Femap v11.1.2 and NX Nastran v9.1

Technical Seminar for Femap and NX Nastran Users

Hosted by: Adrian Jensen, BSME, P.E., Sr Staff Mechanical Engineer

George Laird, PhD, PE, Principal Mechanical Engineer

FINITE ELEMENT ANALYSIS
Predictive Engineering

Applied **CAx**
We do this *every day*

FEA, CFD & LS-DYNA Training, Support and Consulting

Siemens NX CAD, CAM, CAE, Teamcenter and Femap and NX Nastran Sales

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.

FINITE ELEMENT ANALYSIS
**Predictive Engineering**
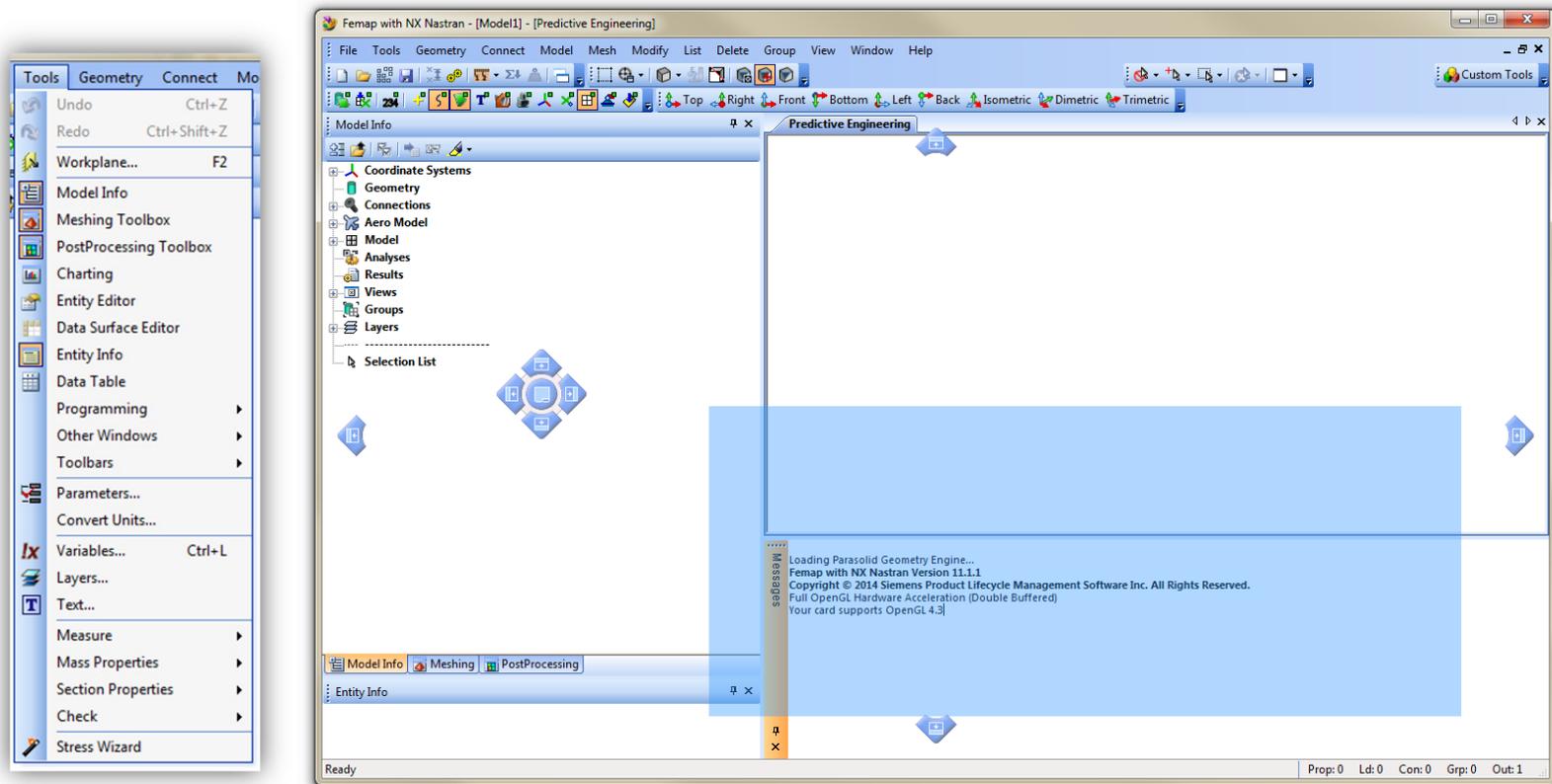
1.2   TOOLBARS

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.
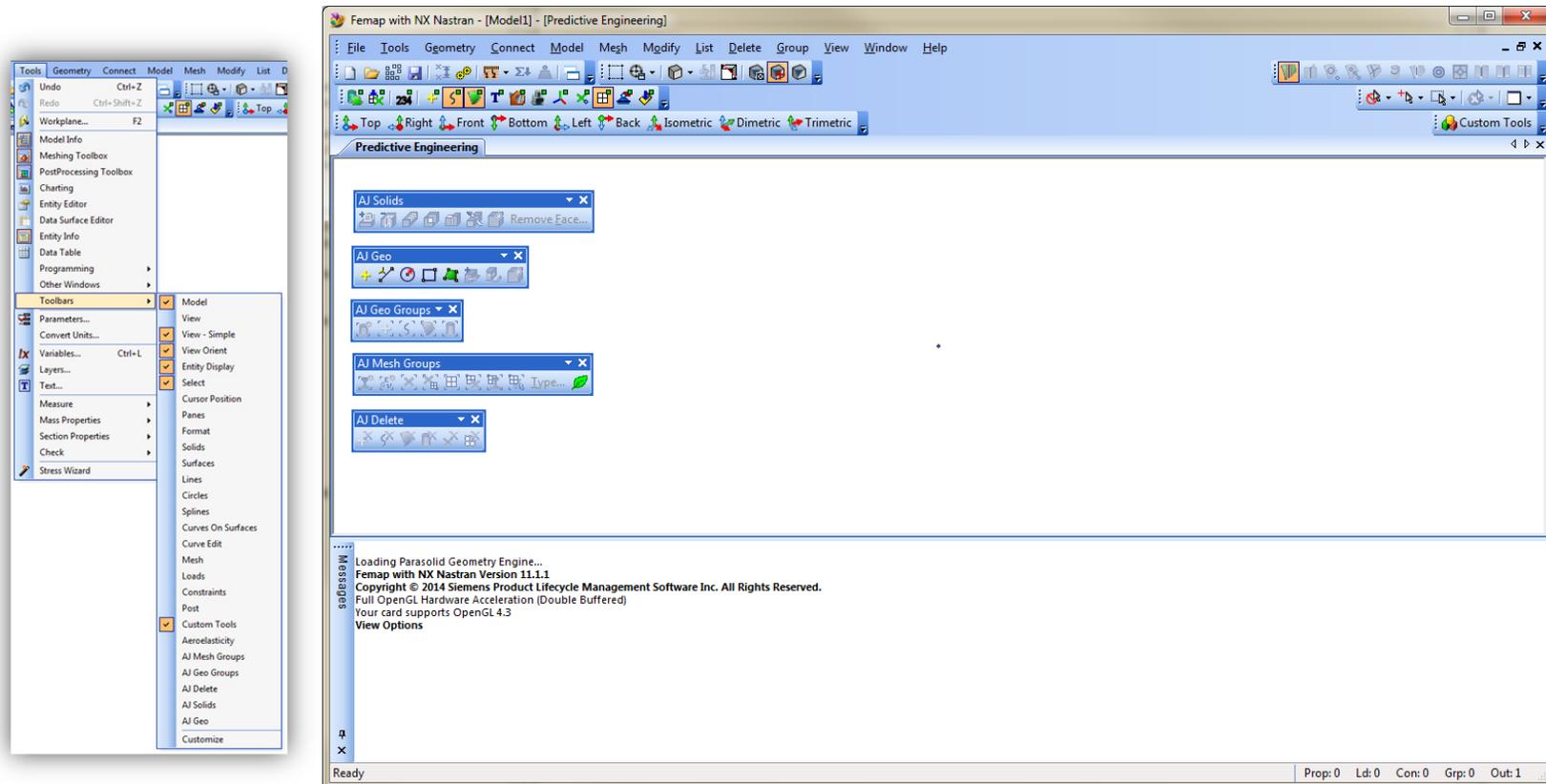
1.3   PREFERENCES

