

Webinar: **What's New in Femap 2020.2** July 30th, 2020

Host: Jonathan Hill ATA Engineering, Inc.

Presenter: Andy Haines Siemens Digital Industries Software

5



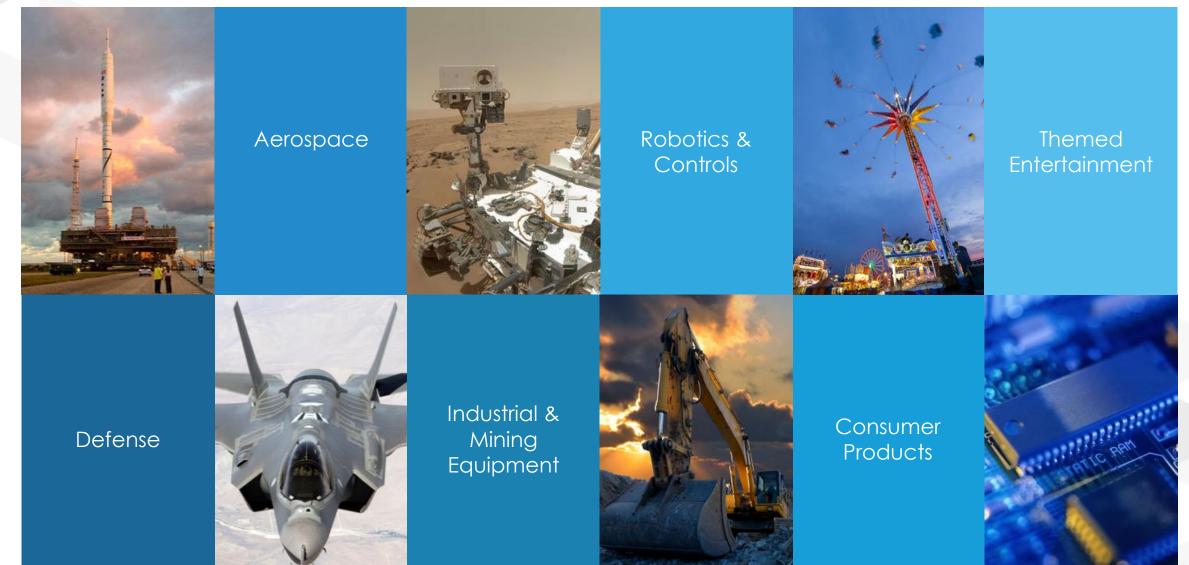
13290 Evening Creek Drive S, San Diego CA 92128

in ata-engineering



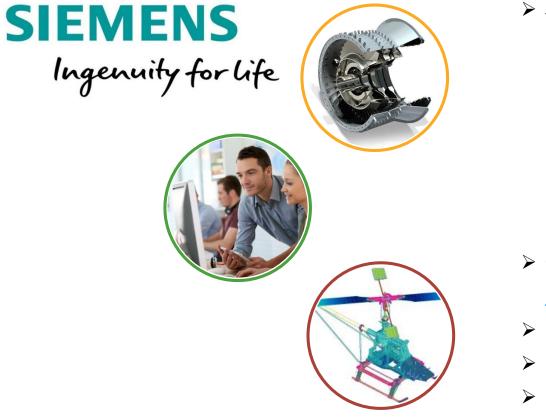
ATA Provides High-Value Engineering Services With Expertise in Design, Analysis, and Test

ATA Engineering helps to overcome product design challenges across a range of industries



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.



- > Siemens product lines we support include:
 - Simcenter STAR-CCM+
 - ➢ Simcenter Femap
 - Simcenter Nastran (formerly NX Nastran)
 - ➢ Simcenter 3D
 - ➢ NX CAD & CAM
 - ➤ Teamcenter
 - ➢ Solid Edge
- Contact the hotline at 877-ATA-4CAE or

http://ata-plmsoftware.com/support

- > Developer of the official Simcenter Nastran training materials
- Preferred North American provider of Simcenter Nastran training
- > Recognized as Smart Expert Partner with validated expertise in

Femap, STAR-CCM+, and Simcenter 3D





Visit Our Website for Product Information and Free Resources

www.ata-plmsoftware.com

E Support ATA X			Resources ATA X	
← → C ① www.ata-pimsoftware.c	com/support/	☆ □ 9	← → C ① www.ata-pimsoftware.com/resources/?pc=NX	x 0
Support	PRODUCTS ABOUT EVENTS & TRAINING			
	Support ATA's team of experts provide compre	hensive technical support for all of the Siemens software	Resources are password protecte	
		: of our technical team will be in touch shortly. 32-4223. Hours of operation (8a.m. to 8p.m. Eastern) Your Email*	Whitepapers Macro	25
				mber Groups In NX MACRO
	Company*	Phone Number*		k Element Quality MACRO
	Software Product*		Using Synchronous Modeling to Manipulate Solid Bodies in NX Renu PDF	mber Labels in NX MACRO
		Y	Beam Post-Processing with Cross-Section Views in NX PDF	1 200 00 00 00 00 00 00 00 00 00 00 00 00
	Please describe the issue you are h	aving	Basic FEM Checks in NX Por Presentations Other	r Resources
	Submit			emand Webina Simcenter Femap
				emand Webina version 2020.2
			Workflows in NY: Product and Manufacturing Information (PMI) for Design and Analysis PDF On-D	a update for keys ^a , hauge ^a as 15 Ontes, manual Wohlen Maximizing simulation productivity on biomed update is for more biograf
			Comparison of Composite Modeling Techniques PDF Patte	Summary Summary 2020.2. The software is now refer Pertormance enhancements through Sincenter Fertap th software is a stand- and the audication to increase "audity Appe Snite element modeling are and and and in the formation of the software is a stand- and the audication to increase "audity
© 2017 ATA Engineering. All rights reser PRODUCTS ABOUT EVENTS & TRAININ	rved. Web Design by Eyesparks NG SUPPORT NEWS CONTACT SEARCH		Training Videos Tutor	 of Biff to some all was not and allowing and allowing and allowing to some and allowing and allowing and allowing to some allowing allowing allowing allowing allowing allowing to some allowing allowing allowing allowing allowing to some allowing allowing allowing allowing allowing allowing allowing allowing allowing allowing allowing allowing allowing allowing allow
				wind taking the Schemes August types of Schemes Integration and the Schemes August memory and





Webinar: **What's New in Femap 2020.2** July 30th, 2020

Host: Jonathan Hill ATA Engineering, Inc.

Presenter: Andy Haines Siemens Digital Industries Software

5



13290 Evening Creek Drive S, San Diego CA 92128

in ata-engineering





Simcenter Femap v2020.2 What's New

NIAUAU

Unrestricted © Siemens 2020

Where today meets tomorrow.

Simcenter Femap Direction



35+ Years Dedicated to the Needs of the FEA Analyst!

Maximize efficiency of FEA tasks

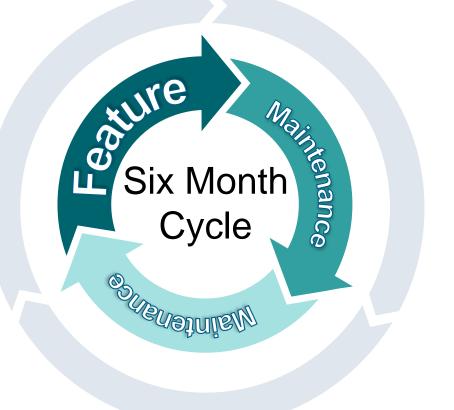
- Efficient creation of high fidelity FE models that accurately represent real-world engineering problems
- Intuitive interpretation of analysis results to improve the design and performance of engineered products

Build upon strong Simcenter Femap capabilities

- Geometry idealization and processing for FE models
- Powerful meshing, model creation and interactive editing
- In-depth support for industry standard solvers
- Flexible customization tools to streamline analysis processes

Simcenter Femap Roadmap – Update Release Cadence





Feature Release Every Six Months Database Change, UI Updates including Localization, Licensing Updates – April - October

Maintenance Release – If Required Database, UI, Licensing Unchanged – Bug Fixes/Critical Updates Only

Maintenance Release – If Required Database, UI, Licensing Unchanged – Bug Fixes/Critical Updates Only

Simcenter Femap Roadmap – Update Release Cadence

Feature Release – April/October

- New Features
- Expanced Functionality
- Database Changes
- API Expansion
- Will Require Licensing Update, Localization Update

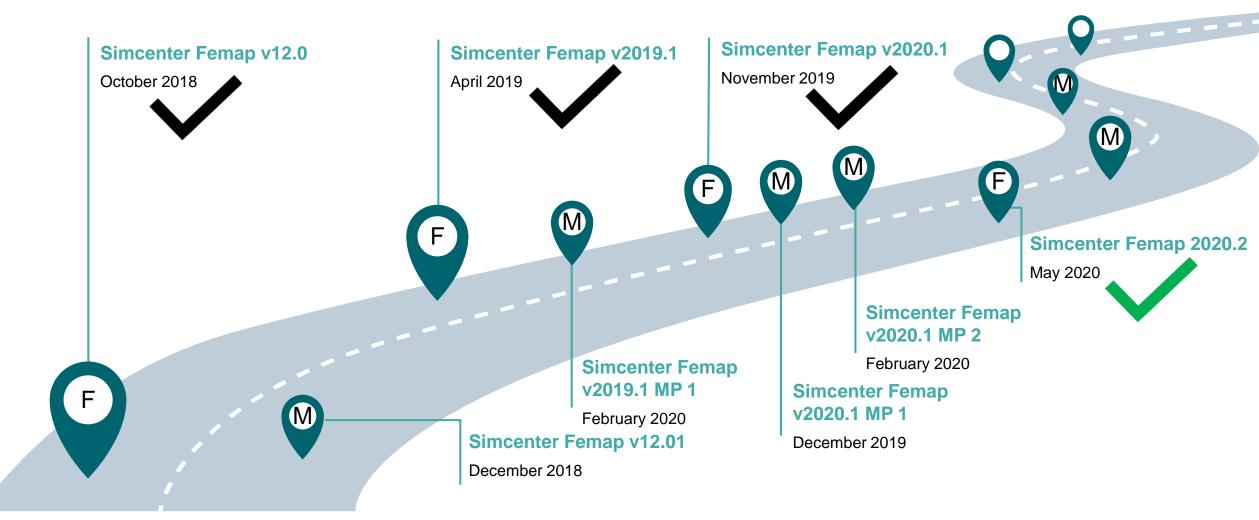
Maintenance Release

- Bug Fixes
- Minor Enhancements without User Interface Changes
- Working to Modernize Installer Maintenance Releases as Patches, not full Installs
- Simcenter Nastran Update



Simcenter Femap v12.0 and Beyond Roadmap





Agenda: Performance Improvements UI and Visualization Preprocessing Meshing Post Processing Solver Support Miscellaneous

Simcenter Femap v2020.2 – What's New Performance Improvements



- Connect->Automatic command
 - 3x-5x improvement in models with a large number of connections
- Geometry, Copy/Rotate/Reflect: Mesh, Copy/Rotate/Reflect; and Move commands
 - Significant improvements have been made to both the copy/move phase of these commands and the redrawing phase
 - Most noticeable improvement will be in large models when copying small portions of the model...potentially 40x faster.
- Length-based operations on Curves
 - Impacts Mesh Sizing and Coordinate methods such as Along Curve, Length Along and Midpoint by making these methods both Faster and More Accurate
 - Mesh Sizing 2x-3x faster on large model

Simcenter Femap v2020.2 – What's New Performance Improvements

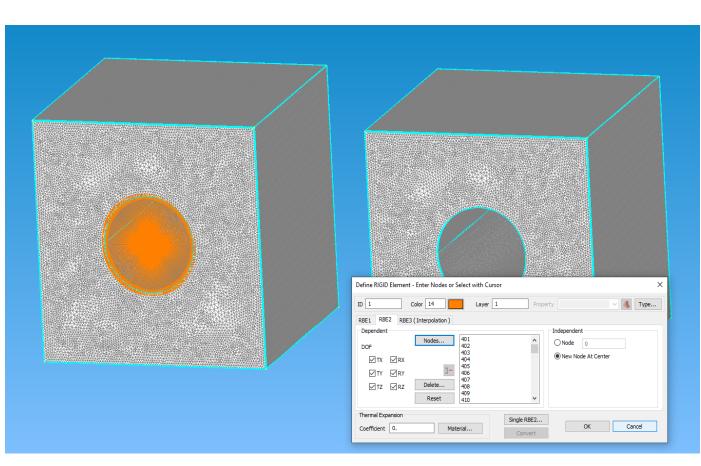


- Commands which can be "Aborted"
 - Many Checking and Listing commands can be aborted while doing computations or listing results.
 - Anywhere from 2x-100x improvement in speed of these commands with only slightly slower abort reaction time
 - Largest improvements in the Tools->Check and Tools->Mass Properties commands
- Output Process
 - **30%+** improvement in the Envelope, Linear Combinations, and RSS combinations
- Nastran Model Read
 - Almost **2x** improvement in importing large input files
- ANSYS Results Read
 - 2x-3x improvement when reading results from non-compressed RST files

