

Advanced Post Processing

A Seminar for FEMAP v11.2.2 Users

Adrian Jensen, PE | Senior Staff Mechanical Engineer





Siemens PLM Software Sales & Support CAD I CAM I CAE I Teamcenter www.AppliedCAx.com



TABLE OF CONTENTS

1.	0	UTPU	JT MANAGEMENT
	1.1	SCRAT	TCH AND OUTPUT FILE ORGANIZATION
	1.2	NASTE	ran Output Requests
	1.3	File C	OPTIONS
	1.	.3.1	Attach to Results Files
	1.	.3.2	Create (Analysis) Studies
2.	Ρ	OSTPI	ROCESSING TOOLBOX
	2.1	DEFOR	rm View Style9
	2.	.1.1	Deformed9
	2.	.1.2	Animate9
	2.	.1.3	Vector
	2.2	CONT	OUR VIEW STYLE
	2.	.2.1	Contour
	2.	.2.2	Beam Diagram
	2.	.2.3	Vector
	2.3	Freeb	ODY VIEW STYLE
	2.	.3.1	Freebody
	2.	.3.1	Interface Load
	2.	.3.2	Section Cut
3.	D	ΑΤΑ Ι	ГАВLЕ
	3.1	Addin	NG ENTITIES
	3.2	Addin	NG OUTPUT DATA



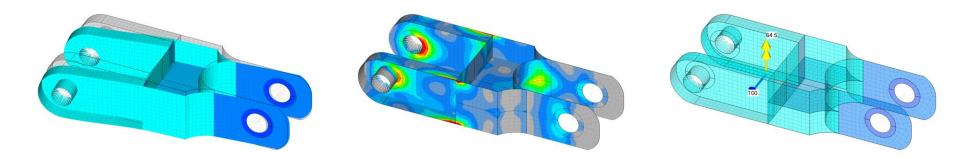
	3.3	INTERFACING WITH FREE BODY DIAGRAMS
4.	FI	EMAP API
	4.1	EXPORTING DATA TO EXCEL
	4.2	CREATING CUSTOM OUTPUT VECTORS
	4.3	Modifying View Settings



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.

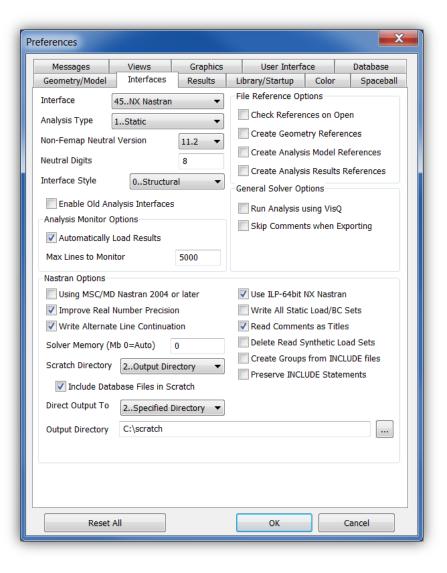
Next, we will take a tour of the PostProcessing Toolbox. Any experienced FEMAP user will know their way around the Toolbox but it is a worthwhile talking point because it's always changing and improving. More so, if you are like some of the FEMAP veterans in our office, it is time to let go and embrace 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.