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.
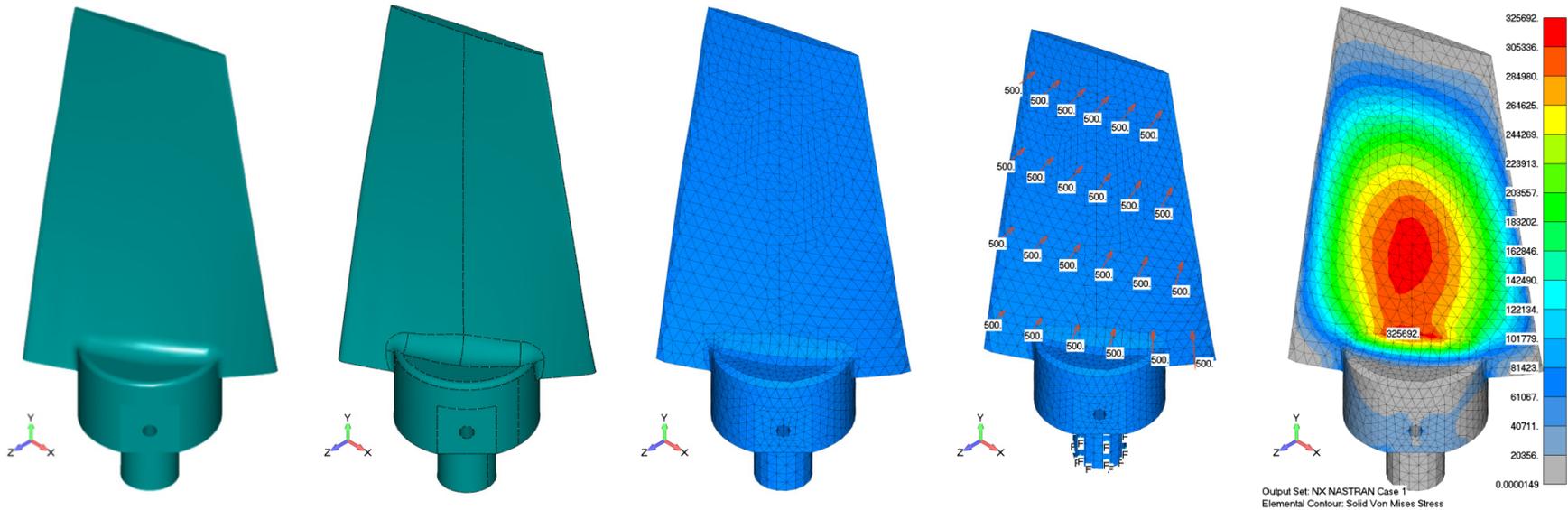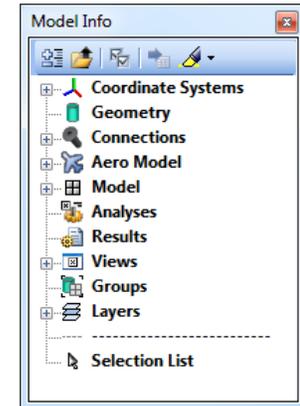
FINITE ELEMENT ANALYSIS
Predictive Engineering
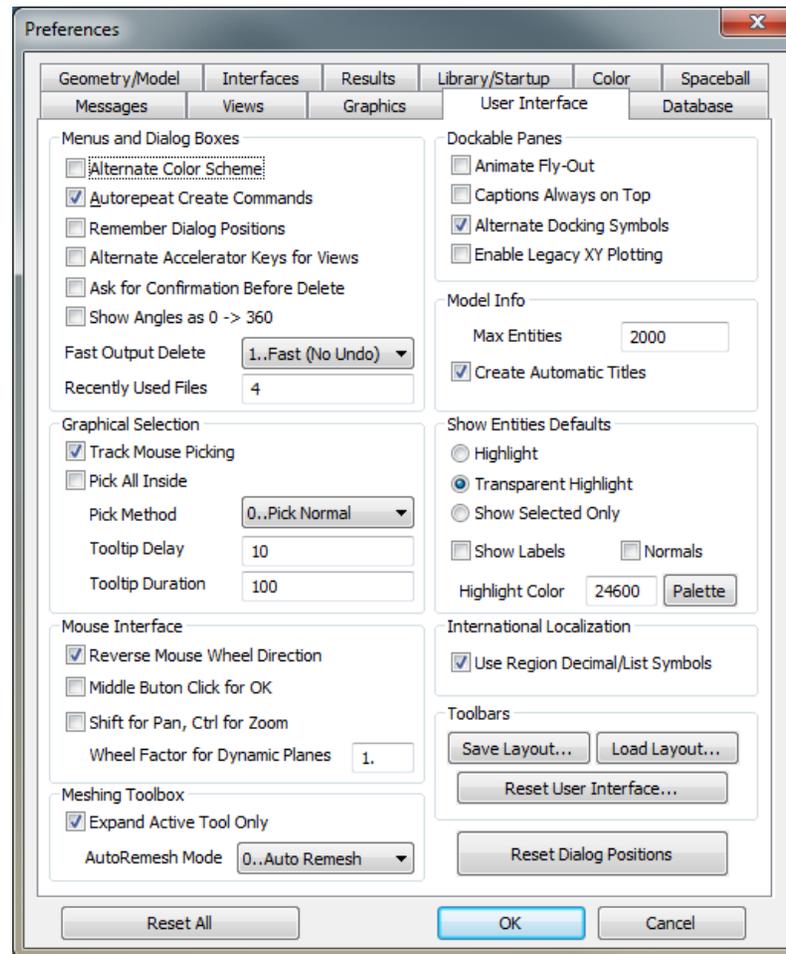
2. ANALYSIS WORKFLOW

- Geometry
- Material
- Property
- Mesh Sizing
- Meshing
- Loads
- Constraints
- Analyze
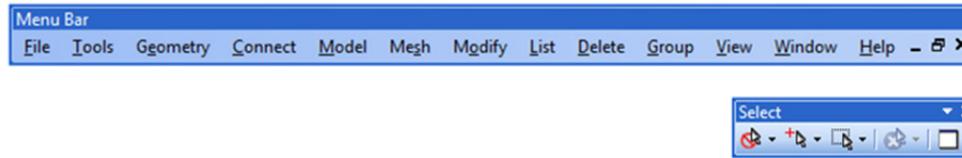
Regardless of the technique used (Menus, Tree, Selector), the workflow always follows the same sequence.


