



Advanced Post Processing: How to Efficiently Share Your FEA Results

A Simcenter Femap Seminar for Simulation Engineers

Geoffrey Bee, PE – Application Engineer, CAE

Adrian Jensen, PE, MBA – Director of Engineering, CAE





AppliedCAx

CAD • CAE • CAM • PLM

NX • Teamcenter • Simcenter Femap

Simcenter 3D • Simcenter STAR-CCM+ • Amesim

Portland, OR

WE DO THIS EVERY DAY

Since 2008 Applied CAx has guided companies to realize their investment in digital engineering tools.

NX CAD

SIMCENTER FEMAP

NX CAM

SIMCENTER 3D

TEAMCENTER

SIMCENTER STAR-CCM+

SOLID EDGE

SIMCENTER 3D · FEMAP · STAR-CCM+
NX CAD-CAM · TEAMCENTER · SOLID EDGE

Our Next Femap Training Opportunity

October 31st – November 10th, 2022

Live, Online

AppliedCAx.com/Training

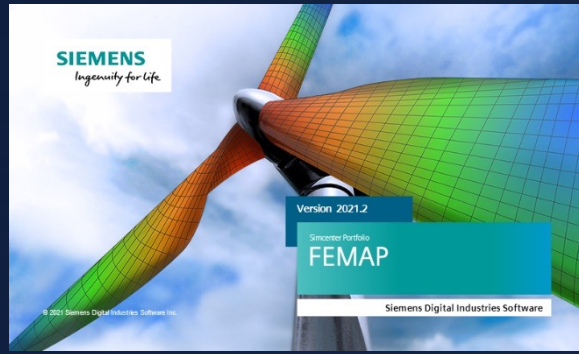
CAE Support Review:

As far as tech support is concerned, I have had fast and top-quality responses. The awesome thing is, I get a lot of information during the support communication, but I also receive the full concept and learn a lot. Even if the issue is very simple, I get a quick response. If someone asks me about buying Siemens products, I will surely recommend Applied CAx.

Srivatsa Pradeep, MSME
Project Consultant (Structures & FEA)
Hatch LTK Engineering Services

HATCH LTK

Positive Change for the Next Century



Simcenter Femap Practices: Advanced Post Processing in Femap v2022.2

A Seminar for Simulation Engineers

Geoffrey Bee, PE – Application Engineer, CAE

Adrian Jensen, PE, MBA – Director of Engineer, CAE

Output Management

- Scratch and Output File Organization
- Nastran Output Requests
- File Options
 - Attach to Results
 - Create (Analysis) Studies
- Things to look for in the f06 file
 - OLOAD
 - SPCForce

PostProcessing Toolbox

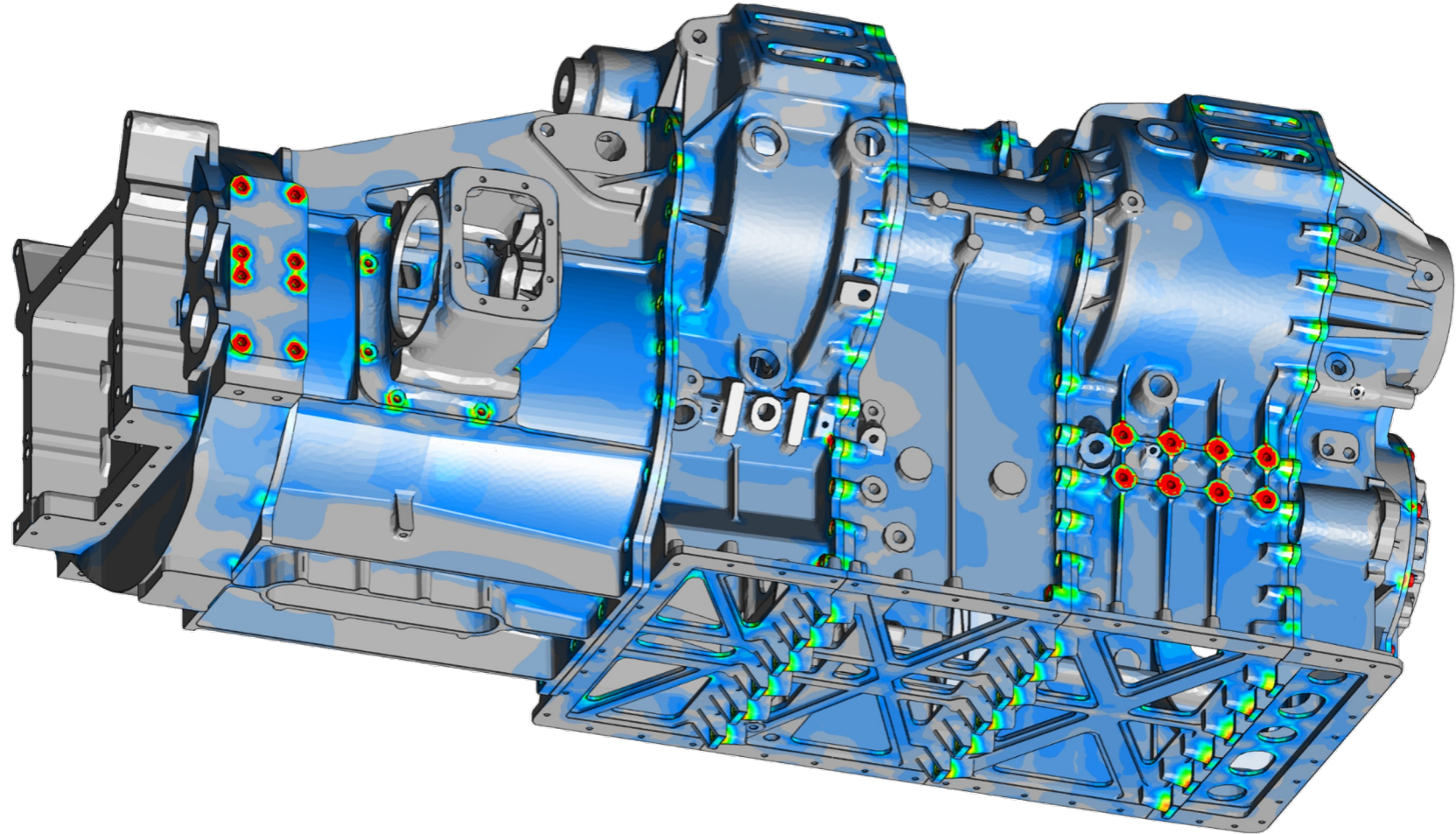
- Deform View Style
 - Deformed
 - Animate
 - Vector
- Contour View Style
 - Contour
 - Beam Diagram
 - Vector
- Freebody View Style
 - Freebody
 - Interface Load
 - Section Cut

