

FEMAP Version 12 What's New

FEMAP Release Schedule



Regular release schedule

- v12.0 August 2018
- v11.4.2 December 2017
- v11.4.1 September 2017
- v11.4 May 2017
- v11.3 May 2016
- v11.2 March 2015
- v11.1 November 2013
- v11.0: January 2013











FEMAP Direction



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 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

FEMAP 12.0 Overview



- UI and visualization
- Geometry enhancements
- Preprocessing
- Postprocessing
- Solver support





UI and Visualization New Functionality and Updates



New functionality and updates

- Live Screen Entities
 - View Axes
 - Contour Legend
 - Post Titles
 - View Titles
- Silhouette and Feature Lines
- Postprocessing
- Performance



UI and Visualization Interactive Screen Entities



Interactive screen entities

- View Legend
- View Axes
- Post Titles
- Contour Legend

Mouse over the areas indicated and each item becomes "live"

- Drag to reposition
- Other specific functionality on each item



UI and Visualization Interactive Screen Entities

View Legend and Post Titles

- Pick to select
- Move
- Text auto justification
 - Left
 - Center
 - Right
- Quick access to applicable
 View Option for further control





Unrestricted © Siemens AG 2018 Page 8 2018-07-23

Ul and Visualization Interactive Screen Entities

View Axes

- Pick to activate
- Cube for accessing standard orientations
- Face Pick for Top, Bottom, Left...
- Edge pick for 45 Degree Left/Top Top/Back…
- Corner pick for isometric views
- Arrows rotate about Screen Z
- Home to autoscale
- Quick access to applicable View Option for further control



SIEMENS



UI and Visualization Interactive Screen Entities

Contour Legend

- Pick to activate
- Drag to move
- Pick the down arrow to bring up the standard View Option dialog
- Control options
 - Max/Min
 - Smooth/Level
 - Number of Levels





Best Possible Graphics

UI and Visualization

- Queries the available graphics hardware to determine best options to get best performance
- AMD and NVIDIA work well but Intel is not fully supported by Performance Graphics option

Preferences	-	~			×
Geometry/Model Interfa	ces Results Graphics	Library/Sta	artup C	Color	Spaceball
Graphics Options	⊂ Include In Dvr	namic Rotation -	. Data	buse	Solvers
Hardware Acceleration	V Point	Coordina	ate Svs	✓ Fill	
Best Possible	Curve	✓ Node		✓ Shading	
Performance Graphics	Surface	Element		Smooth	Lines
0No Vertex Arrays 🔻	Boundary	Constrai	int 🛛	Filled Ed	oes
Max VBO MB 1074	Solid	✓ Load		Mesh Lo	cations
	V Text	Connect	tions	√ Undefor	med
Min VBO B 1024 -	I abels	Elemente	s as Free Ed	ne	
Memory Optimization	Workplane	Element	Symbole	gc.	
Use Midside Nodes	w workplane	Lienient	Symbols		
Multi-Model Memory	Textures		Advanced /	Debug Opt	ions
Beam Facet Edges	🔽 2D Mapping	,	0No Debu	ug Message	es 🔻
Smooth Lines	Smooth Tex	xtures	📃 Elap	osed Time	
Auto Regenerate	Force All Tr	iangles	Frame R	ate	
Fast Picking	Max Size 4	096 🔻	0No Oper	nGL Errors	-
Fast Pick Visible			0No VBO	messages	•
XOR Picking Graphics	Block Control		Bitmap Align	ment	4
Edges Using Lines	Block Size	32 🔻	Divel Forma	t (0-Auto)	
Dialog Refresh	Search Depth	10	Pixel office	(her:)	
Trailing Zeros			BitBlt Delay	(ms/MPix)	0
Max Mag 10000.			TDR protect	tion 0	
Reset All		0	к	Cano	el



UI and Visualization Feature and Silhouette Lines



Feature Lines and Silhouette Lines

- Mesh based displays hard edges
- Controlled with break angle

View Options	— 🗆 X				
View 1 Untitled					
Category O Labels, Entities and Color Tools and View Style O PostProcessing Options	Draw Entity Element Control and Silhouette 0All Elements 1All Elements, Silhouette 2Orawn Elements				
Free Edge and Face Shrink Elements Fill, Backfaces and Hidden Filled Edges					
Graphics Options	Surface Division				
Performance Graphics Transparency Shading Perspective Axisymmetric Axes View Legend View Axes	0None 1Property 2Material				
Workplane and Rulers	Color				
Workplane Grid Group Clipping Planes	28676 Palette				
Symbols Symbols Preview View Aspect Ratio Model Clipping Plane	Angle 50.				
	Apply OK Cancel				



UI and Visualization Feature and Silhouette Lines

Feature Lines and Silhouette Lines

Control options

Silhouette Lines On/Off

Feature Lines for All Elements, or the current Drawn Elements







All Elements, Silhouette

Drawn Elements, Silhouette

Unrestricted © Siemens AG 2018 Page 12 2018-07-23

Siemens PLM Software

UI and Visualization UI Themes

Updated UI theme

- Fresh and modern look and feel
- Clean UI appearance
- Clear model viewing

Also available in Custom Tools:

- Selection of predefined themes
- Create your own theme and save it





Load View Theme

Exit

Geometry Enhancements Geometry Processing

Sewing uses new Multi-Body processing

• Old algorithm made copies of all sheet bodies, and sewed them together

New algorithm keeps original surfaces, including

- Surface based loads and BCs
- Surface based contact
- Composite curves
- Composite surfaces

Also uses tolerance algorithm, specify a larger tolerance, and surfaces will be sewn at full tolerance first, working its way up to the larger tolerance