Output Set: NX NASTRAN Case 1
Elemental Contour: Solid Von Mises Stress

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.*
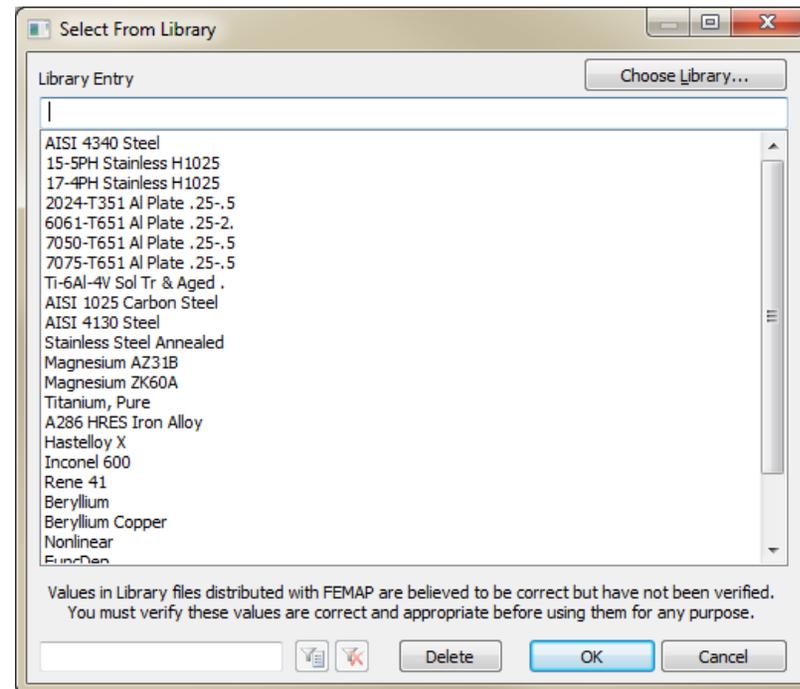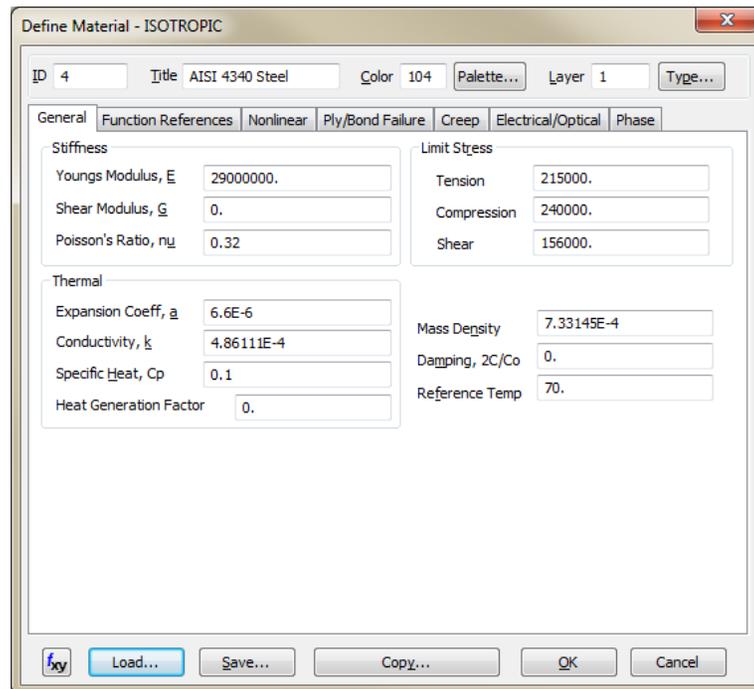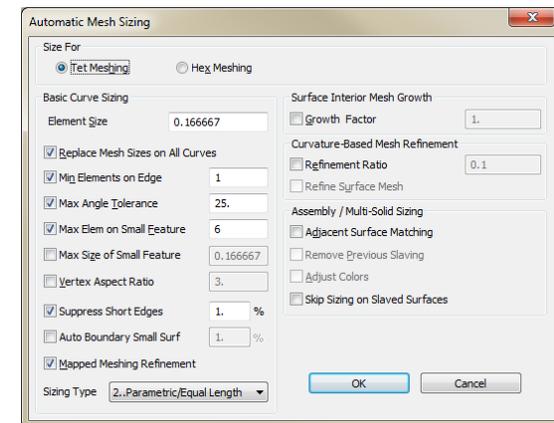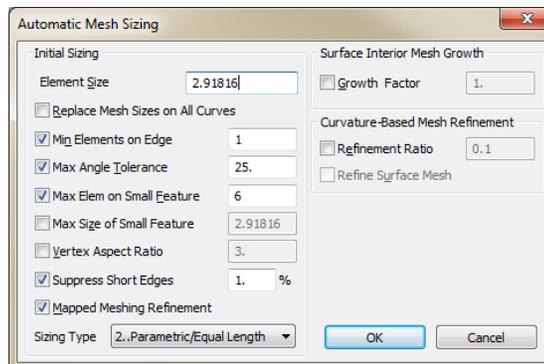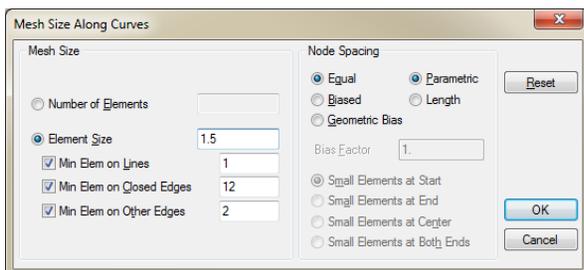
2.3 PROPERTY

Generate property types appropriate for your geometry.
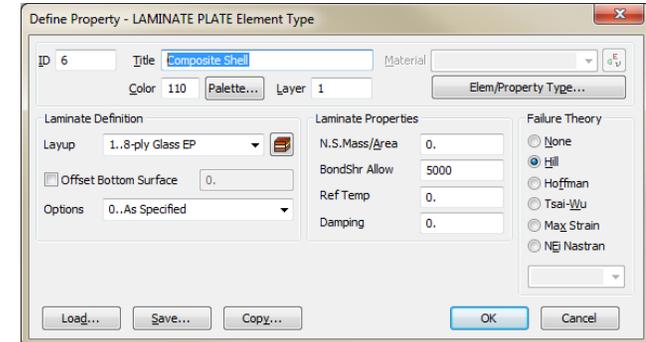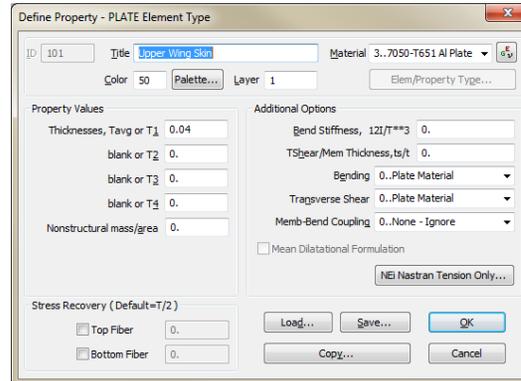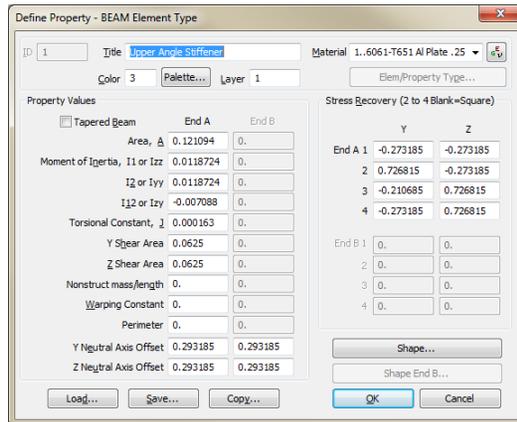