Simcenter Femap v2020.2 – What's New Performance Improvements – Graphics



- Performance
 - Entity creation (especially in PG*)
 - Minimize free face evaluation
 - Minimize full graphics regenerations
- Creating a single entity when the model is already large was slow in PG*.
 - Creating a single rigid element in a model with 3.3 million 10-noded tetrahedral elements took 29 seconds in 2020.1, but now takes 0.3 seconds (~95x improvement)
- Some improvement to entity copy in large models but not as significant as in creation



*PG = Performance Graphics in Simcenter Femap

Simcenter Femap v2020.2 – What's New Performance Improvements – Graphics

- Added Crtl-G Group Evaluate option to Graphics tab of File->Preferences, which performs Group->Operations-> Evaluate on all groups currently visible when using Window-> Regenerate (Ctrl+G Shortcut Key). Helps when creating entities into the displayed group.
- Added *Disable Abort* option to *Graphics* tab of File->Preferences to disable checking for user aborts while drawing
- Max VBO MB and Min VBO B are available to specify when Best Possible is enabled
- Improved performance of thumbnail generation for large models from 30 seconds to 10 seconds

Geometry/Mod	el	Interfa	aces	Resu	ts	Library	/Startup		Color	Spaceba
Messages	Vier	NS	Grap	hics	U	lser Interf	ace	Da	atabase	Solvers
Graphics Opti	ons		Ind	ude In D	ynar	nic Rotatio	n			
Hardware /	Acceler	ation	\checkmark	Point		Coor	dinate S	iys	🗹 Fill	
Best Possib	ole		\checkmark	Curve		✓ Node			🗹 Shadir	ng
Performant	ce Grap	ohics		Surface		🗹 Elem	ent		Smoot	th Lines
3Vertex Buf	ffer Ob	je 🖂		Boundar	y	Cons	traint		✓ Filled I	Edges
Max VBO MB	1024	+ ~		Solid		🗹 Load			Mesh	Locations
Min VBO B	1024	+ ~		Text		Conr	ections		Undef	formed
Memory Op	otimiza	tion		Labels		Elem	ents as	Free	Edge	
Use Midside			\checkmark	Workpla	ne	Elem	ent Sym	bols	Cente	er
Multi-Mode	l Memo	ry	Tex	tures			Adva	anced	l / Debug C	ptions
Beam Face	t Edge	s		2D Mapp	ing		0	No De	ebug Messa	ages 🗸
Smooth Lin	es			Smooth T	Textu	ires		E	lapsed Tim	e
Auto Rege	nerate		F	orce All	Triar	ngles	F	rame	Rate	
Fast Picking	9		Max	Size	409	5 ~	0	No Op	enGL Erro	rs 🗸
Fast Pick V	isible		Bloc	k Contro	ol		1	Excee	ed VBO limit	t v
XOR Picking	g Grapi	nics	Bloc	k Size	3	2 ~	Bitm	an Al	ianment	4
Edges Usin	-		Cont	ch Dept		10			-	
Dialog Refr			<u> </u>		L		Pixe	I Forr	nat (0=Aut	to) 0
Ctrl-G Grou		uate	Digi	t Contro			BitB	lt Deli	ay (ms/MPi	x) 0
Disable Ab			J	Loca	le		TDR	prot	ection	0
Max Mag	100	00.		Optio	ons			Openi	MP Op	otions

Agenda:

Performance Improvements **UI and Visualization** Preprocessing Meshing Post Processing Solver Support Miscellaneous

Simcenter Femap v2020.2 – What's New UI and Visualization – High DPI Support



New Icons

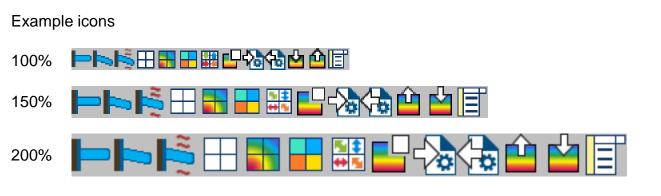
- Icons were redrawn with a modern style and color palette
 - Colors selected to provide enhanced contrast making selection easier
 - Consistent theme throughout Femap
- Designed to maximize familiarity to minimize learning curve

		· · · · · · · · · · · · · · · · · · ·	······································
夑 Simcenter Femap with Nastran - [wingpost.modfem] -	[Untitled]	夑 Simcenter Femap with Nastran - [wingpost.modfem] - [Untitled]
Eile Iools Geometry Connect Model Mesh	M <u>o</u> dify <u>L</u> ist <u>D</u> elete <u>G</u> roup <u>y</u>	 Simcenter Femap with Nastran - [wingpost.modfem] - [File Tools Geometry Connect Model Mesh Control to the second s	M <u>o</u> dify <u>L</u> ist <u>D</u> elete <u>G</u> roup [⊕
铝 Model Info 👩 Meshing 📻 PostProcessing		🎦 Model Info 📸 Meshing 📲 PostProcessing	
Old		New	

Simcenter Femap v2020.2 – What's New UI and Visualization – High DPI Support

New Icons – All 1600+ Completed in v2020.2!

- Vector assets of all icons allow for Femap to fully support high resolution (4K) displays
- UI scaling Full high DPI support in 2020.2 now available!!!
- Other options can be set with Preference:
 - Disabled Femap is drawn at 100% resolution, however text and icons can be small and hard to read
 - Handled by Windows Femap is scaled at the Windows scaling setting via extrapolation; leads to blurry text and blurry text and icons



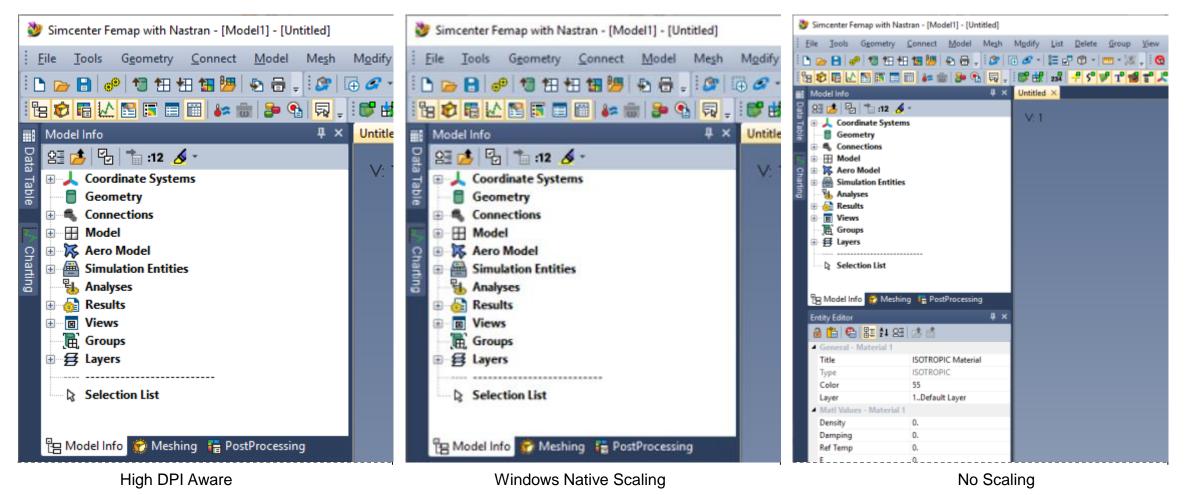
- 100%
- 150%



Simcenter Femap v2020.2 – What's New UI and Visualization – High DPI Support



High DPI Support - v2020.2 examples on 4k monitor



Unrestricted © Siemens 2020

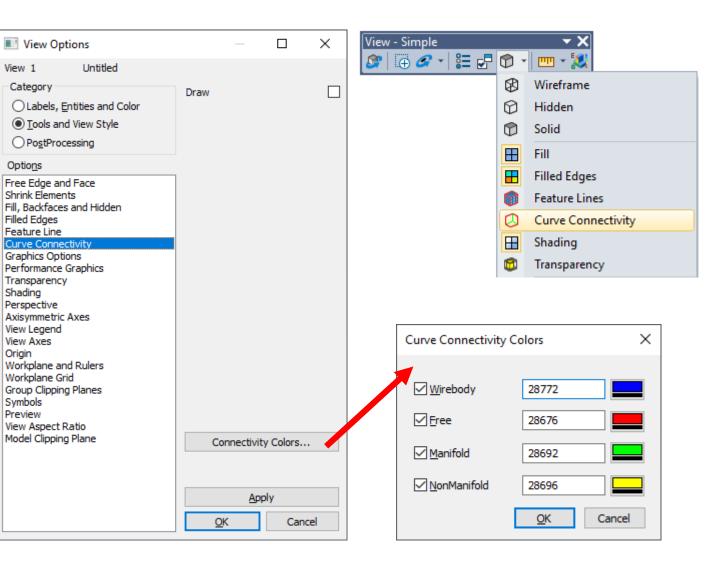
Siemens Digital Industries Software

Simcenter Femap v2020.2 – What's New UI and Visualization – Curve Connectivity



Curve Connectivity in View->Options colors Curves by level of connectivity

- Override curve color based on surface connectivity
 - Free 1 surface attached
 - Manifold 2 surfaces attached
 - NonManifold more than 2 surfaces attached or "T-junction"
 - Wirebody 0 surfaces attached (not common as Wirebody entities cannot be created in Femap)

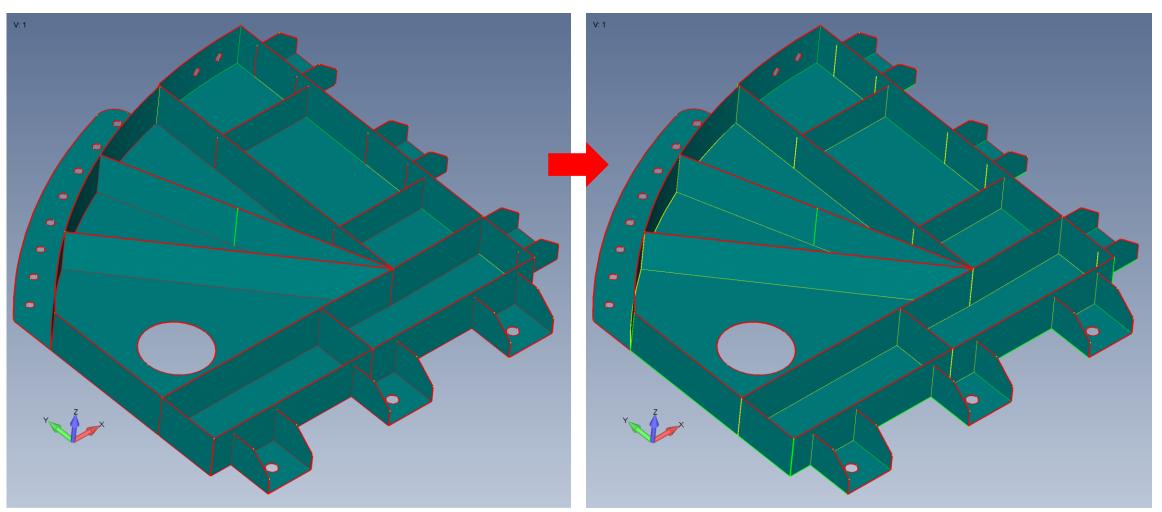


Simcenter Femap v2020.2 – What's New UI and Visualization – Curve Connectivity



Before NonManifolod Add – No NonManifold Curves Exist

After NonManifolod Add – NonManifold Curves Shown in Yellow



Simcenter Femap v2020.2 – What's New UI and Visualization – Renumbering



Improved Renumbering commands to provide better control when choosing to offset IDs

- Replaced old "Constant Offset" option with new *Renumbering Options* section
 - Compress Specify Starting ID and optionally Increment
 - Offset To Specify Starting ID
 - Offset By Specify ID Offset

Renumber To	×
Starting ID 1 Increment Sort Renumbered Order by Original ID Selection Order	1 Sort Order () <u>A</u> scending () <u>D</u> escending
 ○ <u>C</u>olor ○ <u>L</u>ayer ○ <u>T</u>ype ○ Property ○ Minimum <u>N</u>ode ID 	Renumbering Options Compress Offset To Offset By
○ Coordinates	Verify <u>R</u> enumbering
C≦ys 0Global Rectangular ✓ Order +X ✓ +Y ✓ +Z ✓	
Tolerance 1.E-8	<u>O</u> K
Absolute <u>V</u> alue	Cancel

Simcenter Femap v2020.2 – What's New UI and Visualization – Copy/Save Formats



Enhanced Copy to Clipboard and Save to File commands for the Data Table, Connection Editor, Mesh Point Editor, Function/Table Editor and Entity Editor to copy/save data using formatted HTML.