Geometry Enhancements Geometry Processing

Single call to stitch -0.1 tolerance - fully connected and ready to mesh







Some surfaces are composite overlapping edges with composite curves



Various gap distances – some quite large

Unrestricted © Siemens AG 2018 Page 15 2018-07-23

Geometry Enhancements Boolean Operations





Two overlapping solids

Cylinder is split at 0, 90, 180, and 270. Two combined surfaces and four combined curves Face of block is split into 8 faces, recombined with a single composite surface and four composite curves – internal curves highlighted

Geometry Enhancements Boolean Operations





Boolean Add, block composite face split into four, cylinder composite faces updated Boolean Subtract block from cylinder, split faces of block are recombined on resulting cylinder **Boolean Common**

Geometry Enhancements Boolean Operations







Solid – Embed cylinder embedded Into block Solid – Embed cylinder embedded Into block exploded view

Geometry Enhancements Combined Curve and Combined Surface Preservation



Preservation of combined/boundary curves and surfaces in solid operations



Geometry Enhancements Combined Curve and Combined Surface Preservation

Preservation of combined/boundary curves and surfaces in solid operations





Geometry Enhancements Combined Curve and Combined Surface Preservation



Preservation of combined/boundary curves and surfaces in solid operations





Starting with this divided surface, the Combined/Composite Curve Tool and Combined/Boundary Surfaces Tool in the Meshing Toolbox is used to create a single surface, with 4 edges and four corners















Menu based "Curves from Surfaces" and Meshing Toolbox splitters now work with Combined Curves and Combined Surfaces

Project Along Vector







Edges of the underlying surfaces are being displayed – note that where required by the underlying splits, a Combined Curve has been created





Surface and solids model with Combined Curve and Combined Surface now support all slicing operations – consider this single solid, that was created by a Boolean Add of two solids both with Combined Geometry





Planar Slice



Unrestricted © Siemens AG 2018 Page 29 2018-07-23



Spline Slice



Unrestricted © Siemens AG 2018 Page 30 2018-07-23



Parallel Planar Slices



Unrestricted © Siemens AG 2018 Page 31 2018-07-23

Geometry Enhancements Surface Ruled/Between Curves



Geometry – Surface – Ruled/Between Curves replaces Geometry – Surface – Ruled command

Curve to Curve Surface Surface Between Curves Х Option has been From То Parameters... Shape 2.. Along Vector Shape 1...Tangent to Surfac V enhanced Curve Curve <u>O</u>K Surface Factor Cancel At Each curve • Ruled (as before) • Tangent To Surface Along Vector Starting geometry as shown

Geometry Enhancements Surface Ruled/Between Curves





Geometry Enhancements Surface Alignment for Meshing



Modify – Update Other – Align Surface Parameters implemented as a GUI command (available via API in v11.4.x)

FEMAP splits all periodic surfaces at 0° and 180°

In some cases, with data coming in from various CAD systems, the surface representation may not be exactly aligned in connected faces, this results in short edges in the geometry near adjacent splits

Meshing tools in FEMAP can overcome this, but this command provides a more exact solution



Geometry Enhancements Surface Alignment for Meshing



Entity Selection:

 Auto-Mode pick All on Solids, and the Automatic Align(ment) Mode – FEMAP will rotate connected periodic surfaces to match

Align Surface Parameterization		
Entity Selection O Surfaces All on Solids		
Align Mode	ОК	
O Along Vector	Cancel	



Splits aligned, no more short edges – however the small hole is now very close to one of the splits lines, better option, select the Along Vector option for full control

Geometry Enhancements Performance Improvement

Feature Removal – recoded to maximize performance

Removing 50+ small holes from wing spar

30+ seconds in v11.4.x

Less than 1 second in v12!








Washer has been expanded to include non-circular holes

Burns a perfect offset and with pre-established mapped meshing constraints creates a perfect mesh around any cutout





On non-tangent corners performs different splits on inside, outside and sharp corners to create the best possible mesh



SIEMENS Ingenuity for life

Solid Washer

Auto connects and remeshes with tetras or bricks





SIEMENS Ingenuity for life

Solid Pad

Auto connects and remeshes with tetras or bricks





Updated Copy/Rotate/Reflect commands

- Copy/Rotate/Reflect Geometry with associated mesh or vice versa
- Options to also Copy/Rotate/Reflect:
 - Loads
 - Constraints
 - Connections/Regions
- Block and Offset Numbering options
- Copy using a Pattern or in the Same Location, create Multiple Repetitions, and/or AutoRepeat using currently selected entities and specified options

Unrestricted © Siemens AG 2018









Updated Copy/Rotate/Reflect Commands

- Copy Orient/Transform methods:
 - Move Along Vector
 - Move to Location
 - Between Coordinate Systems
 - Between Vectors
 - Between Planes
- Rotate Orient/Transform methods :
 - Rotate Around Vector
 - Rotate to Location
- Reflection Option to specify Trap Width



Copy Geometry and Mesh

Copy Solid geometry with associated mesh using Move to Location method and AutoRepeat Copy



Page 43 2018-07-23



Reflect Geometry and Mesh

• Using Surface geometry with associated mesh, Offset existing Entity IDs by 100



Page 44 2018-07-23

Example:

- geometry associated surface mesh
- complex nodal temperature distribution
- geometry based radial constraint in a non-global cylindrical coordinate system





Reflect – select single solid

- Geometry reflected
- Mesh still associated
- Load reflected
- Dependent coordinate systems reflected
- Geometry based constraint reflected and attached





Rotate – 7 copies

Everything rotated, associated, and consistent





Copy Mesh

Copy Mesh with associated Geometry, Loads, Constraints, and/or Connections/Regions