2.4 MESH SIZING

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

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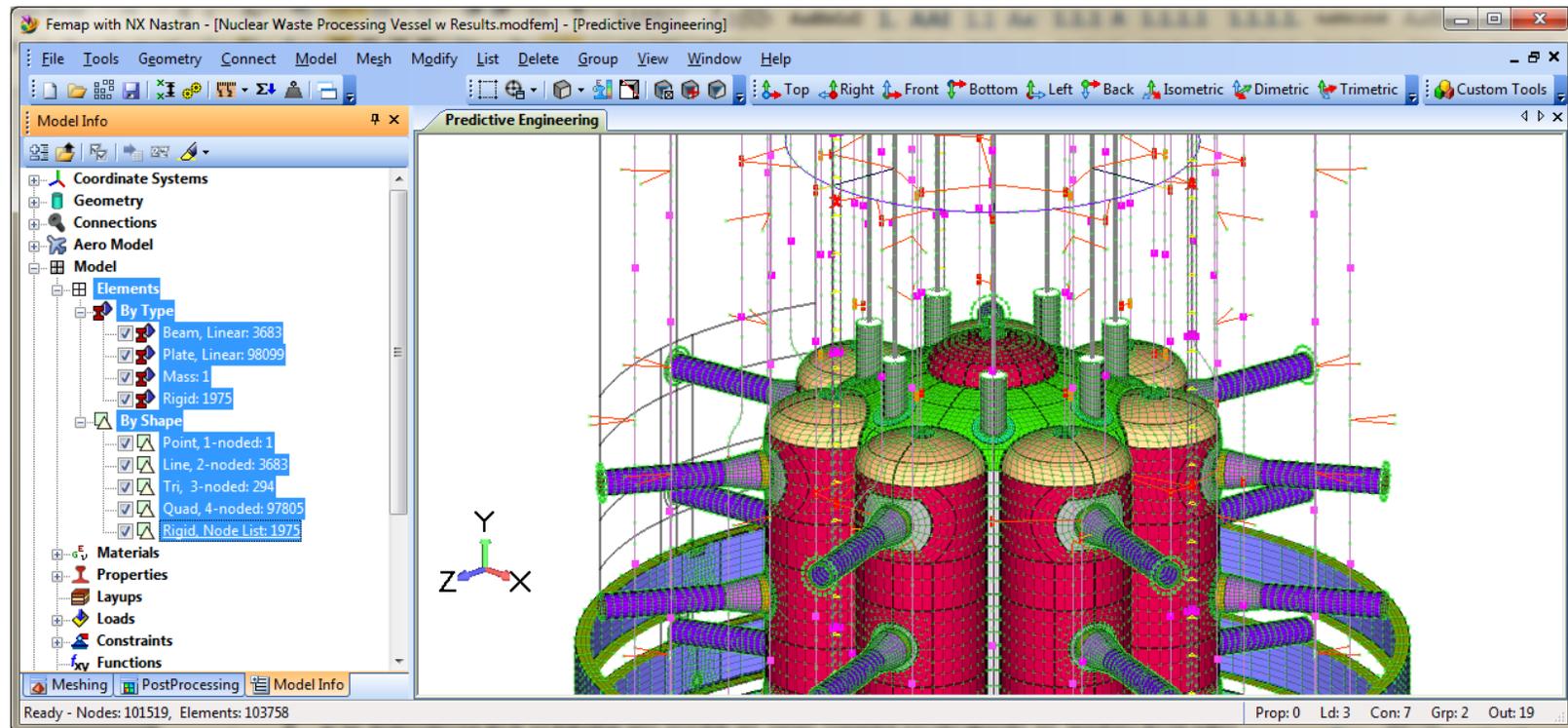
2.6   LOADS
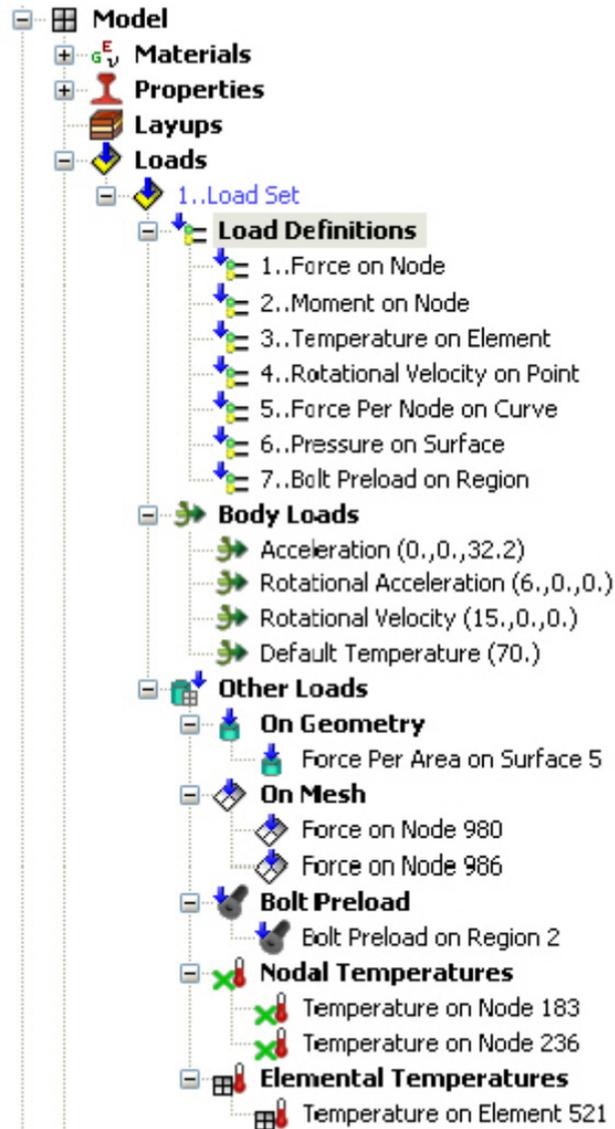
Every time a load is created on finite element entities (i.e., Model, Load, Nodal; Model, Load, Nodal on Face; and Model, Load, Elemental) or geometry (Model, Load, On Point; Model, Load, On Curve; Model, Load, on Surface) a "Load Definition" will also be created in Femap. A Bolt Preload will also create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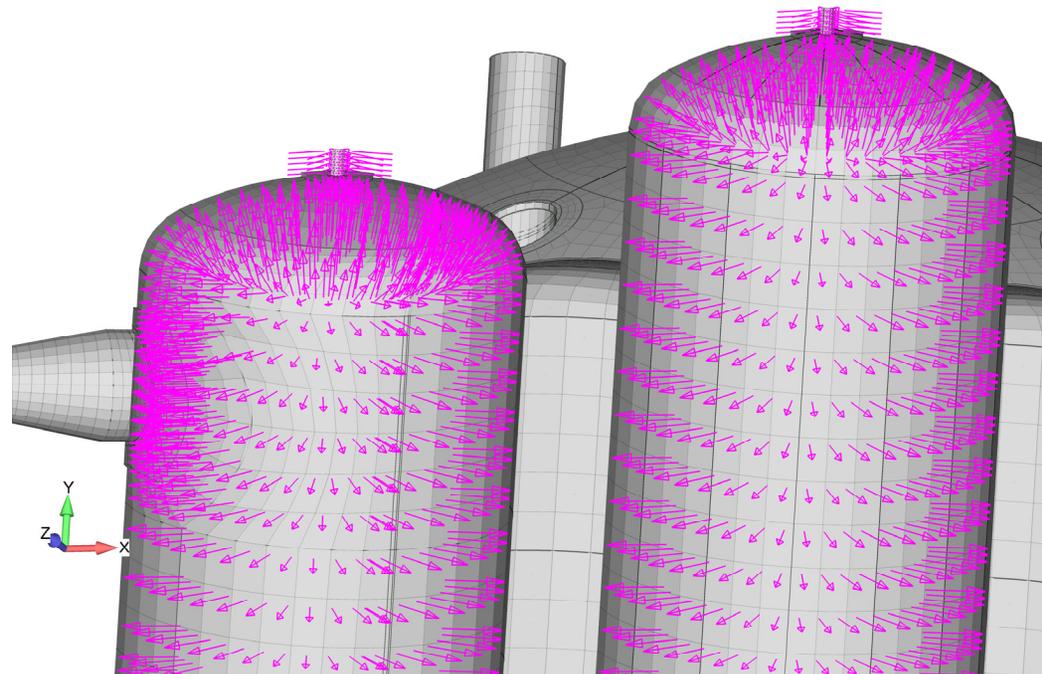
2.7 CONSTRAINTS

