow Stiffness (FOLLOWK)	
<input type="checkbox"/> Spin Softening (SPINK)	
<input checked="" type="checkbox"/> Include Stress Stiffening at First Iteration (STKFRST)	
<input type="checkbox"/> Update Stiffness (TSTEPK)	
<input type="checkbox"/> Enable Stiffness Matrix Stabilization (MSTAB)	
Matrix Stabilization Factor (MSFAC)	1.E-10

Simcenter Femap 2401

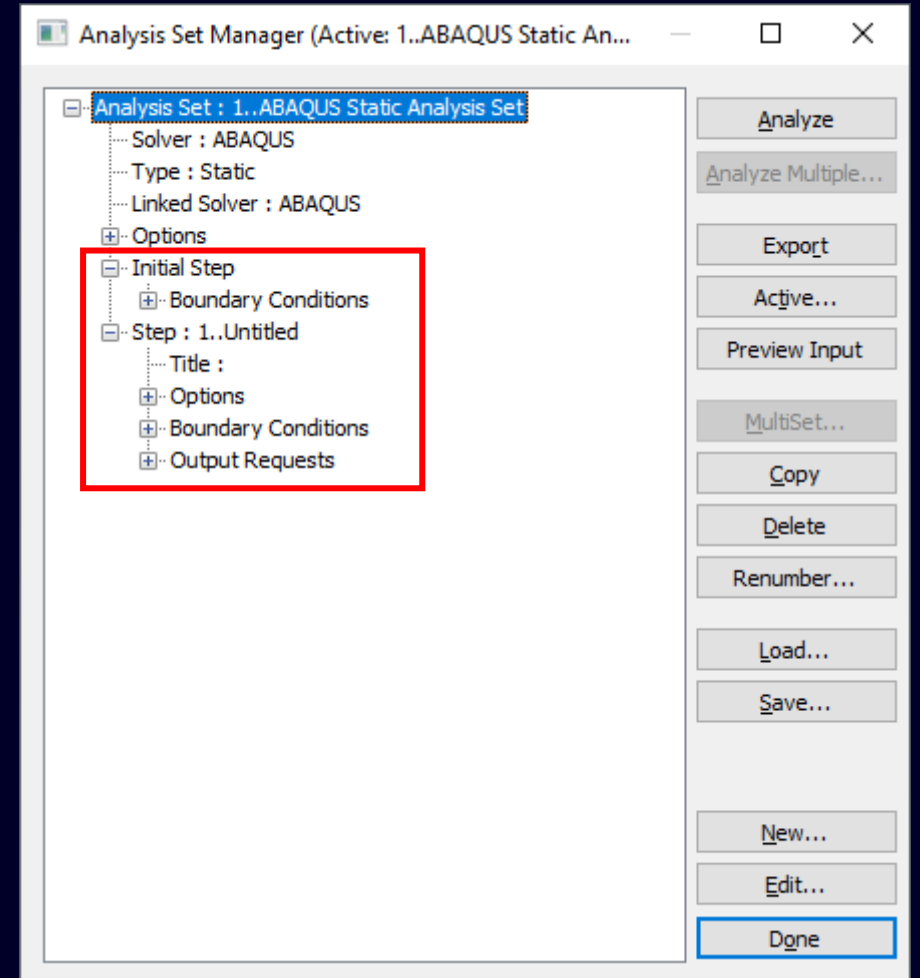
ABAQUS – Implementation of Initial Step and Steps

To allow ABAQUS users to work in a more familiar environment when setting up an analysis, Femap 2401 now uses terminology and methodologies similar to those found in other Pre-and-Post Processors and the solver itself

As a first step, a newly created Analysis Set for ABAQUS will now always include:

- **Initial Step** – Used to set Default Constraints, Default Connectors, and Initial Conditions for the Analysis
- **Step 1** – Used to specify Options, Boundary Conditions, and Output Requests for the Analysis Step