Move Geometry and Mesh

- Option to include Mesh associated to Geometry and vice versa available for selected:
 - Modify ->Move By commands
 - Modify ->Rotate By commands
 - *Modify ->Reflect* commands (All New for 12.0!)
 - Modify, Align commands
 - Connections/Regions, Loads, and Constraints automatically moved with entities





Preprocessing Updated File – Merge Command

Added Copy in Current Model option to Merge/Extract section

Added options for transformation of merged entities in addition to Between Coordinate Systems in the new Orientation/Transform section:

- Move Along Vector
- Rotate Around Vector
- Reflect Across Plane
- Between Vectors
- Between Planes

Unrestricted © Siemens AG 2018

Page 50

Align between Vectors and Planes

2018-07-23

Model Merge Manag	ger							– 🗆 X	
/lerge/Extract									
O Merge Across Mode	els F	rom Model	Base_Mod	del.modfem				~	
Copy in Current Ma	odel	To Model	Base_Mod	del.modfem					
Entity Type	Current IDs	Merge	e IDs	Renumber To	Ren	umbering	^	All On	
Point	18	1.	.8	916		3 of 8		All Off	
Curve	112	1	12	1324	1	2 of 12			
Surface	16	1.	.6	712		5 of 6		Select All	
Solid	11	1.	.1	22		1 of 1		Colort Name	
Coordinate System								Select None	
✓ Node	1374	13	374	194700	35	350 of 350		Update Selected	
Element	217438	217.	.438	1444	22	2 of 222			
Material	11	1.	.1	22		1 of 1	~	Update All	
enumbering and Duplica	ates Strategy		Entit	y Selection					
○ None	Renumber D	uplicates	C	None 💿 Al	In Model	Select			
Minimal Renumbering	Overwrite D	uplicates	0	From Group					
O Block Renumbering	0								
			C) ID Range From	1		To	99999999	
	Desumber To	•	1						
Compress	Renumber to	1		Add Related and Asso	ociated Entities	A	dd R	elated Entities	
rientation / Transform			Optio	ons					
None				Create Group for Me	erged Model				
O Move Along Vector	O Between Vectors			Create Parent CSvs for Merged Model					
ORotate Around Vector	or OBetween Planes			Condense Transferred Groups					
O Reflect Across Plane	Between Coordina	te Systems		Limit Loads, Constraints and Contact to Merged Entities					
From 0. Global B	ectangular			Keep Loads and Con	straints in Orig	inal Sets			
onabbane					_				
To 0Global R	ectangular	\sim	Du	inlicates to Data Table	a	OK		Cancel	



Unrestricted © Siemens AG 2018

Preprocessing Mesh Point Editor

Accessed via the Tools – Mesh Point Editor command (and for version 12, also the Mesh – Mesh Control – Mesh Points on Surface command), the Mesh Point Editor dockable pane is an enhanced tool for creating Mesh Hard Points

Mesh Point Editor

×v ~ 0.	.Global Rect	tangula 👻 📔	જ 🐮 😚	킠 • [뜯 앮] ~ `	Vg 🐨 😽 🇭	<u></u> .							
ID	On/Off	Locked	Node O	Definition	On Geometry	On ID	Distance To	Geometry X	Geometry Y	Geometry Z	Point X	Point Y	Point Z
1				0None	3Surface	1	0.	3.879518	2.143998	0.	3.879518	2.143998	0.
2				0None	3Surface	1	0.	7.92898	3.483364	0.	7.92898	3.483364	0.
3				0None	3Surface	1	0.	2.228281	6.004525	0.	2.228281	6.004525	0.
4				0None	3Surface	1	0.	8.046925	7.146926	0.	8.046925	7.146926	0.
5				0None	3Surface	1	0.	3.958148	8.643865	0.	3.958148	8.643865	0.
6				0None	3Surface	1	0.	5.530755	4.743945	0.	5.530755	4.743945	0.

- Used to create mesh hard points at a specified coordinate, using an existing point or node, loaded (imported) via a file, or copied from the clipboard
- Mesh Points can now be o" a surface, curve, or point
- Mesh Points now also have their own Symbol (shown on the right)
 - View Options can be used to control display of mesh point symbols





Preprocessing Cohesive Elements and Properties



New Property and Element Type

- Used in NX Nastran SOL 401 and SOL 402
- Property creates PSOLCZ entry
- Thickness on the Property is always used, which allows user to create a Solid Cohesive Elements with Zero Physical Thickness
- Similar to Solid Laminate elements, Brick (CHEXCZ) and Wedge (CPENTCZ) only

Define Property - COHESIVE SOLID Element Type	>	×
ID 1 <u>T</u> itle Color 110 Palette Layer 1	<u>M</u> aterial ✓ ^{GE} _ν Elem/Property Type	
Material Axes	Cohesive Values	
● Align to CSys 0Global Rectangular ∨	Thickness 0.	
◯ Align to <u>E</u> lement		
Loa <u>d</u> <u>S</u> ave Cop <u>y</u>	OK Cancel	



Preprocessing Cohesive Material

Cohesive Properties and may reference an *Isotropic* (MAT1), *Orthotropic (3D)* (MAT11), or "Cohesive" (MATCZ) material

Material

0

0

Support for Cohesive material available via Other Types

	Define Material		×
	ID 1 Ittle Cohesive Material	Color 55 Palette Layer 1	Ту <u>р</u> е
Type X	Material Type 511NX Nastran Cohesi	ve (MATCZ Sol 401,402) ~	
sotropic Orthotropic (2D)	Out of Plane NormStiff - K03S	Critical Energy Release	Loa <u>d</u>
Dr <u>t</u> hotropic (3D)	In-Plane ShearStiff - K02 0. Transverse ShearStiff - K01 0.	Mode 1 - GIC 0. Mode 2 - GIIC 0.	<u>S</u> ave
A <u>n</u> isotropic (2D)	Damage Law	Mode 3 - GIIIC 0.	
Anisotropic (3D)	0=POLY,1=BITR,2=EXPO 0	Coupling Coefficient - DOCU 0.	
<u>nyperelastic</u>	1=TanPos,2=Secant,3=Tan 0	Exponent - EXPN 0.	Next >≥
Oth <u>er Types</u>	Time Delay - TAU 0.]	<u><</u> < Prev
Cancel	Delay Param - ADEL 0.		
			<u>0</u> K
			Cancel