Nastran Output Requ	ests		X
Nodal		Elemental	
☑ <u>D</u> isplacement	0Full Model 🔹	Force	0Full Model <
Applied Load	0Full Model 🛛 👻	✓ Stress	0Full Model <
✓ Constraint Force	0Full Model 🛛 👻	St <u>r</u> ain	0Full Model 🔻
Equation Force	0Full Model 🛛 👻	Strain Energy	0Full Model 👻
Force <u>B</u> alance	0Full Model 🛛 👻	Heat Flux	0Full Model 🔻
Velocity	0Full Model 🔻	E <u>n</u> thalpy	0Full Model 🔻
Acceleration	0Full Model 💌	Enthalpy Rate	0Full Model 🔻
Kinetic Energy	0Full Model 💌	Temperature	0Full Model 🔻
<u>T</u> emperature	0Full Model 💌	Kinetic Energy	0Full Model 👻
		Energy Loss	0Full Model <
		Fluid Pressure	0Full Model 👻
Customization			
Element Corner Re	esults R	esults Destination	Prev
Output Modes (a,b,c)	·	2PostProcess Only	•
	E	0Default	<u><u>o</u>ĸ</u>
Magnitude/Phase	Real/Imaginary	1Print Only 2PostProcess Only	Cancel
Relative Enforced I		3Print and PostProcess 4Punch Only 5Punch and PostProcess 6XDB	



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.

Manage Res	ults Files			Femap Database	External Results	Femap with Inter
1 /	Attached File(s)					
ID	Filename	File Path	Attach Options File Format			
1	space003	C:\Users\Tumalo\Desktop\Webinars\Femap v11 Release Webinar	ME Format ● <u>N</u> X Nastran ● <u>M</u> SC/MD Nastran ● NEi Nastran ■ Memory Mapped File Attach File <u>Unload</u>			
0		III	Reload Detach			
alysis Type: de / Labels:			Locate			
File Date: Memory Mappe	ed:		Save To Model			

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.

Preferences										
Messages Geometry/Model	Views Interfaces	Graphics Results	s User Interface Database Library/Startup Color Spaceball							
File Options	2.110110000	J	General Solver Optic							
Attach to Resul	t Files		Compute Principa	al Stress/S	train					
Use Memory M			Assume Enginee	2						
Create Studies			Read DirCos for	Solid Stres	s/Strain					
Study Titles										
Nastran Output Se		•								
4SUBTITLE		•								
Append F	emap Title									
Auto Answer NAST	RAN F06 Read	Questions								
Output Contain	s QUADR Eleme	nts	0No		•					
Output Contain	s Corner Output	t	1Yes 👻							
Read PSD/Freq	Functions		1Yes 🔻							
Read Corner O	utput		1Yes		•					
Hide f06 Warni	ng Dialog									
Reset A	ll		ОК	(Cancel					

Model Info
经 👌 🖓 🐂 🖙 🕖 •
E 🕹 Coordinate Systems
Geometry
Connections
🗄 😿 Aero Model
🕀 🖏 Analyses
Analysis Study
[1.Clevis - 7 - Results-000 [
all IFIXED - POLL
a 3_FIXED - TWIST
□ 2.Clevis - 7 - Results-001 - Updated Thickness
a _ 2eevis / Results oor opdated filledicss
5.FIXED - TWIST
6.FIXED - SIDE LOAD
□ III 3Clevis - 7 - Results-002 - Alt Constraints
all 7FIXED - PULL
al 8FIXED - TWIST
🚽 🗃 9FIXED - SIDE LOAD
🖻 🍓 All Results
🚽 🗐 1FIXED - PULL
al 2FIXED - TWIST
al S.FIXED - SIDE LOAD
al 4FIXED - PULL
al 5FIXED - TWIST
a 6FIXED - SIDE LOAD
- J.FIXED - PULL
a.FIXED - TWIST
9FIXED - SIDE LOAD
🕀 🔂 Groups
⊕ ∰ Layers
Selection List

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.

💩 • 📩 🚯 🖏 🞼 • 🖽 • 🗆 •	a 🕹 🇳	
Deform		×
Style	Animate	•
Results		
Output Set	3NX NASTRAN Case 1 🖛 🚯	+
Output Vector	1Total Translation 🔶 🚯	+
Options		
Transform	None	Ŧ
Scale	Actual Deformations	•
Scale Actual By	25.	
Scale Based on Group		
Deform Relative To	Origin	•
Deformed Model		
Undeformed Model		
Animation		
Shape	Sine - Full Abs	•
Frames	16	
Delay	101	
Animation Control	>>>	
Contour		×
Freebody		×
		_