		Data Table										3				
	i	a 🛅 🛱	3 🖬 🚍 - 4	: 🗣 E	염 🖌 🔀 😽	8	•									
	10	D Prop IE)	Тур	e Topology	Color	Layer	Formulation	C1 C	2 C3	C4 Orien	tation Vecto	r			
		06 101Up	oper Wing Skin	PLA	TE Quad, 4-noded	0	1	None	204 2	05 209	10					
	2	14 5Upp	er Stringer - T Se	ction BEA	M Line, 2-noded	124	1	None	200 2	01	0., -1.	, 0.				
	9	7 101Up	oper Wing Skin	PLA	TE Quad, 4-noded	15	1	None	63	22 197	7					
ID	Prop ID		Туре	Торо	ology	Со	lor	Layer	Fo	rmu	latio	n C1	C2	C3	C4	Orientation Vector
106	101Upper Wi Skin	ng	PLATE	Qua	d, 4-nodeo	0 b		1	No	ne		204	205	209	10	
214	5Upper String T Section	ger -	BEAM	Line,	2-noded	124	4	1	No	ne		200	201			0., -1., 0.
97	101Upper Wi Skin	ng	PLATE	Qua	d, 4-nodeo	d 15		1	No	ne		6	322	197	7	

Improves formatting when pasting into Office Tools and allows viewing of saved files with standard browsers.

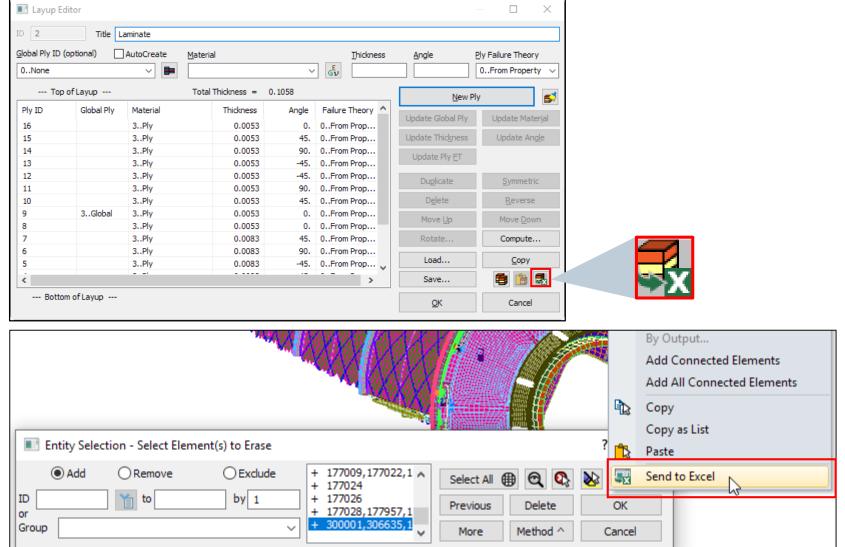
Simcenter Femap v2020.2 – What's New UI and Visualization – Send to Excel



New technology in Simcenter Femap offers ability to transfer data from dialog boxes and panes directly to Microsoft Excel

Examples of dialog boxes which have *Send to Excel* commands or icons to transfer data are:

- Layup Editor
- Standard Entity Selection



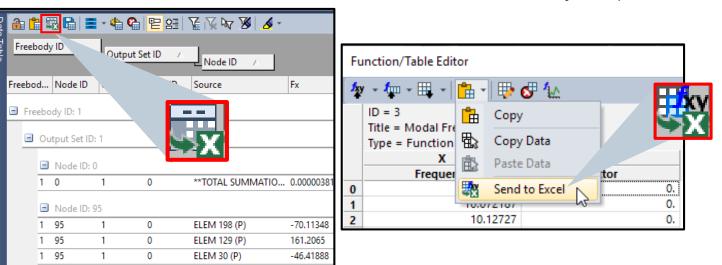
Simcenter Femap v2020.2 – What's New UI and Visualization – Send to Excel

Various dockable panes in Femap are now outfitted with *Send to Excel* commands or icons

Typically located next to Copy/Save icons or on *Copy to Clipboard* menu

Data is sent with HTML Formatting

- Organized
- Clean look and feel
- Easy to manipulate column width and other features of the table



J	AutoSave 💽 Off) 📙 🏷 - 🖓 - 🗧										
	Α	В	С	D	E	F	G	Н	- I	J	К
1	Freebody ID	INODE ID	Output Set ID	CSys ID	Source	Fx	Fy	Fz	Мх	Му	Mz
2	Freebody ID): 1									
3	Output Set	ID: 1									
4	Node ID: 0										
5	1	0	1	0	**TOTAL SUMMATIO N**	3.815E-06	1500	7.629E-05	-58050.01	0.0018311	23253.75
6	6 Node ID: 95										
7	1	95	1	0	ELEM 198 (P)	-70.11348	155.5345	462.3213	0.134426	0.953465	0.341325
8	1	95	1	0	ELEM 129 (P)	161.2065	11.38988	203.6129	-0.0564618	-0.0990452	-0.931557
9	1	95	1	0	ELEM 30 (P)	-46.41888	18.41415	326.4073	79.94971	98.96718	6.704117
10	Node ID: 17	4									

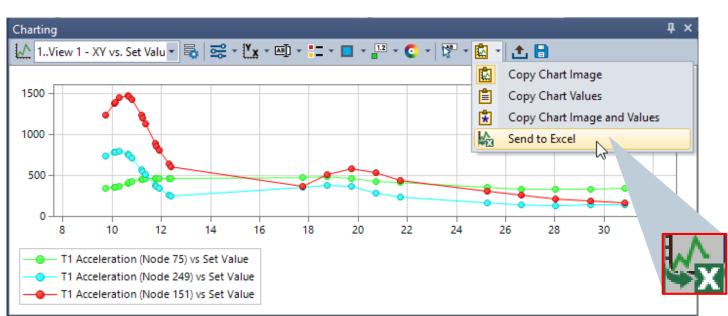


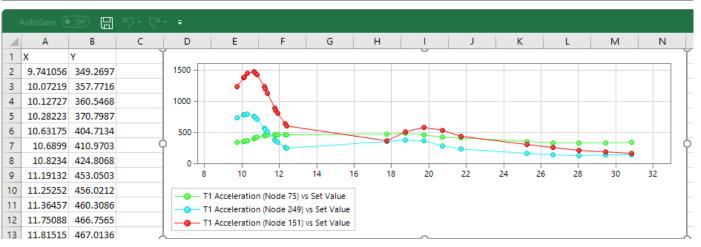
Simcenter Femap v2020.2 – What's New UI and Visualization – Send to Excel



Send to Excel command in Charting pane simultaneously transfers:

- XY Data Pairs for ALL Data Series currently visible in the *Charting* pane
- Image of Chart





Siemens Digital Industries Software

Simcenter Femap v2020.2 – What's New UI and Visualization – Miscellaneous



Updated *Equation Editor* dialog box to be resizable, added context-sensitive help by pressing *F1 key*, and reordered items in the *Functions* list to group similar items together.

Added ELTHK(elemID ; cornerID), which returns planar element corner thicknesses.

Updated View->Options command to automatically switch *Color Mode* to appropriate "View Color" option after using *Color Palette* to select a color, then clicking *OK*

Updated selection of solids to allow use of the Select Visible icon, but only selects visible solids, not "Simcenter Femap volumes"

0	<u>S</u> tored Cu <u>r</u> rent <u>E</u> quation	+ - / 0	SIN() COS() TAN() ASIN() ACOS() ATAN()	^	1Coordinate System 2Point 3Curve 4Surface	<u>O</u> K Cancel
			LOG0 LN0 INT0 ABS0 RND0 SQR0 SQRT0 EXP0 POW(;) VEC(;;) XPT0 YPT0 ZPT0 XCUCEN0 YCUCEN0 ZCUCEN0 XCUCEN0 XCUMID0 ZCUMID0	~	5Solid/Volume 6Text 7Boundary 8Node 9Element 10Material 11Property 12Load Set 13Constraint Set 14View 15Output Set 16Output Format 17Connector 18Connection Property	Ca <u>l</u> culate S <u>a</u> ve Variab
Insert Variab	le	Inser <u>t</u>		In	isert Fu <u>n</u> ction	

Simcenter Femap v2020.2 – What's New UI and Visualization – Miscellaneous



Updated List->Geometry->Curve command to include Arc/Circle center coordinates when *Advanced* option is enabled.

ssages												
C urves ve(s) Sele	ected											
ID	Type	Color	Layer	Bias	MeshElem		Points		#Surface	Length	Radius	Angle
13	Edge	120	1	0.	0	14	13		0	0.392699	0.125	180
		Seg	ments:	17	Address: 410							
		Cen	ter:		X=0.		Y=4.	Z=1.				
15	Edge	120	1	0.	0	16	15		0	0.589049	0.1875	180
		Seg	ments:	17	Address: 391							
		Cen	ter:		X=0.		Y=5.	Z=1.				
18	Edge	120	1	0.	0	18	17		0	0.785398	0.25	180
		Seg	ments:	17	Address: 389							
		0	ter:		X=0.		Y=6.	Z=1.				

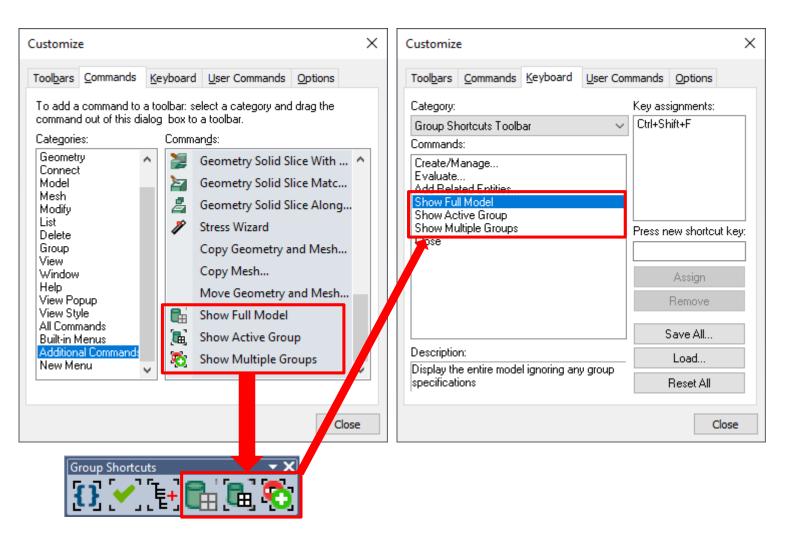
Improved Model->Merge to more cleanly handle cases where user intentionally chooses to overwrite existing entities in the current model.

Simcenter Femap v2020.2 – What's New UI and Visualization – Miscellaneous



Added ability to use commands on "floating" toolbars (i.e., not "docked") while using another command. It was already possible to use commands on "docked" toolbars in previous versions.

Added "Group Display Mode" (i.e., *Show Full Model, Show Active Group*, and *Show Multiple Groups*) commands under "Additional Commands" on the *Commands* tab of the *Customize* dialog box. Once added to any toolbar, they can also have shortcut keys assigned



Agenda:

Performance Improvements UI and Visualization **Preprocessing** Meshing Post Processing Solver Support Miscellaneous

Simcenter Femap v2020.2 – What's New Preprocessing – Beam Properties



Added Write Zeros at End B (Off=Blank) option for Nastran solvers to Beam Property

Define Property - BEAM Elemen	nt Type				×	If option is
ID 2 <u>T</u> itle Taper	ed I-Beam		<u>M</u> aterial 1	1AISI 4340 S	teel $\sim G_{\nu}^{E}$	file for that
Color 110		ayer 1		Elem/Pro	perty Ty <u>p</u> e	
Property Values			Stress <u>R</u> ec	overy (2 to 4	Blank=Square)	Preview Analysis Inpu
☑ Tapered <u>B</u> eam ☑ Writ	e Zeros at End End A	B (Off=Blank) End B		Y	z	< 1 >< 2 >>
Area, <u>A</u>	0.205	0.16605	End A 1	-0.375	-0.261473	\$ Femap Property
Moment of Inertia, 11 or Izz	0.00462396	0.00303378	2	0.375	-0.261473	 \$ Femap PropShap \$ Femap PropMeth
I2 or Iyy	0.029066	0.0190702	3	0.25	0.738527	\$ Femap PropOrie \$ Femap PropShap
I <u>1</u> 2 or Izy		0.	4	-0.25	0.738527	\$ Femap PropMeth \$ Femap PropOrie
Torsional Constant, <u>J</u>	7.07667E-4	0.0004643	End B 1	0.0075		PBEAM 1 +375
Y S <u>h</u> ear Area	0.103553	0.		-0.3375	-0.235325	+ YES +3375
Z Shear Area	0.0885987	0.	2	0.3375	-0.235325	+ .5051366
Nonstruct mass/length	0.	0.	3	0.225	0.664675	\$ Femap Property \$ Femap PropShap
Warping Constant	0.	0.	4	-0.225	0.664675	\$ Femap PropMeth \$ Femap PropOrie
Perimeter	4.3	3.87				\$ Femap PropShap \$ Femap PropMeth
Y Neutral Axis Offset	0.	0.		Shape		\$ Femap Prophet PBEAM 2
Z Ne <u>u</u> tral Axis Offset	0.183649	0.165284		Shape En		+375 + YES
Loa <u>d</u> <u>S</u> ave	(Сор <u>у</u>		<u>o</u> k	Cancel	+3375 + .5051366 +

If option is disabled, nothing is written to Nastran input file for that field, which means End B = End A for field