Preprocessing Cohesive Meshing



Mesh Cohesive Elements			×
ID 2521 Color 124 Element Edge to Split	Palette Layer 1 Property	y 2COHESIVE SOLID Property	Resize Original Elements
Erom Node 154 To Node 1057	Top Face Percent Along 50. At Location Location Bottom Face	 Zero Thickness Match Cohesive Property Other Thickness 0. 	 Reduce Size Offset by Thickness Both Sides Top Only
Limit Region <u>M</u> idside	Nodes	<u>О</u> К	Cancel

Works similar to Mesh – Editing – Edge Split command, but inserts a layer of Cohesive elements at the Split Location, using the specified Element Thickness and Resize Original Elements options

Preprocessing Beam Centerline Finder



Geometry – Curve Centerline command implemented for v12 (was available via API in v11.4)

Choose or create material to be connected to new beam properties

Material can optionally override any pre-defined solid attributes

Geometry Selection options:

- Curves interactively pick curves, the mid-point of that curve is calculated, and a cross-section cut of that solid is used to calculate beam properties
- Solids choose solids, two algorithms are run, one that looks for cylindrical and toroidal surfaces, and finds circular cross sections, if no circular cross section is found, the curve method above is used on the longest edge of the solid
- Tubes Only turns off the secondary long edge check

Centerline from Solid Geometry X					
Geometry Attributes Material 1AISI 4340 Steel ✓ ⓐ Apply to Geometry with no Attributes ○ Apply to All Geometry					
Geometry Selection Curves Solids Tubes Only	<u>O</u> K Cancel				

Preprocessing Beam Centerline Finder





Preprocessing Beam Centerline Finder



Circular cross-sections created as FEMAP circular tubes

All others treated as general sections





Preprocessing Data Mapping Enhancements

Model – Load – Map Output From Model implemented customer request for Criteria based mapping onto elements

 All Compatible Elements option available when mapping data from Source model to Target model (i.e., no Group needed in Source model)

Data Surface Editor

- Criteria option available when defining an Output Map Data Surface
- Arbitrary 3D Data Surface utilizes new high speed mapping algorithm



Map from Model Output							
Source							
From Model	Model 1		•				
Results on Group	0All Comp	oatible Elements	•				
Output Set	1pressure	2	•				
Output Vector			•				
Data Conversion							
Contour							
Conversion	Туре	0From View	-				
Unmapped	Values	4No Output	-				
X or C	onstant						
	Y						
	Z						
Target To Model Loads To Data Surface Nodal Elemental							
OK Cancel							

Preprocessing Data Mapping Enhancements - Example





Unrestricted © Siemens AG 2018

Page 59 2018-07-23

Preprocessing Discrete Value Plots



- Colored based on element value or range of values
- Covers all material, property and entity ID available in Model Data Contour
- Auto creation of list of values or list of ranges based on each model's data



Postprocessing Dynamic Criteria

Accessed via the View – Advanced Post – Dynamic Criteria command or in the PostProcessing toolbox

- Interactively adjust Criteria Limits Mode and Limit Values
- Use slider bar(s) to update display
- Optionally display Criteria Labels and/or use the Absolute Value of current Output Vector





Postprocessing Report Generator

- Tools Report Generator command:
- Checks to see if Microsoft Word is available
- Dialog box with various tabs allows user to control settings
- FEMAP creates report directly within Microsoft Word
- Uses special functionality to create report seamlessly
- Organized for the technical analyst
- Summarize Model Data

SI	Ε	Μ	Ε	N	S	
	Ing	pen	aity	1 fe	orl	ife

fo Entities Picture	
Name	Stress Analyst
Company	FEMAP
Organization	Siemens PLM
Description	
Summary Report for F	inite Element Model

Postprocessing Report Generator

- Entities tab
 - Choose Entity Types to include in report (load sets, constraint sets, connections, output sets, groups, and/or layups)
 - Include the active entity of a certain type, all, or select a subset
- Picture tab (View Settings)
 - Choose Active View, Multi-View, or Current Layout (additional options available for each)
 - Choose to include deformation and/or contour in pictures of model
 - Specify orientation
 - Specify picture size (height/width)



Report Generator			×]	
Info Entities Picture	Constraint S	ets Connections			
Active All Select	Constraint 3 Active All Select	Connections			
✓ Output	Groups	Layups			
Active All Select	Active All Select	Report Generator			>
		1Active View Image: Comparison of the second	□ Deformed [□ Contoured [ulti-View [□ 2⊻ [□ 2½ [Landscape He ✓ Level Colors W 3 Image: Second seco	eight 730 lidth 630 .ayout Active View All Views No Changes
				Create Report	Done



Postprocessing Report Generator

Navigation Search document HEADINGS PAGES RESULTS . Information Geometry Mesh Material Structural Thermal Analyses Load Set 1..Bearing Load Constraint Set 1..Fixed Base ▲ Layup

1..Composite Layup Example 2..Composite Layup Example 2

- Results Summary
 1..NX NASTRAN Case 1
- 1..NX NASTRAN Case 1, 7026..Plate Top MajorPrn Stress Group 2..Arm
- Appendix Model Files (*.modfem)|*.modfem|
 - Results Recovery

 NX NASTRAN Case 1

 Notes:

Material

 $\star \mathbf{X}$

P -

Structu	Structural								
ID	Title	<u>Youngs</u> Modulus, E	Shear Modulus, G	Poisson, Nu	Tension Limit	Compression Limit	Shear Limit		
1	MAT_1	9900000	3800000	0.33	35000	35000	27000		
2	MAT_2	16000000	6200000	0.31	145000	15000	93000		
3	MAT_3	10300000	4000000	0.33	66000	65000	44000		
Total	3								

Results Summary

1..NX NASTRAN Case 1

Output Vector		Node	Element	Max	Node	Element	Min
1	Total Translation	126		0.330235	1		0.000000
2	T1 Translation	321		0.008644	383		-0.012213
3	T2 Translation	309		0.329352	1		0.000000
4	T3 Translation	220		0.018874	126		-0.025825
5	Total Rotation	131		0.008847	1		0.000479
6	R1 Rotation	256		0.003492	131		-0.008846
7	R2 Rotation	383		0.002254	382		-0.002274
8	R3 Rotation	386		0.005688	382		-0.003387

SIEMENS Ingenuity for life

1..NX NASTRAN Case 1, 7033..Plate Top VonMises Stress Output Vector Element Element Max Plate Top 136 5546.248535 159 290.303314 VonMises Stress 5563. 5563. 4877 4877 3505 3505. 2134. 2134. 1448. 1448. <u>↓</u>___× 2 🛶 📈 762.1 762.1 76.31 5563. 5220. 4877. 3505. 2819. 2134. 1791. 1448. 1105. 762.1 419.2

Unrestricted © Siemens AG 2018 Page 64 2018-07-23

76.31



Added Previous On icon buttons to Select Output and Select Output Sets to Process dialog boxes used in a number of different commands (for example, List – Output – Results to Data Table and Model – Output – Process when processing Complete Output Sets)





Added ability to send transformed results to the Data Table using the List – Output – Results to Data Table command and added option to Show Summary Table

Send Results to Data Table					
Report Style			Transform		
Output Sets Output Vectors Nodes/Elements	◯ in Rows ◯ in Rows ◉ in Rows	 in Columns in Columns in Columns 	None From Active View Custom		
	Alternat	e Column Order Immary Table	Transform <u>Q</u> K Cancel		

Divided the Mesh – Edge Members command into two commands:

- Mesh Edge/Skin Elements Line Elements on Edges
- Mesh Edge/Skin Elements Planar Elements on Faces

Added Group – Operations – Generate From Criteria command – only adds elements which currently pass the Criteria (only available when a Criteria plot is being displayed in the graphics window)

Unrestricted © Siemens AG 2018

Page 66 2018-07-23



Added ability to edit the currently specified Layup when creating or editing a Laminate or Solid Laminate Property

Define Property - LAMINATE PLATE Element Type		×
ID 1 Title LAMINATE Property - 7 Color 30 Palette Layer 1	Material Elem/Prop	γ σ ^ε ν erty Ty <u>p</u> e
Laminate Definition Layup 17 Plies Offset Bottom Surface 0. Options 0As Specified	Laminate Properties N.S.Mass/ <u>A</u> rea 0. BondShr Allow 0. Ref Temp 0. Damping 0.	Failure Theory None Hill Hoffman Tsai-Wu Max Strain NEi Nastran
Loa <u>d</u> <u>S</u> ave Cop <u>y</u>	<u>O</u> K	Cancel



Calculation of Laminate Equivalent Properties now done with both Full Membrane/Bending Coupling Included and Full Membrane/Bending Coupling Excluded (informational purposes only)



Solver Support Solver Updates

NX Nastran

- SOL 401 Multi-step Nonlinear Structural
- SOL 402 Multi-step Nonlinear Kinematic solver support
- SOL 200 Optimization
 - Enhanced design optimization support
 - Topology optimization support

ANSYS

- 100% completely new translator
- Modern ANSYS element types
- ANSYS organized input file creation

Abaqus

- Expanded support of rigid analytic surfaces
- Support export of CBUSH as CONN3D2 element

SIEMENS

Ingenuity for life

Solver Support Memory Management (NX Nastran 12)

SIEMENS Ingenuity for life

Nastran memory split into 2 parts:

- Fixed allocation
 - Database functions
- Dynamic allocation
 - Used for memory intensive modules, i.e. sparse solver, etc.
 - Can be released when not needed
 - ISHELL execution of external application
 - SOLs 402, 601 & 701
 - API call to external DLL.
- No UI controls. Implementation is transparent to the user.
- Value specified on MEMORY keyword is now a hi-water mark, not an allocation

💐 Windows Task N	Manager			
<u>File Options Vie</u>	ew <u>H</u> elp			
Applications Proces	sses Services	Performance Netw	working Users	
CPU Usage	CPU Usage H	listory		
0 %	- Physical Mer	mory Usage History		
6.78 GB				
Physical Memory	(MB)	System	504-5	
l otal Cached	326/3 24405	Handles Threads	53887 1882	
Available	25720	Processes	120	
Free	1414	Up Time	2:07:31:15	
Kernel Memory (MB)	Commit (GB)	7 / 39	
Paged	1596	6		
Nonpaged	508	Resource	e Monitor	
Processes: 120	CPU Usage: 0	% Phys	ical Memory: 21	%