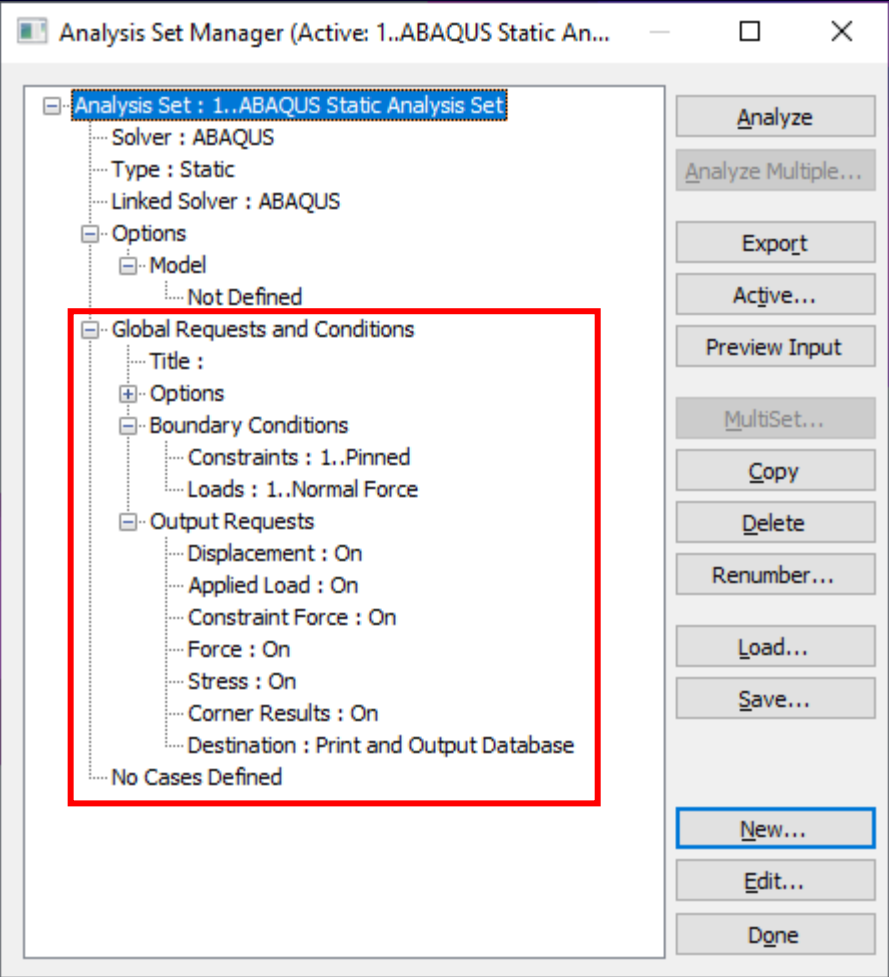
Additional “Steps” can then be created, as needed, and the type of analysis to perform in each step can be specified



