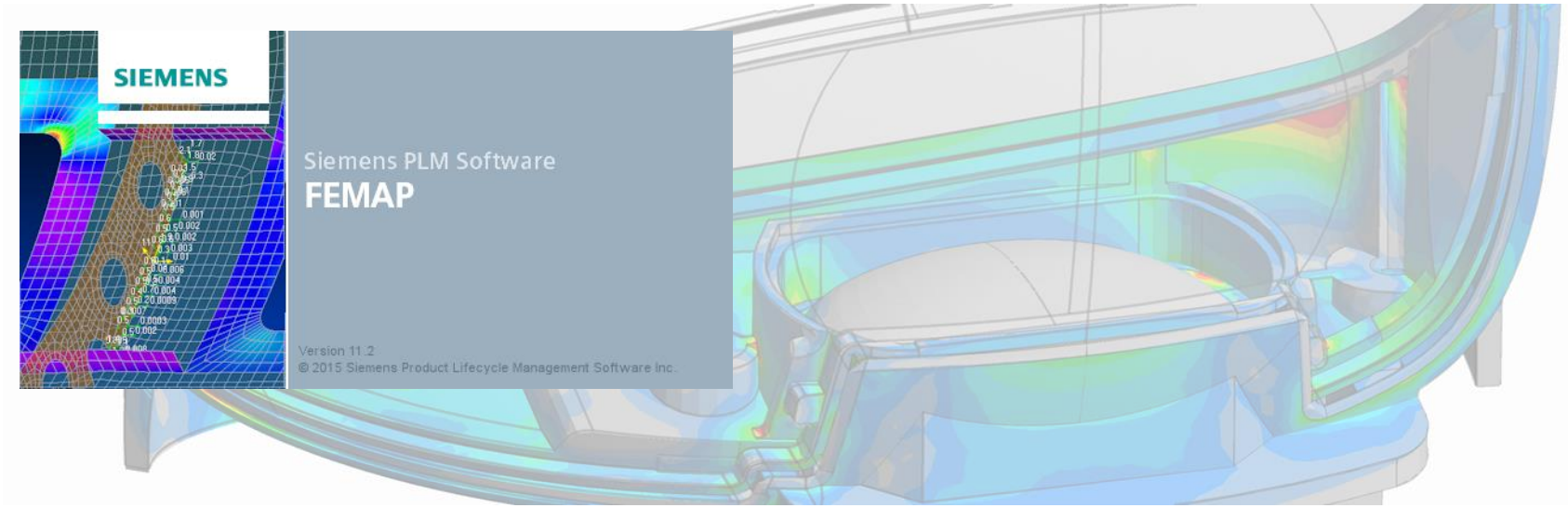


Advanced Post Processing

A Seminar for FEMAP v11.2.2 Users

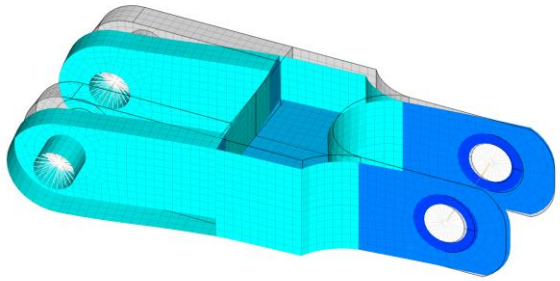
Adrian Jensen, PE | Senior Staff Mechanical Engineer

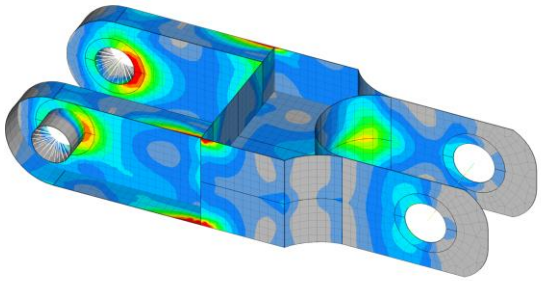
TABLE OF CONTENTS

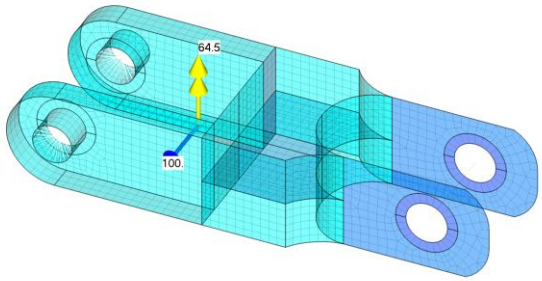
There's a lot of hard work that goes into getting a quality set of FEA results – geometry prep, meshing, materials, properties, boundary conditions, contact and analysis settings – and that's if everything goes smoothly. But what do you do when the analysis results are finally ready? For some, that's when the real work begins. Post processing is the art of taking a mountain of data and turning it into something easy to present and easy to understand. It has been our experience that a well-crafted combination of images and tabular data will quickly make your point and engage the reviewer far better than paragraphs of text..

In this seminar, we will start with file management. Although a bit boring, it is a critical first step, especially if you are working with gigabytes of data. We will discuss where the output data goes; in which files and where those files are stored. Additionally, we will discuss how your FEMAP model accesses that data.

Next, we will take a tour of the PostProcessing Toolbox. Any experienced FEMAP user will know their way around the Toolbox but it is a worthwhile talking point because it's always changing and improving. More so, if you are like some of the FEMAP veterans in our office, it is time to let go and embrace the new post-processing workflow. We will explore "post" outside of the Toolbox by exploring the Data Table. This is our preferred method for collecting tabular output data for nodes and elements and is often overlooked. Finally, we will move on to customized and automated post with the FEMAP API. We will use existing APIs (programs, scripts, macros…whatever you want to call them) to export data to Excel, create custom output vectors and modify the view settings within FEMAP.