Preview Analysis Input File		
< 1 >< 2 >< 3 >< 4 >< 5 >< 6 >< 7 >< 8 >< 9	><	
\$ Femap Property 1 : Tapered I-Beam		^
\$ Femap PropShape 1 : 9,0,1.,0.5,0.75,0.1,0.1,0.1		
\$ Femap PropMethod 1 : 9,0,1,0.3		
\$ Femap PropOrient 1 : 9,0,0.,1.,2.,7.,8.,-1.,0.,0.		
\$ Femap PropShape 1 : 9,1,0.9,0.45,0.675,0.09,0.09,0.09		
<pre>\$ Femap PropMethod 1 : 9,1,1</pre>		
\$ Femap PropOrient 1 : 9,1,0.,1.,2.,7.,8.,-1.,0.,0.		
PBEAM 1 1 .205 .004624 .029066 0.7.0767-4 +375261473 .375261473 .25 738527 - 25 738	0.+	
	5771	
+ YES 116605.0030338.0190702 4.643-4 +3375235325 .3375235325 .225 .664675225 .664		
+ .5051366.4321888	10/34	
+ 0183649 0165	284	
\$ Femap Property 2 : Tapered I-Beam	,201	
\$ Femap PropShape 2 : 9,0,1.,0.5,0.75,0.1,0.1,0.1		Edit <u>P</u> revi
\$ Femap PropMethod 2 : 9,0,1,0.3		
\$ Femap PropOrient 2 : 9,0,0.,1.,2.,7.,8.,-1.,0.,0.		
\$ Femap PropShape 2 : 9,1,0.9,0.45,0.675,0.09,0.09,0.09		Analyz
\$ Femap PropMethod 2 : 9,1,1		Analyz
\$ Femap PropOrient 2 : 9,1,0.,1.,2.,7.,8.,-1.,0.,0.	I	- Europe
PBEAM 2 1 .205 .004624 .029066 0.7.0767-4 +375261473 .375261473 .25 <u>738527 - 25 738</u>	0.+	Expor
+375261473 .375261473 .25 <u>738527 - 25 738</u> + YES 116605.0030338.0190702 0. 4.643-4	0	
+3375235325 .3375235325 .225 .664675225 .664		
+ .5051366.4321888	.0701	<u>D</u> one

Simcenter Femap v2020.2 – What's New Preprocessing – Tools

Tools->Parameters

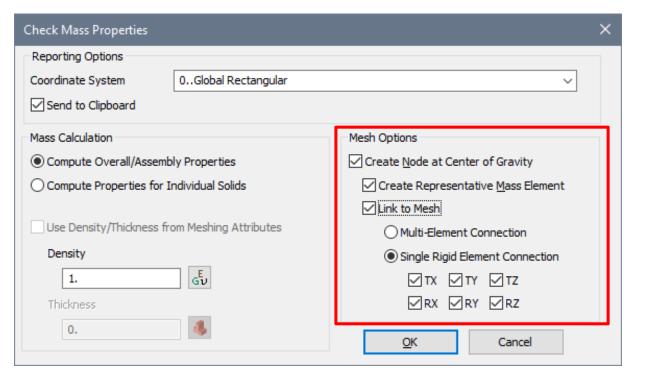
- Added ability to specify Output Coordinate System to use for all newly created nodes
- Added ability to specify Color, Next ID, and Inc for Monitor Points
- Add ability to specify Next ID and Inc for Matrix Input entities

Model Parameters										×
Color and ID Contro										
	Color	Next ID	Inc		Color	Next ID	Inc		Next ID	Inc
Coord S <u>y</u> s	60	3	1	Connection Prop	110	1	1	Layup	1	1
Point	24	1	1	Region	20488	1	1	Load Set	1	1
Curve	120			Connector	14	1	1	Constraint Set	1	1
Combined Curve	20590	1	1	<u>N</u> ode	46	1	1	Function	1	1
Sur <u>f</u> ace	60	1	1	<u>E</u> lement	124	1	1	Group	1	1
<u>B</u> oundary	24642	1	1	Material	55	1	1	Output Set	1	1
Sol <u>i</u> d	66	1	1	Property	110	1	1	Output Format	1	1
Mesh Point	24578	1	1	Aero Panel	124	1	1000	View	2	1
Te <u>x</u> t	124	1	1	Aero Property	110	1		Matrix Input	1	1
Monitor Point	105	1	1	Aero Spline	24696	1	1		-	-
				Aero Surface	24590	1	1			
Active Entities				Acro Surface	24330	1		Marca Talanana		
	0 Clab	al Rectangular	~					Merge Tolerance	1.73205E-4	
Coord S <u>y</u> s		_	~	Loa <u>d</u> Set			~	0		
Output <u>C</u> Sys	0Glob	al Rectangular	~	Cons <u>t</u> raint Set			~	O Specified	0.	
				<u>L</u> ayer	1. Defa	ult Layer	~	Curve Faceting		
Connection Prop			~					Angle Error	1.5	%
				<u>G</u> roup			~	Chord Error	0.1	%
<u>M</u> aterial			~	Output <u>S</u> et			~	Curve Factor	3.	
P <u>r</u> operty			~	Output Form <u>a</u> t			~	<u>0</u> K	G	ancel

Simcenter Femap v2020.2 – What's New Preprocessing – Tools

Tools->Mass Properties->Solid Properties

- Added support for multiple solids
- Support copying the listing to the Clipboard in Text, RTF, and HTML format
- Update the listing format to be similar to Tools->Mass Properties->Mesh Properties
- Supported computing overall properties and properties for individual solids.
 "Individual Solids" is only available if you select multiple "true solids" ... no general or sheet bodies.



Added Mesh Options section to connect newly created representative mass element to any number of nodes in an existing mesh using either a single rigid element or multiple elements generated by Mesh->Connect->Closest Link functionality



Simcenter Femap v2020.2 – What's New Preprocessing – Tools



Tools->Mass Properties->Mesh Properties

- Added support to copy the listing to the Clipboard in Text, RTF, and HTML format
- Added support for Solid Laminates Elements
 - Laminate Layup thicknesses are scaled to match the length of each edge in the layup direction. Each ply is then computed as a "thin" single material brick/wedge with corners at the correct top/bottom ply locations.
- NOTE: CG computations will not match Nastran unless you use PARAM,COUPMASS,1. This is an error in Nastran, not Simcenter Femap. See PR# 9631432. Also, inertias will not match, just like all of the Simcenter Femap inertia computations, we simply use a lumped mass at the centroid and do not account for the inertia of the individual element geometry.

Check Mass Properties		×
	s are based on a point mass irate for small numbers of e	
Reporting Options		
Coordinate System	0Global Rectangular	~
List Properties for 1	Individual Elements	
Send to Clipboard		
Mesh Options		
Create Node at To	tal Center of Gravity	<u>о</u> к
Create <u>M</u> ass El	ement	Cancel

Simcenter Femap v2020.2 – What's New Preprocessing – Loads and Boundary Conditions



Added ability to create Follower Forces and Follower Moments (Nastran Only)

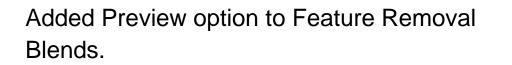
- Define a force or moment in terms of a magnitude and direction nodes
- 2 nodes/points can be specified to define the direction of the of the load terms of a vector.
- 4 nodes/points can be specified to define the direction as normal to a plane
- Both nodal and geometric follower loads can be defined

Create Loads on Nodes			×
Load Set 1 Untitled	ł		
Title	Layer 1	Coord S <u>v</u> s	0Global Rectangular 🗸
Force Follower Force Moment Follower Moment Displacement Enforced Rotation Velocity Rotational Velocity Acceleration Rotational Acceleration 	Direction Magnitude Only Vector Along Curve Normal to Plane Normal to Surface Load Val Magnitude 0. Node 1 (G1)		Method Constant Variable Data Surface Data Surface Data Surface
Static Fluid Pressure Total Fluid Pressure General Scalar Steam Quality Relative Humidity Fluid Height Condition Unknown Condition	Node 3 (G3) Node 4 (G4)		<u>O</u> K Cancel

Agenda:

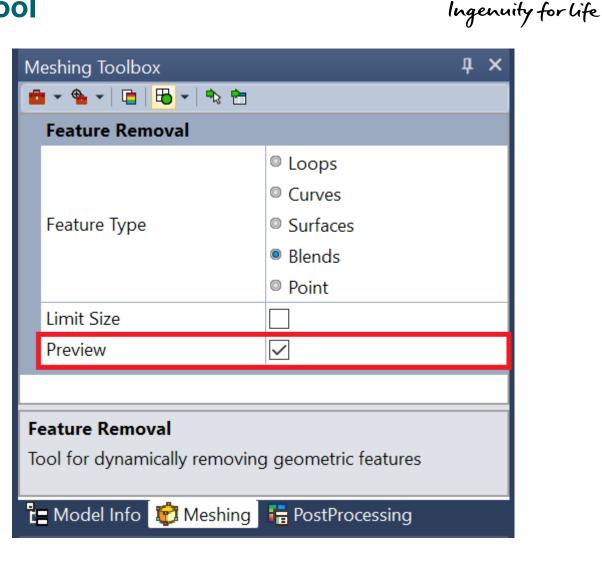
Performance Improvements UI and Visualization Preprocessing **Meshing** Post Processing Solver Support Miscellaneous

Simcenter Femap v2020.2 – What's New Meshing Toolbox – Feature Removal Tool



- Brings up surface entity select dialog with all found blends highlighted.
- Ability to add or remove specific blends.
- · Can be used with or without Limit Size.

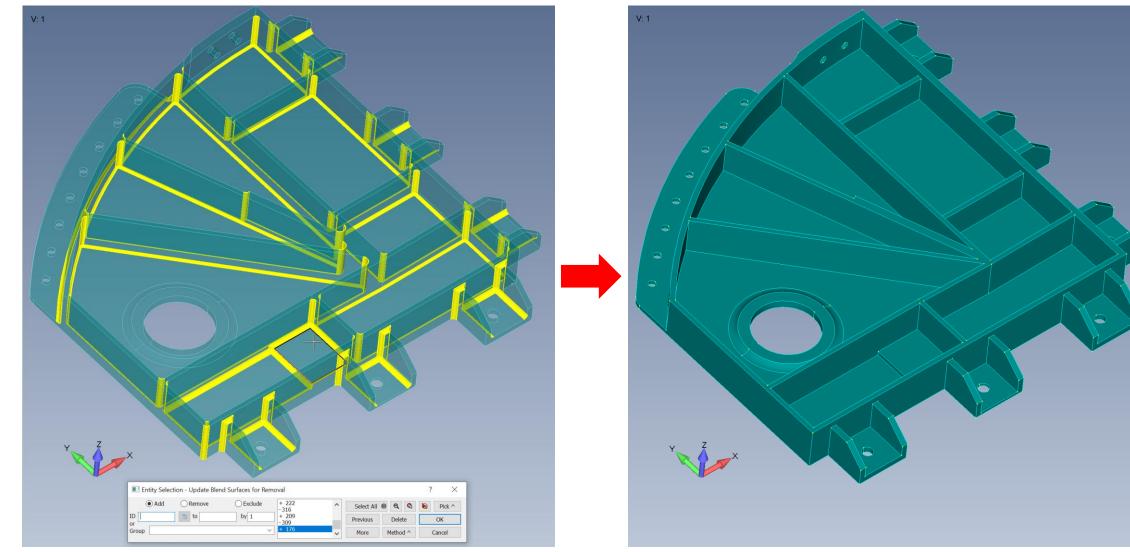
Unrestricted © Siemens 2020



SIEMENS

Simcenter Femap v2020.2 – What's New Meshing Toolbox – Feature Removal Tool





Simcenter Femap v2020.2 – What's New Meshing Toolbox – *Geometry Editing* Tool

Added Surface/Replace Face capability to Extend under Geometry Editing.

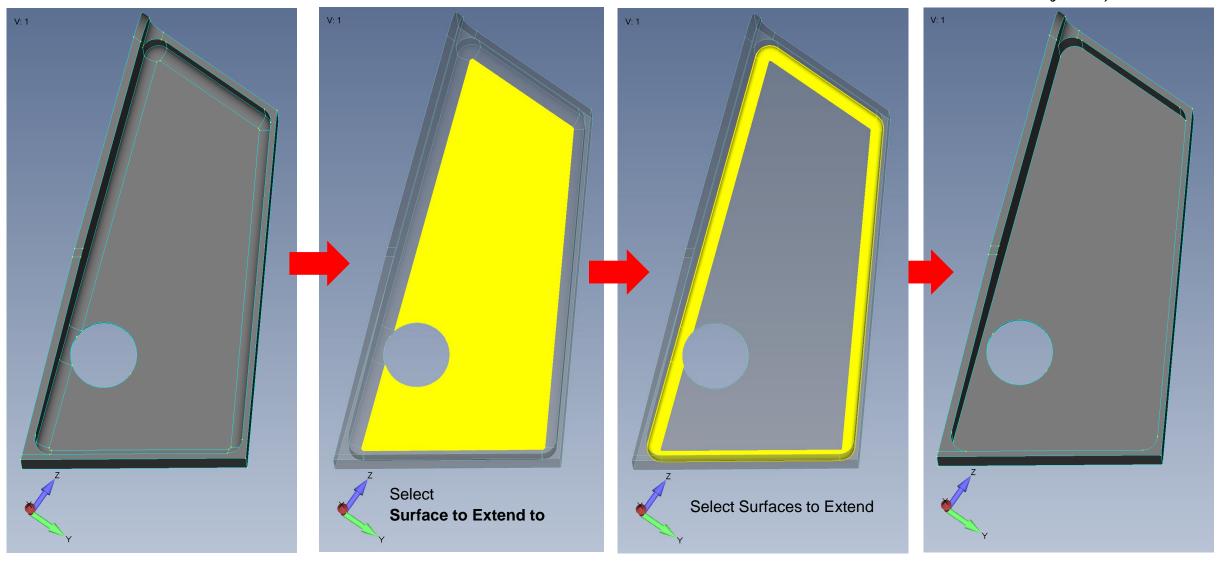
- Extend faces of solid up to selected surface.
- Remove fillets that have failed from blend removal.
- Raise or retract surfaces to main solid to remove features.
- Existing extend commands are now under Curve on Surface.