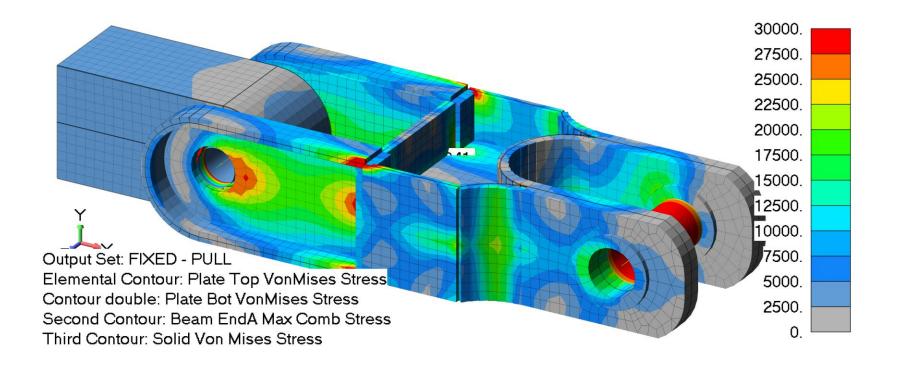


2.2 CONTOUR VIEW STYLE

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.



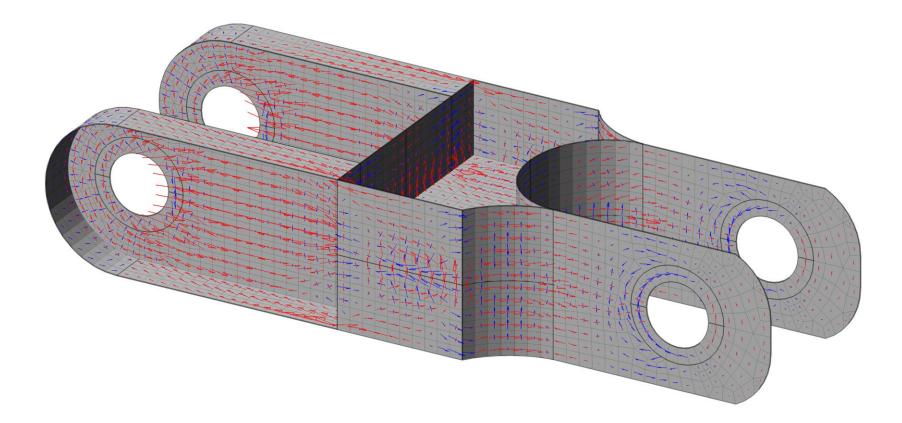


2.2.2 BEAM DIAGRAM

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.



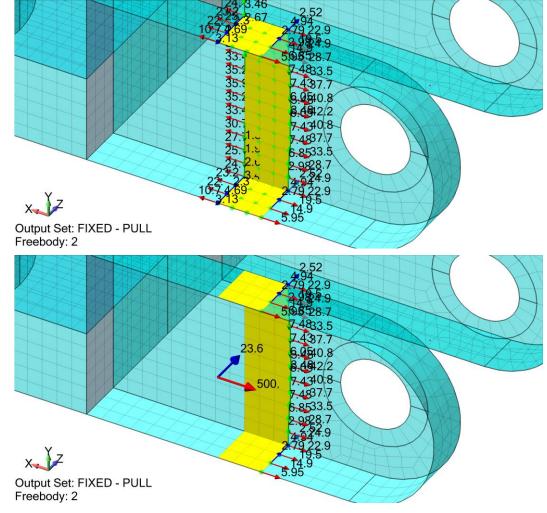


2.3 FREEBODY VIEW STYLE

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.



