



1. WELCOME TO FEMAP

1.1 FEMAP DOCKABLE PANES

Femap contains several "Dockable Panes" that offer different tools used to create and modify models, evaluate and sort data, create reports, and view info of specific entities. Each dockable pane can be either visible or hidden by using the **Tools** menu command corresponding to the specific pane.







1.2 TOOLBARS

Femap has many useful toolbars that help you perform a variety of different functions. The toolbars contain icons representing certain commands and are grouped together by functionality. Each toolbar can be made visible or hidden using the **Tools > Toolbars** menu command, then choosing a particular toolbar from the list. When the toolbar is visible, it will have a check mark next to the toolbar name on the menu. Once the toolbars are visible, they can either be "docked" around the edge of the Femap interface or "floating" somewhere inside the Femap interface.







1.3 PREFERENCES

This command allows you to customize the operation of Femap. These options control how certain commands will operate, set defaults, and define disks or files to be used. This command bring up a "tabbed" dialog box with 10 tabs, each tab representing the type of entity you want to modify.

Preferences	-			×			
Geometry/Model	Interfaces	Results	Library/Startup Color	Spaceball			
Messages	Views	Graphics	User Interface	Database			
Menus and Dialog B	oxes		Dockable Panes				
Alternate Color	Scheme		Animate Fly-Out				
Autorepeat Cre	ate Commands		Captions Always on To	p			
Remember Dialo	g Positions		Alternate Docking Sym	ools			
Alternate Accele	erator Keys for	Views	Enable Legacy XY Plott	ing			
Ask for Confirm	ation Before De 0 -> 360	lete	Model Info				
Fast Output Delete	1Fast ()	No Undo) 🔻	Max Entities 20	5000			
Recently Used Files	s 4						
Graphical Selection			Show Entities Defaults				
Track Mouse Pic	king		Highlight				
Pick All Inside			Transparent Highlight				
Pick Method	0Pick No	ormal 🔻	Show Selected Only Show Labels Normals Highlight Color 24600 Palette				
Tooltip Delay	10						
Tooltip Duration	100						
Mouse Interface			International Localization				
Reverse Mouse	Wheel Direction ick for OK	ı	☑ Use Region Decimal/List Symbols				
Shift for Pan. C	trl for Zoom		Toolbars				
Wheel Factor fo	Wheel Factor for Dynamic Planes 1.			Save Layout			
Meshing Toolbox			Reset User Interface				
Expand Active 1	Fool Only						
AutoRemesh Mo	de 0Auto R	emesh 🔹	Reset Dialog Posit	ions			
Reset A			ОК	Cancel			





2. ANALYSIS WORKFLOW

- Geometry
- Material
- Property
- Mesh Sizing
- Meshing
- Loads
- Constraints
- Analyze









2.1 GEOMETRY

Import geometry or create your own within Femap.

- ACIS Solid Model Files *.SAT files
- Parasolid Solid Model Files *.X_T files
- IGES Files *.IGS files
- STEP Files *.STP files (AP203 and AP214 geometry)
- Stereolithography Files *.STL files
- Wireframe Files *.DXF files.
- CATIA V4 Models *.MDL files
- CATIA V4 Express Files *.EXP, *.DLV files
- CATIA V5 Files *.CATP files
- I-DEAS Files *.IDI files
- Pro/ENGINEER Models *.PRT and *.ASM files
- Solid Edge Models *.PAR, *.PSM, *PWD, and *ASM files
- Unigraphics Models *.PRT files







2.2 MATERIAL

Load a material from one of several Femap libraries or create your own. Don't include unnecessary material data. Be mindful of your unit system. *We recommend that you change the default material library to one of the more specialized libraries located in the Femap directory.*

Define Material - ISOTR	OPIC			Select From	Library	
ID 4 Title	AISI 4340 Steel <u>C</u> olor	104 Palette	Layer 1 Type	Library Entry		Choose Library
Stiffness Youngs Modulus, <u>E</u> Shear Modulus, <u>G</u> Poisson's Ratio, n <u>u</u>	29000000. 0. 0.32	Limit St <u>r</u> ess Tension Compression Shear	215000. 240000. 156000.	AISI 4340 Stee 15-5PH Stainle 17-4PH Stainle 2024-T351 AI 6061-T651 AI 7050-T651 AI 7075-T651 AI	el ss H1025 ss H1025 Plate .255 Plate .255 Plate .255 Plate .255	
Thermal Expansion Coeff, <u>a</u> Conductivity, <u>k</u> Specific <u>H</u> eat, Cp Heat Generation Fac	6.6E-6 4.86111E-4 0.1 tor 0.	Mass De <u>n</u> sity Da <u>m</u> ping, 2C/Co Re <u>f</u> erence Temp	7.33145E-4 0. 70.	Ti-6AI-4V Sol T AISI 1025 Carl AISI 1125 Carl Stainless Steel Magnesium AZ Magnesium ZK Titanium, Pure A286 HRES Irc Hastelloy X Inconel 600 Rene 41 Bervilium	r & Aged . Joon Steel el Annealed 31B 50A in Alloy	E
fxy Load	Save Co	ρχ	QK Cance	Values in Libra You must v	er ry files distributed with FEMAP a erify these values are correct ar via	re believed to be correct but have not been verified. Id appropriate before using them for any purpose. Delete OK Cancel