SIEMENS Ingenuity for life

Meshing Toolbox	ť	ιx			
💼 🕶 💁 🕶 🛅 🔂 🕶 🗞 🛅					
Geometry Editing					
Operation	 Curve Break Point to Point Point to Edge Edge to Edge Slice Pad Washer Extend Project/Move Point Project Curve 				
Geometry Type	Curve on Surface				
	Surface/Replace Face				
Surface to Extend To	0	V 2			
Geometry Type					
🗄 Model Info 😰 Meshing	PostProcessing				

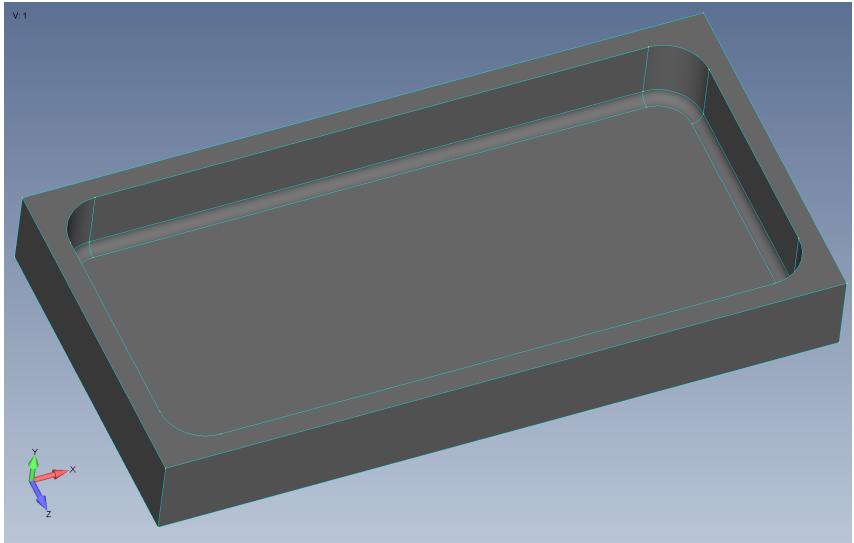
Simcenter Femap v2020.2 – What's New Geometry Editing Tool – Surface/Replace Face Workflow





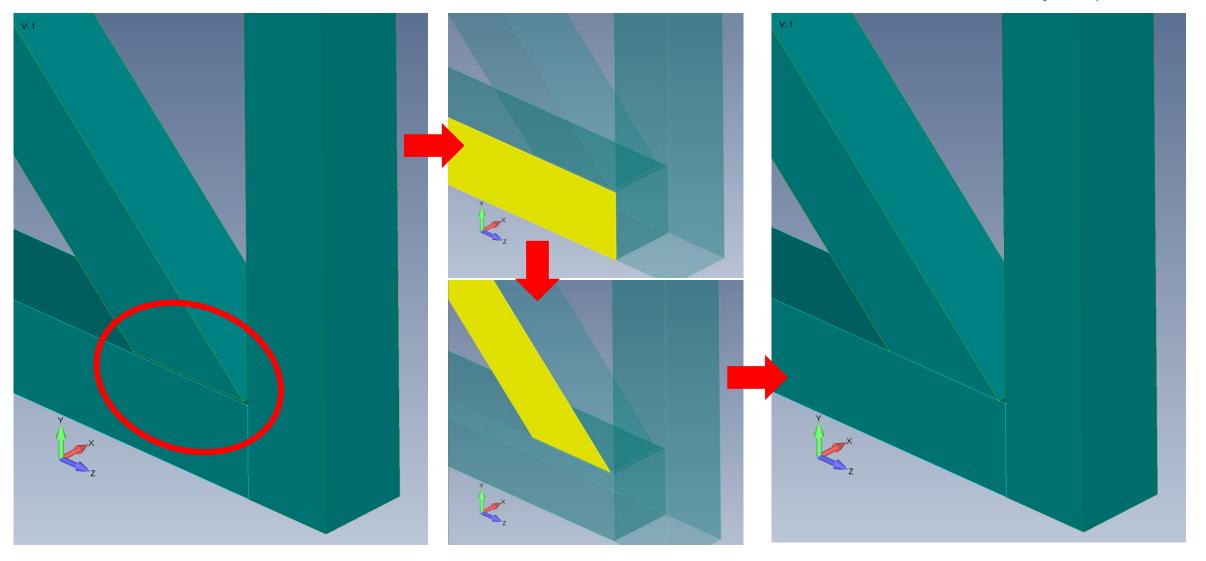
Simcenter Femap v2020.2 – What's New Geometry Editing Tool – Surface/Replace Face Workflow





Simcenter Femap v2020.2 – What's New Geometry Editing Tool – Surface/Replace Face Workflow





Simcenter Femap v2020.2 – What's New Meshing – Mesh Editing



- Updated various Mesh Editing commands to more fully and consistently handle the other entities associated and/or related to the elements being edited
- Commands that have been updated include:
 - Mesh->Editing->Interactive
 - Mesh->Editing->Split
 - Mesh->Editing->Element Refine
 - Mesh->Editing->Edge Split
- Features that are supported across these commands include:
 - Updating Nodal Constraints and Nodal Permanent Constraints
 - Updating Loads (On Faces, Corner Pressures, On Edges, Distributed Line)
 - Updating Regions (both Nodal and Elemental)
 - Updating Tapered Beam Shapes, Beam Offsets, Plate Thickness and Material Direction
 - Updating Groups
 - Updating Geometric Associativity

Simcenter Femap v2020.2 – What's New Meshing – Miscellaneous



- Mesh Sizing
 - Improved consistency when setting Mesh Sizes on curves using Element Size
 - Previously two curves of seemingly similar length could end up with a different number of elements (+/- 1) due to very small length differences.
- Combined Curve Creation (typically done to improve mesh quality)
 - Only allow creation of Combined Curves if all underlying curves have the same manifold type (i.e., all curves are Free Edge, Manifold, NonManifold or Wirebody).

Agenda:

Performance Improvements UI and Visualization Preprocessing Meshing **Post Processing** Solver Support Miscellaneous

Simcenter Femap v2020.2 – What's New Postprocessing – Send to Excel



- List->Output->Results to Excel
- Automatically opens Excel to transfer a variety of Results very quickly
- Take advantage of:
 - Row / Column control
 - Alternate column order
 - Formatting style
 - Transformation of Output Data
- Transfer Nodal and/or Elemental Results at same time

	Tools 🕨 📔 🔞 - 🚉 - 🕞	* 🖸 * 📭 * 📮 i 🔜 🖉 🖉	Send Results to Excel				
		🦑 . i ⊨ ⊨ 🏂 🖽 🎛 🖽	Report Style			Transform	
	Connection		Output Sets	In Rows) in Columns	○ None	
	Model •		Output Vectors	O in Rows	in Columns	O From Active	View
	Output 🔸 🛃 Query		Nodes/Elements	in Rows	O in Columns	Custom	
B			Houespeichertes	C III Kows	On Columns	Custom	
ď	Group Compare			Alternat	te Column Order	Transf	orm
e	View Summary to D	ata Table					
	Model Info Results to Data	a Table		Formatt	ing	<u>O</u> K	Cancel
	Destination Results Rankin	ig to Data Table					
		es to Data Table					
	Contoured Re	sults to Data Table					
	Results to Exce						
	Results to Exce						
			_		Results Transform	ation	?
Mod	del - TowerLMM.modfem				Nodal Vector Out	put	
	tput Sets	Output Vectors			None		
	6. •¤ b= p= Y 🖌 🕼	All Output Vectors From Output	Set 60Case 1 Freq 30.83381	~	O Into CSys	0Global Rectang	r v
			Ouick Filter 0None - Ignore	~	O Into Node Ou	tput CSys	
	♠ 1Mode 1, 10.8234 Hz ✓ ♠ 20Case ♠ 2Mode 2, 11.19132 Hz ✓ ♠ 21Case				Line Element Out	put	
	(a) 2,Mode 2, 11.19132 HZ (a) 21Case (b) 3Mode 3, 11.25252 Hz (c) (b) 22Case		3003Bar EndB Plane2 Momer 3004Bar EndA Pl1 Shear For		None	· • • • • • • • • • • • • • • • • • • •	
	🔂 4Mode 4, 19.72662 Hz 🛛 🗹 🔂 23Case	1 4T3 Translation	3005Bar EndA Pl2 Shear For		O Into CSys	0Global Rectangula	r v
	👰 5Mode 5, 28.03074 Hz 🛛 🗹 👰 24Case		✓ 3008Bar EndA Axial Force ✓ 3010Bar EndA Torque				
	6Mode 6, 29.21541 Hz ✓ 25Case 7Mode 7, 29.8124 Hz ✓ 26Case	8R3 Rotation	3075Bar EndA Pt1 Bend Stre		Shell Element Out	tput	
	(m) 7Mode 7, 29.8124 Hz ♥ (m) 26Case (m) 8Mode 8, 29.83483 Hz ♥ (m) 27Case		3076Bar EndA Pt2 Bend Stre 3077Bar EndA Pt3 Bend Stre		O Into Mati Dire	stian	
	[a] 9Mode 9, 30.89938 Hz [] ✓ [] 28Case	1 24T3 Acceleration	🗹 3078Bar EndA Pt4 Bend Stre	ess	_	0Global Rectangula	r v
	🚯 10Mode 10, 33.84494 Hz 🛛 🗹 🚯 29Case	1 26R1 Angular Acceleration	3083Bar EndB Pt1 Bend Stre 3084Bar EndB Pt2 Bend Stre		◯ Into CSys		
	▲ 11Case 1 Freq 9.741056 ▲ 30Case	2 28R3 Angular Acceleration	3085Bar EndB Pt3 Bend Stre	ess		⊚x ⊖y ⊖z	
	 ⁶/₁12Case 2 Freq 10.07219 ⁷/₆ ⁶/₁31Case ⁶/₁32Case ⁶/₁32Case 		✓ 3086Bar EndB Pt4 Bend Stre ✓ 3107Bar EndA Axial Stress	ess	O Along Vector	Vecto	
	🙀 14Case 4 Freg 10.28223 🗸 🕼 33Case	2 54T3 Constraint Force	🗹 3452Bar EndA Pt1 Bend Stra			Minimum Angle From Normal	0.5
	🔂 15Case 5 Freq 10.63175 🛛 🗹 👧 34Case	2 56R1 Constraint Moment	3453Bar EndA Pt2 Bend Stra 3454Bar EndA Pt3 Bend Stra		Solid Element Out	tput	
1	🙀 16Case 6 Freq 10.6899 🛛 🏹 🏭 35Case	2 58R3 Constraint Moment	🗹 3455Bar EndA Pt4 Bend Stra	ain	None		
Ľ	🙀 17Case 7 Freq 10.8234 🛛 🗹 👰 36Case		3456Bar EndB Pt1 Bend Stra 3457Bar EndB Pt2 Bend Stra		O Into CSys	0Global Rectangula	r v
	18 Care 8 Fred 11 10132		3458Bar EndB Pt3 Bend Stra		O Into Matl Dire	ction	
	18Case 8 Freq 11.19132 🗹 🍓 37Case						
		1	3459Bar EndB Pt4 Bend Stra	ain >	Output Orient	ation <u>O</u> K	Cancel

Siemens Digital Industries Software

Simcenter Femap v2020.2 – What's New Postprocessing – Send to Excel

File

R9

1 2 3

4

5 6 7



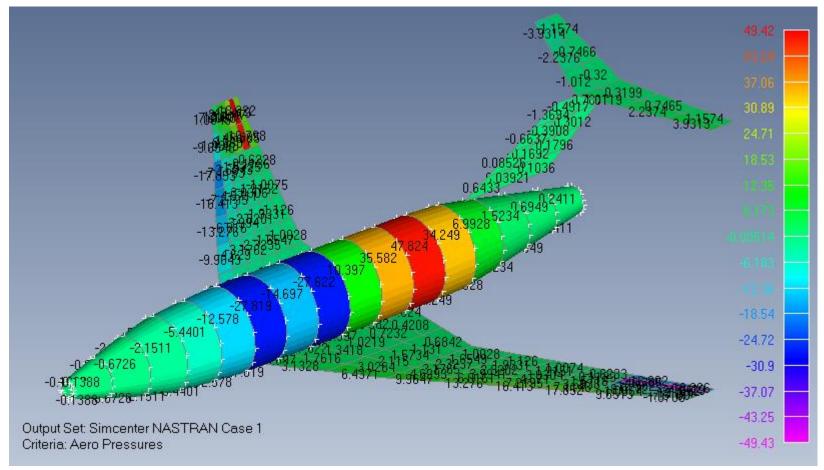
When *Formatting* option is enabled, each worksheet in Excel will include:

- Column Filters
- Frozen Row Heading
- Frozen ID Column
- Nodal and Elemental results organized by different sheets

			9- C-									T] —		×
ile	Home	e Insert P	age Layout	Formulas	Data	Review Vie	w Develope	r Add-ins	Help	ACROBAT	PDF-XChang Tear	m Design 🖓 🔎	Tell me	ß	P
)		<u>-</u> = L×	· · J	fsc											~
	A	В	С	D	E	F	G	н		1	J	к		L	
ID	-	CSys ID 💌 🕽	K 🔽 Y	ا 🔽 ۲	z	💌 Set ID 💌	Set Value	Set Title	 Stud 	ly ID 💌 2T	1 Translation 💌 3	T2 Translation	▼ 4T3	Translation	ا ب ا
	1	0	60	0	₽↓	<u>S</u> ort A to Z				0	0.02625739	0.0169593	51	-3.041476	-05
	2	0	60	0	Z↓	S <u>o</u> rt Z to A				0	0.02586838	0.0170971	51	-0.000918	686
	3	0	30	0		Sor <u>t</u> by Color		,		0	0.004329831	0.0055207	52	0.002758	089
	4	0	-30	0		-	NO A THE R			0	0.004329831	-0.0055207	53	-0.002758	089
	5	0	-60	0	- 10	<u>C</u> lear Filter Fro	m "Set Litle"			0	0.004806403	-0.011814	43	0.001381	083
	6	0	-60	0		Filter by Color		•	·	0	0.02586838	-0.0170971	51	0.000918	686
	7	0	-60	0		Text <u>F</u> ilters		•	·	0	0.02625739	-0.0169593	51	3.041478	-05
	8	0	-30	0		Search		2		0	0.005149405	-0.0054488		0.003258	
	9	0	30	0		Select A	VID			0	0.005149405	0.0054488		-0.003258	
	10	0		240.4366			req 30.83381	^		0	-0.08096946	-0.033967	99	0.004498	
4	•	Nodes	Elements	5 (+)		10.8234 Hz								Þ
							0, 33.84494 Hz					▣ □ -		+	100%
_						🗹 Mode 2,	11.19132 Hz								
						 Mode 3,	11.25252 Hz								
							19.72662 Hz								
							28.03074 Hz								
						✓ Mode 6,	29.21541 Hz	~							
						Г	OK	Cancel							
							UK	curreer							

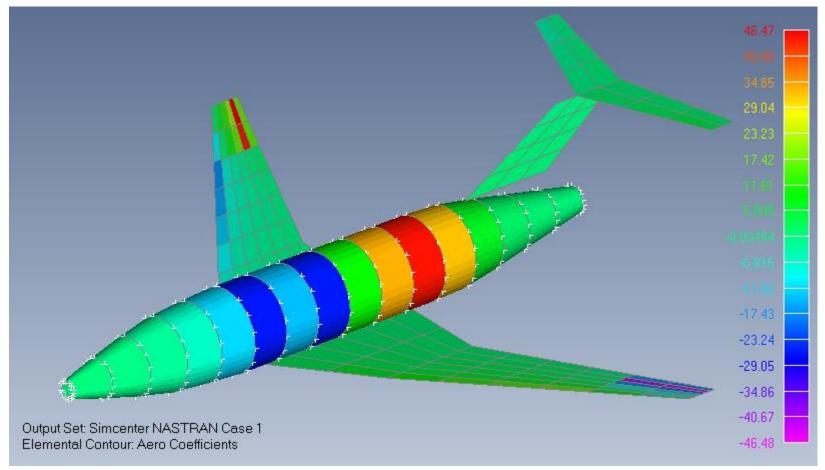


Added support to display Aero Pressures from Static Aeroelasity (SOL 144) analysis (*.f06 file only)



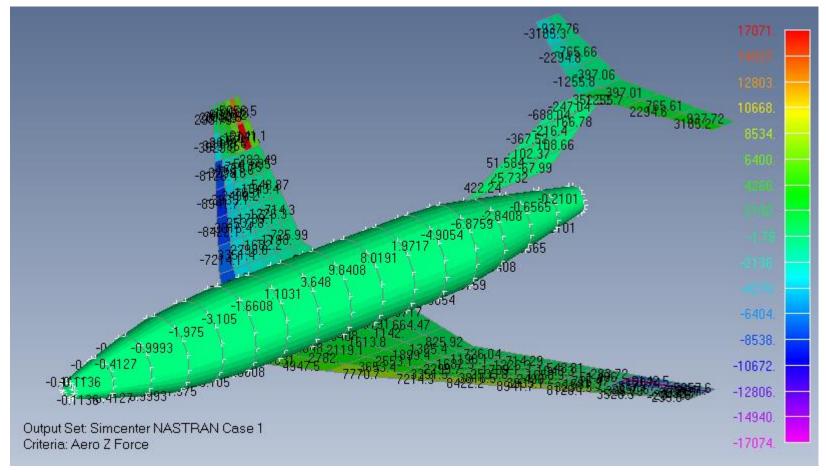


Added support to display Aero Coefficients from Static Aeroelasity (SOL 144) analysis (*.f06 file only)





Added support to display Aero Forces generated by Static Aeroelasity (SOL 144) analysis (*.f06 file only)



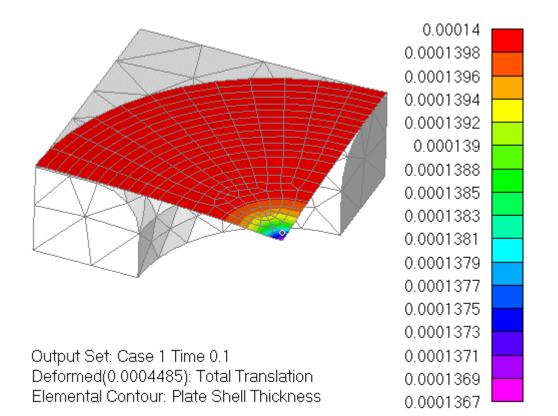
Siemens Digital Industries Software

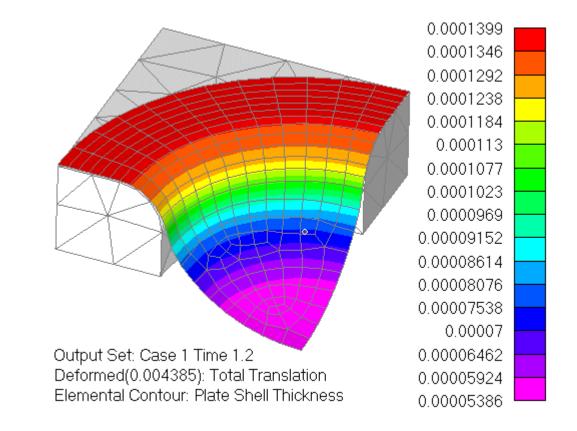


Added support for Shell Thickness Output (requested by SHELLTHK) from Simcenter Nastran SOL 402

Step 1 Thickness: Max = 1.4E-04; Min 1.367E-04

Step 12 Thickness: Max = 1.399E-04; Min 5.386E-05



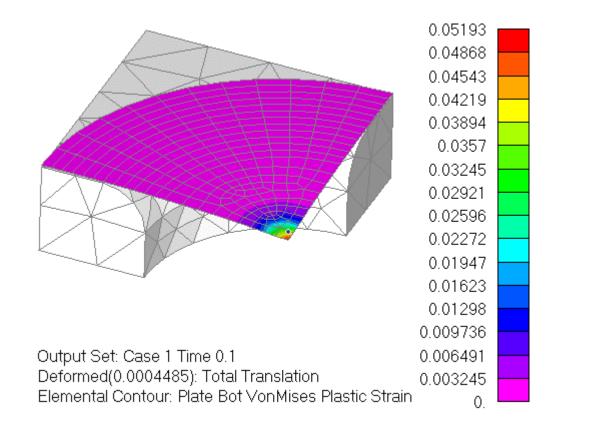


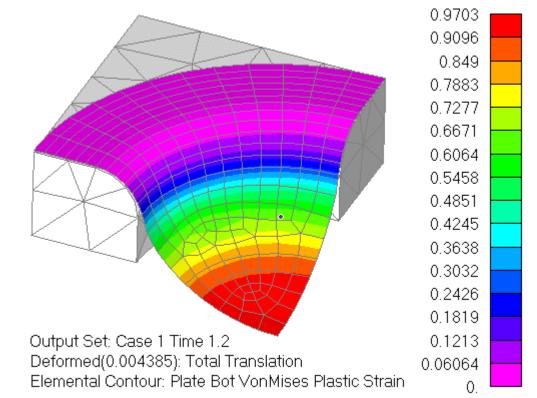


Added support for Plastic Strain Output (requested by PLSTRN) from Simcenter Nastran SOL 401/402

Step 1 Plastic Strain: Max = 0.05193; Min 0.0

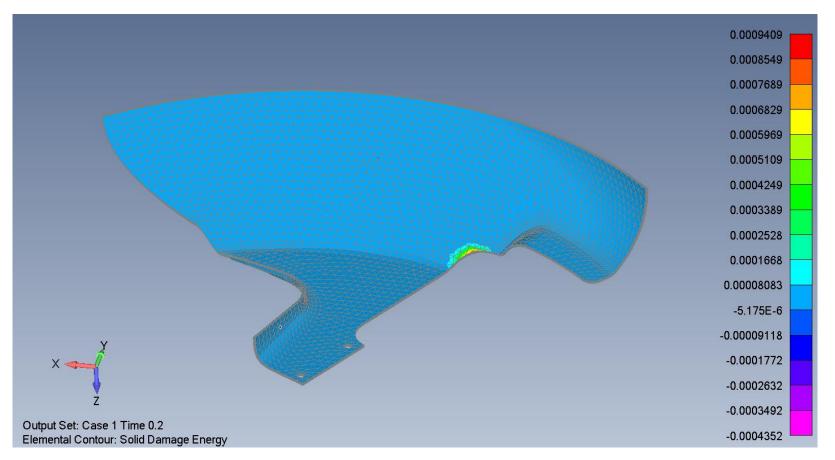
Step 12 Plastic Strain: Max = 0.9703; Min 0.0







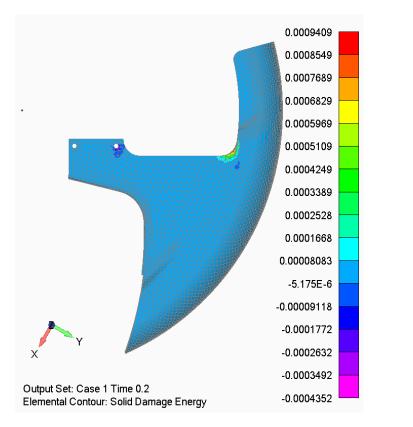
Added support for progressive failure results output (requested by PFRESULTS) from SOL401/SOL402

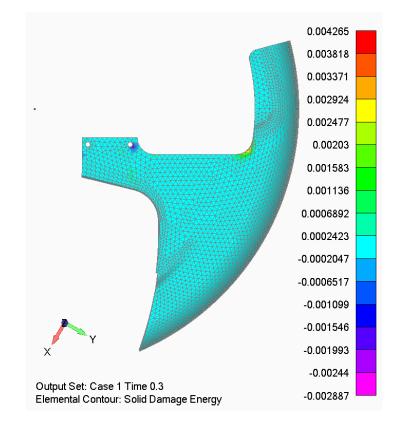




Failure results include Damage Status, Damage Values, Damage Energy, and Crack Density

Time 0.2 Damage Energy: Max = 9.41E-4; Min -0.435E-4 Time 0.3 Damage Energy: Max = 4.27E-3; Min -2.89E-3

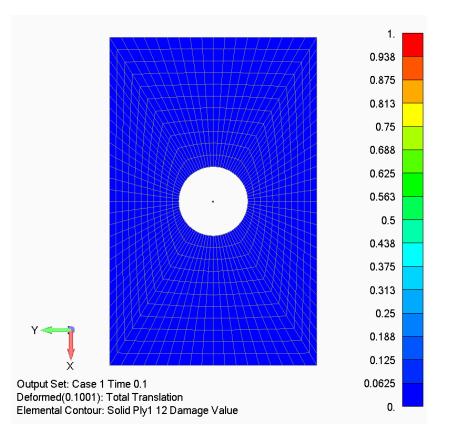




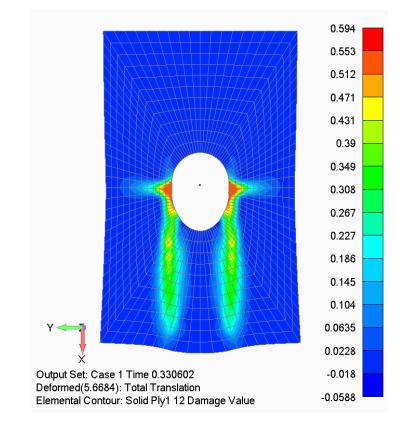


Failure results include Damage Status, Damage Values, Damage Energy, and Crack Density