Charting

- Expanding Complex Results
- Charting Complex Results

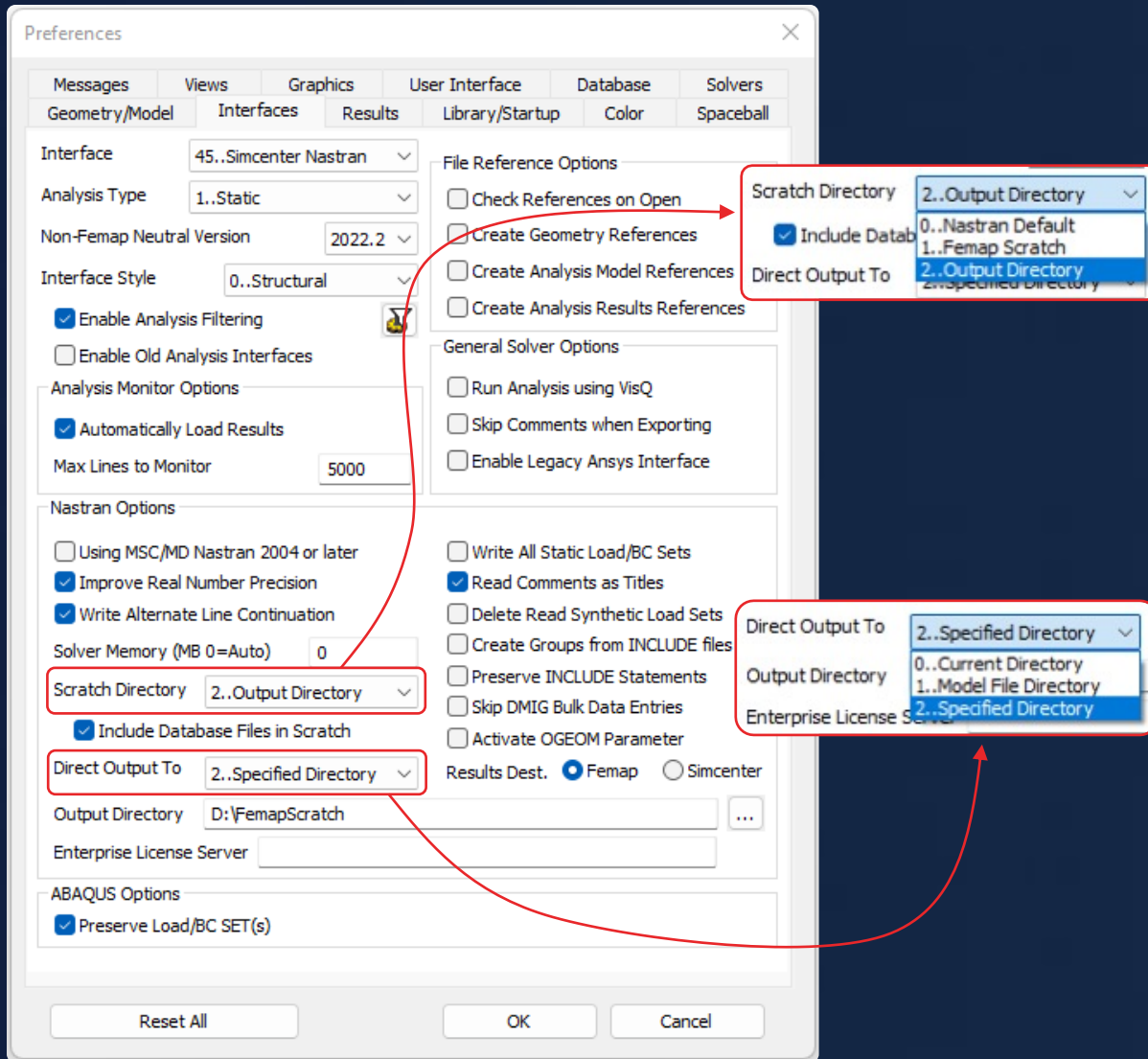
Femap API

- Group_Post_FNO
- ASME Stress Intensity Calculator



Output Management

Scratch and Output File Organization



Scratch Directory

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

1..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.

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

Direct Output To

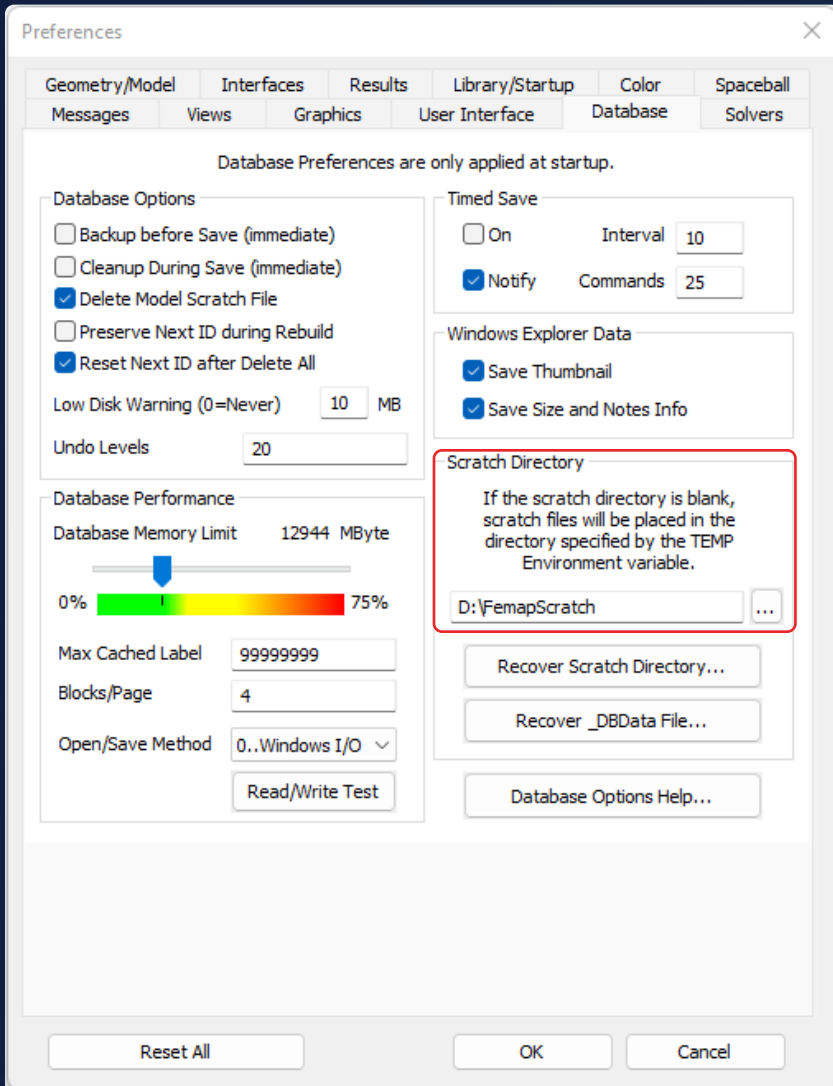
0..Current Directory: Last used directory by FEMAP. If a model has been saved to a directory, the output will be directed to that directory when this option is on.

1..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.

2..Specified Directory: This option allows you to send all Simcenter Nastran output to a directory that you have specified. This can help 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.

Output Management

Femap Scratch Files



Preferences dialog box showing Database Options, Timed Save, Windows Explorer Data, and Scratch Directory settings.

Database Preferences are only applied at startup.