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.
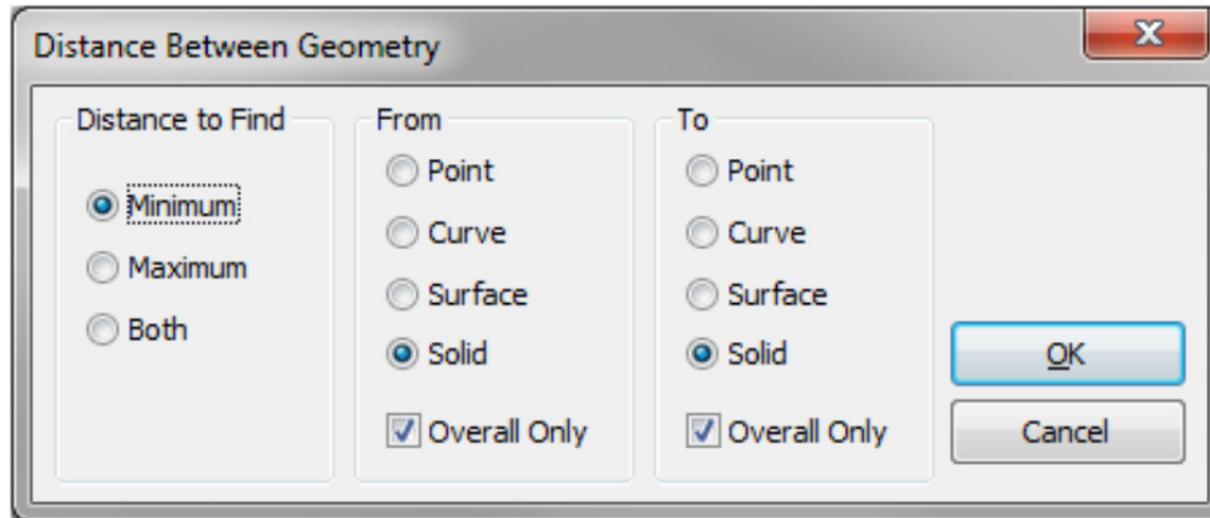
FINITE ELEMENT ANALYSIS
Predictive Engineering
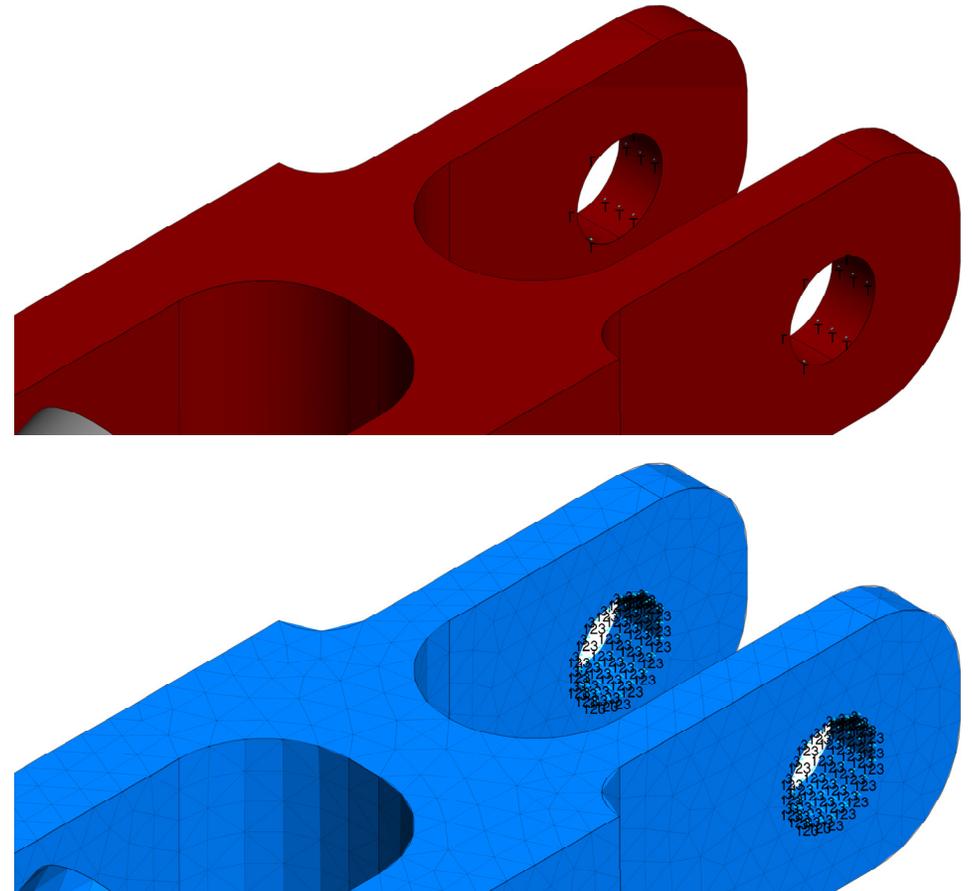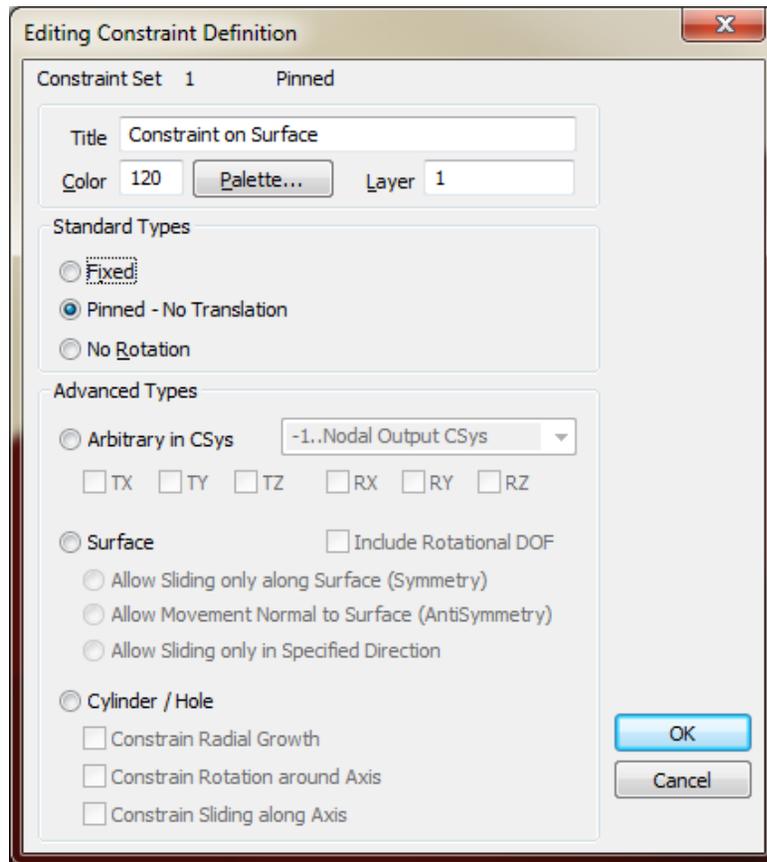
2.8   ANALYZE