Step 1 Ply D12: Max = 0.0; Min 0.0



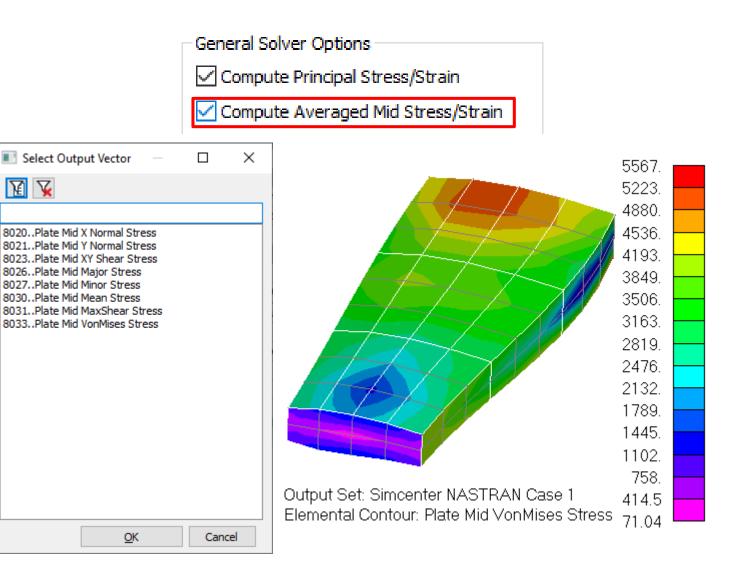
Time 0.33 Ply D12: Max = 5.94E-1; Min -5.88E-2





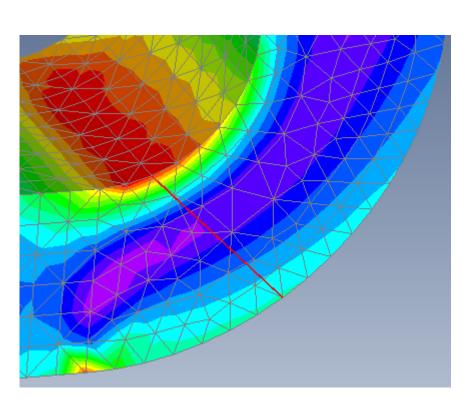
Added an option to *Results* tab of File-> Preferences to allow computing Averaged Midplane Stress and Strain.

- Simple average of Top and Bottom Results.
- Includes computation of Invariants from averaged mid values.
- Supported for Results either Read into the Database or Attached.
- Not supported for Complex Results.



Stress Linearization

- ASME Boiler and Pressure Vessel Code, 2007 SECTION VIII, DIVISION 2, ANNEX 5.1 LINEARIZATION OF STRESS RESULTS FOR STRESS CLASSIFICATION
- Calculates a Linearized Membrane, Linearized Bending, and Peak Stresses along a Stress Classification (SCL)



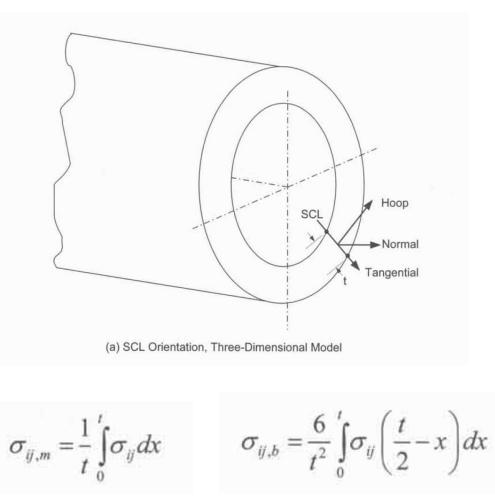


Control Options	
Output Set	1Simcenter NASTRAN Case 1 🗸 👔
	Use nonlinear results
Element Type(s)	O Shell Solid Axisymmetric
Stresses	● von Mises Stress ○ Tresca Stress
	Full Component Bending
Nodes	2789 2898
Output	Calculate
M] - 💽 - 🚏	- 💼 -
10000	
10000	
10000	
	0.2 0.4 0.6
0	
0	0.2 0.4 0.6 es Stress, Max Value = 17800.64 me: 1896.69
0 - 0 0 - 0	es Stress, Max Value = 17800.64
0 - 0 von Mise Membra Maximu	es Stress, Max Value = 17800.64 ine: 1896.69
0 - 0 0 Von Mise Membra Maximu	es Stress, Max Value = 17800.64 ine: 1896.69 m Membrane and Bending: 11736.13 ine and Bending
0 - 0 von Mise Membra Maximu Membra	es Stress, Max Value = 17800.64 ine: 1896.69 m Membrane and Bending: 11736.13 ine and Bending
0	es Stress, Max Value = 17800.64 ine: 1896.69 m Membrane and Bending: 11736.13 ine and Bending



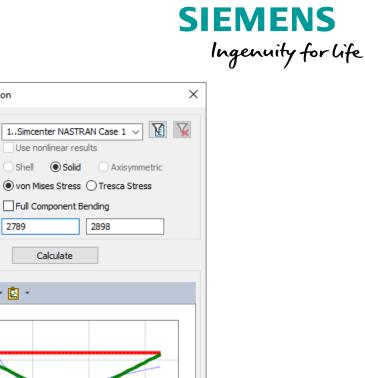
Stress Linearization Process per the ASME Code

- Choice of the SCL is critical, user should be familiar with criteria for a proper SCL as defined in the ASME standard
- Stress tensor extracted from the FEA results is transformed to be aligned with the SCL
- Linearized Membrane Stress tensor components are the average of each along the SCL
- Linearized Bending Stress tensor only includes Hoop and Normal axial stress, and Hoop/Normal shear stress



Stress Linearization Process in Femap

- Choose Output Set
- Choose Linear or Nonlinear Results for the Stress Tensor
- Element Type for Stress Data
- Von Mises or Tresca Stress
- Pick Nodes for the Stress Classification Line (SCL)
- Calculate



Stress Linearization Control Options

Output Set

Stresses

Nodes

Output

10000

0

Copy to Clipboard

🔟 - 💽 - 🔀 - 🔯 -

Element Type(s)

2789

0.2

Membrane: 1896.69

Membrane and Bending Bending: 10530.05

SCL from node 2789 to 2898 von Mises Stress

Membrane Stress: 1896.69 | Bending Stress: 10530.05 Maximum Membrane and Bending: 11736.13

von Mises Stress, Max Value = 17800.64

Maximum Membrane and Bending: 11736.13

0.4

0.6

Close

~

Stress Linearization Results

- Chart showing the actual stress along the SCL
- Calculated Membrane, Bending, Membrane + Bending, Maximum Membrane + Bending
- Copy to Clipboard Option
- Exact Data values can also be copied from the Femap Chart

Stress Linearization

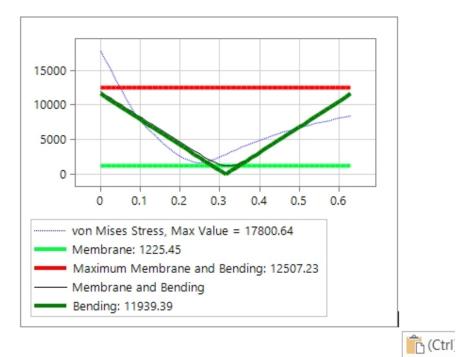
SCL from node 2789 to 2900 von Mises Stress

Membrane Stress: 1225.45 | Bending Stress: 11939.39

Maximum Membrane and Bending: 12507.23

Using Solid Linear Stress Data

Stresses have been transformed from the Material Direction to the SCL.



SIEMENS

Ingenuity for life

Stress Linearization Notes

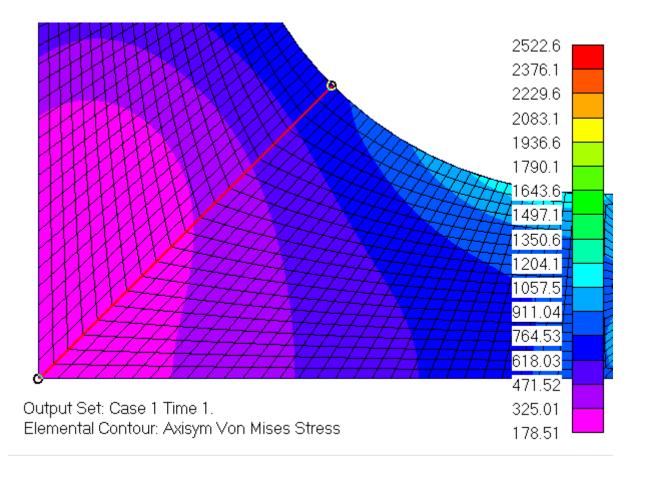
- Full Component Bending option allows you to process the linearization with all components of the stress tensor considered for the bending stress calculations
- Tresca Stress calculates a "Tresca" failure criterion stress value, which is based on the maximum shear stress, the Tresca Stress calculated by the tool is 2 times maximum shear

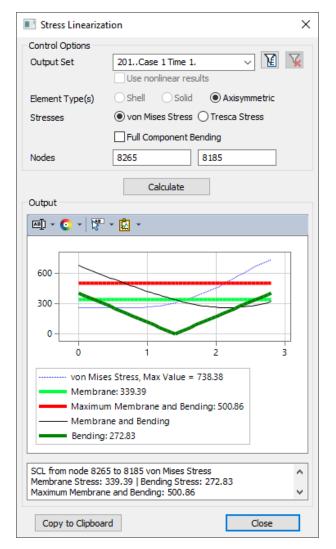
\times Stress Linearization Control Options ¥ ¥ Output Set 1...Simcenter NASTRAN Case 1 Use nonlinear results Shell Solid Axisymmetric Element Type(s) von Mises Stress O Tresca Stress Stresses Full Component Bending 2789 2900 Nodes Calculate Output 📼 - 💽 - 🞇 - 🔯 -15000 10000 5000 0 0.2 0.3 0.4 0.5 0 0.1 0.6 von Mises Stress, Max Value = 17800.64 Membrane: 1225.45 Maximum Membrane and Bending: 12507.23 Membrane and Bending Bendina: 11939.39 SCL from node 2789 to 2900 von Mises Stress \wedge Membrane Stress: 1225.45 | Bending Stress: 11939.39 Maximum Membrane and Bending: 12507.23 Copy to Clipboard Close





Stress Classification Line (SCL) from Node 8185 to Node 8265





Agenda:

Performance Improvements UI and Visualization Preprocessing Meshing Post Processing **Solver Support** Miscellaneous

Simcenter Femap v2020.2 – What's New **Solver Support – Nastran Solvers**



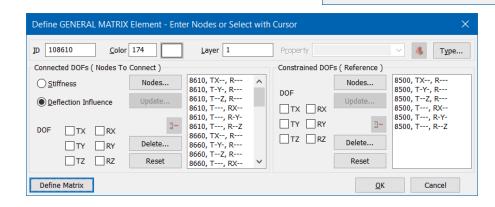
Canc<u>e</u>l

<u>0</u>K

Added support for the General Element capability (GENEL)

- Define a substructure in terms of empirically-determined structural data
- Table interface for defining deflection influence or stiffness matrixes
- Graphical representation drawn that represents the connected DOFs connected to the constrained DOFs

8610, 1	8610, 2	8610, 3	8610, 4	8610, 5	8610, 6
1.8496098E-7					
4.3683599E-9	1.9827996E-7				
8.1577909E-9	1.35986E-7	1.1678696E-7			
-9.122513E-9	-5.994352E-7	-4.578160E-7	3.4509594E-6		
1.6985996E-7	5.1056084E-9	1.0292702E-8	-9.295370E-9	2.0552197E-6	
3.3427403E-7	1.7277304E-9	3.6303409E-9	-5.179189E-9	-1.138419E-6	2.890009
-2.691350E-8	2.093800E-10	-1.052230E-9	-2.457711E-9	-1.734820E-7	1.191730
-5.184617E-9	-5.776719E-8	-4.858070E-8	1.3634997E-7	-2.668321E-9	-7.11560
8.6998675E-9	5.7968101E-8	6.1694948E-8	-1.25253E-7	9.8123394E-9	5.754991
-1.108710E-8	-1.505300E-7	-1.227620E-7	3.4884397E-7	-7.602768E-9	-1.36325
1.7726296E-7	6.2000609E-9	1.0802104E-8	-1.18226E-8	1.0989999E-7	2.824899
1.01767E-7	1.0023598E-8	8.3744673E-9	-2.611611E-8	-2.96721E-7	5.733358
-5.910898E-9	2.9146394E-9	5.2385793E-9	-6.111996E-9	-2.143601E-8	3.599730
-2.382830E-9	1.8070596E-9	2.5149203E-9	-3.389921E-9	-7.099821E-9	1.750280