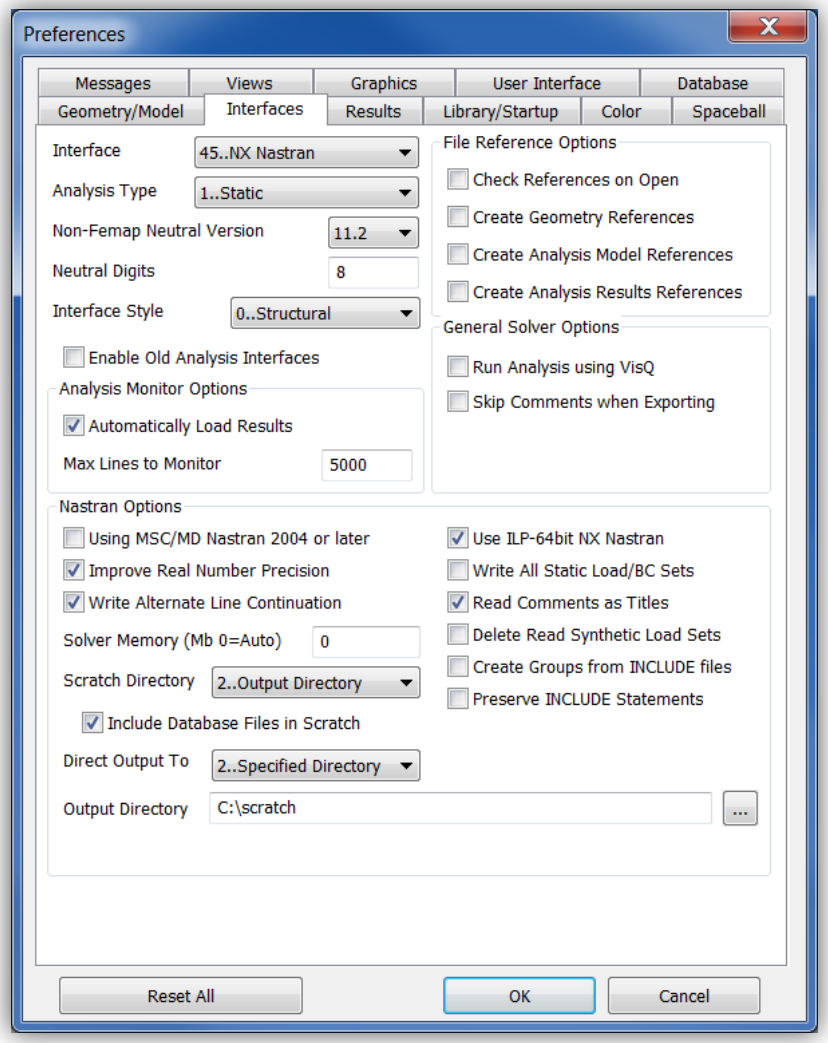






1. OUTPUT MANAGEMENT

1.1 SCRATCH AND OUTPUT FILE ORGANIZATION



Scratch Directory

Nastran Default: Directory chosen during installation to use for creating NX Nastran scratch files.

Femap Scratch: Directory specified in the Database tab of the Preferences dialog box, where the FEMAP has been directed to place the FEMAP scratch file.

Output Directory: Directory specified by the Direct Output To option on this tab of the Preferences dialog box.

Direct Output To

Current Directory: Last used directory by FEMAP. If a model has been saved to a directory, the output will be directed to that directory when this option is on.

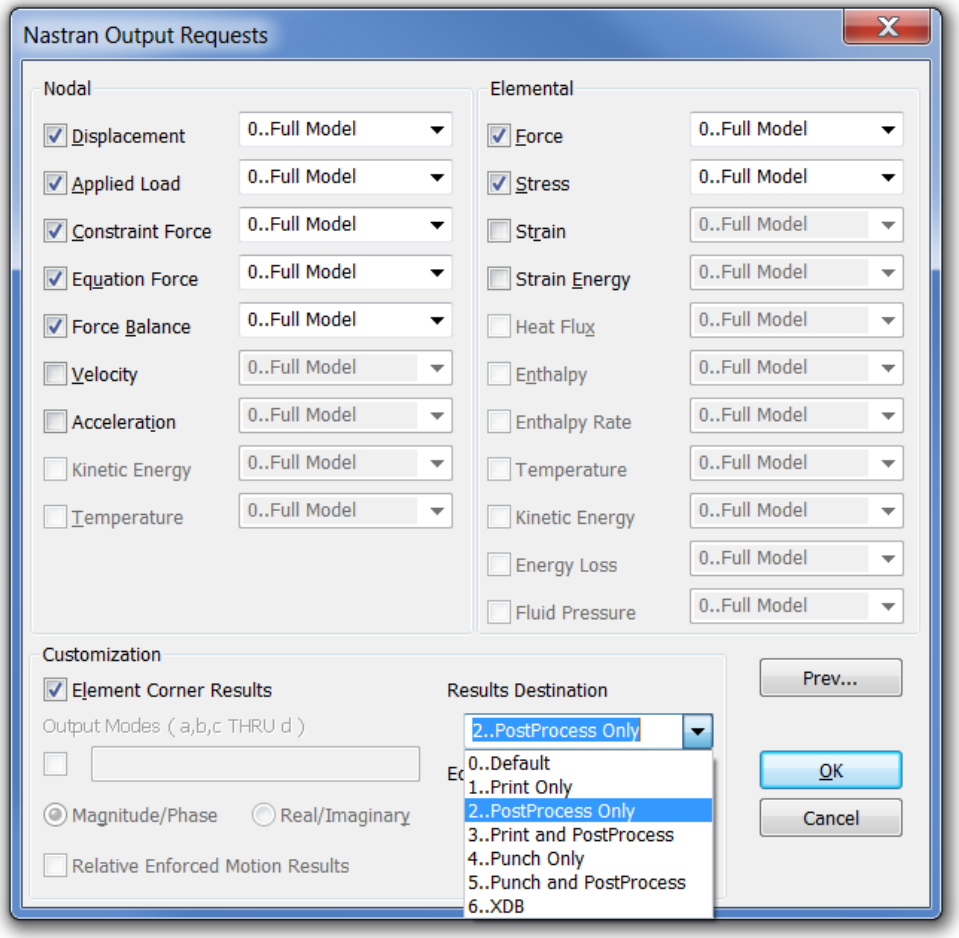
Model File Directory (default): The directory where the model file is located. All output will go into this directory until the model is saved somewhere else.

Specified Directory: This option allows you to send all NX Nastran output to a directory that you have specified. This can helpful because your output will always be in the same place if you need to view the files or “clean-up” leftover output files from old analysis runs.

1.2 NASTRAN OUTPUT REQUESTS

Results Destination

Customization also allows you to select a results destination file (PostProcess only = *.op2, Print Only = *.f06, XDB = *.xdb, etc.).



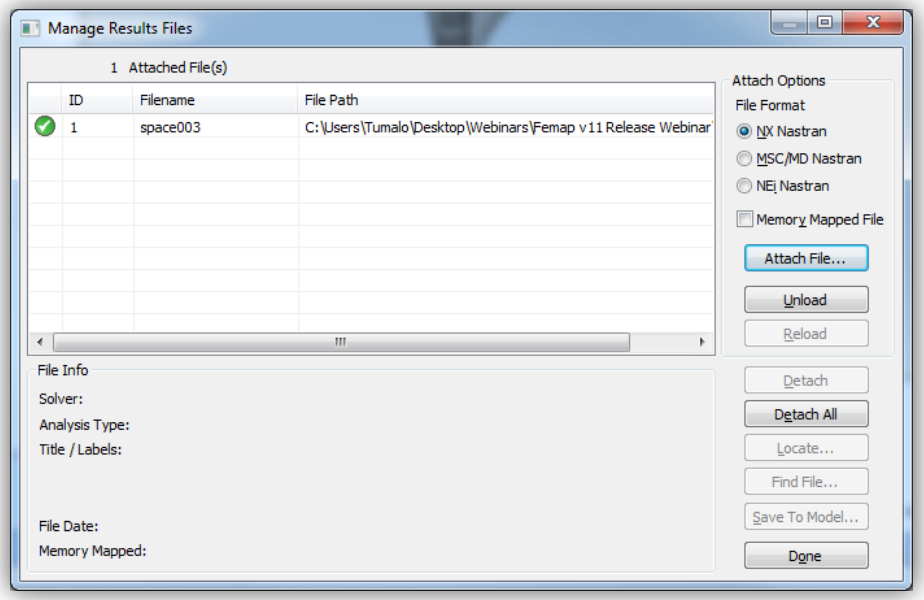
Note: When you select "3..Print and PostProcess" as the Results Destination, you are sending the results to both the .f06 and the .op2 file. Normally, you would not want to do this, but the option is there to complete all the possible combinations for requesting output. When FEMAP runs NX Nastran, it automatically reads the results (you can change this with a preference: File, Preferences, then click Interfaces, then uncheck box "Automatically Load Results"), but it does it by first reading the .f06 file. FEMAP reads the .f06 file first to attain any error, warning, or information messages that might have occurred during the analysis.

