

Summary of New Features

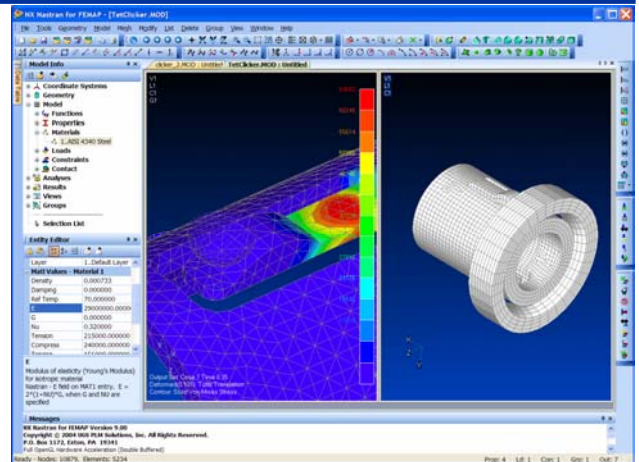
- **New GUI**
 - Floating or Dockable Fully Customizable Toolbars
 - Multiple Model Interface
 - Model Info Tree
 - Entity Editor
 - Entity Selector Toolbar
 - Data Table
- **NX Nastran for Femap**
 - Support for new NX Nastran Iterative Solver
 - NX Nastran 3.0 Advanced Nonlinear Solver
 - XDB Output Reader
- **Other New Features**
 - Meshing Enhancements
 - Geometry Enhancements
 - Analysis Program Updates

More Information available via the web:

Femap Home Page
Femap v9 Demo Video

Femap v9 Introduces New GUI

Femap v9 includes an all new Microsoft Office like Graphical User Interface (GUI). The focus of the new interface is to provide streamlined, direct access to Femap functionality. All elements of the new interface are optional. Existing Femap users can use the software as before and learn to take advantage of the increased productivity offered in the new interface as they become more familiar with v9.



Floating Dockable Toolbars

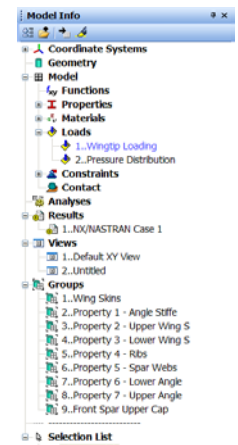
17 standard toolbars ship with Femap v9. Almost every aspect of each toolbar is fully customizable. It is possible to change the icon and text of every button, add or remove buttons, and rearrange them. New buttons can be added that access Femap API functionality or external programs.

Multiple Model Interface

New in v9 is the ability to work with multiple Femap models in the same session. Each open model can have multiple views. There are no limits on the number of windows which can be opened, although only 20 can be saved with each model. Also, based on numerous user requests – you can now cut and paste entities from one model to another!

Model Info Tree

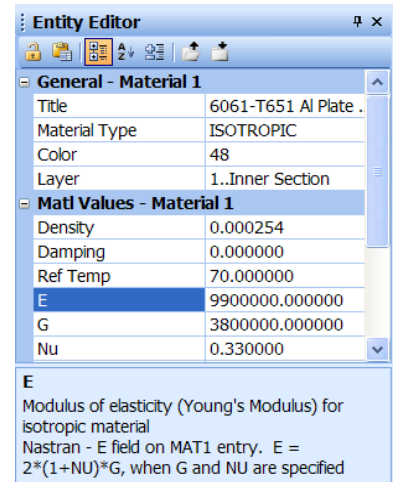
The Model Info tree provides direct access to higher level entities in your Femap model. All set-based entities, such as load sets, constraint sets, and output sets appear in the tree. Through right-click access, entities can be activated, listed, renamed, deleted, etc. Specific functionality has been added to highlight entities in a group, as well as optionally select and combine multiple groups. Selecting individual items in the Model Info Tree also loads them into the Entity Editor where details can be edited and your model updated more efficiently.