Simcenter Femap v2020.2 – What's New Solver Support – Nastran Solvers



Added support for Follower Forces and Follower Moments

- When 2 nodes/points are specified to define the direction of the the load in terms of a vector, writes FORCE1 or MOMENT1
- When 4 nodes/points are specified to define the direction as normal to a plane, writes FORCE2 or MOMENT2
- Typically used by SOL 106, SOL 401, and SOL 402 for analysis which includes large deformation

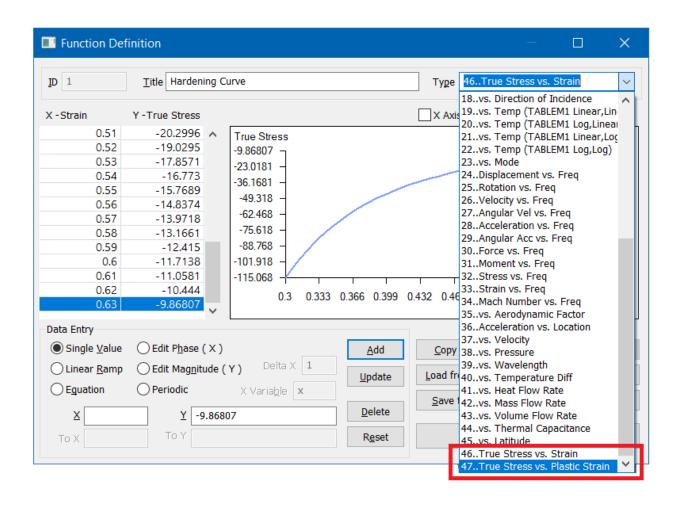
	<u>V</u> alue	Time/Freq Dependence	Data Surface
Magnitude	10.	0None ~ 1	xy
Node 1 (G1)	24]	
Node 2 (G2)	57]	
Node 3 (G3)	27]	
Node 4 (G4)	54]	
∕ Normal To	Plane	[<u>O</u> K Cancel
23456			

Simcenter Femap v2020.2 – What's New Solver Support – Simcenter Nastran



Added new nonlinear contact and control parameters for SOL402

- New function types to specify type of exported stress-strain measures for SOL 402 (engineering vs. true)
- New option to consider thickness and Z-offset for shell elements involved in contact



Simcenter Femap v2020.2 – What's New Solver Support – Simcenter Nastran



Added support for modeling progressive ply failure for solid laminates (MATDMG)

- New material types to specify unidirectional fiber reinforced ply damage model and enhanced unidirectional ply damage model
- New function type to specify nonlinear damage evolution law for in-plane shear (shear damage vs thermodynamic force)
- Compute damage values, energy, status and crack density to predict failure

Define Material				×
ID 2 Title Sim	. Nastran UD F	Ply Failure (MATDMG) Layer 1		Material Ty <u>p</u> e
Material Type 512Sim	n. Nastran UD I	Ply Failure (MATDMG Sol 401,402)	~	
Material ID - MID	1	Transverse Limit E YS22	0.2	Loa <u>d</u>
Max Damage Value - DMAX	0.999	Coupling Coeff B2	0.6	<u>S</u> ave
3D Effect Parameter - PE	0	Damage Var Coeff B3	0.5	Сор <u>у</u>
E. Threshold (+) - Y11LIMT	25.	Coupling Coeff A	0.6	
E. Threshold (-) - Y11LIMC	2.	Initial Plastic Threshold - LITK	0.1	
Coeff. Parameter - PLYUNI	0	Plastic Law Param BIGK	489.	Next >>
Nonlin. Coefficient (+) - KSIT	3.	Plastic Law Exp EXPN	0.293	<< Prev
Nonlin. Coefficient (-) - KSIC	20.	Transition Thickness - HBAR	0.434333	ZCHEV
Shear E. Limit - YS12	3.	Delay Param ADEL	1.	
Damage Evolution - TID	108NASTR/	Delay Time - TAU	1.E-4	
E.Threshold in Shear - Y012	0.			<u>О</u> К
Shear Critical E YC12	0.			Cancel

Simcenter Femap v2020.2 – What's New Solver Support – ANSYS Updates

Added several options to *Model Control* section of *ANSYS Command and Model Control* dialog box

- Added Skip Csys, Skip Nodes, Skip Materials, Skip Properties, and Skip Elements - allows user to skip any combination of these entity types entirely during export
- Added Skip Groups allows user to skip writing ESEL, NSEL and CM entries during export
- Added Skip Misc allows user to skip view settings (/VIEW, /ANGLE, /ZOOM, etc), preprocessor commands (/NOPR, /FCOMP, /PREP7, /GOPR, etc), solution settings (/SOLU, ANTYPE, EQSLV, etc), SOLVE, FINISH, etc
- Changed Skip Beam/Bar Cross Sections to Skip Shapes



ANSYS Command and Model Control X
ANSYS Version 10 ANSYS Product 0Default Command Line Jobname Output Directory Database Memory (MBytes)
Workspace Memory (MBytes) Other Command Line Inputs List Input in Output (-par1 val1 -par2 val2)
Model Control Portion of Model to Write 0Entire Model
Skip Csys Skip Nodes Skip Materials Skip Properties Skip Elements Skip Groups Skip Shapes Skip Misc
Result Control File Compression Level 1No Compression ✓
Manual Control Skip Standard Start Text (Off) End Text (Off) Prev Next QK Cancel

Simcenter Femap v2020.2 – What's New Solver Support – ANSYS Updates



Added *Result Control* section to select the *File Compression Level* for the ANSYS results file (*.RST File)

Added ability to read output data contained in ANSYS results files which use "Version 1" compression (ANSYS default)

Added Manual Control section to:

- ANSYS Command and Model Control
- ANSYS Load Step
- ANSYS Time Step Options
- ANSYS Modal Analysis Options
- ANSYS Harmonic Analysis Options
- ANSYS Random Vibration Analysis Options
- ANSYS Buckling Options
- ANSYS Nonlinear Options
- ANSYS Output Requests

ANSYS Command and Model Com	ntrol X
ANSYS Version 10 ANSY Command Line Jobname	S Product 0Default ~
Output Directory	
Database Memory (MBytes) Workspace Memory (MBytes)	
Other Command Line Inputs	(-par1 val1 -par2 val2)
Model Control Portion of Model to Write	0Entire Model V
Skip Csys Skip Nodes	Skip Materials Skip Properties
Skip Elements Skip Groups	Skip Shapes Skip Misc
Result Control File Compression <u>L</u> evel	1No Compression
Manual Control	Start Text (Off) End Text (Off)
Pre <u>v</u> Ne <u>x</u> t	<u>O</u> K Cance <u>l</u>

Simcenter Femap v2020.2 – What's New Solver Support – LS-Dyna Updates



Added several options to *Model Control* section of *LS-DYNA Model Option* dialog box

- Added Skip Csys, Skip Nodes, Skip Materials, Skip Properties, and Skip Elements - allows user to skip any combination of these entity types entirely during export
- Added Skip Groups allows user to skip writing *SET_ entries during export
- Added Skip Functions allows user to skip writing functions (*DEFINE_CURVE entries) during export

LS-DYNA Model Options	×
Export Options	Material Based Plate Parts
Model Control Portion of Model to Write	0Entire Model ~
Skip Csys Skip Nodes Skip Elements Skip Groups	Skip <u>M</u> aterials Skip <u>P</u> roperties
Manual Control	Start Text (Off) End Text (Off)
Prev Ne <u>x</u> t	<u>O</u> K Cancel

 Added Skip Misc – allows user to skip comments starting with "\$", *KEYWORD, *TITLE with corresponding "title", *CONTROL TERMINATION with corresponding value, *DATABASE_BINARY_D3PLOT with corresponding value, *END, and other entries during export

Simcenter Femap v2020.2 – What's New Solver Support – LS-Dyna and ABAQUS Updates



Added *LS-Dyna Analysis Monitor* to monitor solver progress, review solver files, and optionally import results

Added support Preserve Load/BC Sets option for ABAQUS

 When enabled, causes imported loads and boundary conditions which reference a SET to create a Load Definition or Constraint Definition in Femap

Added support to write all supported load/constraint types that are contained in a Load/Constraint Definition as a SET entry in the ABAQUS input file, if requested by enabling the *Write All Groups as Sets* option in the *ABAQUS Model Options* dialog box

Messages	Views Gra	phics	User Interface	Database	Solvers
Geometry/Model	Interfaces	Results	Library/Startup	Color	Spaceball
nterface Analysis Type Ion-Femap Neutra nterface Style Enable Old Ar Analysis Monitor (45Simcenter N 1Static al Version 0Structur nalysis Interfaces Options	lastran 2020.2 al	File Reference File Reference Create Geo Create Ana Create Ana General Solver Run Analys	Options erences on Oper ometry Referen alysis Model Ref alysis Results Re	n ces erences eferences
Automatically Max Lines to Mor		5000		acy Ansys Inter	-
Improve Real Write Alterna Solver Memory (Scratch Directory	- [ion 0 efault v	Read Comr	atic Load/BC Se nents as Titles d Synthetic Loa ups from INCLU NCLUDE Statem Bulk Data Entrie	d Sets IDE files ents
Direct Output To			~		
Output Directory					
ABAQUS Options					

Agenda:

Performance Improvements UI and Visualization Preprocessing Meshing Post Processing Solver Support **Miscellaneous**

Simcenter Femap v2020.2 – What's New **API – Control Multiple Femap Sessions**



- API Developers can now control multiple Femap sessions simultaneously
- Including Excel VBA Developers
- Forward-compatible

_

ProcessID

19336

22476

13704

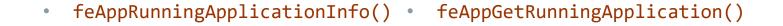
🖳 Choose Running Femap...

Model Name

Bender Model

Pressure Vessel

Wing Test





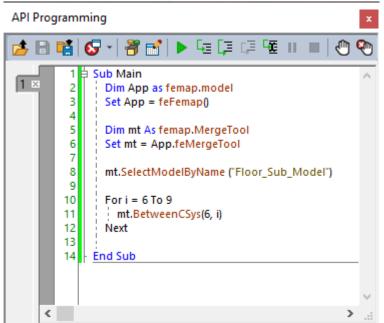
Unrestricted © Siemens 2020

Siemens Digital Industries Software

Simcenter Femap v2020.2 – What's New API

New Tools

- Added a MergeTool Object (feMergeTool) that is similar to the existing CopyTool and MoveTool, but exposes the capabilities of the File->Merge Command
 - Ability to select From Model by name
 - Methods to toggle individual/groups of entity types to include
 - Method to "Add Related" entities with "Add Associated" option
 - Unique methods for each Orientation/Transform option
 - Properties to specify Options
 - Methods to create "Patterns" of merged entities (not in UI)





Simcenter Femap v2020.2 – What's New API



Stress Linearization API Access

- Automate processing of different Stress Linearization scenarios
- Same options as GUI version

Output from API Script:

Maximum Stress = 17800.6378136816 Membrane Stress = 1225.44895224937 Bending Stress = 11939.3928698333

```
1 🗄 Sub Main
       Dim App as femap.model
       Set App = feFemap()
       Dim fSL As femap.StressLinear
       Set fSL = App.feStressLinear
 8
       fSL.ouSetID = 1
 9
       fSL.nElemType = 0
10
       fSL.ndID1 = 2789
11
       fSL.ndID2 = 2900
12
       fSL.bNonlinearMode = False
       fSL.bFullStressTensor = False
13
14
15
       If fSL.CalcStressLinearization() = FE OK Then
         Msg = "Maximum Stress = " +Str$(fSL.dMaxStress)
16
17
         App.feAppMessage( FCM_NORMAL, Msg )
          Msg = "Membrane Stress = " + Str$(fSL.dMembraneStress)
18
19
         App.feAppMessage( FCM_NORMAL, Msg )
20
         Msg = "Bending Stress = " + Str$(fSL.dBendingStress )
21
         App.feAppMessage( FCM_NORMAL, Msg )
22
       End If
23
24
     End Sub
```

Simcenter Femap v2020.2 – What's New API



New Application Methods

- Added feFileGetByName() that returns Model ID of an open model based on its filename.
- Added feEdgesOfFreeFaces() that returns pairs of node IDs along the edges of Free Faces of selected elements.

New Object Methods

- Added NextOnEntity() method to the LoadGeom, LoadMesh and BCGeom objects and NextOnNode() to the BCEqn object. These new methods allow you to retrieve multiple loads/constraints of different types on multiple entities.
- Added AddNodesOnFreeEdges() and AddNodesOnFreeFaces() to the Set object that add nodes on free edges or faces of selected elements to the Set.
- Added ComputeStdShape2() and ComputeGeneralShape2() to the Property object that allow you to specify whether to compute properties at End A or End B of a tapered beam property.

Contact Us



13290 Evening Creek Drive S San Diego, CA 92128

(858) 480-2000

plm_sales@ata-e.com

www.ata-e.com www.ata-plmsoftware.com

@ATAEngineering

ata-engineering