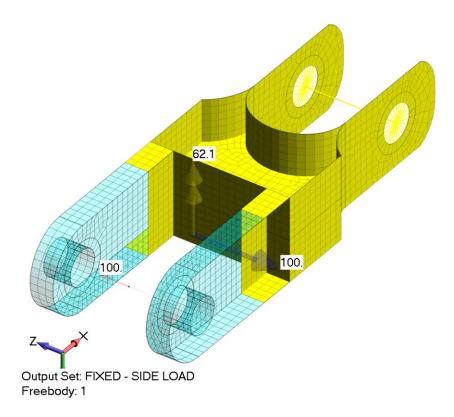
2.3.1 INTERFACE LOAD

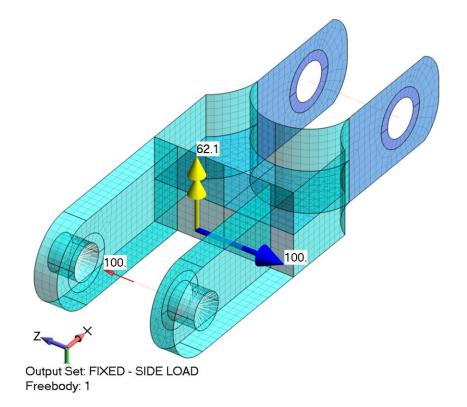
The "Interface Load" display mode will only generate FBDs on selected nodes attached to selected elements. In addition to the nodal FBDs, the Interface Load option allows the user to generate total summation vectors.



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.







3. DATA TABLE

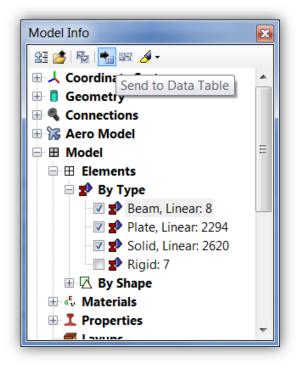
The Data Table pane allows you send data to an interactive, dynamically changing "table" using various methods to fill the table. Each item appears as a single row separated into a number of columns when it enters the Data Table. Once in the Data Table, information of the entities of the same type can 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 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.

Data	Tab	le		100							X
2	🔒 🖷 📑 - 🎜 😝 🔚 XII 🍗 🐨 🖓 🌛 -										
Fre	Freebody ID / ID /										•
Outp	ou	CSys ID	Source	Fx	Fy	Fz	Mx	My	Mz		
		ody ID: 2									
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		
Ξ	ID:	: 307									
7		0	**SUM**	0.	0.	0.	0.	0.	0.		
Ξ	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

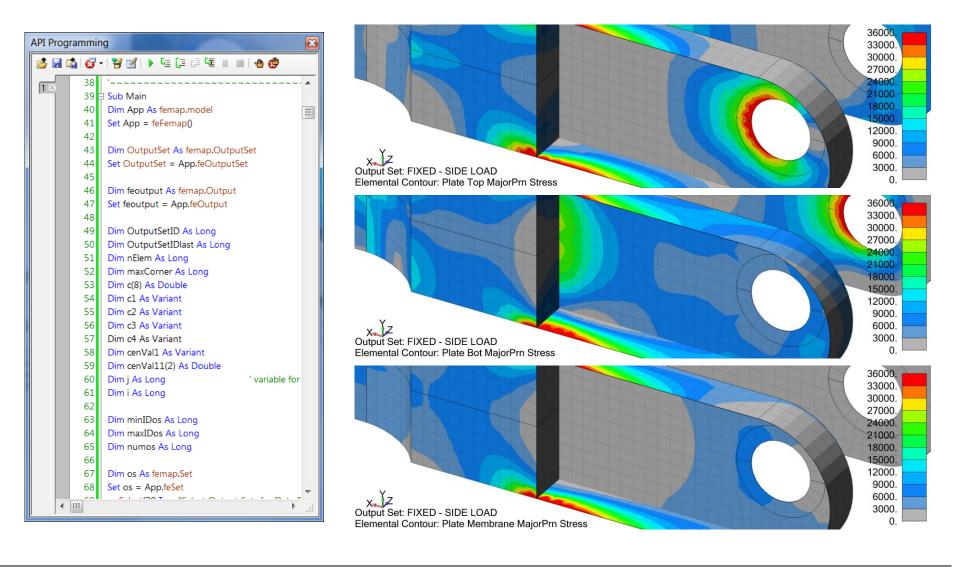
"List Output to Excel" can be found in the Custom Tools toolbar or under the API folder in your FEMAP directory (e.g., "C:\FEMAPv1122\api\PostProcessing\List Output to Excel.BAS"). This tool allows the user to select multiple output sets, output vector and elements. The selected data is dumped directly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