Database Options


- Backup before Save (immediate)
- Cleanup During Save (immediate)
- Delete Model Scratch File
- Preserve Next ID during Rebuild
- Reset Next ID after Delete All

Low Disk Warning (0=Never) MB

Undo Levels

Database Performance

Database Memory Limit MByte

0%  75%

Max Cached Label

Blocks/Page

Open/Save Method ▾

Timed Save

- On Interval
- Notify Commands

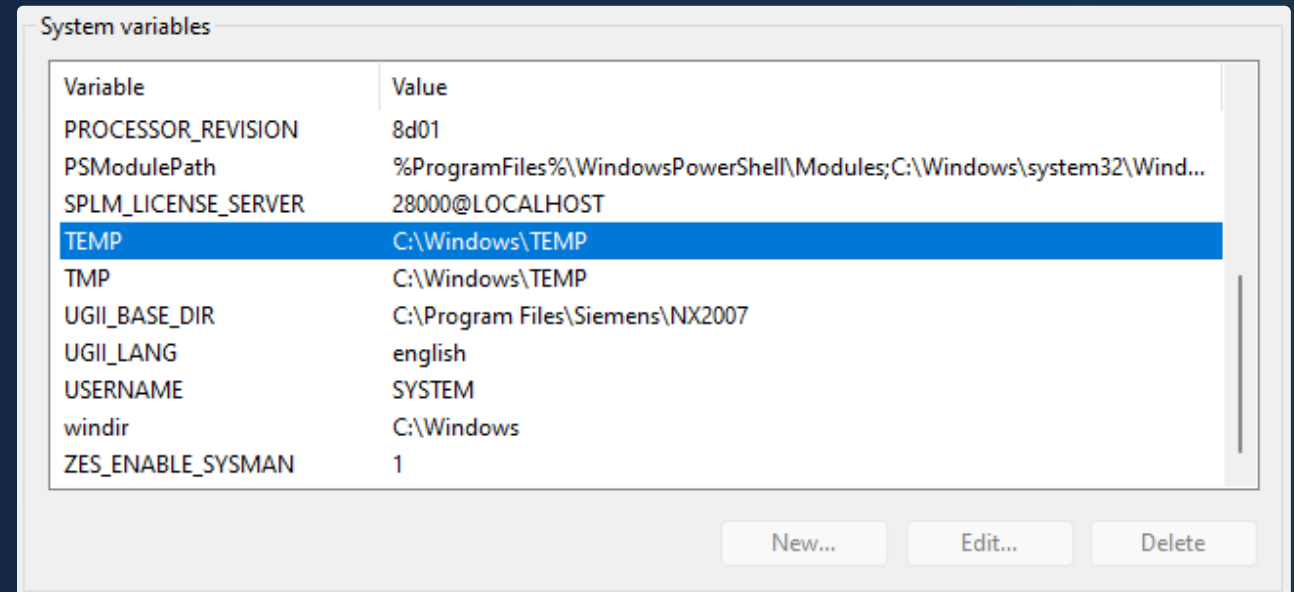
Windows Explorer Data

- Save Thumbnail
- Save Size and Notes Info

Scratch Directory

If the scratch directory is blank, scratch files will be placed in the directory specified by the TEMP Environment variable.

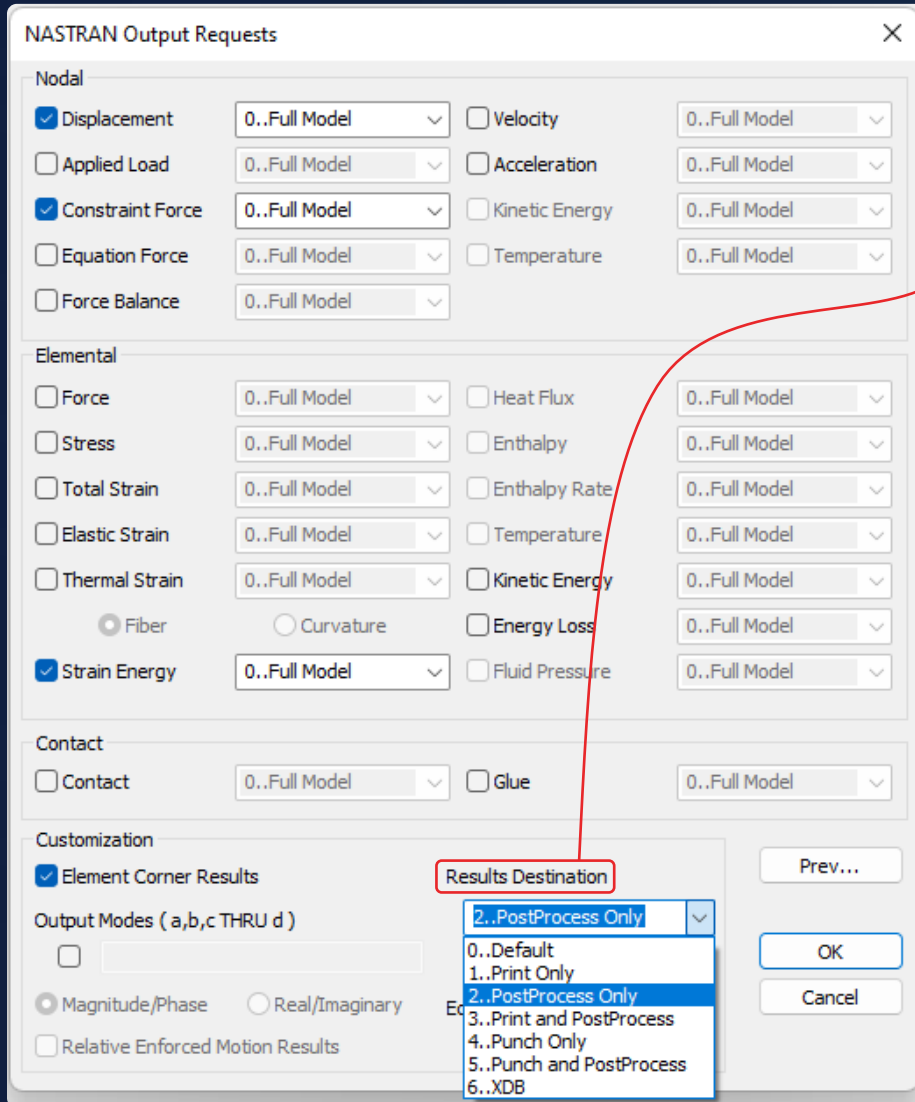
...



System variables dialog box showing a list of variables and their values.

Variable	Value
PROCESSOR_REVISION	8d01
PSModulePath	%ProgramFiles%\WindowsPowerShell\Modules;C:\Windows\system32\Wind...
SPLM_LICENSE_SERVER	28000@LOCALHOST
TEMP	C:\Windows\TEMP
TMP	C:\Windows\TEMP
UGII_BASE_DIR	C:\Program Files\Siemens\NX2007
UGII_LANG	english
USERNAME	SYSTEM
windir	C:\Windows
ZES_ENABLE_SYSMAN	1

Nastran Output Requests



Customization also allows you to select a results destination file type:

Results Destination

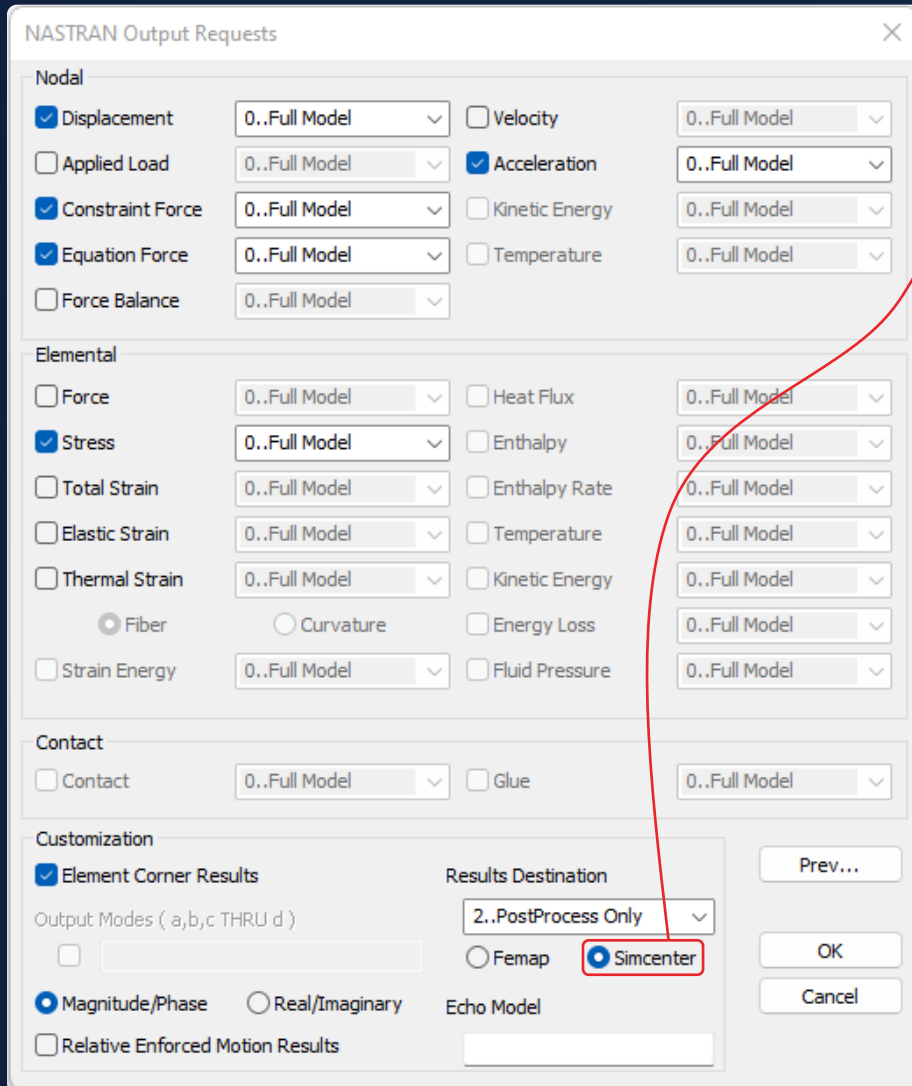
- 0..Default = .op2 files created using PARAM,POST,-1
- 1..Print Only = *.f06 – An ASCII output file you will use most frequently
- 2..PostProcess Only = *.op2 – A binary file known as an “OUTPUT2” file
- 3..Print and PostProcess = *.f06 and *.op2
- 4..Punch Only = *.pch – An ASCII file that contains “punched” output
- 5..Punch and PostProcess = *.pch and *.op2
- 6..XDB = *.xdb – A binary file known as a “results database” file

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 Simcenter Nastran, it automatically reads the results. This can be changed by: File, Preferences, then click Interfaces, then uncheck box “Automatically Load Results”.
- FEMAP reads the .f06 file first to obtain any error, warning, or information messages that might have occurred during the analysis.

Output Management

Nastran Output Requests



NASTRAN Output Requests

Nodal

- Displacement (0..Full Model)
- Applied Load (0..Full Model)
- Constraint Force (0..Full Model)
- Equation Force (0..Full Model)
- Force Balance (0..Full Model)
- Velocity (0..Full Model)
- Acceleration (0..Full Model)
- Kinetic Energy (0..Full Model)
- Temperature (0..Full Model)

Elemental

- Force (0..Full Model)
- Stress (0..Full Model)
- Total Strain (0..Full Model)
- Elastic Strain (0..Full Model)
- Thermal Strain (0..Full Model)
- Fiber Curvature
- Strain Energy (0..Full Model)
- Heat Flux (0..Full Model)
- Enthalpy (0..Full Model)
- Enthalpy Rate (0..Full Model)
- Temperature (0..Full Model)
- Kinetic Energy (0..Full Model)
- Energy Loss (0..Full Model)
- Fluid Pressure (0..Full Model)

Contact

- Contact (0..Full Model)
- Glue (0..Full Model)

Customization

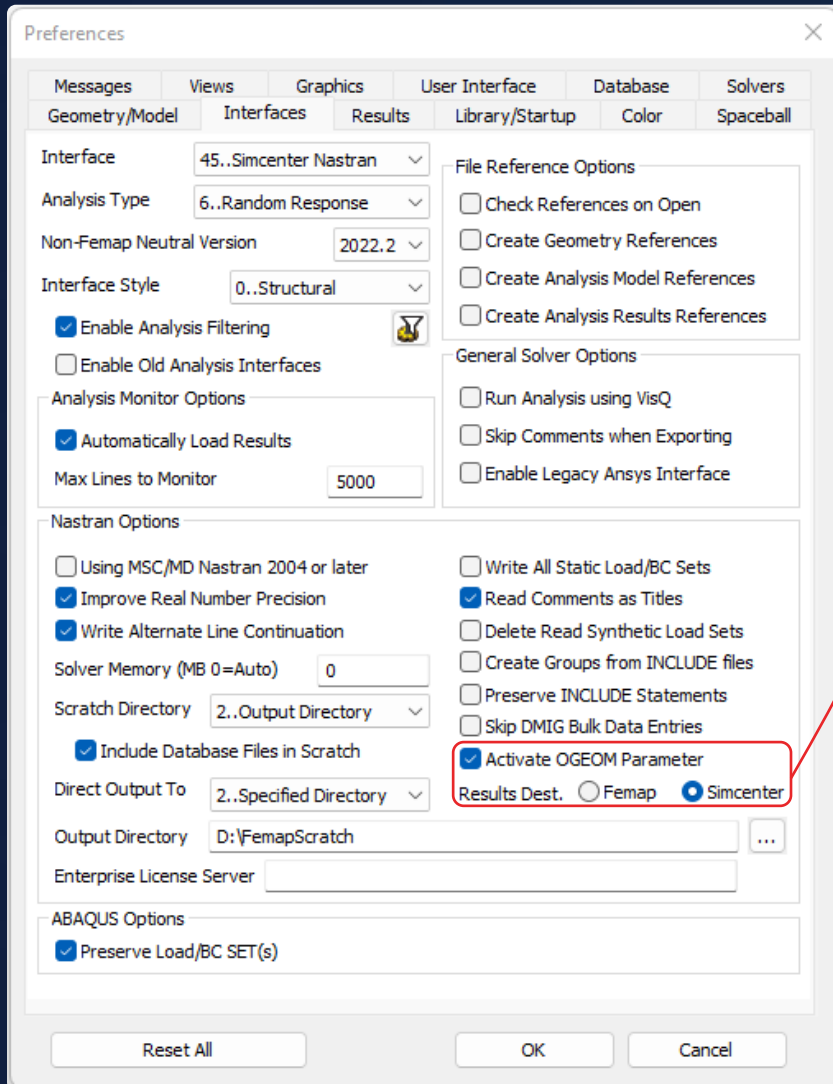
- Element Corner Results
- Output Modes (a,b,c THRU d)
- []
- Magnitude/Phase Real/Imaginary
- Relative Enforced Motion Results
- Results Destination: 2..PostProcess Only
- Femap Simcenter
- Echo Model: []

Prev... OK Cancel

As of version 2022.1 it is now possible to instruct Nastran to create an output file, in particular an .op2 file, which can be used by other applications in the Siemens Software Portfolio, including Simcenter 3D.