2.3 PROPERTY

Predictive Engineering

FINITE ELEMENT ANALYSIS

Generate property types appropriate for your geometry.

efine Property - BEAM Element Type			
ID 1 Title Upper Angle Stiffener	Material 16061-T651 Al Plate .25 ▼ 🚭	Define Property - PLATE Element Type	
Color 3 Palette Layer 1	Elem/Property Type		
Property Values	Stress Recovery (2 to 4 Blank=Square)	10 101 Title Upper Wing Skin Material 37050-T651 Al Plate V 🐨 Define Property - LAMINATE PLATE Element Type	
Tapered Beam End A End B	Y Z	Color 50 Palette Layer 1 Elem/Property Type	
Area, <u>A</u> 0.121094 0.	End A 1 -0.273185 -0.273185	Property Values Additional Options ID 6 Title Composite Shell Material	-
Moment of Inertia, I1 or Izz 0.0118724 0.	2 0.726815 -0.273185	Thicknesses, Tava or T1 0.04 Bend Stiffness, 121/7**3 0. Color 110 Palette Layer 1 Elem/P	roperty Type
I <u>2</u> or Iyy 0.0118724 0.	3 -0.210685 0.726815	Nark or T2 0. TShear/Mem Thickness.ts/t 0.	Cellura Theorem
I <u>1</u> 2 or Izy -0.007088 0.	4 -0.273185 0.726815	Laminate Definition	Pallure Theory
Torsional Constant, <u>1</u> 0.000163 0.		Layup 18-ply Glass EP V S N.S.Mass/Area 0.	
Y Shear Area 0.0625 0.	End B 1 0. 0.	blank or T4_0. BondShir Allow 5000	
Z Shear Area 0.0625 0.	2 0. 0.	Nonstructural mass/grea 0. Ref Temp 0.	Tani Mar
Warping Constant 0		Mean Dilatational Formulation Options 0As Specified	Max Strain
Perimeter 0. 0.	4 0. 0.	NEi Nastran Tension Only	NEi Nastran
V Neutral Avic Officet 0.293185 0.293185	Shana	Stress Recovery (Default=7/2)	
Z Neutral Axis Offset 0.293185 0.293185	Shape End B	Top Fiber 0. QK	
Load Save Copy	QK Cancel	Bottom Fiber 0. Cancel Load Save Copy OK	Cancel

2.4 MESH SIZING

Choose your mesh sizing method according to the element type you want to use.

Mesh Size	Node Spacing
○ Number of <u>E</u> lements	Egual Egual Egrametric Biased Geometric Bias
	Bias Eactor 1. (a) Small Bements at Start
Min Elem on Other Edges 2	Small Elements at End Small Elements at Center Small Elements at Center Small Elements at Path Fold







FINITE ELEMENT ANALYSIS Predictive Engineering

2.5 MESHING

Finite elements can be classified as point (0-D), line (1-D), planar (2-D) and solid (3-D) elements.

- 0-D elements (e.g., point masses) are created on one node and can be meshed on geometric points.
- 1-D elements (e.g., beams) are created on two nodes and can be meshed on geometric lines.
- 2-D elements (e.g., plates) are created on three (triangular) or four nodes (or quadrilateral) and can be meshed on geometric surfaces.
- 3-D elements (e.g., solids) are created on a minimum of four nodes (tetrahedral) or eight nodes (hexahedral) and can be meshed on geometric solids.





FINITE ELEMENT ANALYSIS Predictive Engineering

2.6 LOADS



Every time a load is created on finite element entities (i.e., Model, Load, Nodal; Model, Load, Nodal on Face; and Model, Load, Elemental) or geometry (Model, Load, On Point; Model, Load, On Curve; Model, Load, on Surface) a "Load Definition" will also be created in Femap. A Bolt Preload will also create a Load Definition. These Load Definitions will appear in the Loads branch of the Model Info tree and can be given a title.





FINITE ELEMENT ANALYSIS Predictive Engineering

2.7 CONSTRAINTS

Every time a constraint is created on finite element entities (i.e., *Model, Constraint, Nodal; Model, Constraint, Nodal on Face; and Model, Constraint, Equation*) or geometry (*Model, Constraint, On Point; Model, Constraint, On Curve; Model, Constraint, on Surface*) a "Constraint Definition" will also be created in Femap. These Constraint Definitions will appear in the *Constraints* branch of the *Model Info* tree and can be given a title.