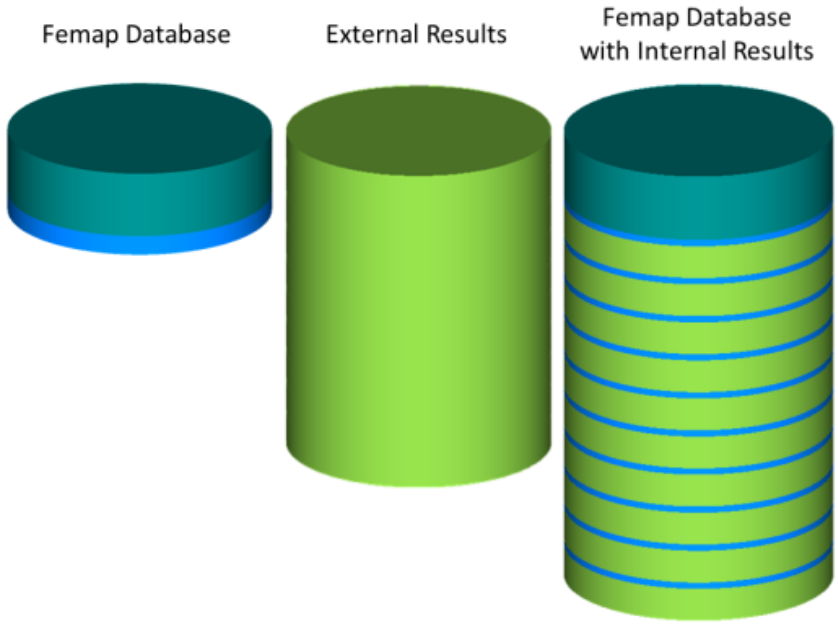
If you are requesting grid point force data to create Freebody plots in FEMAP, you must choose the "2..PostProcess Only" option or the "6..XBD" option, as the grid point force data is not in the .f06 file.

1.3 File Options

1.3.1 Attach to Results Files

Attaching to results files allows post processing to occur without "internalizing" the contents of each file into the FEMAP database. This is especially helpful when you have a large output files. In general, there are two reasons for large output files. First, a large model will typically create a large output file unless care has been taking in requesting output. Second, analysis types which create a large number of Output Sets, such as transient response, frequency response, and nonlinear analysis, can create large output files for even small and medium sized models.

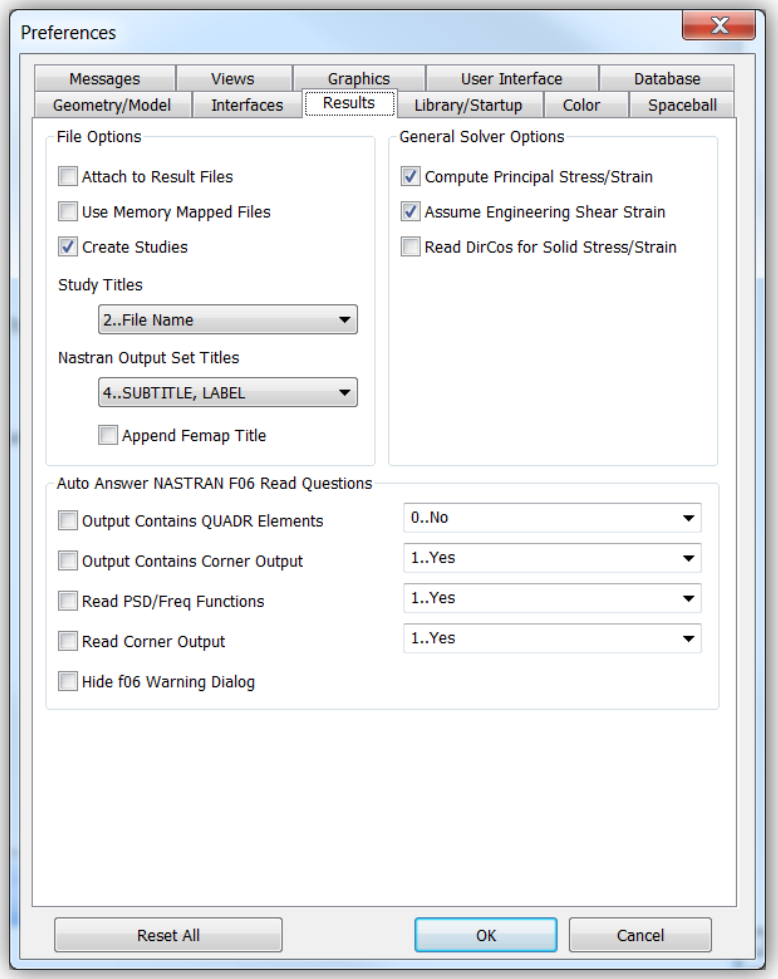


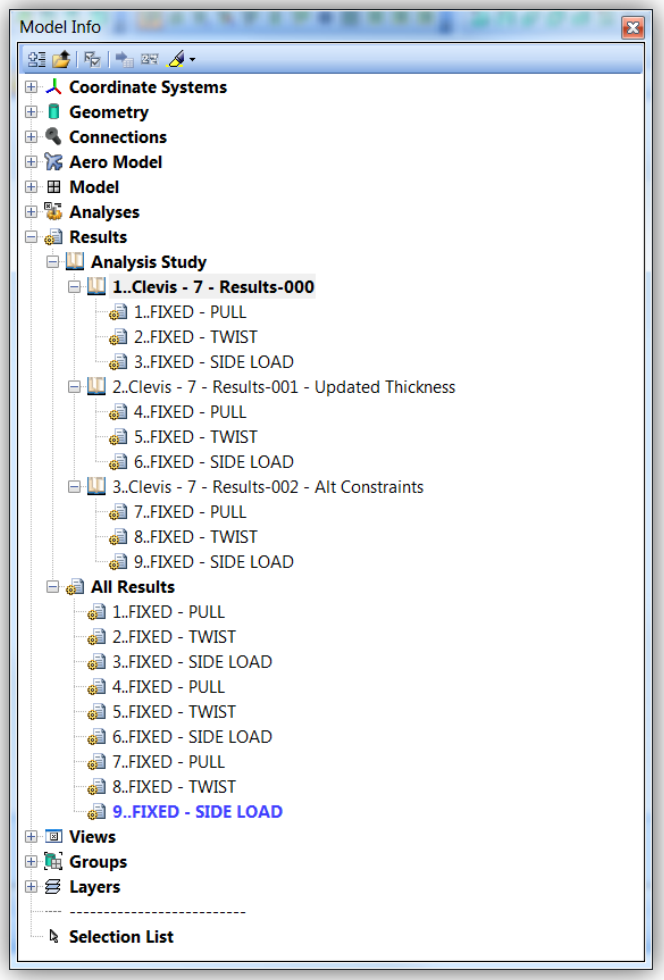


Note: Within ***File > Preferences*** you can choose to automatically attach to results files and use "Memory Mapping" for a potential increase in speed. When Memory Mapping is turned on, FEMAP will attempt to attach to the file using RAM. In order for an attached output file to be properly "Memory Mapped" it must be able to fit into a contiguous block of unused system memory.

1.3.2 CREATE (ANALYSIS) STUDIES

You can use Analysis Studies to group your output sets for better organization and data processing.





2. POSTPROCESSING TOOLBOX

2.1 DEFORM VIEW STYLE

Showing the deformed shape of your model is a standard first step in post processing. It's a quick an easy way to verify the application of your boundary conditions and see how the structure moves under load.