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

Applied CAx
We do this every day
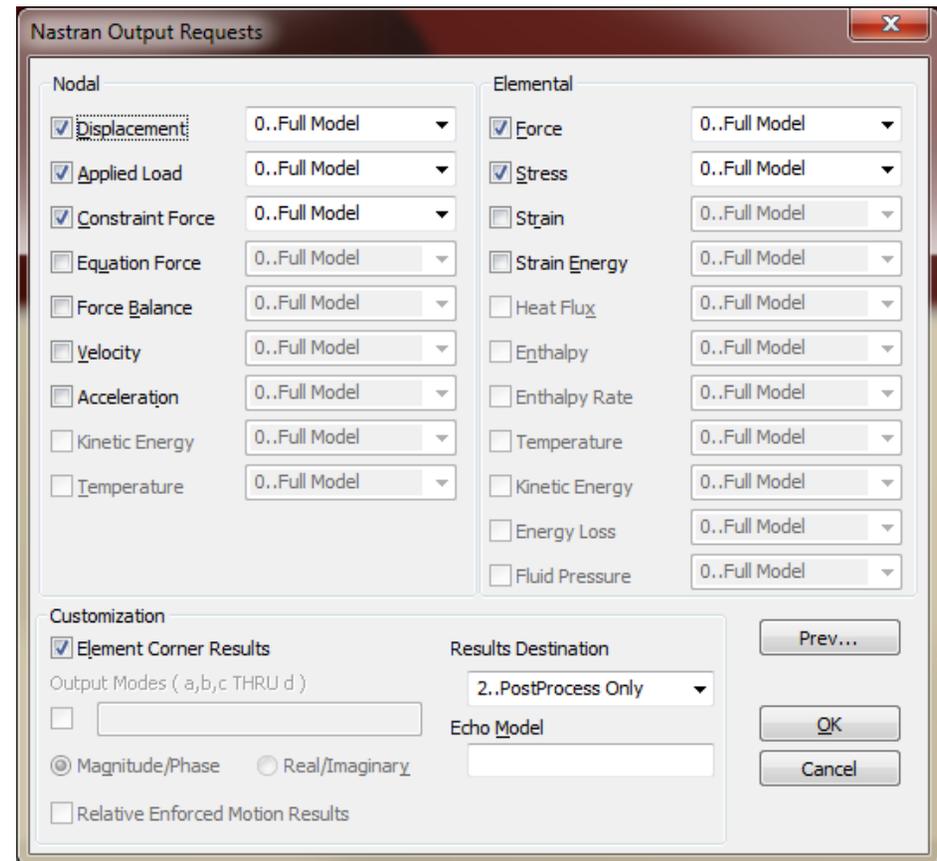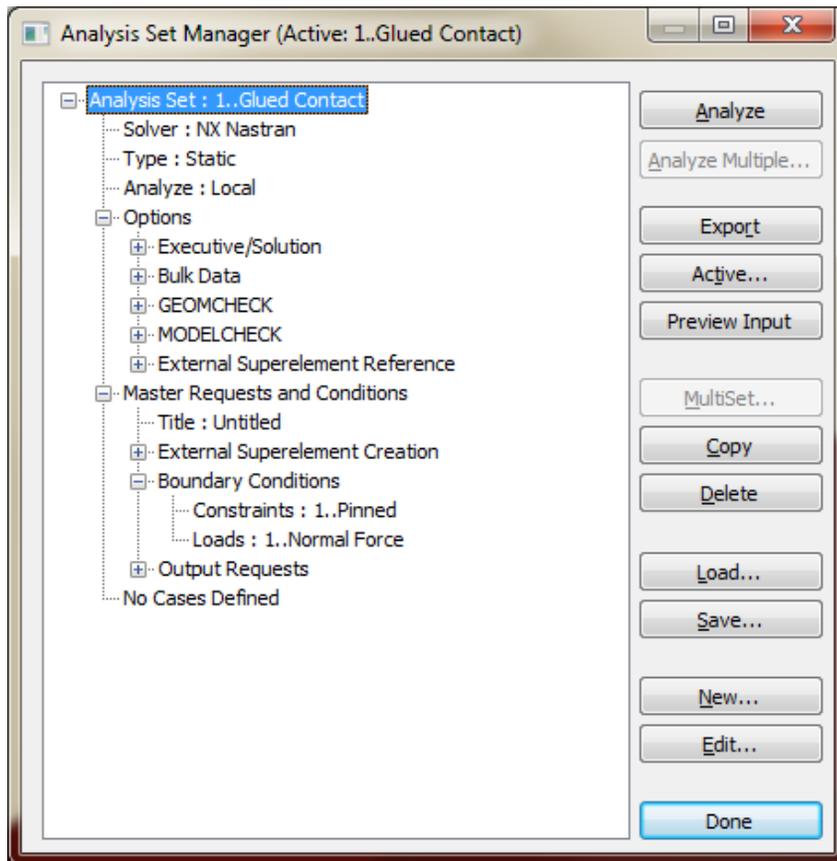
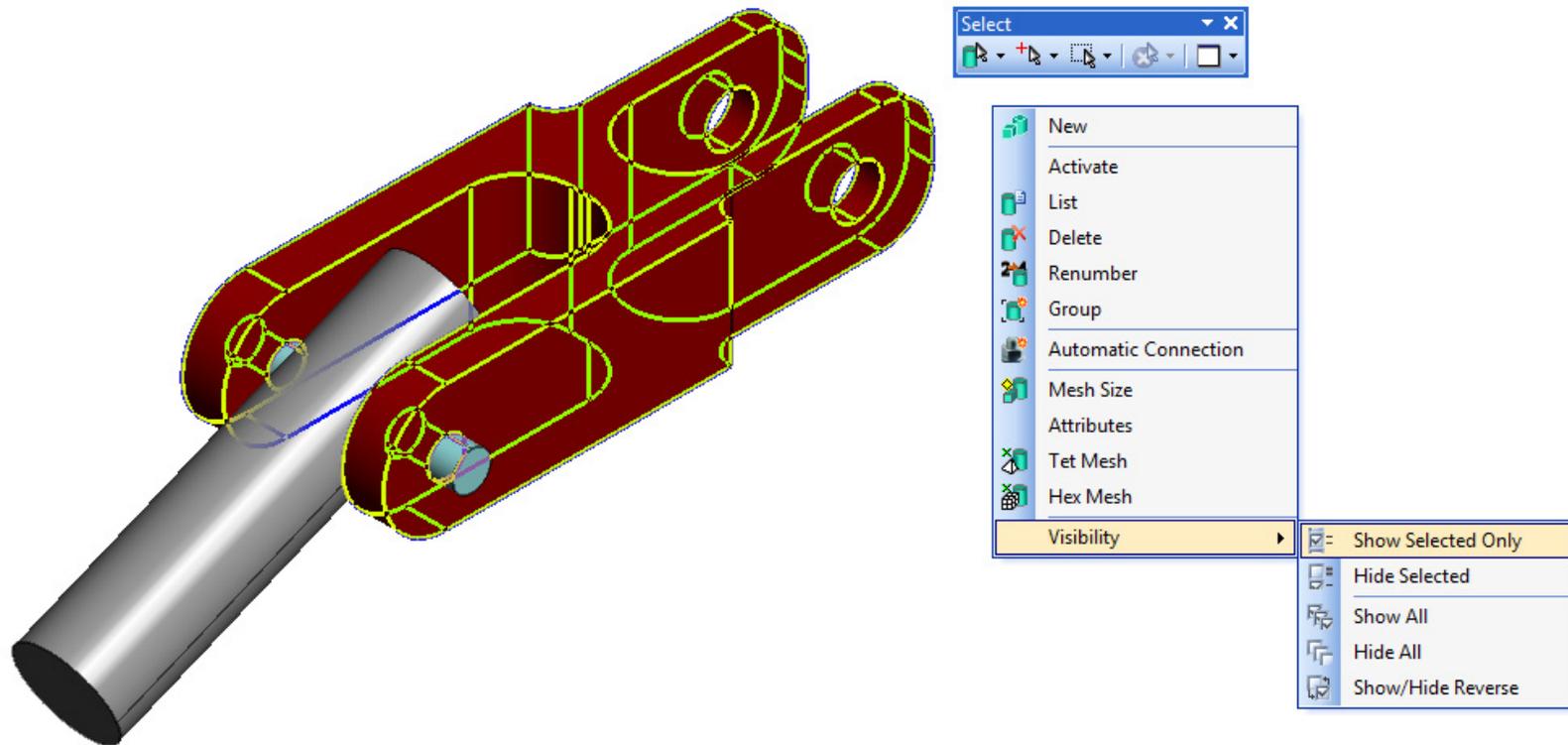# 3. TIP, TRICKS, NEW FEATURES