Once selected, Nastran knows to produce this type of output by having PARAM,POST set to a value of -2 in the input file.

Nastran Output Requests



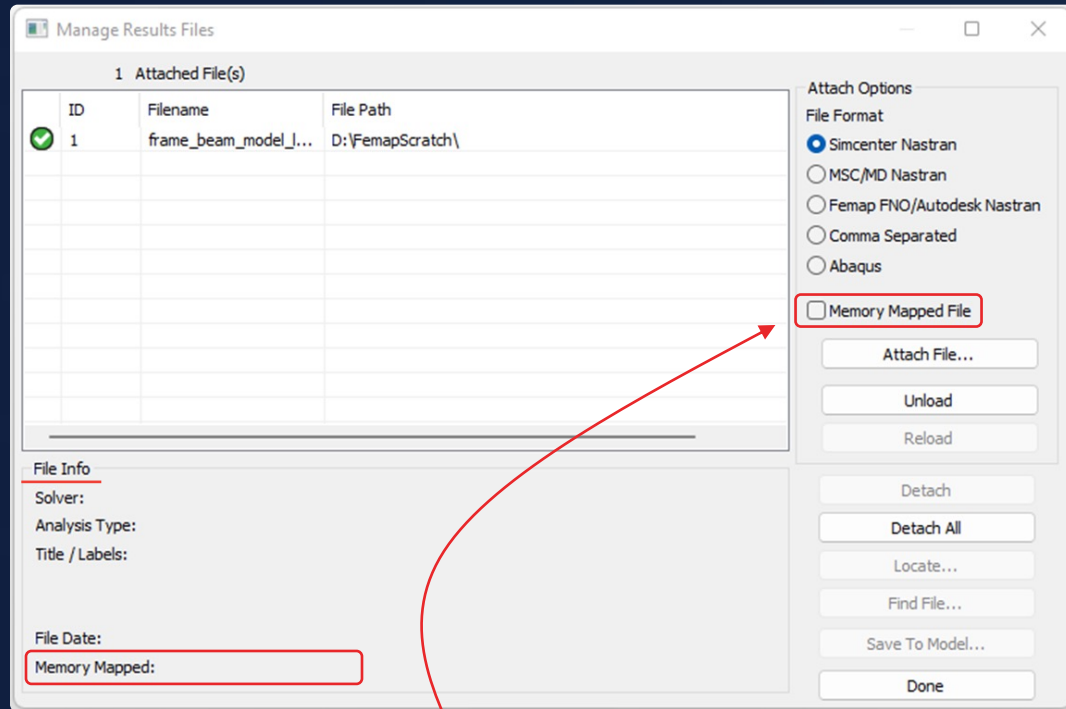
You can also set Femap to always create .op2 files using PARAM,POST,-2 automatically.

To do this: File → Preferences. In the Preferences dialog box, choose the Interfaces tab, and set the Results Destination option to Simcenter.

Some pre and post processors that can read .op2 files created using PARAM,POST,-1 (Femap default) require that the model information also be in the .op2 file, which is done by setting PARAM,OGEOM to a value of YES.

By turning on the Activate OGEOM Parameter option on this same tab, all newly created Analysis Sets will have the OGEOM option turned on automatically.

File Options – Attach to Results

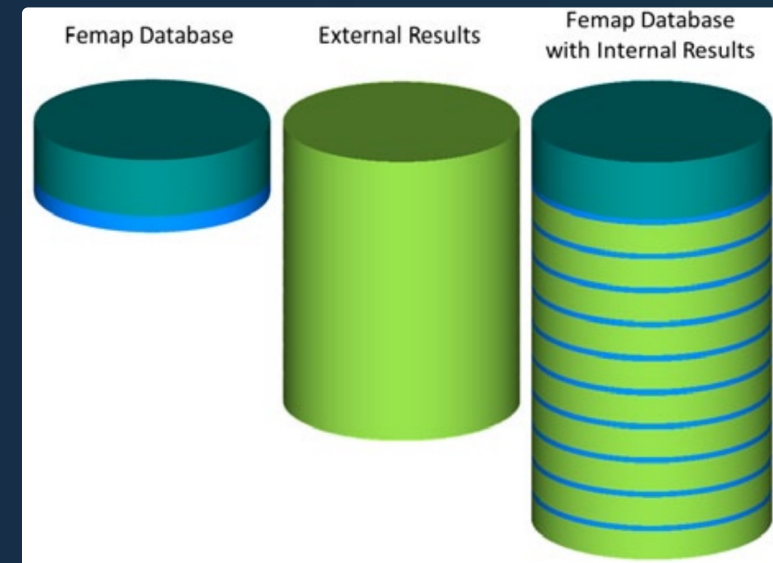


File → Attach to Results... attaches to results files and allows post processing to occur without “internalizing” the contents of each file into the FEMAP database. This is especially helpful when you have a large output file. In general, there are two reasons for large output files:

- First, a large model will typically create a large output file unless care has been taken 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.

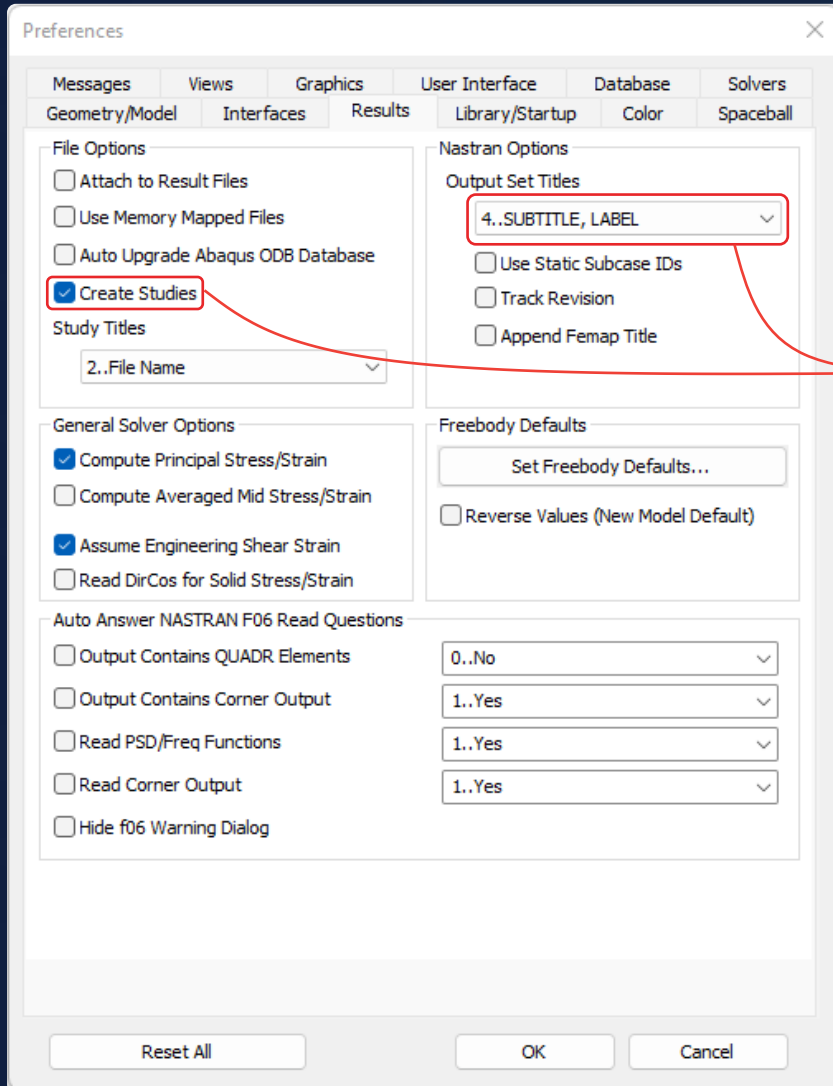
Maintaining external results data in this way minimizes the Femap database size and makes results data handling much more manageable.