2.1.1 DEFORMED

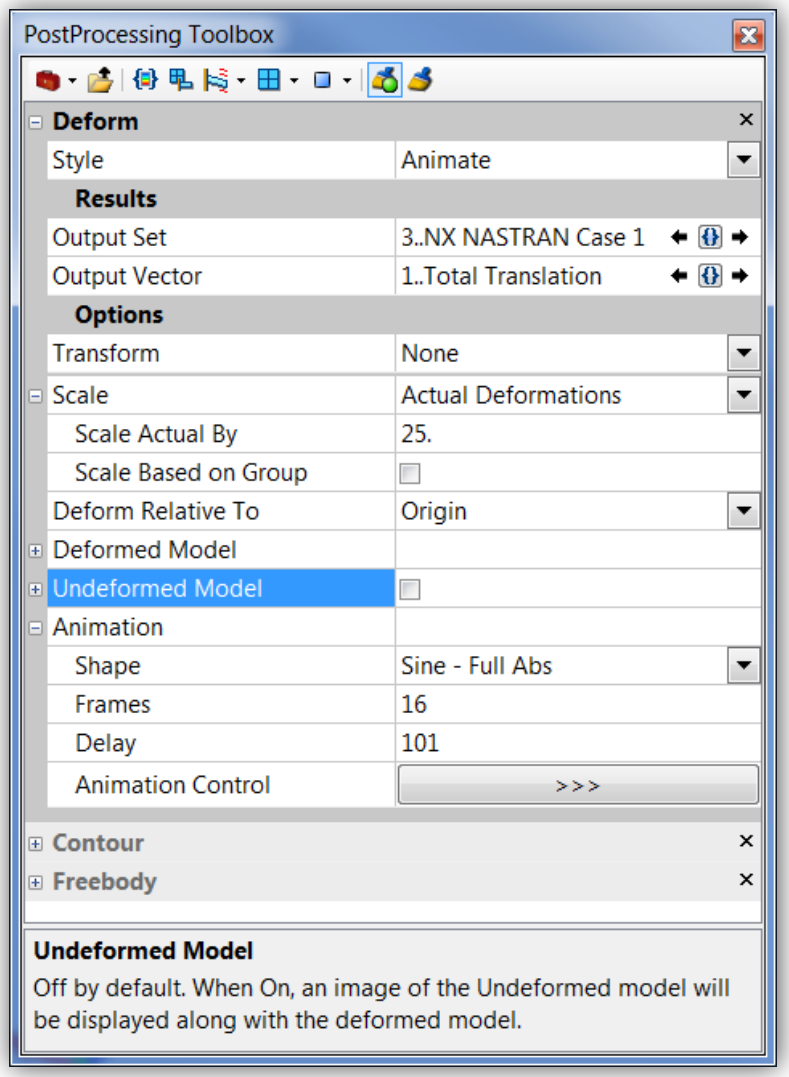
This is the standard option. It provides a static, deformed image of the structure at the chosen scale factor. Be careful when choosing output vectors; you will almost always want to use "1..Total Translation".

2.1.2 ANIMATE

As the name implies, this options animates the deformation of the model. A single output set or, in the case of nonlinear analysis, multiple sets can be animated. The user can control the speed and resolution of the animated view.

2.1.3 VECTOR

This option is useful for detailed investigation of displacements. The Vector style will display displacement vectors on each node. The user can control the magnitude and view style of the vectors.

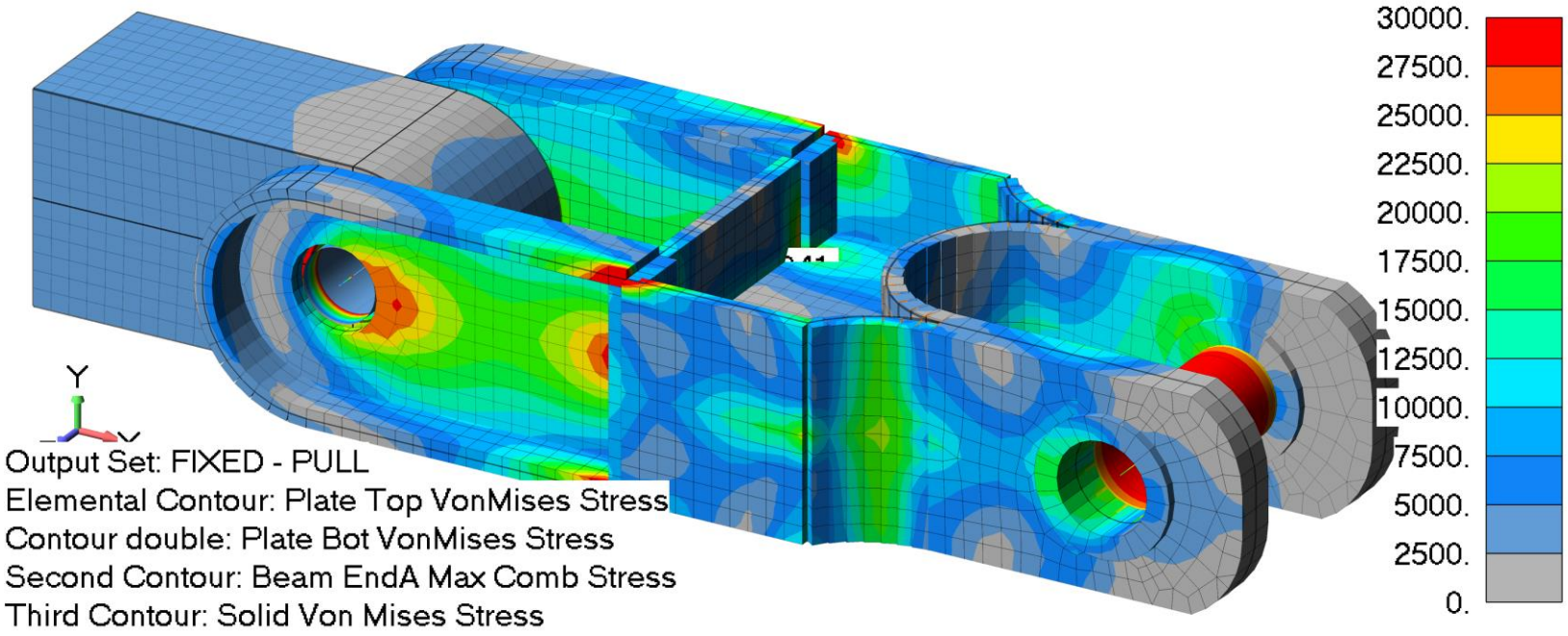


2.2 Contour View Style

A stress contour is probably the most common post processing tool for finite element analysis. Without contour plots, we wouldn't have the GIPPO acronym (Garbage In = Pretty Pictures Out). The Contour view style allows the user to display displacement, force and stress information on the mesh.

2.2.1 Contour

This is the standard option. The user chooses an output set and an output vector. For results from NX Nastran, different element types will have different output vectors (e.g., Beam EndA Max Comb Stress, Plate Top VonMises Stress, Solid Von Mises Stress). With the Contour view style, the analyst can display multiple output vectors at the same time, adjust contour colors, set legend limits and more.