Editing Constraint Definition	x			
Constraint Set 1 Pinned				
Title Constraint on Surface				
Color 120 Palette Layer 1			Ť÷.	
Standard Types				
© Fixed				
Pinned - No Translation				t _t .
No <u>R</u> otation				
Advanced Types				
Arbitrary in CSys -1Nodal Output CSys				
TX TY TZ RX RY RZ				
Surface Include Rotational DOF				
 Allow Sliding only along Surface (Symmetry) 				
 Allow Movement Normal to Surface (AntiSymmetry) 			23 432	
Allow Sliding only in Specified Direction			123 123 123 123 123 123 123 123	123 123 23
🔘 Cylinder / Hole			1233334123 17232023	
Constrain Radial Growth	ОК			
Constrain Rotation around Axis	Cancel			12312123723 12312123723 12312123723
Constrain Sliding along Axis				THE BAS
		XXX		



2.8 ANALYZE

FINITE ELEMENT ANALYSIS
Predictive Engineering

As they say in the Femap help file examples: "THE MODEL IS NOW READY TO BE ANALYZED!"

Analysis Set Manager (Active: 1Glued Contact)		Nastran Output Reque	ests			×
Analysis Set : 1Glued Contact	Analyze	Nodal	0Full Model	Elemental	0. Full Model	Ţ
	Analyze Multiple	Applied Load	0Full Model 👻	✓ <u>Force</u> ✓ <u>S</u> tress	0Full Model	-
Options Executive/Solution	Export	Constraint Force	0Full Model v	Strain	0Full Model	
	Preview Input	Force <u>B</u> alance	0Full Model	Heat Flux	0Full Model	-
	MultiSet	Acceleration	0Full Model	Enthalpy Rate	0Full Model	-
External Superelement Creation Boundary Conditions Constraints : 1 Pipped	<u>C</u> opy Delete	Kinetic Energy	0Full Model V	Kinetic Energy	0Full Model	*
Loads : 1Normal Force ⊕ Output Requests	Load			Energy Loss	0Full Model	• •
I No Cases Defined	<u>S</u> ave	Customization	sults	Results Destination	Prev	
	<u>N</u> ew	Output Modes (a,b,c i	THRU d)	2PostProcess Only Echo Model		
	Done	Relative Enforced N	Motion Results		Can	





3. TIP, TRICKS, NEW FEATURES

3.1 MEASURE BETWEEN GEOMETRY

This function measures the minimum or maximum distance (or both) between two sets of selected geometric entities.

Distance Between Geo	ometry		×
Distance to Find Minimum Maximum Both	From Point Curve Surface Solid Voerall Only	To Point Curve Surface Solid Verall Only	<u>Q</u> K Cancel





3.2 SELECTOR BLANKING

This method allows the user to blank/un-blank entities from the Femap window using the Selector tool.







3.3 LOAD FROM FREEBODY

This technique creates loads directly from a freebody display. The "Multi-Model" option allows the analyst to generate the loads within a separate model file. This option also facilitates the generation of RBE3 elements to assist in applying the freebody loads to a model with a finer mesh.







3.4 MODEL MERGE

The *File > Merge* command allows entities from any model currently open in the same instance of Femap to be "merged" with the active model. At least two models must be open for this command to be available. To facilitate bringing entities into the active model, a number of overall *Renumbering and Duplicates Strategy, Entity Selection,* and *Model Orientation* options are available in the *Model Merge Manager* dialog box.

Model Merge Manager					~	
Merge/Extract From Model Model Merge	e Test 2.modfem	•	To Model Mod	del Merge Test 1.modfem		
Entity Type	Current IDs	Merge IDs	Renumber To	Renumbering		All On
V Point	10121	10121	193	56 of 56		All Off
Curve	638	638	156	28 of 28		
Surface	115	115	1630	15 of 15		Select All
Coordinate System	33	33	44	1 of 1		Select None
Vode Node	1387	1645	15645	72 of 471		Selectivone
Element	1284	11564	2851564	12 of 620		Update Selected
Material	13	13	46	3 of 3		
V Property	1104	1104	614	9 of 9		Update All
 None Minimal Renumbering 	Renumbe Overwrite	r Duplicates	None) All In Model	Add Ref	ferenced Entities
 Block Renumbering Offset Renumbering 	Overwhite	Duplicates	 From Group ID Range 	From 1	То	99999999
Compress	Renumber To	L	Select			
lodel Orientation					Duplicat	es to Data Table
Create Group for Merge	ed Model	Trans	sform Merged Model		Coopicat	
Create Parent CSys for	Merged Model	From	0Basic Rectangular	-		<u>O</u> K
Condense Transferred	Groups	то	0Basic Rectangular			
			a na a ana na a a a a a a a a a a a a a			Cancel





3.5 RENUMBERING ENTITIES FROM THE MODEL INFO TREE

For the analyst with a touch of OCD... You can now renumber selected entities from the Model Info Tree. This method renumbers entities in the model by attempting to use the specified ID to start. If specified ID to start is already used, Femap will use next empty ID instead until all selected entities have been renumbered.







4. GAINING NEW SKILLS

We have lots of seminars, how-to-guides, tutorials and API tips and tricks







5. FATIGUE ESSENTIALS











6. TRAINING OPPORTUNITIES

- LS-DYNA Analysis for Structural Mechanics, January 2015
- Femap and NX Nastran Training, October 2014