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. One can verify “Memory Mapped” status in the File Info section

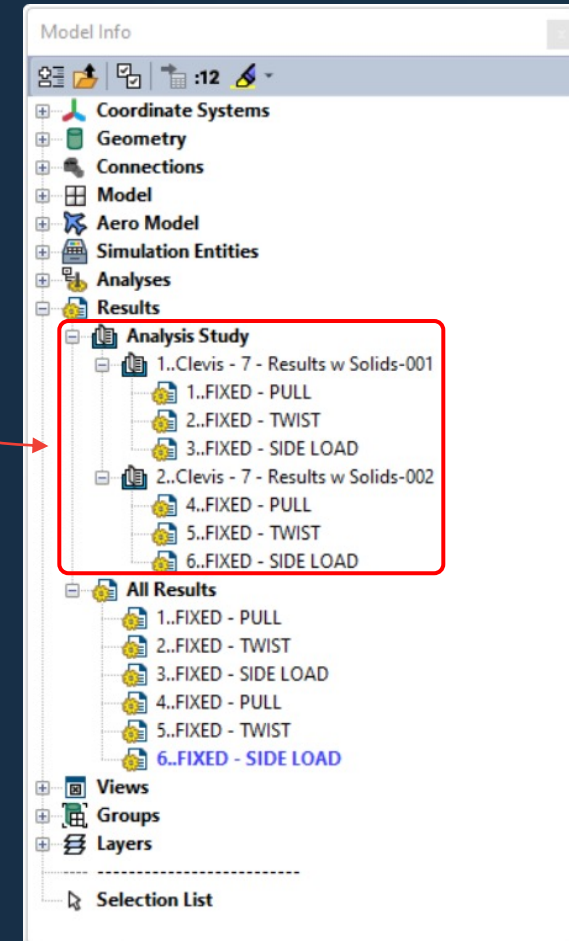


Output Management

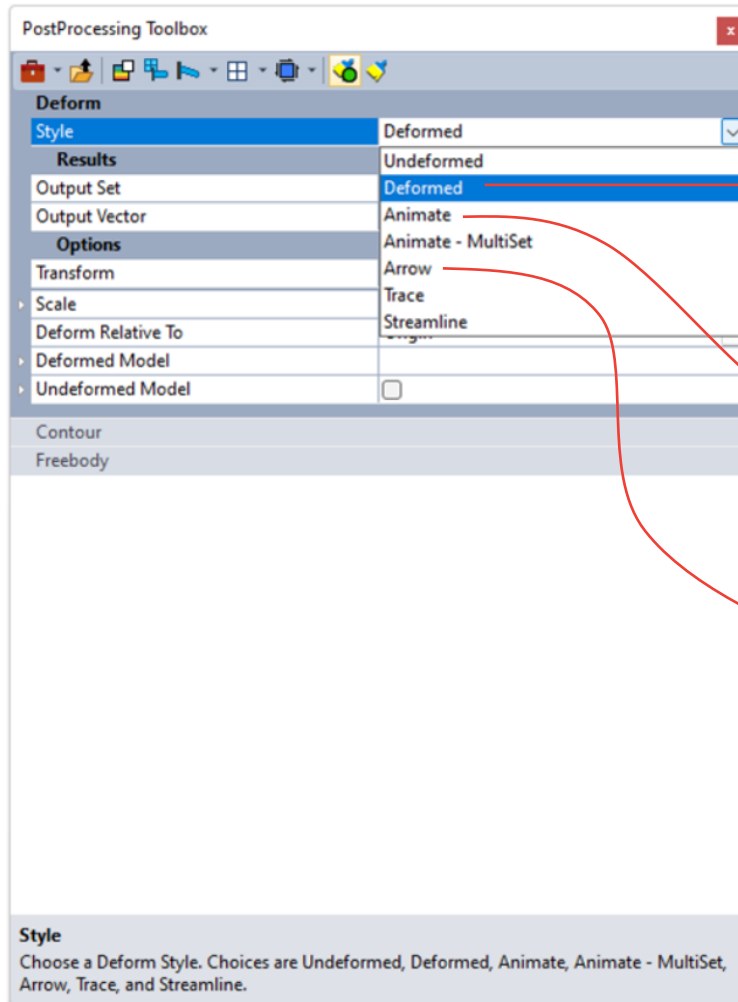
File Options – Create (Analysis) Studies



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



Deform Style



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

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".

Animate

As the name implies, this option 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.

Arrow

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

Contour Style – Contour

PostProcessing Toolbox

Deform

Contour

Style: Contour

Results

Output Set	6..FIXED - SIDE LOAD	↔ ↕ →
Output Vector	7033..Plate Top VonMises Stress	↔ ↕ →
Additional Vector(s)		
Output Vector	3164..Beam EndA Max Comb Stress	↔ ↕ →
Output Vector	60031..Solid Von Mises Stress	↔ ↕ →

Options

Data Conversion	Average	▼
Data Selection	Contour Group	▼
Type	Elemental	▼
Double-Sided Planar	<input checked="" type="checkbox"/>	
Show On Groups	Full Model / Visible Groups	▼
Show As	Filled	▼
Levels		
Legend	<input checked="" type="checkbox"/>	