Solver Support NX Nastran Multi-step Nonlinear



NX Nastran SOI 401/402 Analysis Set Manager (Active: 3..NX Nastran MS NL Analysis Set) X nalvsis Set : 3..NX Nastran MS NL Analvsis Se Solver : NX Nastran ... Type : Multi-Step Structural Subcase sequence dependency selection (SEQDEP) Integrated Solver : NX Nastran ⊕ Options Bolt preload sequence (BOLTSEQ) Global Requests and Conditions - Subtitle : -Label : TSTEP1 Solution time steps intervals E Control Options Analyze Boundary Conditions Analyze Multiple.. х Analysis Case Constraints : 1...Fixed at Ends -Loads : None Analysis Type 29...Bolt Preload -Export 1 Active... Case 1...Static Case Subtitle Bolt Sequence title : Static Case Preview Input Label Available Bolt Preload Bolt Preload Sequence be : Static Step Control 100...NASTRAN TEMPD 100 -- 3..BOLTFOR 10 Force on 3D elements 1 uentially Dependent : YES MultiSet. Sequentially Dependant 200...NASTRAN TEMPD 200 -- 4..BOLTFOR 10_Force on 3D elements(1) 2 htrol Options 300...NASTRAN TEMPD 300 -- 5..BOLTFOR 10 Force on 3D elements(2) 1 Not Sequentially Dependant Copy 400...NASTRAN TEMPD 400 e Steps Indary Conditions Manual Control Delete Start Text (Off) Constraints : None Skip Standard oads : 1 External Force Renumber... End Text (Off) NASTRAN Multistep Nonlinear Time Steps Properties Prev... Ne<u>x</u>t... OK Cance Add Bolt Sequencce Move Up Load... nber of Increm End Time **Output Frequency** Output Increment Number of Increments 1 Solution Step End Time 1 Remove Bolt Sequence Move Dowr 1. Save... 0 0. 0 Solution Step End Time 1. Update Bolt Increments 0. 0 Solution Step End Time 1. <u>N</u>ew.... 0. 0 Solution Step End Time -1. Pre<u>v</u>... Ne<u>x</u>t... OK Cancel 0. 0 Solution Step End Time 1. Edit... Pre<u>v</u>... Next... <u>0</u>K Cancel Done

Solver Support NX Nastran Multi-step Nonlinear

SIEMENS Ingenuity for life

Support for new NX Nastran solution sequences:

- SOL 401 Multi-step Structural
- SOL 402 Multi-step Kinematics

Analysis Set Manager updated for new solution workflows

- Includes robust selection of element types With NX Nastran 12
- New multi-step workflow allows change of analysis type in subcase (statics, modes)
- Subcases can be dependent or independent of previous subcase
- Change contact/glue and boundary conditions in subcases
- Iteration/solution control changes allowed in subcases




NX Nastran SOL 401 – bolt preload example



Unrestricted © Siemens AG 2018

Page 73 2018-07-23



SOL 401, NLSTEP, is a multi-step, structural solution which supports a combination of static (linear or nonlinear), modal (real eigenvalue), bolt preload, and buckling subcases

- Geometric nonlinearity large displacement and rotation
- Geometric stiffening due to initial displacements and stresses
- Follower forces
- Large deformation effect resulting in large strains has not been implemented
- Material nonlinearity
- Plasticity
- Creep
- Primary solution operations are time increments, iterations with convergence tests for acceptable equilibrium error, and stiffness matrix updates

SOL401 FEMAP support - elements

- 3D solid elements
- Axisymmetric
- Plane strain
- RBE2 and RBE3
- CQUADR/CTRIAR (CQUAD4/TRIA3 can still be used, but internally converted to CQUADR/CTRIAR
- Shell elements support OFFSETs
- CBAR/CBEAM PBAR/PBARL/PBEAM/PBEAML all supported
- Beam end offsets are supported
- Geometric nonlinear large displacement/large rotation
- Material nonlinearity not supported in BEAMS
- Beam distributed loads are follower forces
- CELAS1/PELAS
- CBUSH/PBUSH (no damping)
- Cohesive





SOL401 FEMAP Support – Materials

- Isotropic (MAT1)
- Orthotropic (MAT8, MAT11)
- Anisotropic (MAT2, MAT9)
- Plasticity (MATS1)
- Creep (MATCRP)

SOL401 analysis setup

- SOL401 is multi-step
- Each case can optionally be dependent on the previous case or cases
- Nonlinear static
- Bolt preload
- Modal
- Buckling





SOL401 FEMAP support – multi-step control

- Setup in the FEMAP Analysis Set Manager
- Time Step and Solution Control in each Subcase
- Example
 - Case 1 ramps the structure up through a nonlinear static case
 - Case 2 uses the stiffened results of case 1 and runs a modal solution







- Setup in the FEMAP Analysis Set Manager
- Time step and solution control in each subcase
- Example
 - Case 1 ramps the structure up through a nonlinear static case
 - Case 2 uses the stiffened results of case 1 and runs a modal solution





SIEMENS

Ingenuity for life









SOL401 support – contact

- Handled just like contact for SOL101, SOL601/701
- New tab for NX multi-step nonlinear

Define Connecti	on Property					>
ID 🛃	Title SOL401	Contact Palette <u>L</u> aye	er 1	Connec	t Type Contact	~
ANSYS	MSC Nas	tran	LS-DYNA	NEi Na	stran	MARC
NX Linear	NX Multister	Structural	NX Adv Nonl	in NX	(Explicit	ABAQUS
Contact Pair (B	SCTSET)	I	Glued Con	tact Propert	y (BGSET)	
Friction		0.	Search Dis	tance	C).
Min Contact Sea	arch Dist	0.				
Max Contact Se	earch Dist	17.68391				
Contact Proper	ty (BCTPARM)	* Can be define	d on local entrie	es		
Convergence C	riteria 1PTC	DL Convergen 🗸	* Initial Pe	netration	0Calcula	ated \checkmark
Force Conver	gence Tol	0.05	* NOSE	P		
* PTOL		0.	Geometry	Updates	-1Defau	lt v
RCTOL		0.05	* GUPTOL		1	0.
Max Status Iter	ations	20	Tangential	Stiffness	1Adapti	ve v
* OPNSTF		1.E-6	* SCRIT		1	0.005
* OPNTOL		1.	* FRIC	DLY		
* GAPTOL		1.E-10		DISTO	0.	
* TANSCL		1.				
Common Conta	ct (BCTPARM)	and Glue (BGPA	RM) * Can be	defined on lo	ocal entries	
Eval Order	2Medium	~	* Penalty F	Factor Units	11/Leng	ith ~
Refine Source	2Refinement	t Occurs 🗸 🗸	* Penalt	ty Autoscale		1.
Generate Co	ontact Preview I	ile	* Nor	mal Factor		0.
* Glue Factor			✓ * Tan	igential Fact	or	-1.E-3
Sliding Glue						
Defaults	Loa <u>d</u>	<u>S</u> ave	Cop <u>y</u>		<u>о</u> к	Cancel