Entity Editor

Through toolbar selection or via the Model Tree, the Entity Editor is automatically loaded. In the sample on the right, a material has been selected and is available for review or editing. Content appropriate help is displayed at the bottom the Entity Editor for each individual field. Entity information can be copied to the clipboard and pasted into other Windows applications with full retention of formatting. A sample is shown below.

General - Material 1	
Title	6061-T651 Al Plate .25-2.
Material Type	ISOTROPIC
Color	48
Layer	1..Inner Section
Matl Values - Material 1	
Density	0.000254
Damping	0.000000
Ref Temp	70.000000
E	9900000.000000
G	3800000.000000



Toolbar Entity Selector

One of Femap v9's new toolbars is an advanced entity selector. Femap entities can now be selected from the top level of the GUI, before a command is chosen. Future commands can also use the pre-selected entities instead of prompting you to select again. The selector includes options to automatically select associated entities (nodes on an element, the material and property associated with an element, etc.) Once a selection has been made, it can be automatically turned into a Femap group.

Data Table

The Femap v9 Data Table provides a convenient grid based tabular representation of model data. The Data Table gets filled-in through selections made using the Entity Selector. Once populated, the table can be sorted by any column and grouped. Columns can be hidden or removed, and columns of Output Data associated with selected entities can be added.

In addition, rows can be deleted, and the Selection Set updated to use only the remaining rows.

ID	Prop ID	Eleme...	Element Topology	Orient...	Orientation Vector	Color	Layer	Formul...	C1	C2	C3	C4
266	1..Angle Stiffener	BEAM	Line2	0	0., 0., 1.	124	3	0	204	10		
265	1..Angle Stiffener	BEAM	Line2	0	0., 0., 1.	124	3	0	200	204		
264	1..Angle Stiffener	BEAM	Line2	0	0., 0., 1.	124	3	0	7	200		
263	1..Angle Stiffener	BEAM	Line2	0	0., 0., 1.	124	3	0	6	7		
115	3..Lower Wing Skin	PLATE	Quad4			124	3	0	219	220	225	224
111	3..Lower Wing Skin	PLATE	Quad4			124	3	0	121	120	220	219
33	1..Angle Stiffener	BEAM	Line2	0	0., 1., 0.	124	3	0	363	312		

NX Nastran for Femap

Introduced with Femap v8.3, NX Nastran for Femap adds the time-proven NASTRAN solver to your Femap Pre- and Post-Processor. NX Nastran for Femap v9 includes version 3 of the NX Nastran solver. V3 of NX Nastran adds Advanced Nonlinear Analysis and a fast Iterative solver to the most robust finite element solver available today.

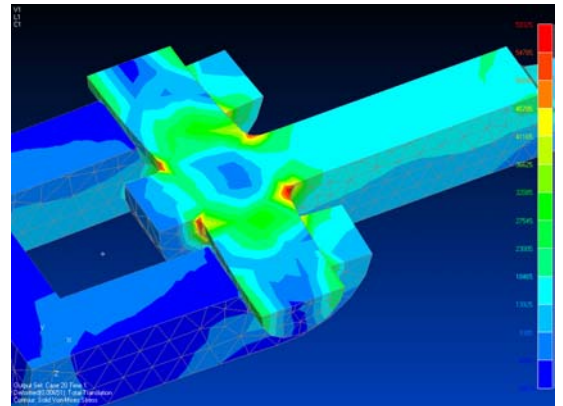
NX Nastran Version 3 Iterative Solver

The latest version of NX Nastran for Femap now includes a powerful Pre-Conditioned Conjugate Gradient (PCG) solver. Available for Linear Static analyses, the PCG solver in NX Nastran v3 can solve large tetrahedral models up to 12 times faster than other versions of NASTRAN.