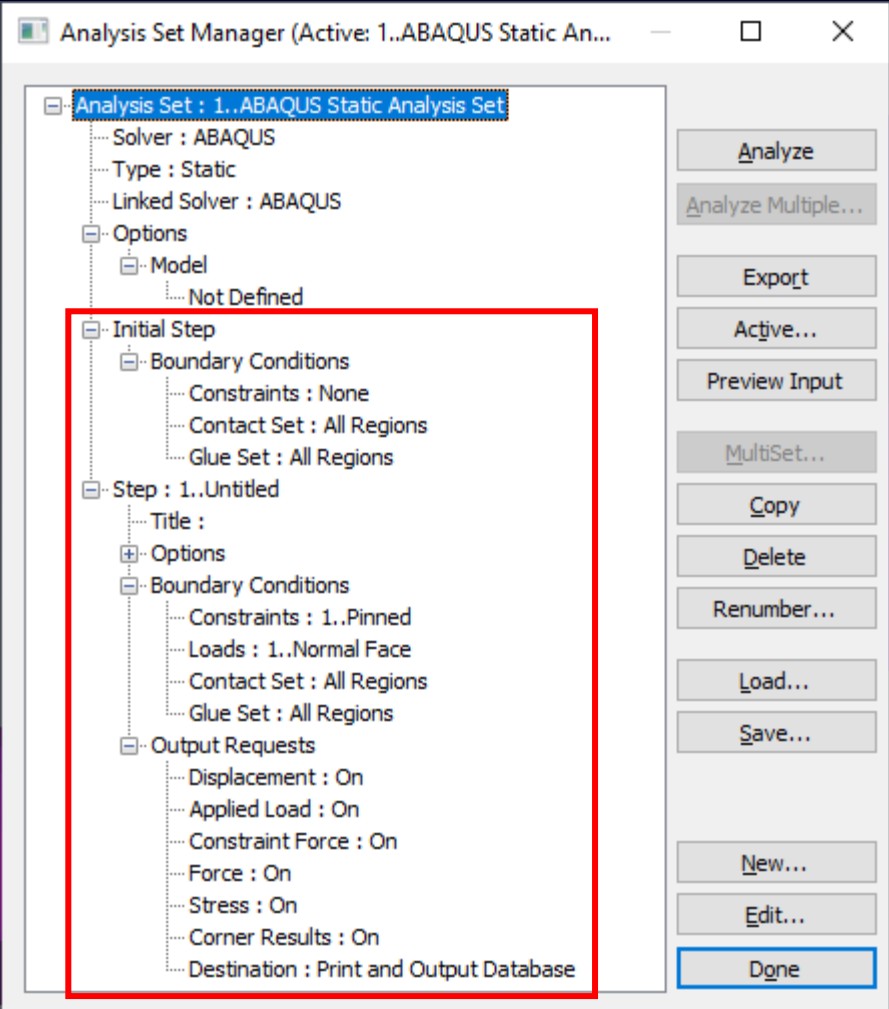
Simcenter Femap 2401

ABAQUS – Implementation of Initial Step and Steps

2306 and Below



2401 and above



Simcenter Femap 2401

ABAQUS – Implementation of Initial Step and Steps

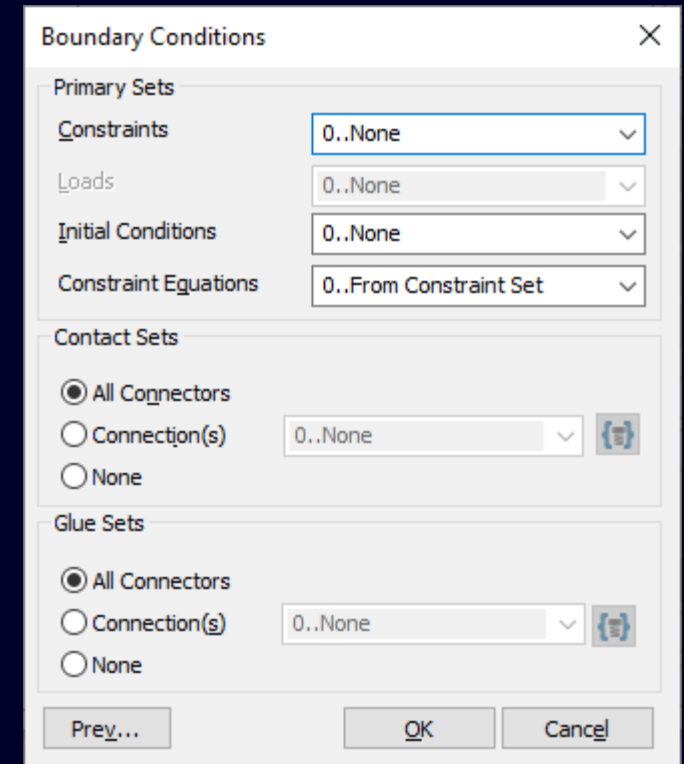
Summary of “new” vs “old” Analysis Set paradigm

Initial Step – Did not exist before and is used to specify overall Boundary Conditions for the entire analysis

- Constraints* – Specify Default Constraint Set
- Initial Conditions* – Specify Initial Conditions
- Constraint Equations* – Specify if constraint equations for the Default Constraints come from the Constraint Set specified in Constraints or a different Constraint Set
- Contact Sets and Glued Sets** – Connectors to use, choose between:
 - All Connectors – Use all Connectors of corresponding Type
 - Connection(s) – Use selected Connector Set of corresponding Type
 - None – Use no Connectors of corresponding Type

*Note: Previously done in “Global Requests and Conditions”

**Note: Could not be specified in previous versions of Femap



Simcenter Femap 2401

ABAQUS – Implementation of Initial Step and Steps

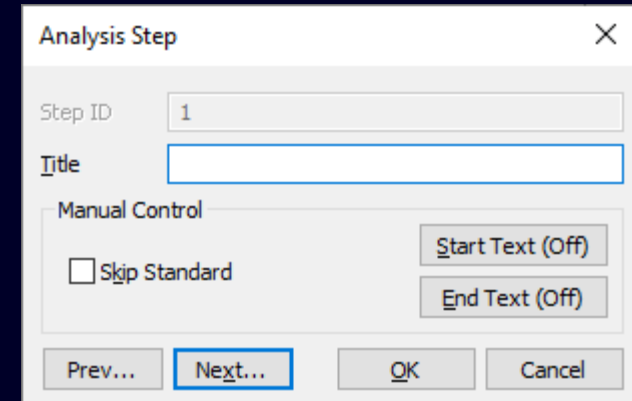
Summary of “new” vs “old” Analysis Set paradigm

Step “n” – Used to specify *Options*, *Boundary Conditions*, and *Output Requests* for a “Step”

- “Step 1” automatically created with new Analysis Set
- To create a “Step”, highlight existing “Step” and click “New” button
 - If there are no “Steps” defined, “No Cases Defined” is shown

Analysis Step – Top-level of each “Step”

- Step ID – Automatically increments to next available ID in Femap, but can be changed to another value
 - “Step ID” only used to control order of “Steps” in Analysis Set
- Title – Optional, but if Analysis Set is “Untitled”, Title of Load Set selected in *Boundary Conditions* is used as NAME for *STEP
- Manual Control – Can be used to add text to Start/End of “Step”