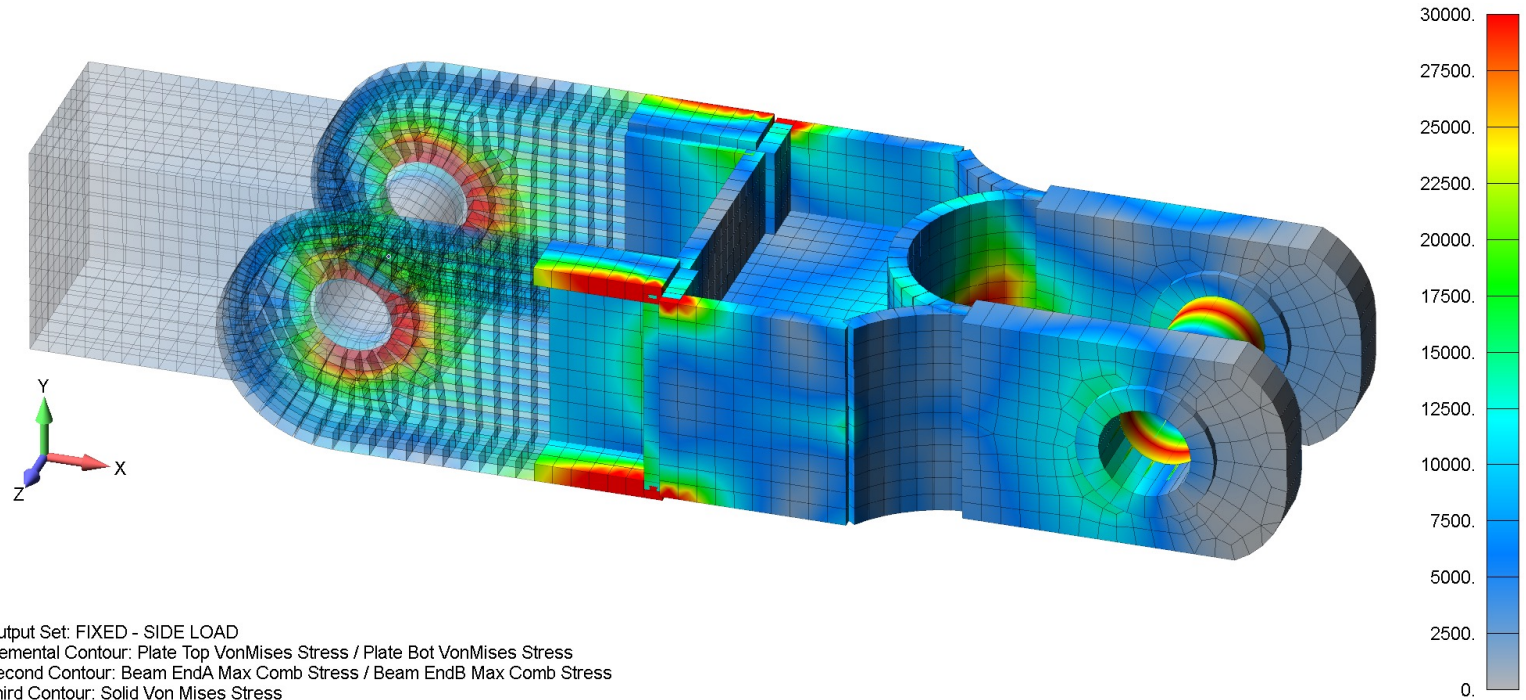
Freebody

Style

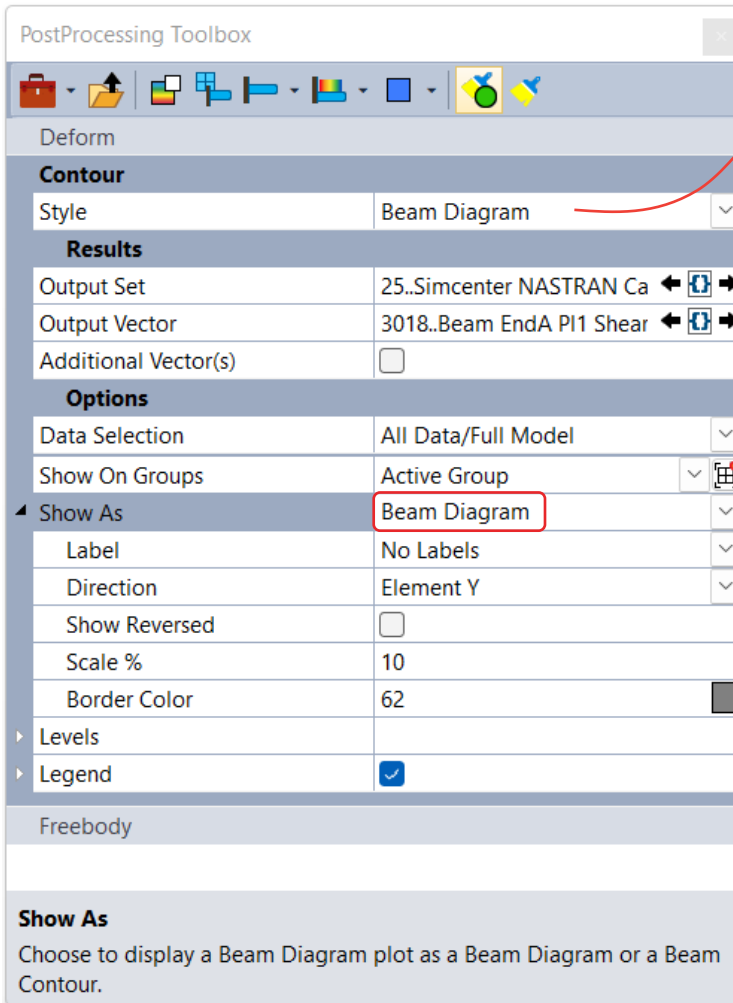
Choose a Contour Style. Choices are No Contours, Contour, Criteria, Beam Diagram, IsoSurface, Section Cut, or Contour Arrow.

Contour

The Contour style allows the user to overlay displacement, force, strain, and stress information on the mesh



Contour Style – Beam Diagram

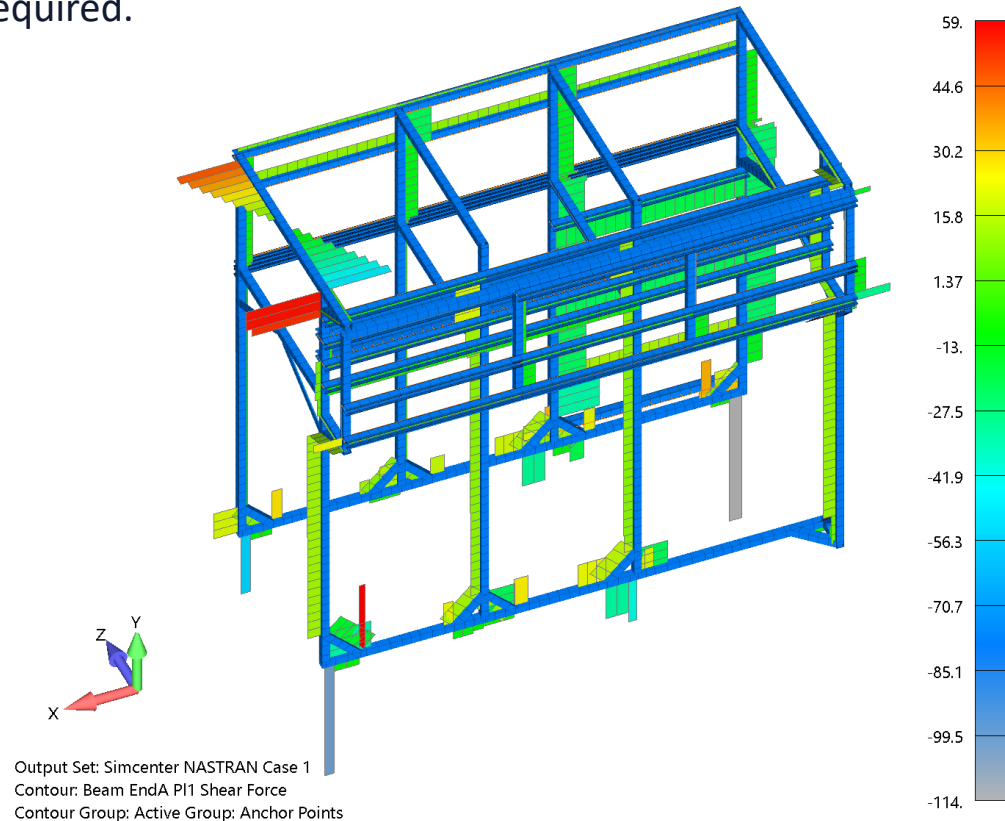


The screenshot shows the 'PostProcessing Toolbox' window. Under the 'Contour' section, the 'Style' is set to 'Beam Diagram'. In the 'Options' section, 'Show As' is set to 'Beam Diagram' (highlighted with a red box). Other options include 'Data Selection' (All Data/Full Model), 'Show On Groups' (Active Group), 'Label' (No Labels), 'Direction' (Element Y), 'Show Reversed' (unchecked), 'Scale %' (10), and 'Border Color' (62). A 'Show As' section at the bottom explains the choice between 'Beam Diagram' and 'Beam Contour'.