Advanced Nonlinear

New in this release is the availability of the NX Nastran for Femap Advanced Nonlinear module. The Advanced Nonlinear Module adds powerful, easy-to-use nonlinear functionality to Femap. Some of the features include:

- Nonlinear Elastic-Plastic Material Properties
- Large Deformation Analysis
- Rigid Contact
- Deformable Contact
- Tied Contact
- Extensive control over load time stepping
- Multiple friction models



XDB Output Data Reader

See what data is available and import only what you want.

Import post-processing information for selected groups of nodes or elements

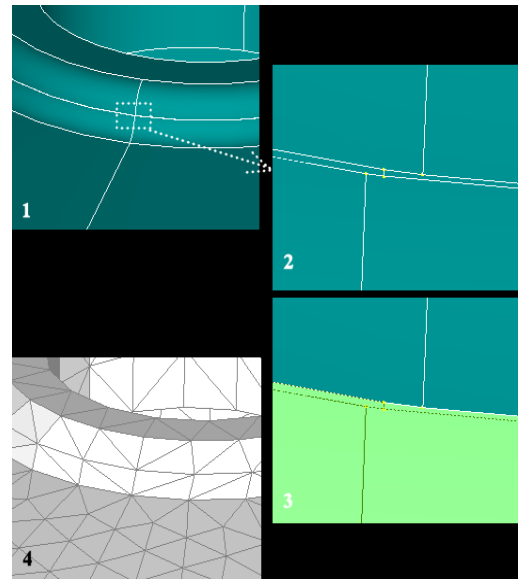
Import output data from specific sets, based on natural frequency, time step, or output set.

Other New Features

Meshing Enhancements

New "auto-boundary" functionality groups sliver surfaces with adjacent surfaces to effectively remove the sliver surface from the mesh. In this example:

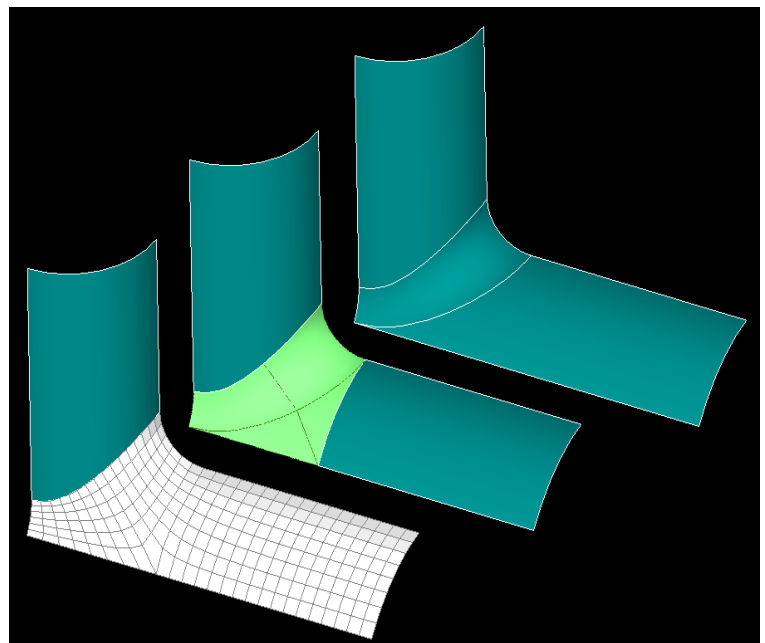
- 1) everything looks good from far away
- 2) up close we find extremely short edges and sliver surfaces
- 3) during mesh sizing, sliver surfaces are automatically connected to neighbors and short edges are suppressed
- 4) a quality mesh results.



Geometry Enhancements

The following picture, from right to left, shows:

- 1) surfaces being split between two points
- 2) surfaces being combined in new meshing regions, and
- 3) the resulting mesh.



Analysis Program Updates

Femap v9 now supports:

NX Nastran 3.0
ABAQUS 6.4
MSC.Marc 2005

MSC.Nastran 2005
NE/Nastran 8.4
ANSYS 8.0