Simcenter Femap 2401

ABAQUS – Implementation of Initial Step and Steps

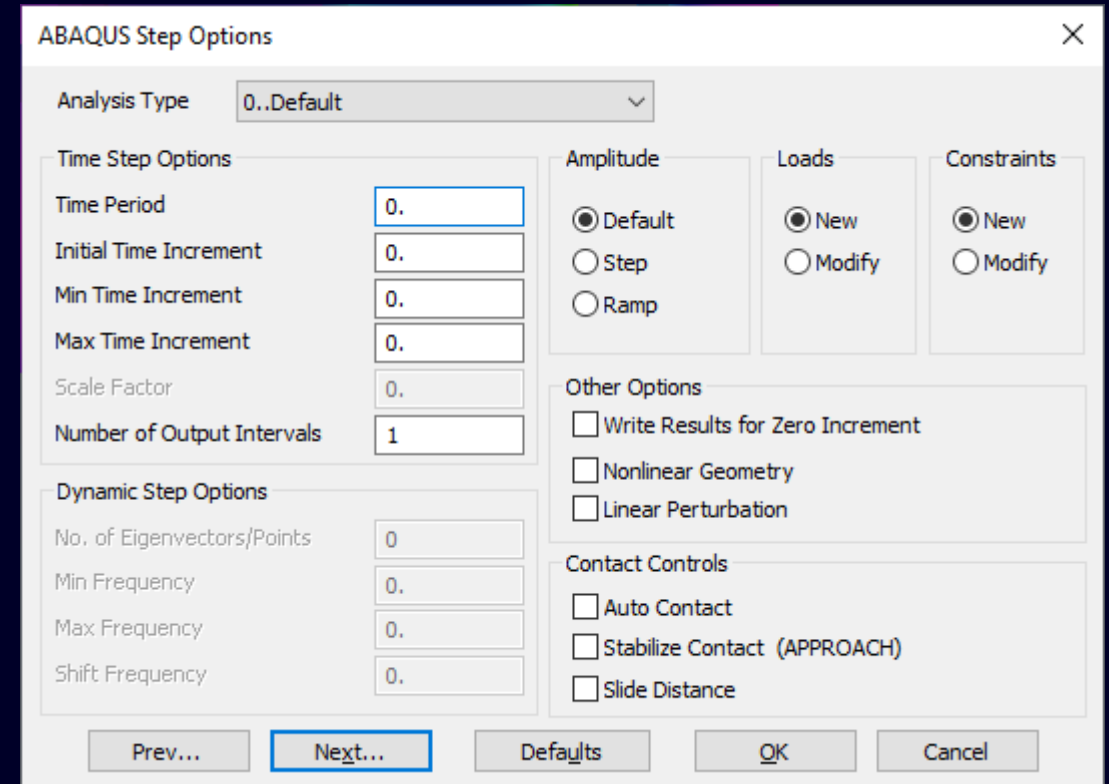
Summary of “new” vs “old” Analysis Set paradigm

Step “n” – Used to specify *Options*, *Boundary Conditions*, and *Output Requests* for a “Step”

*Options** – ABAQUS Step Options dialog box allows user to:

- Enter *Time Step Options*
- Enter *Dynamic Step Options* (when applicable)
- Choose an option for *Amplitude/Loads/Constraints*
- Select various *Other Options*
- Select various *Contact Controls*

*Note: This dialog box did not change for 2401 and values previously specified in ABAQUS Step Options dialog box in Global Requests and Conditions must now be specified in ABAQUS Step Options dialog box in “Step 1”



Simcenter Femap 2401

ABAQUS – Implementation of Initial Step and Steps

Summary of “new” vs “old” Analysis Set paradigm

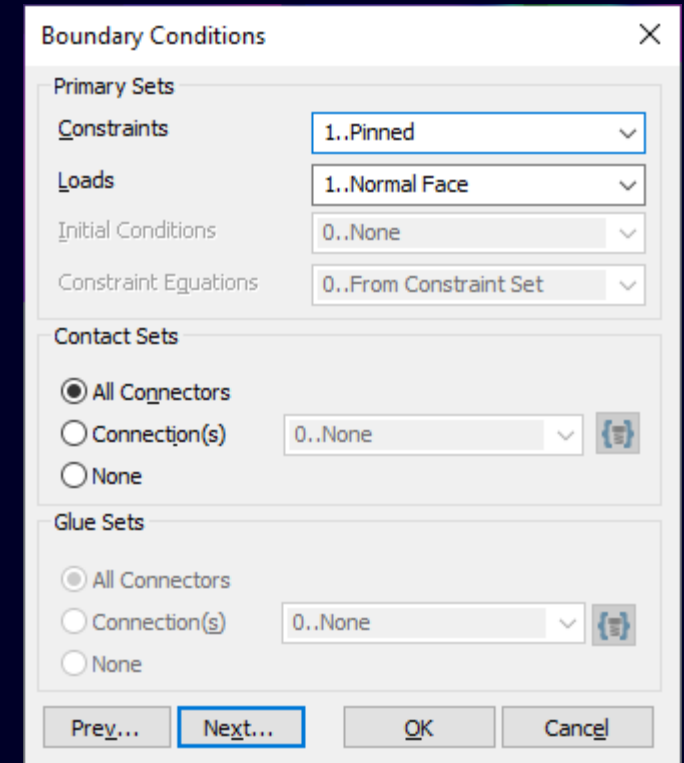
Step “n” – Used to specify *Options, Boundary Conditions, and Output Requests* for a “Step”

Boundary Conditions

- Constraints* – Specify Constraint Set to use for Step
- Loads* – Specify Load Set to use for Step
- Contact Sets** – Non-Glue Connectors to use for Step, choose between:
 - All Connectors – Use all Connectors of corresponding Type
 - Connection(s) – Use selected Connector Set of corresponding Type
 - None – Use no Connectors of corresponding Type

*Note: *Boundary Conditions* which would previously have been specified in Global Requests and Conditions must now be specified in “Step 1”