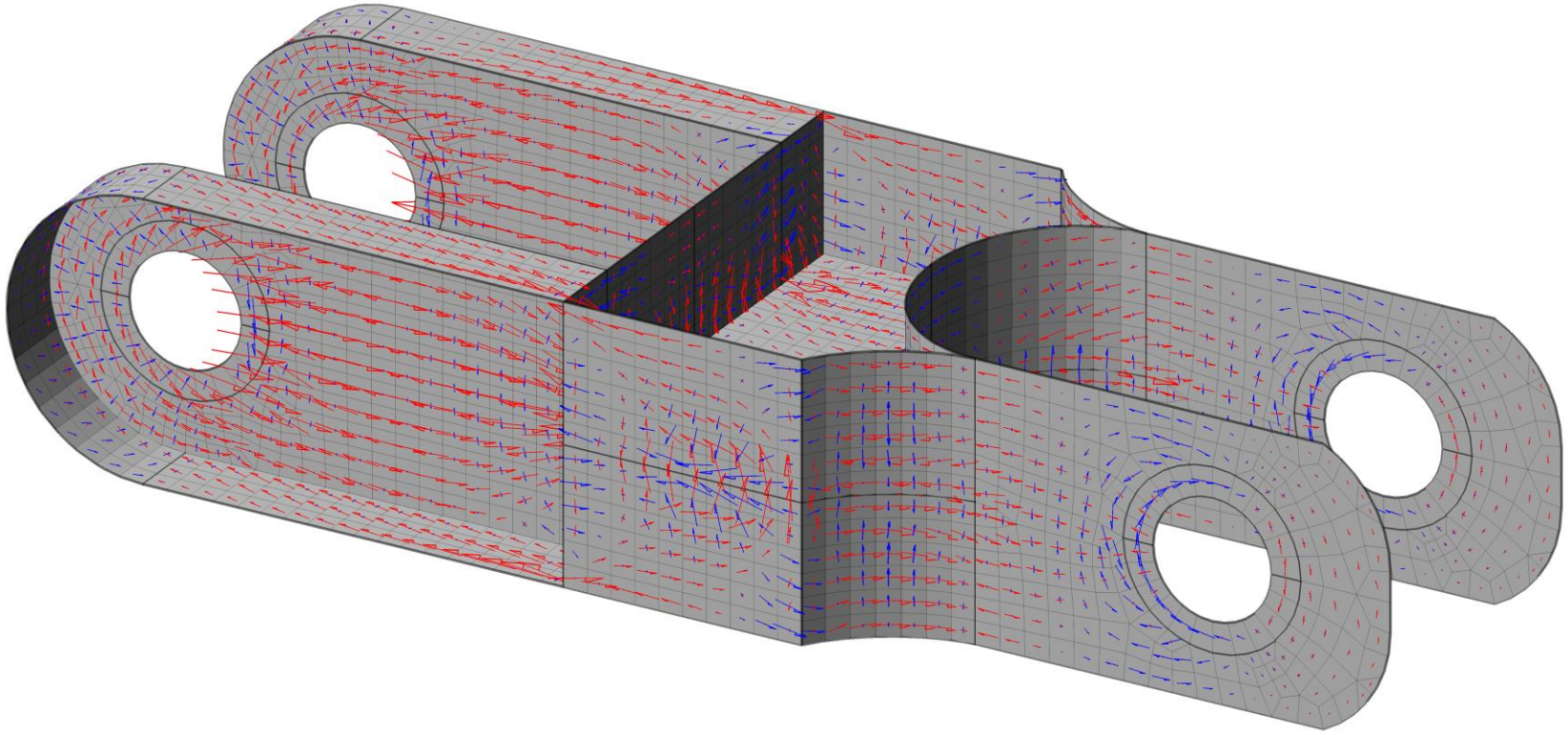


2.2.2 Beam Diagram

Remember those shear-moment diagrams from all those years ago? The Beam Diagram view style keeps the tradition alive! Display forces, moments, displacements and stresses contoured over the mesh or plotted beam diagram style.

2.2.3 Vector

This view style is most useful for looking at the directional flow of stress with the principal stress output vectors. The Vector view style places vectors on the elements to show the direction and magnitude of stresses and forces.

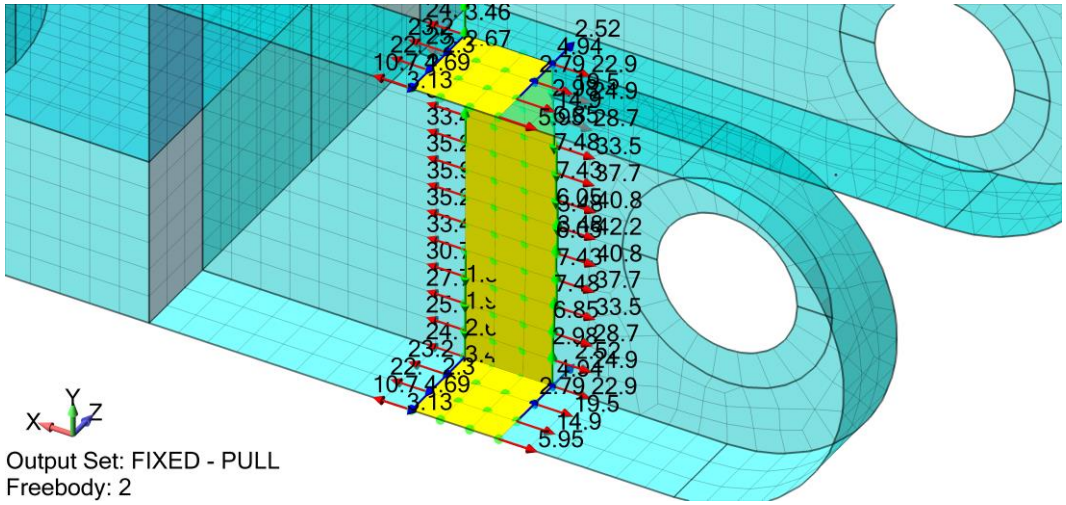


2.3 Freebody View Style

Freebody Diagrams (or FBDs) are an advanced post technique, yet they are beautifully simple at the same time. It's all about summing the forces and moments for a selection of elements and nodes. The trick is carefully selecting the entities and making sure that the "Freebody Contributions" are logical.

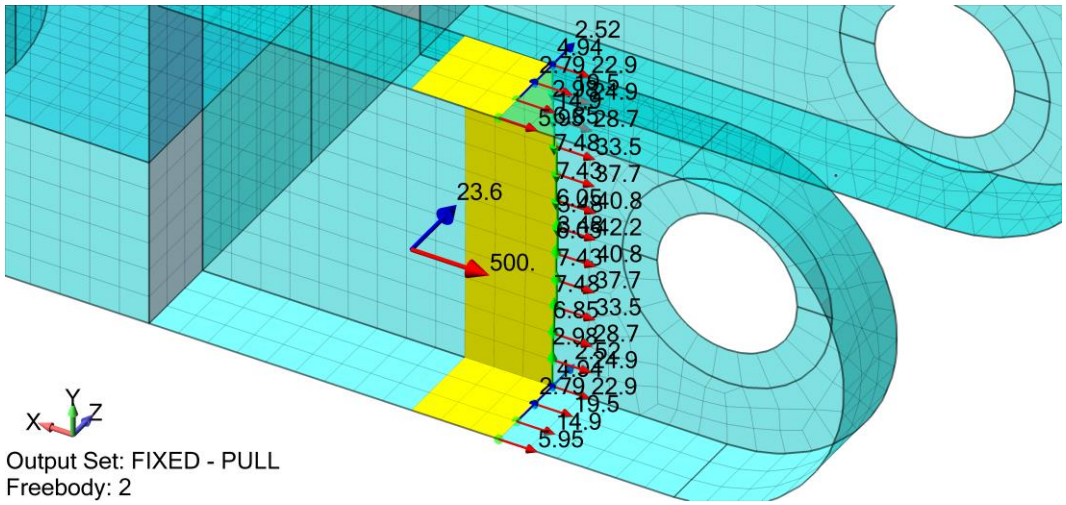
2.3.1 Freebody

The default display mode for the FBD toolbox is simply called "Freebody". This display mode only requires the user to select elements. FBDs will be generated on any on the nodes of the selected elements that are connected to loads, constraints, RBEs, or other elements.