Select Output Vectors Model - Clevis - 7 - Results w Solids.modfem Output Sets Output Sets	Output Vectors All Output Vectors From Output Set 3.FIXED - SIDE LOAD With Seam Ends Plane1 Moment 3014Beam Ends Plane1 Moment 3015Beam Ends Plane2 Moment 3014Beam Ends Plane2 Moment 3015Beam Ends Plane2 Moment 3016Beam Ends Plane2 Moment 3017Beam Ends Plane2 Moment 3018Beam Ends Plane2 Moment 3018Beam Ends Plane2 Moment 3016Beam Ends Plane2 Moment 3017Beam Ends Plane2 Moment 3018Beam Ends Plane2 Moment 3018Beam Ends All Shear Force 3020Beam Ends All Shear Force 3021Beam Ends All Shear Force 3022Beam Ends All Force 3023Beam Ends All Force 3024Beam Ends All Force 3025Beam Ends All Force 3026Beam Ends All Force 3027Plate Top V Normal Stress 3026Beam Ends All Force 3027Plate Top MajorPm Stress 3131Beam Ends Plat Comb Stress
Add Similar Layer/Ply Results Add Com	nponent/Corner Results OK Cancel

	Book1 - Microsoft Excel														
Fi	ie +	lome	Insert Pag	ge Layout	Formu	las	Data	Reviev	/ View	Developer			∝ ?	- 7	x
Past	te	Calibri B	i v 11 <i>I</i> <u>U</u> v A [*] → A [*] Font	→ = ∧* = 10 10 10	F 🚍 🚍 F 🐺 🗐 Alignment	ай 1914 - Га	Gene \$ *0	% ,	Styles	Insert → Delete → Format → Cells	Σ . 	Sort & Filter * Editing	Select *		
	A1		• (=	f_{x}	Output	Set ID)								~
	A		В		С				D			I	E		
1	Output	Set ID	Element ID	Plate T	op X Nor	mal S	tress	Plate T	op Y No	rmal Stress	Plate	Тор ХҮ	Shear S	Stress	
2							7020			7021				7023	
3		1	428		-16	533.17	70532		9	816.058594		-	1044.2	70264	
4		1	431		-3	3403.7	78833		1	2996.62891			-815.3	43811	
5		1	432		-26	528.24	19023		1	1593.95898		-	1091.1	63818	
6		1	445		11	1762.6	52402		-2	202.559814			4152.3	52051	
7		1	446		11	1634.7	71875		-	2676.91748			3017.4	96826	
8		1	449		-	-2750	.9375			11545.8584			-2003.	38623	
9		1	450		-35	558.56	53232		1	3400.14551		-	1450.6	74072	
10		1	477		-72	22.251	16479		3	539.484131		-	3279.6	54785	
11		1	478		-13	353.79	90649		7	681.730957		-	4109.9	00879	
12		1	480		-41	2.111	18469		-1	80.2740784			1688.	64856	
13		1	481		45	5.5054	11306		2	388.655029			-1072.	20752	
_14 ∢ _∢	► ► 5	1 Sheet1	482 Sheet2	Sheet3		59.665	50391		5	300.358398 IIII		_	2424.2	25586 ▶ [•
Read	dy 🛅									Ⅲ □ Ⅲ 1	.00% (-)	-0	-+)

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.