**Note: Could not be specified in previous versions of Femap



Simcenter Femap 2401

ABAQUS – Implementation of Initial Step and Steps

Summary of “new” vs “old” Analysis Set paradigm

Step “n” – Used to specify *Options, Boundary Conditions, and Output Requests* for a “Step”

*Output Requests** – Select output quantities and specify other options in *Output Requests* dialog box

- Nodal – Select output to recover for nodes
- Elemental – Select output to recover for elements
- Customization – Specify if output should be recovered at Element Corners and/or Integration Points as well as specify which output files should be created by ABAQUS

*Note: This dialog box did not change for 2401 and values previously specified in *Output Requests* dialog box in Global Requests and Conditions must now be specified in *Output Requests* dialog box in “Step 1”

The screenshot shows the 'Output Requests' dialog box with the following settings:

- Nodal:**
 - Displacement (0..Full Model)
 - Applied Load (0..Full Model)
 - Constraint Force (0..Full Model)
 - Equation Force (0..Full Model)
 - Force Balance (0..Full Model)
 - Velocity (0..Full Model)
 - Acceleration (0..Full Model)
 - Kinetic Energy (0..Full Model)
 - Temperature (0..Full Model)
 - Heat Flux (0..Full Model)
- Elemental:**
 - Force (0..Full Model)
 - Stress (0..Full Model)
 - Strain (0..Full Model)
 - Strain Energy (0..Full Model)
 - Plastic Strain (0..Full Model)
 - Heat Flux (0..Full Model)
 - Enthalpy (0..Full Model)
 - Enthalpy Rate (0..Full Model)
 - Temperature (0..Full Model)
 - Contact (0..Full Model)
- Customization:**
 - Element Corner Results
 - Integration Point Results
 - Results Destination: 8..Print and Output Data
 - Echo Model: (empty)
 - Magnitude/Phase
 - Real/Imaginary

Buttons: Prev..., OK, Cancel



What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

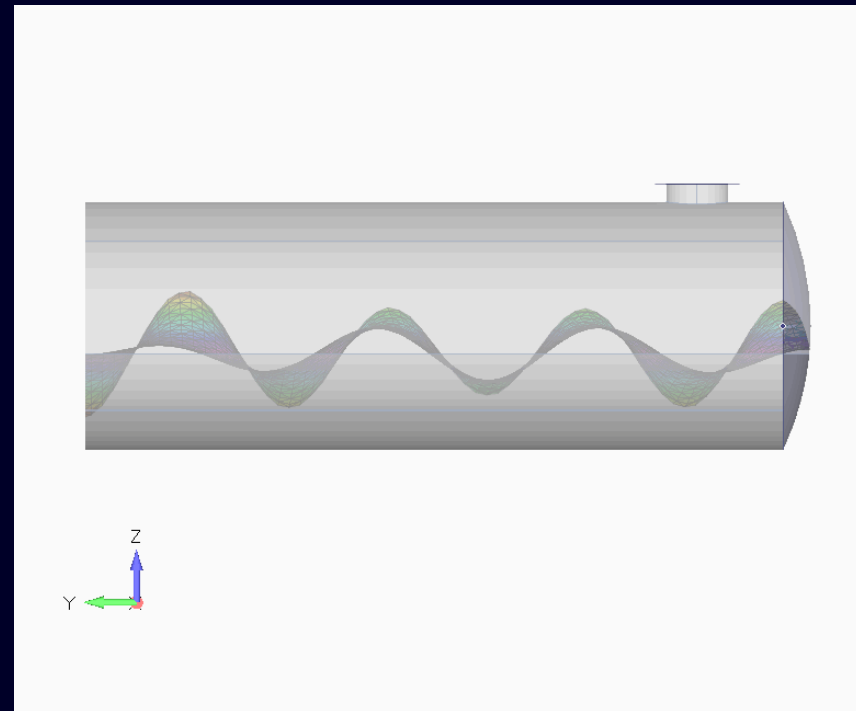
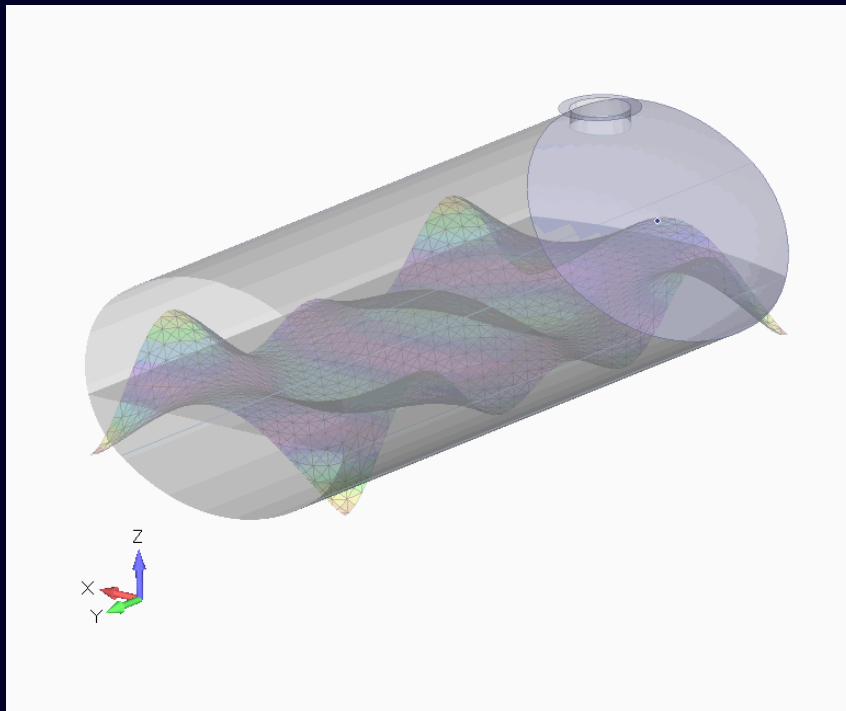
Postprocessing