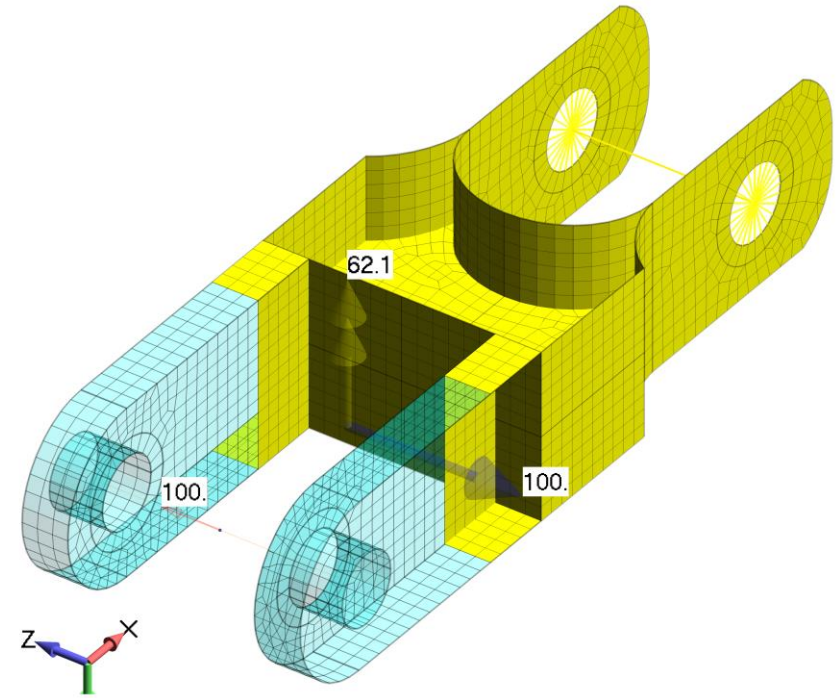
2.3.1 Interface Load

The "Interface Load" display mode will only generate FBDs on selected nodes attached to selected elements. In addition to the nodal FBDs, the Interface Load option allows the user to generate total summation vectors.

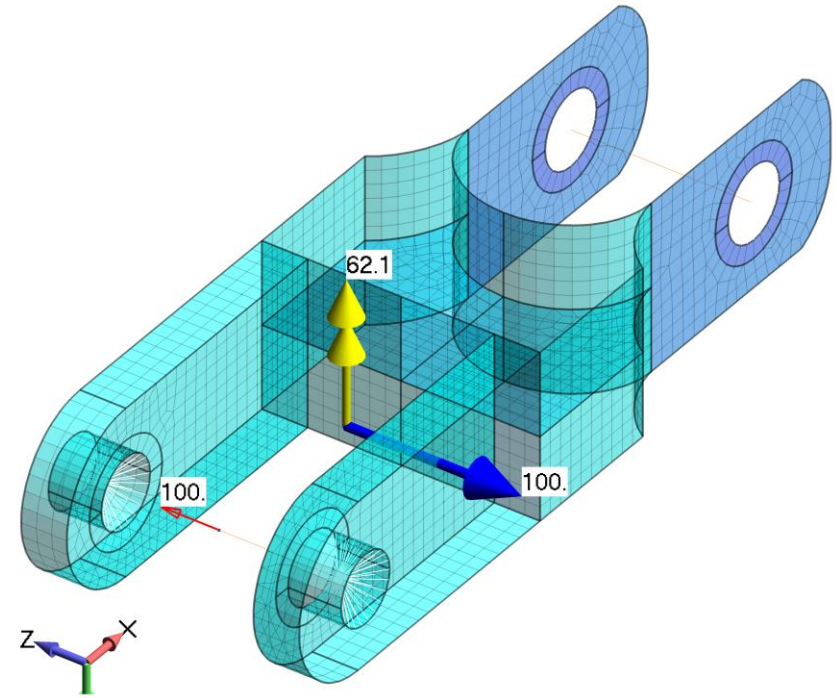


2.3.2 SECTION CUT

"Section Cut" display mode operates in the same manner as the Interface Load, but rather than selecting elements and nodes, the user simply selects a cutting plane. FEMAP will automatically select nodes along the cutting plane and elements on one side of the plane.



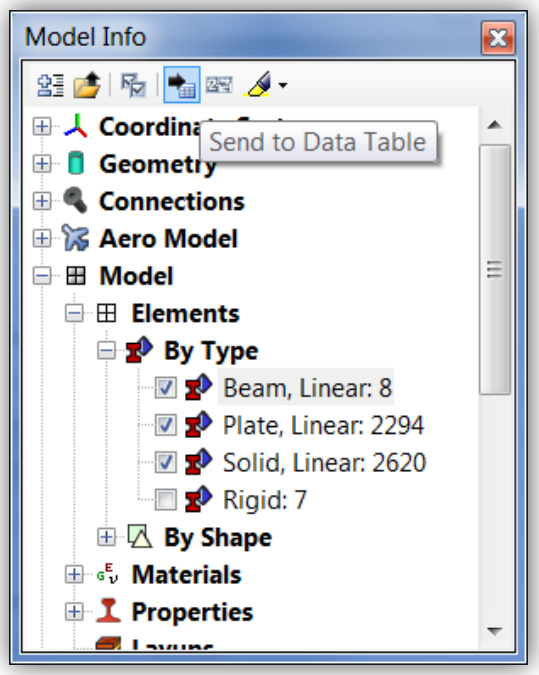
Output Set: FIXED - SIDE LOAD
Freebody: 1



Output Set: FIXED - SIDE LOAD
Freebody: 1

3. DATA TABLE

The Data Table pane allows you send data to an interactive, dynamically changing "table" using various methods to fill the table. Each item appears as a single row separated into a number of columns when it enters the Data Table. Once in the Data Table, information of the entities of the same type can be sorted, filtered, and evaluated to help you understand what is occurring in your model before and after analysis.