3.1   MEASURE BETWEEN GEOMETRY

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

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.
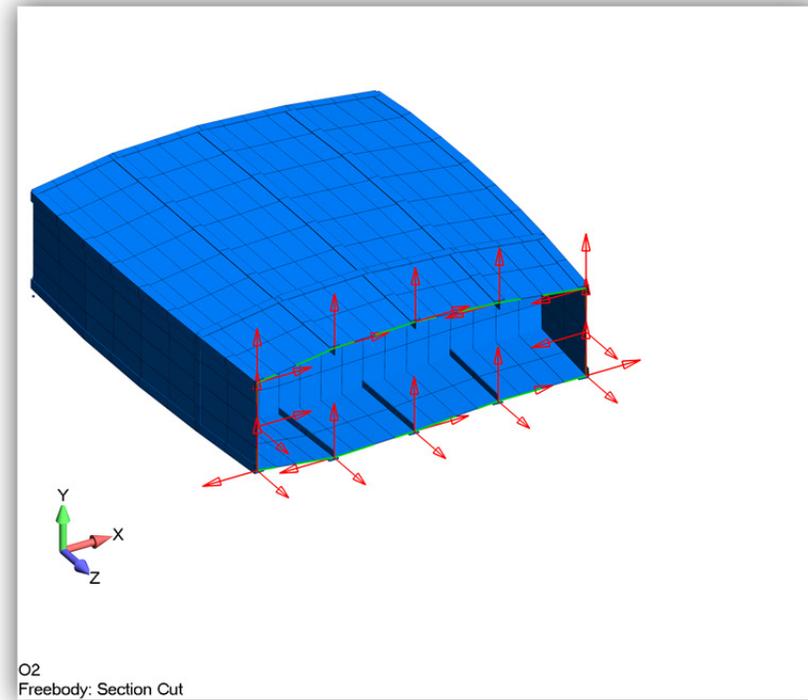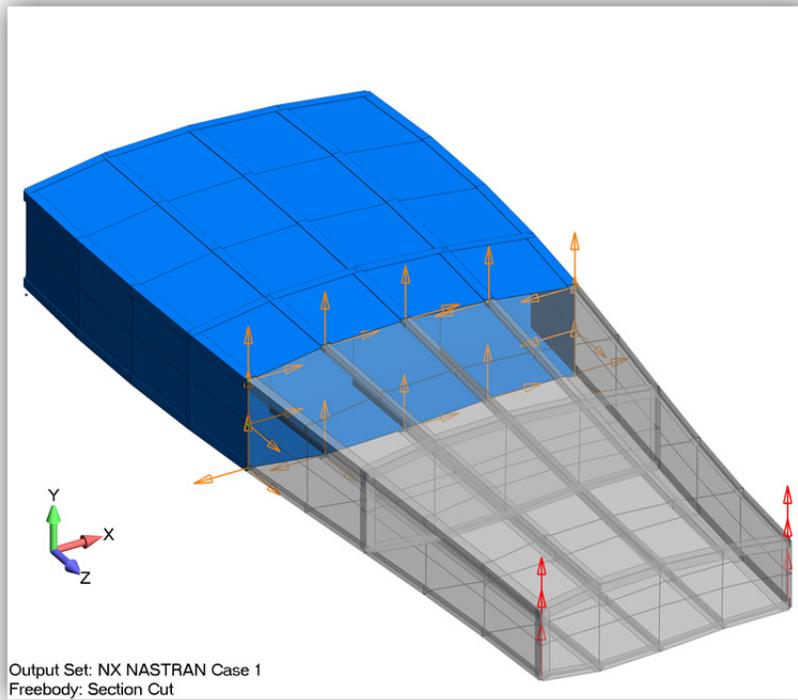
3.4   MODEL MERGE