Miscellaneous and API

Simcenter Femap 2401

Postprocessing – Dynamic Analysis with Defined Fluid Mass

Once Simcenter Nastran has completed a modal or frequency response analysis which included an Incompressible Fluid Surface, the sloshing modes can be postprocessed by using the *File -> Import -> Analysis Model* command to import the 2D plot-only mesh internally generated by Nastran (PLOTTEL elements which cover the sloshing free surface), then animate the modes





What's new **Simcenter Femap 2401**

Managed Environments

Geometry

Preprocessing

Meshing

Performance Improvements

Analysis and Solver Support

Postprocessing

Miscellaneous and API

Simcenter Femap 2401

Command Finder

Femap 2301:

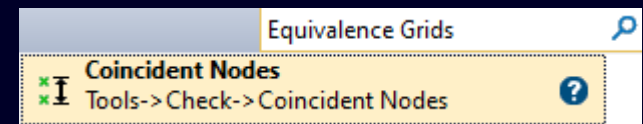
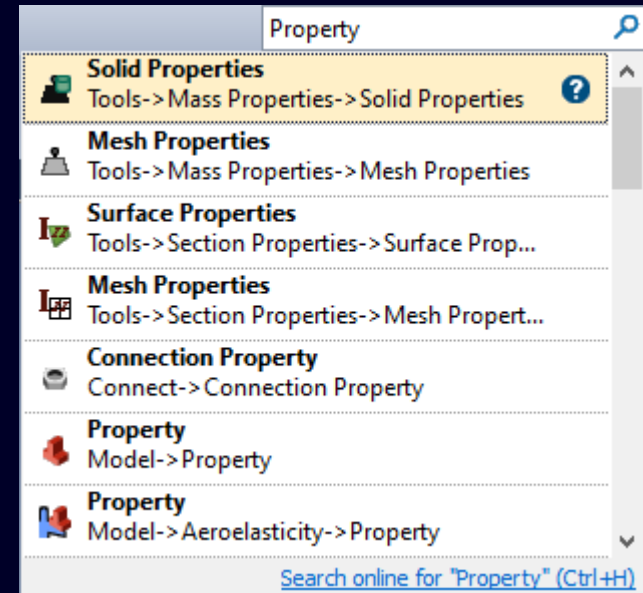
- Initial Implementation

Femap 2306:

- Find Tools in Toolboxes/Panes and Use “Keywords”

Femap 2401:

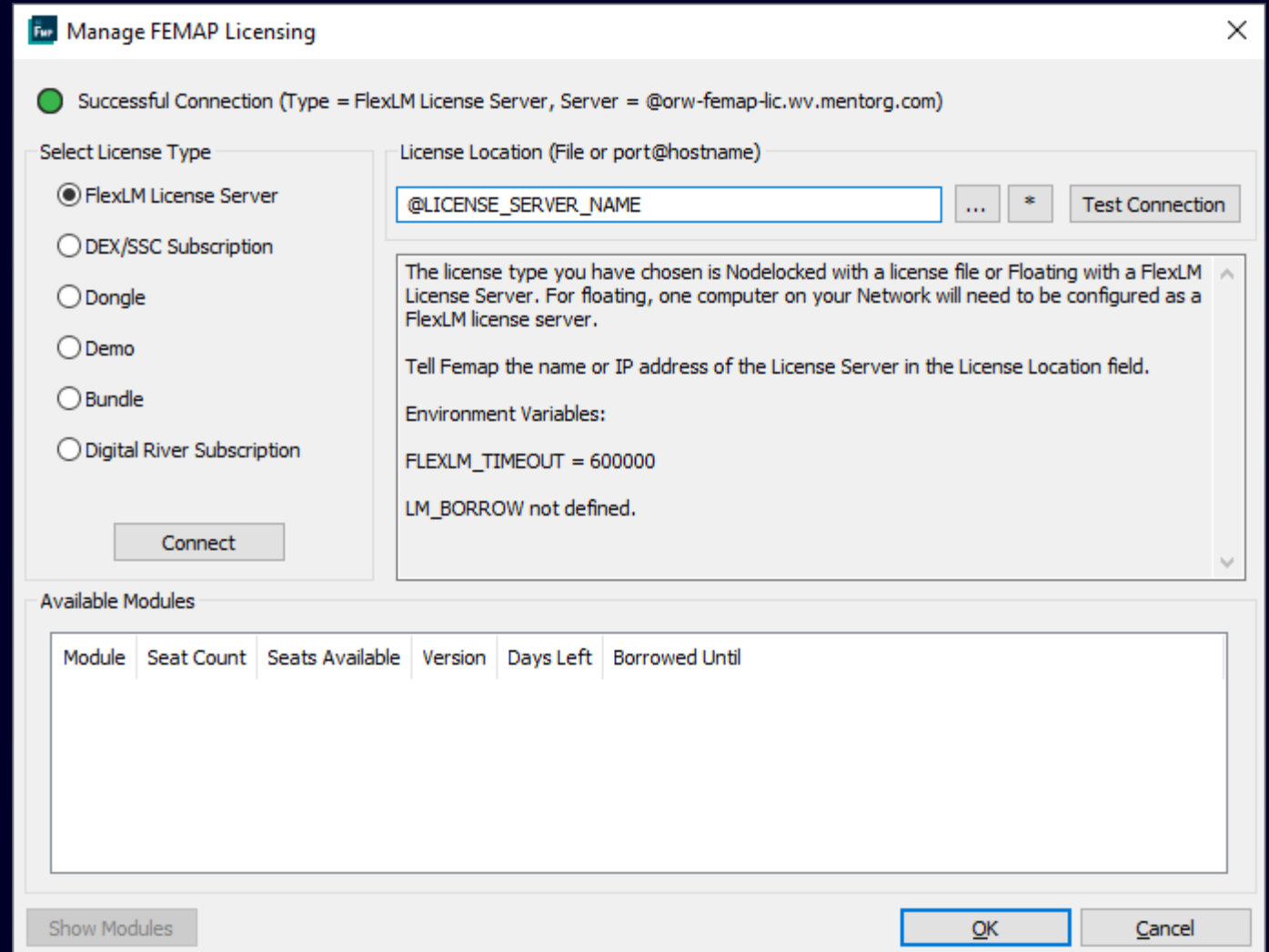
- Feature to pop-up a command when user “hovers-over” a matched command which is useful to identify location of command in Femap user interface
- Allow users to see matched commands as they type text in Command Finder edit box
- Improved layout of the matched command list to provide extra information, such as currently assigned shortcut, location of the command, and access help
- A hyperlink to allow users to search online for the query
- Support for searching within toolbars and Custom/User Tools



Simcenter Femap 2401 License Management

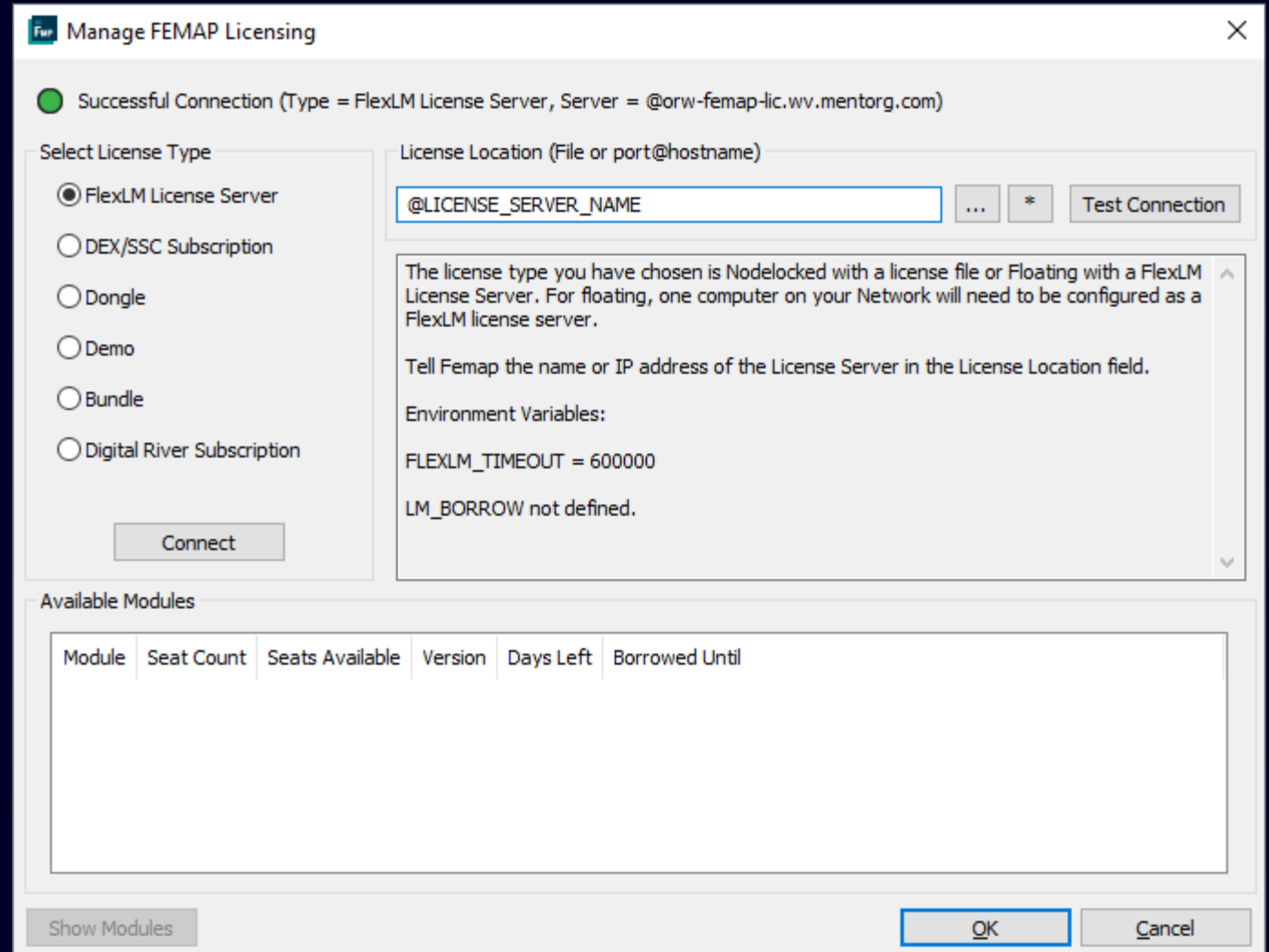
Added Help -> Manage Licensing command which can be used to change the licensing method for Femap at any time