SOL 402, the multi-step nonlinear solution sequence combines the advantages of SOL 401 and SOL 601

- Large displacements, large rotation
- multi-step
- Similar element support as SOL401
 - Solid laminates
 - Shell laminates
- More nonlinear material laws
- Linear elastic
- Plasticity
- Hyperelasticity
- Creep
- Composite
- Damage interface (cohesive zones)
- Based on SAMCEF technology
- Advanced composite failure models

Unrestricted © Siemens AG 2018

SOL 402 support in FEMAP will be exactly like SOL 401, including these additional features:

- SOL 402 is highly compatible with SOL 401 input files
- SOL 402 creates standard Nastran .OP2 files for results processing

Solver Support NX Nastran Design and Topology Optimization



- Support for SOL 200 design and topology optimization
 - Updated work flow for traditional SOL 200 optimization
 - Added support for topology optimization



Options	-	-		
Optimization Type	Oesign	Topology		
Analysis Type	1Static	•		
	Track M	odes		
Goal	Weight	▼		
	Minimize	Maximize		
Cycles	25			
Output Interval	0First and l	ast 🔹		
Optimization Variab	les			
Optir	mization Variab	les (3 Selected)		
Manufacturing Con	straints			
Sele	ect Manufacturi	ing Constraints		
Select Manufacturing Constraints PARAM BDMNCON 10				
PARAM BDMNCON 10				



Solver Support NX Nastran Design Optimization



New optimization interface with expanded support for NX Nastran design optimization

III Analysis Set Manager (Active: 2NX Nastran DesTopOpt A	
Analysis Set Wanagel (Active: 2.100 Wastan Desropopt A. Analysis Set : 1NX Nastran Linear Statics Analysis Set : 2NX Nastran Solver : NX Nastran Type : Design / Topology Optimization Integrated Solver : NX Nastran Options Executive/Solution Bulk Data Optimization Type : Design Analysis Type : Static Goal : Minimize Weight Cycles : 25 Output Interval : First and Last Output Interval : First and Last Output Interval : Selected	Analyze Analyze Multiple Export Active Preview Input MultiSet Copy Delete
	Renumber
MODELCHECK External Superelement Reference Master Requests and Conditions Subtitle : Label : Optimization Limits : None Boundary Conditions Output Requests No Cases Defined	Load Save New Edit

NASTRAN Optimiza	tion Options	83		
Options				
Optimization Type	Design			
Analysis Type	1Static	•		
	Track Modes			
Goal	Weight	•		
	Minimize			
Cycles	25			
Output Interval	0First and Last 🔹			
Optimization Variab	les mization Variables (3 Selected)			
Manufacturing Con	straints			
Select Manufacturing Constraints				
PARAM BDMNC	сом 10 ext ОК (Cancel		

ID Title		Туре	Subtype	Entities	Lower Bound	Upper B
I tip d	isplacement	Nodal Displacement	ТХ	1 Node(s)	-0.525	1.56
R.F.		الله المراجع ال Prev.	Ne	ext	OK	Can
roperty Rel	ation Title			X		
roperty Rel ID 5 Type 0. Options	ation Title .Property		•	X		
roperty Rel ID 5 Type 0. Options Property	ation Title .Property 101Upper	Wing Skin	•	×		
roperty Rel D 5 Type 0. Options Property	ation Title .Property 101Upper	Wing Skin Select Multiple	•	≍ •		
Type O. Type O. Property Type	ation Title .Property 101Upper	Wing Skin Select Multiple	• 	▼ 💽 Edit		
Type 0. Options Property Type Name	ation Title .Property 101Upper PSHELL Thickness	Wing Skin Select Multiple	• ••• •••	▼ 💽 Edit		
Type 0. Options Property Type Name	ation Title .Property 101Upper PSHELL Thickness	Wing Skin Select Multiple	• 	▼ 💦		

Solver Support NX Nastran Design Optimization



New features

- Interface redesigned for ease of use and enhanced compatibility with NX Nastran capabilities
- Better integration with Analysis Manager allows for multiple design studies within a single model

Expanded support for NX Nastran solution sequences

- Existing: linear static, normal modes, linear buckling
- New: frequency response, modal transient, steady aeroelastic
- Now supports different solution sequences for each subcase

Greatly enhanced support for design variables and relationships

- New support for material relationships (E, nu, rho, etc)
- Expanded list of supported property relationships, including user-defined relationships
- Discrete design variable values can now be specified
- Multiple independent DESVAR cards for complex design relationships

Solver Support NX Nastran Design Optimization



Expanded list of supported design responses

- Supported element types now includes beams and solid elements
- User defined responses

Constraints are now decoupled from responses

- Allows for trade studies where constraints vary between subcases
- Disable / enable constraints within the Analysis Manager