3.1 ADDING ENTITIES

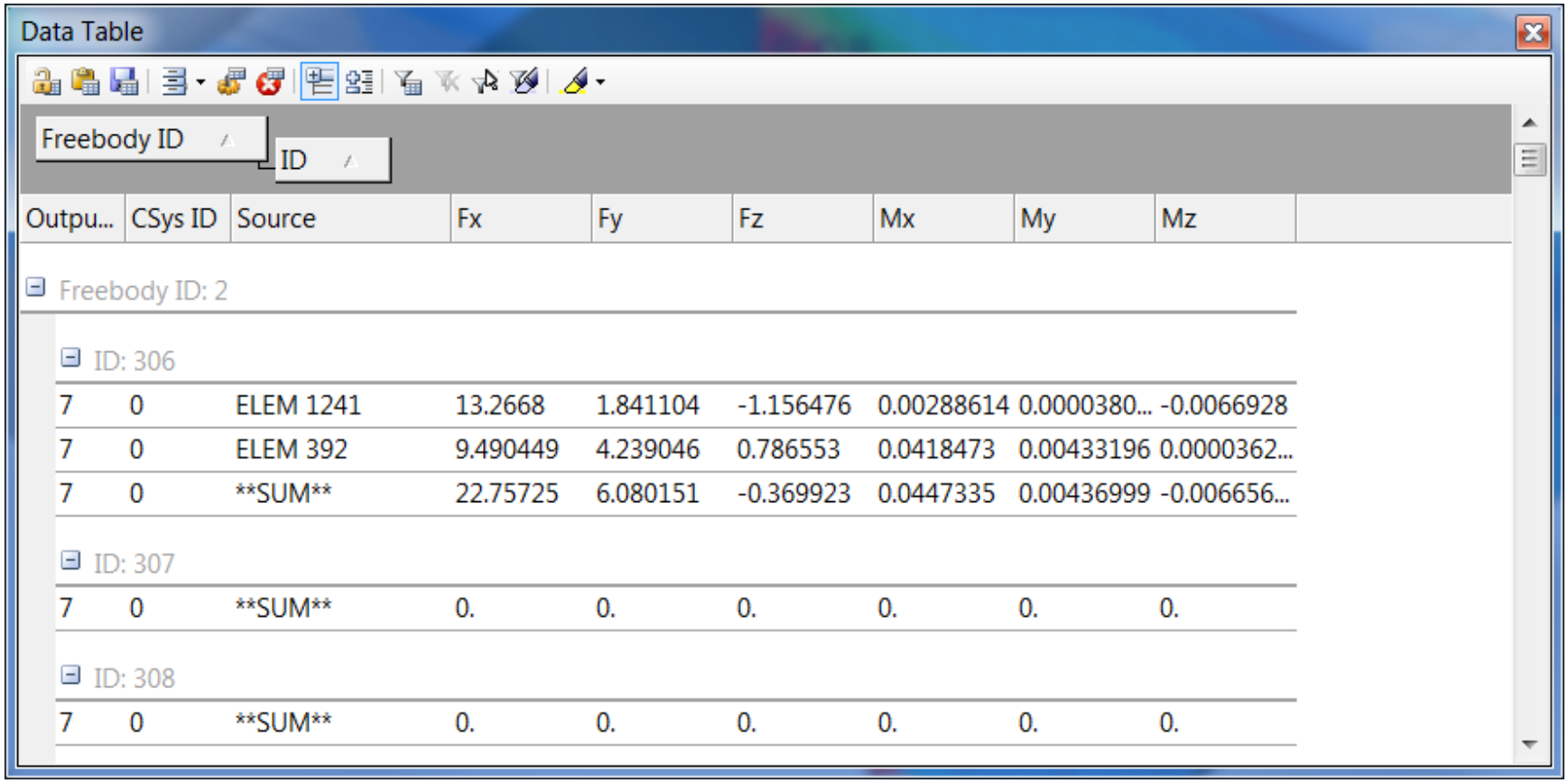
For post processing, it is most common to add nodes and elements to the data table. The Select toolbar is one method used to choose entities to be placed in the Data Table. You can also select entities in the Model Info tree and send them to the Data Table by clicking an icon or send all the output vectors from an entire output set over using a Context Sensitive menu.

3.2 ADDING OUTPUT DATA

One can create an output summary in the data table which can be quickly scanned to determine max/min values. There are many options for creating a customized output summary using ***List > Output > Summary To Data Table / Results To Data Table / Results Ranking To Data Table / Nodal Changes To Data Table / Contoured Results To Data Table***. Additionally, one could use the Add Output Columns button to add output data for entities already in the Data Table

3.3 Interfacing with Free Body Diagrams

Once you get comfortable with FBDs, you can print that information to the Data Table using "List Current Freebody to the Data Table" command near the top of the FBD toolbox.



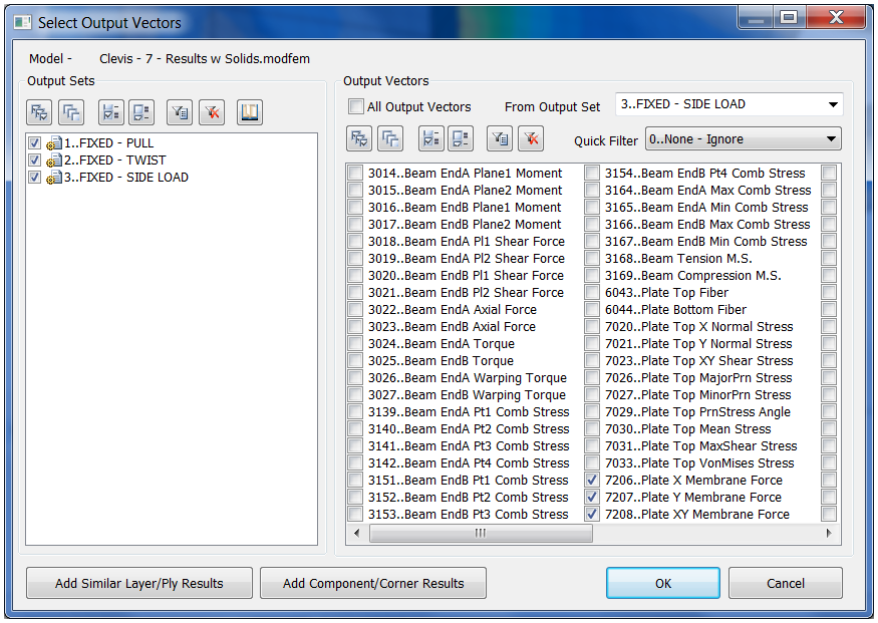
Outpu...	CSys ID	Source	Fx	Fy	Fz	Mx	My	Mz
Freebody ID: 2								
ID: 306								
7	0	ELEM 1241	13.2668	1.841104	-1.156476	0.00288614	0.0000380...	-0.0066928
7	0	ELEM 392	9.490449	4.239046	0.786553	0.0418473	0.00433196	0.0000362...
7	0	**SUM**	22.75725	6.080151	-0.369923	0.0447335	0.00436999	-0.006656...
ID: 307								
7	0	**SUM**	0.	0.	0.	0.	0.	0.
ID: 308								
7	0	**SUM**	0.	0.	0.	0.	0.	0.

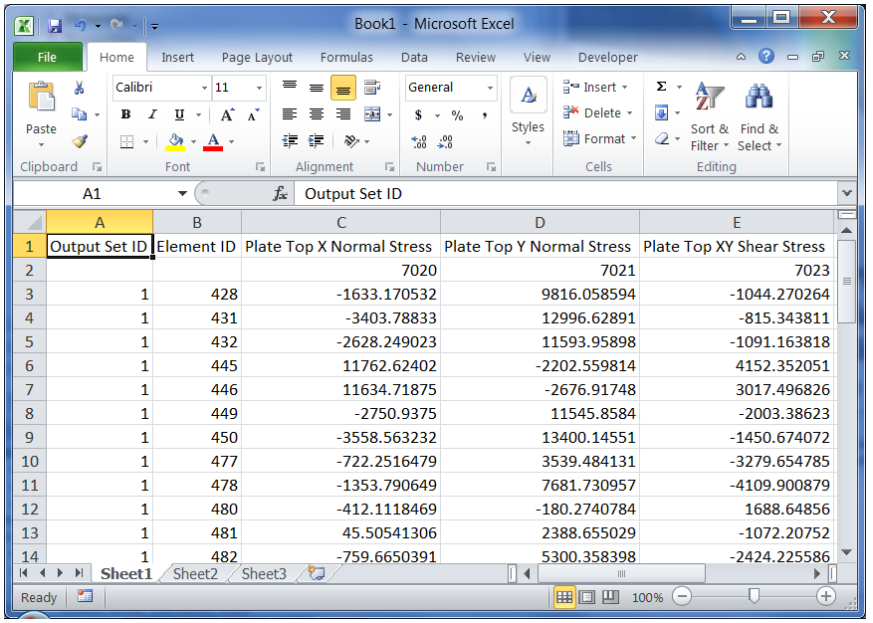
4. FEMAP API

Last but not least, the FEMAP Application Programming Interface (API). APIs have a wide variety of uses but one of the most useful is their applications in post processing. While the programs presented here are quite simple, the use of FEMAP APIs opens up new world of advanced post processing. Just ask the team at Abengoa Solar: http://proceedings.asmedigitalcollection.asme.org/proceeding.aspx?articleid=1797360

4.1 Exporting Data to Excel

"List Output to Excel" can be found in the Custom Tools toolbar or under the API folder in your FEMAP directory (e.g., "C:\FEMAPv1122\api\PostProcessing\List Output to Excel.BAS"). This tool allows the user to select multiple output sets, output vector and elements. The selected data is dumped directly to a new MS Excel workbook.