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.
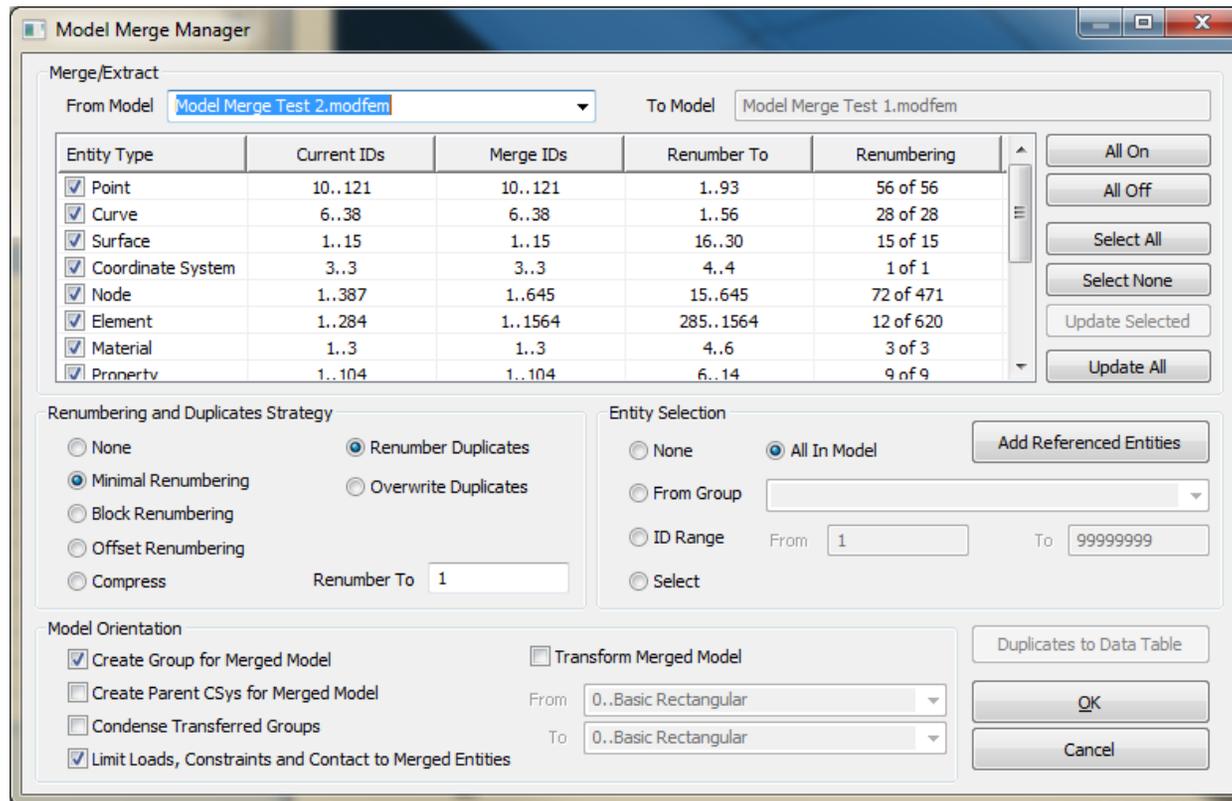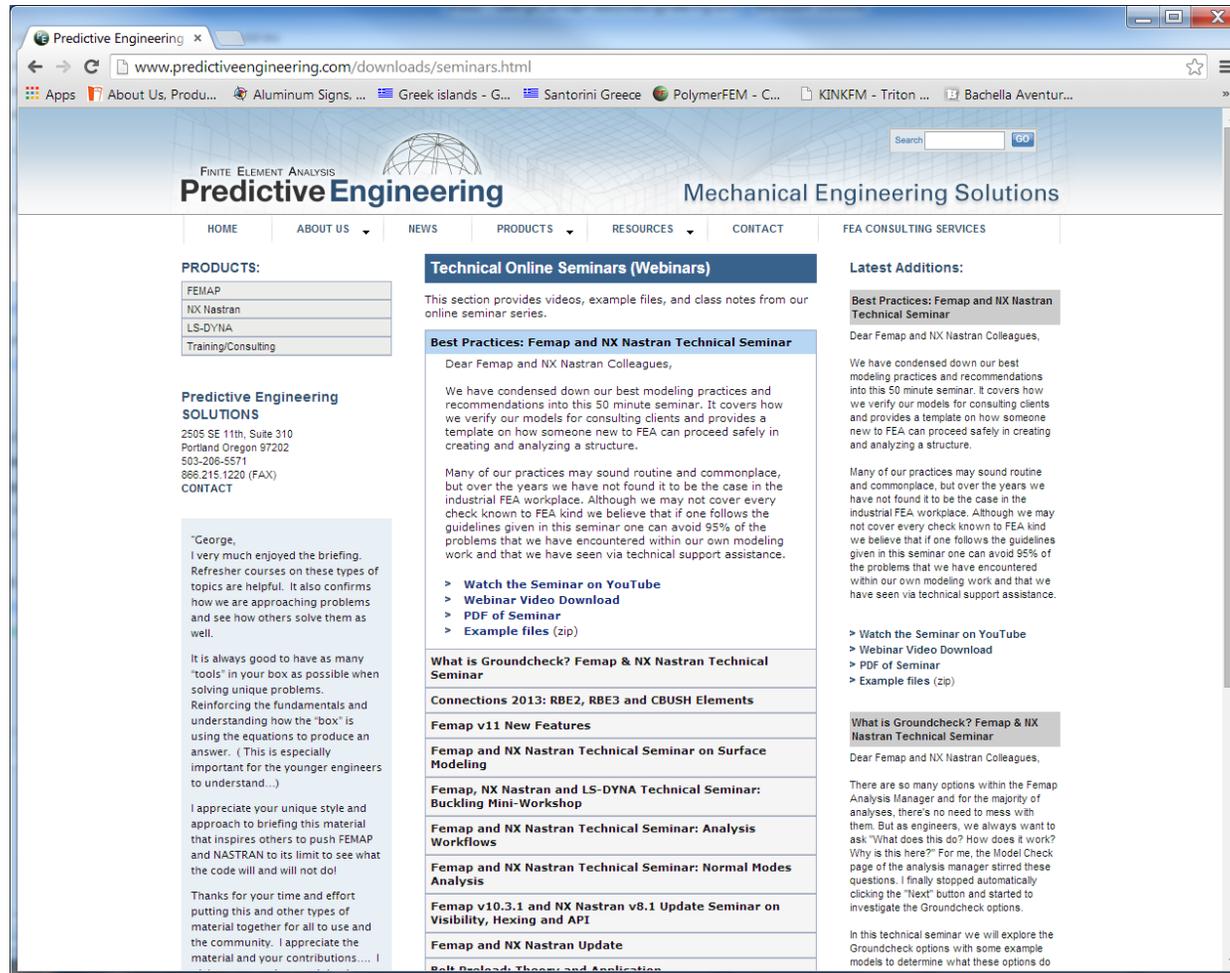
3.5   RENUMBERING ENTITIES FROM THE MODEL INFO TREE

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.
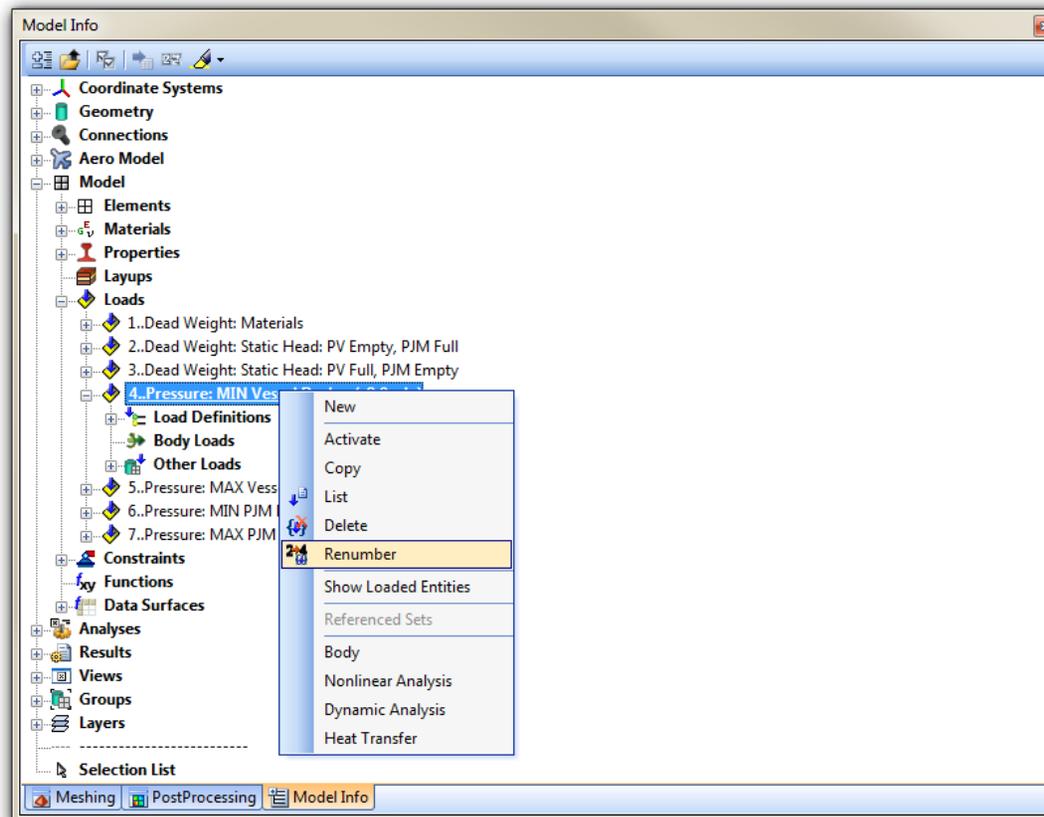
4. GAINING NEW SKILLS

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