- Eliminates need to shut down Femap and use one of the go_*.bat files in the Femap install directory to change licensing method
- The *Manage FEMAP Licensing* dialog box changes depending on which option is chosen in *Select License Type*
- An indicator at the top of the dialog box provides a “visual cue” to the user that Femap is:
 - Properly Licensed (Green-Filled Circle)
 - Not Properly Licensed (Red-Filled Circle)...along with other information



Simcenter Femap 2401 License Management

- Helpful information about each license type is now provided in the text section below the *License Location* field
- When using either *FlexLM License Server* or *Bundle*, additional icons and buttons are available
 - “...” – Browse to license file location
 - “*” – Generate FQDN (Fully Qualified Domain Name)
 - “Test Connection” button – Performs a “ping” of the server machine
- Show Modules/Show Users button toggles and will list Modules or Modules and Users to the Available Modules section



Simcenter Femap 2401

API – New Properties/Attributes

- Added NasMsnlCntRTOLB and NasMsnlCntSTKFRST to Analysis Case Object
- Added NasMsnlCntRTOLB, NasMsnlCntSTKFRST, NasBulkSLOSHCMS, NasBulkSLOSHCMSVal, NasACMODLExport, NasACMODLNORMAL, NasACMODLOVLPANG, NasACMODLSRCHUNIT, NasACMODLINTOL, NasACMODLAREAOP, NasACMODLCTYPE, NasACMODLFSETEntityOpt, NasACMODLFSETOpt, NasACMODLFSET, NasACMODLSSETEntityOpt, NasACMODLSSETOpt, NasACMODLSSET, NasFLSTCNTExport, NasFLSTCNTACSYM, NasFLSTCNTACOUT, NasFLSTCNTASCOUP, NasFLSTCNTREFDB, NasFLSTCNTKINOUT, NasFLSTCNTAGGPCH, NasFLSTCNTFEF70, NasINCMOPTOn, NasINCMOPTExport, NasINCMOPTETHOD, NasINCMOPTSF, NasINCMOPTSFTOL, NasINCMOPTBSIZE, NasINCMOPTNITER, and NasINCMOPTRBFLMASS to Analysis Manager Object
- Added IsINCM PFL, INCM PFLSurfType, INCM PFLGrav, and INCM PFLCmsModes to Connection Region Object
- Added ViewAxesSize to View Object

Simcenter Femap 2401

API – New Methods, Functions, and Global Variables

- Added InnerLoopCurvesAsSet Method to Solid Object
- Added feCheckCoincidentElemQuick function
- Added Pref_UnifiedArchitectureOpenGL and Pref_UnifiedArchitectureOpenGL to specify the options in the Unified Architecture Options dialog box which is accessed by the Unified Architecture button on the Graphics tab of File -> Preferences

Contact

Published by Siemens DI Software

Andy Haines

Principal Applications Engineer
Simcenter Femap Development
411 Eagleview Blvd
Exton, PA 19341
United States of America

Phone +1 610-458-6505

Mobile +1 610-500-1002

E-mail andy.haines@siemens.com



Contact Us

San Diego • Huntsville • Washington D.C.
Denver • Los Angeles • Berkeley • Albuquerque



San Diego
13290 Evening Creek Drive S
San Diego, CA 92128
(858) 480-2000



sales@ata-e.com



www.ata-e.com

Follow ATA Engineering