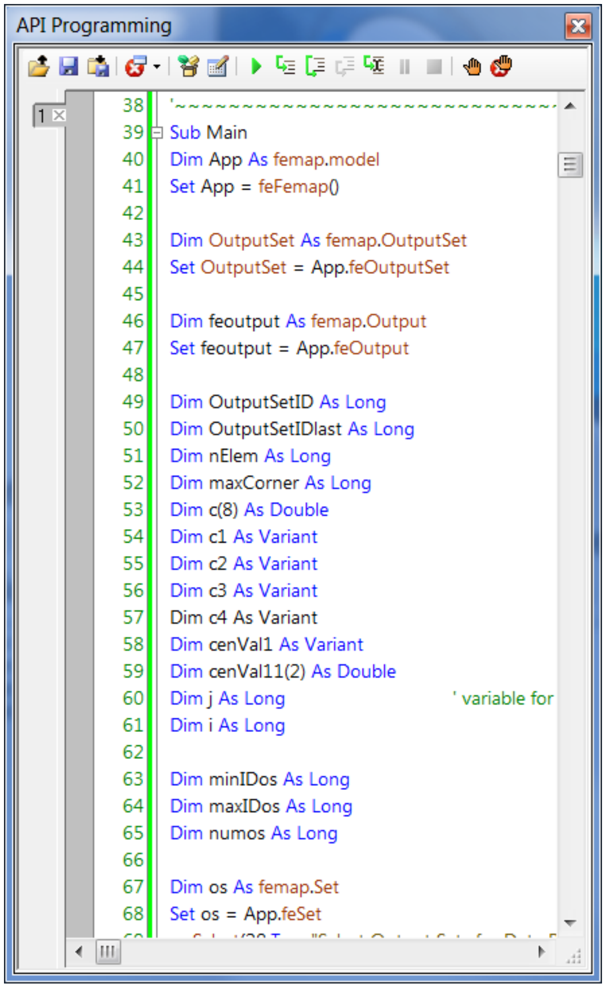


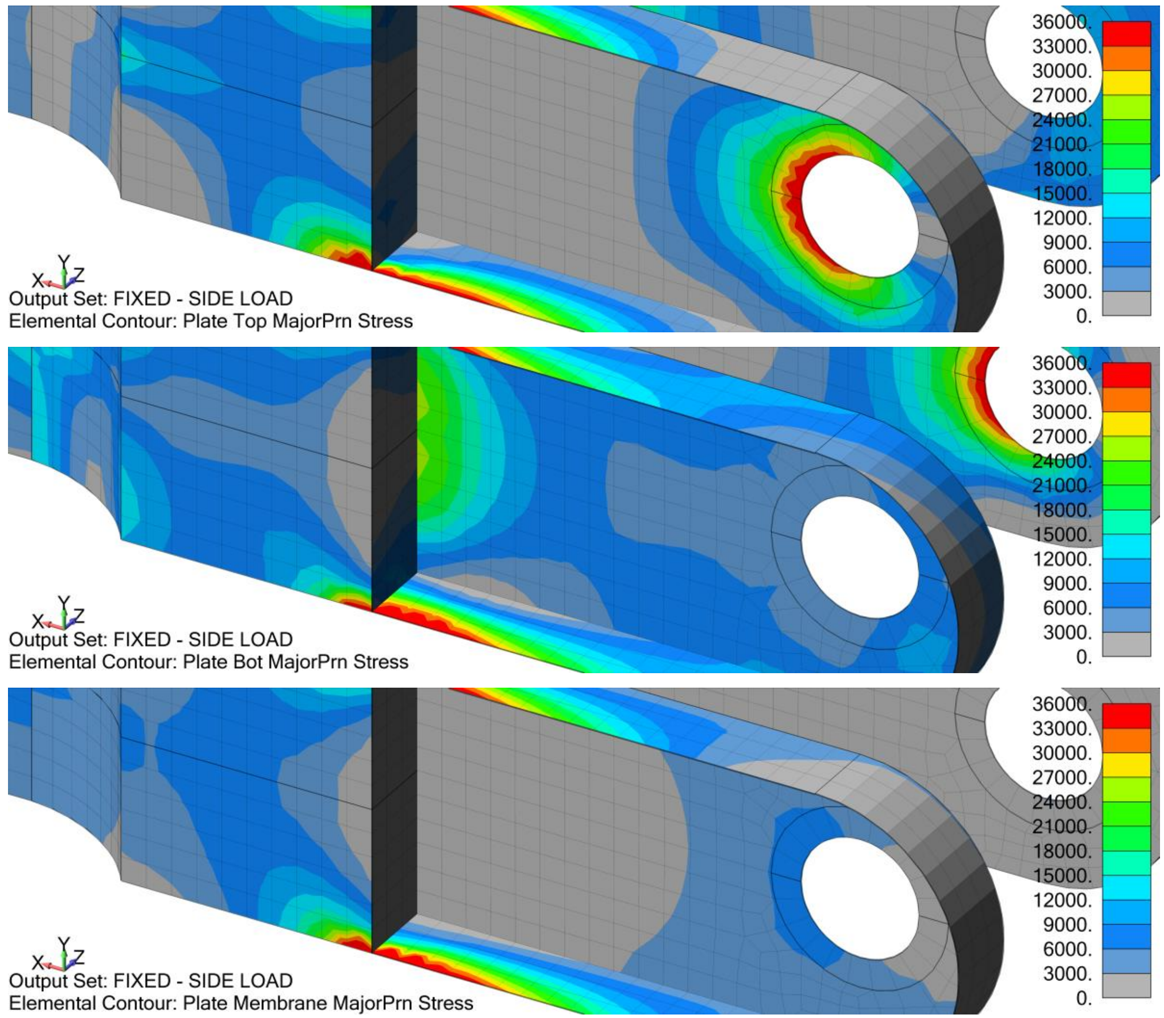


	A	B	C	D	E
1	Output Set ID	Element ID	Plate Top X Normal Stress	Plate Top Y Normal Stress	Plate Top XY Shear Stress
2			7020	7021	7023
3	1	428	-1633.170532	9816.058594	-1044.270264
4	1	431	-3403.78833	12996.62891	-815.343811
5	1	432	-2628.249023	11593.95898	-1091.163818
6	1	445	11762.62402	-2202.559814	4152.352051
7	1	446	11634.71875	-2676.91748	3017.496826
8	1	449	-2750.9375	11545.8584	-2003.38623
9	1	450	-3558.563232	13400.14551	-1450.674072
10	1	477	-722.2516479	3539.484131	-3279.654785
11	1	478	-1353.790649	7681.730957	-4109.900879
12	1	480	-412.1118469	-180.2740784	1688.64856
13	1	481	45.50541306	2388.655029	-1072.20752
14	1	482	-759.6650391	5300.358398	-2424.225586

4.2 Creating Custom Output Vectors

"ASME Stress Intensity Calculator" can also be found in the Custom Tools. This API calculates membrane stresses and stress intensity (think Tresca, not fracture mechanics) for plate elements. These output vectors are commonly used in the pressure vessel industry. If you need to create your own custom output vectors, this API could be a good starting point.





4.3 Modifying View Settings

This little program (Output Vector Switch.BAS) simply cycles contours between Plate VonMises Stress, Plate MajorPrn Stress and Plate MinorPrn Stress. It also flips the contour legend for minor principal stress to show peak compressive stresses in red.