New optimization interface with support for NX Nastran Topology Optimization

Analysis Set Manager (Active: 3NX Nastran DesTopOpt A	
 Analysis Set : 1. NX Nastran Linear Statics Analysis Set : 2. NX Nastran DesTopOpt Analysis Set Analysis Set : 3. NX Nastran DesTopOpt Analysis Set Analysis Set : NX Nastran Type : Design / Topology Optimization Integrated Solver : NX Nastran Options Executive/Solution Bulk Data Design / Topology Optimization Optimization Type : Topology Analysis Type : Static Goal : Minimize Compliance Cycles : 25 Output Interval : First and Best Topology Regions : 1 Selected Manufacturing Constraints : 1 Selected PARAM BDMNCON : Off GEOMCHECK MODELCHECK Master Requests and Conditions Subitle : Label : Optimization Limits : 1 Selected Boundary Conditions Output Requests No Cases Defined 	Analyze Analyze Multiple Export Active Preview Input MultiSet Copy Delete Renumber Load Save New Edit Done

Options Optimization Type Design Optimization Type Topology Analysis Type 1Static Track Modes Goal Compliance Omplia
Analysis Type 1Static Track Modes Goal Ompliance Minimize Maximize
Goal Compliance Maximize
Goal Compliance Minimize Maximize
Minimize
Cycles 25
Output Interval 0First and Best
Topology Regions
Topology Regions (1 Selected)
Manufacturing Constraints
Manufacturing Constraints (1 Selected)
PARAM BDMNCON 10
Prev Next OK Cancel

ID	Title	Туре	Subtype	Entities	Lower Bound	Upper Bo
▼ 1	tip displacement	Nodal Displacement	ТХ	1 Node(s)	-0.525	1.56
R						
		Prev		ext	ОК	Cance
Sele	ct Topology Regio	ns(s)	Ne	ext	OK	Cance
ID	ct Topology Regio Title S active elements A	Prev ns(s) tate # Element(s) ctive 36	Ne	ext	ОК	Cance
] Sele	ct Topology Regio Title S active elements A	ns(s) tate # Element(s) ctive 36	Ne	ext	ОК	



Shared interface with Design Optimization

- Adapt the same workflow
- Reuse optimization constraints and responses between both design and topology optimization

Compatible Solution Sequences

- Statics
- Normal modes
- Modal frequency
- Modal transient
- Linear buckling

Supported Element Types

- 2D: CTRIA3, CTRIA6, CQUAD4, CQUAD8, CQUADR
- 3D: CHEXA, CPENTA, CTETRA

Support for manufacturing constraints





Original model 33,046 nodes, 20,595 elements



Unrestricted © Siemens AG 2018

Page 88 2018-07-23



Added option to File – Export – Geometry command to export STL file based on Criteria Plot of Normalized Mass Density

 Clicking Options button opens dialog similar to the new View – Advanced Post – Dynamic Criteria command and slider bar can be used to export the optimized topology at the desired level

Translate		×
Format O Parasolid XMT ACIS SAT Stereolithography Topology Optimization VRML STEP IGES JT	<u>V</u> ersion	
Options OK	Cancel	

Topology Optimization STL File Output						×
No Limits Above Between Labels Below Outside	Maximum Minimum	«	>	0.306152 0.306152	<u>0</u> K	Cancel

Solver Support ANSYS Enhancements



Seamless model input data transfer between FEMAP and ANSYS structural/mechanical analysis
40 out of 42 FEMAP elements and their properties are mapped to modern ANSYS elements

- 30 FEMAP materials are mapped to modern ANSYS material models constant or tabular
- Complete support of different boundary and loading definitions nodal, elemental, surface or volume, constant or tabular
- FEMAP Connection entities are mapped to modern ANSYS contact definitions
- Data defaults are provided to allow minimal user intervention, while also allowing users to change the defaults, when required
- Error or warning messages are printed the translator works like a compiler to let the user know where the input errors are in either FEMAP model or ANSYS input file

Solver Support ANSYS Enhancements



Equivalence of physical models are maintained via bi-directional data transfer to make sure close computational results are obtained in NX Nastran and ANSYS

Enhanced code architecture allows easy future extensions, including additional ANSYS material models and element properties, some of which may not currently exist in FEMAP interface

Solver Support ANSYS Enhancements





Miscellaneous FEMAP Product Excellence Program



- This program is designed to give us additional insight into how FEMAP users use the product
 - Without telemetry data, we only have subjective and qualitative insights into how FEMAP is used
- Telemetry data provides quantitative usage metrics
- Development actions will be directly influenced by usage data
- FEMAP performance is not affected
- Data is encrypted being to the Siemens cloud
- Participation in the program is <u>optional</u>
 - Your voice will not be heard if you choose not to participate

NO DATA IS COLLECTED THAT IS:

- Personally identifiable
- Intellectual property (e.g. NO model data)





Miscellaneous FEMAP Product Excellence Program



What data is collected	How we plan to use that data
Machine information (CPU, GPU, RAM, OS version, etc)	By having a better understanding of the types of machines used in the field, we can better ensure FEMAP is optimized for the majority of hardware configurations
Program information (version / license type)	Version usage and license type (no serial numbers) gives us insight into how FEMAP is deployed, including adoption rate. This can impact release timing
User preferences	We aim to improve the FEMAP out-of-box experience by knowing which preferences are always set. Do we need to change defaults? <i>Note: Preference data containing any directory or filename information is not collected</i>
Command history	This data gives us insights into which are our most-used and least-used commands, as well as which ones are undone most frequently. This can help us allocate development resources
Pane and Toolbox usage	How are panes and toolboxes used, and how are they used when duplicate menu functionality exists. Do we need better education programs?

In future releases of FEMAP we will disclose the direct impact usage metrics has on development and how it has been used to improve the user experience

FEMAP 12.0 Q and A





Unrestricted © Siemens AG 2018 Page 95 2018-07-23