Section	Property	Value
Contour	Style	Beam Diagram
	Results	
Results	Output Set	25..Simcenter NASTRAN Ca
	Output Vector	3018..Beam EndA P11 Shear
	Additional Vector(s)	<input type="checkbox"/>
Options	Data Selection	All Data/Full Model
	Show On Groups	Active Group
	Show As	Beam Diagram
	Label	No Labels
	Direction	Element Y
	Show Reversed	<input type="checkbox"/>
	Scale %	10
	Border Color	62
	Levels	
	Legend	<input checked="" type="checkbox"/>
Freebody		
Show As		
Choose to display a Beam Diagram plot as a Beam Diagram or a Beam Contour.		

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. Note, beam elements required.



Contour Style – Contour Arrow

PostProcessing Toolbox

Deform

Contour

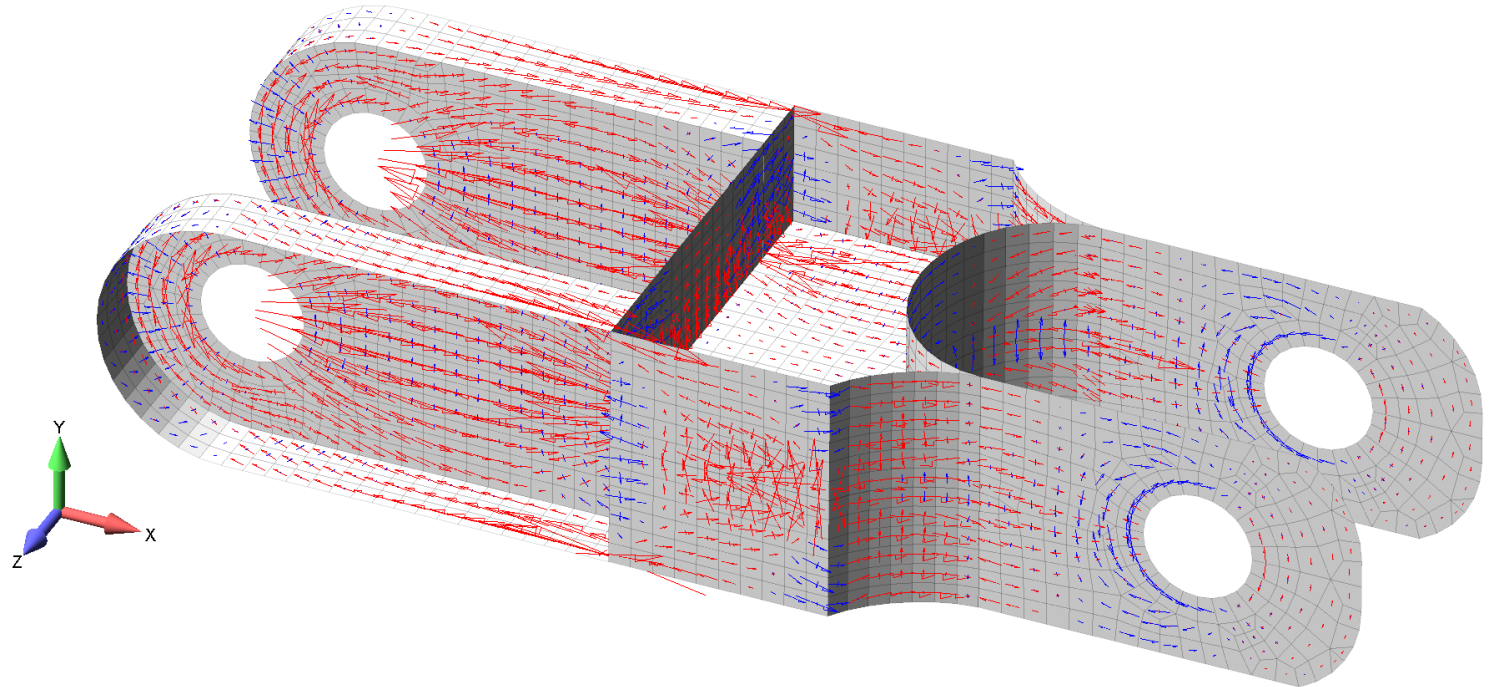
Style	Contour Arrow
Results	
Output Set	1..FIXED - PULL
Output Vector	7027..Plate Top MinorPrn Stress
Options	
Select Arrows from Contour Vector	<input checked="" type="checkbox"/>
Arrows	
Arrow Type	2D Components
X Arrow Display / Vector Select	<input checked="" type="checkbox"/> 7026..Plate Top MajorPrn Stress
Y Arrow Display / Vector Select	<input checked="" type="checkbox"/> 7027..Plate Top MinorPrn Stress
Transform	
Data Selection	Contour Group
Show On Groups	Full Model / Visible Groups
Arrow Head and Color	<input type="checkbox"/> Auto
Solid Arrows	<input type="checkbox"/>
Arrow Length	0.01
Min Vector Magnitude	<input checked="" type="checkbox"/> 1.E-8
Arrow Labels	Off
Levels	
Legend	<input checked="" type="checkbox"/>

Freebody

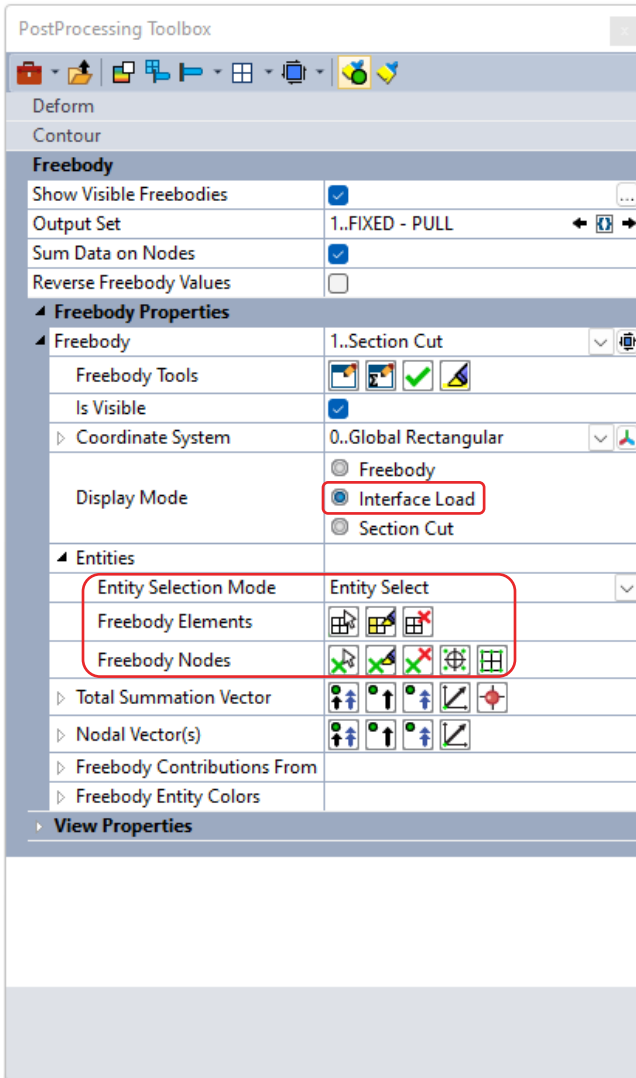
Legend
On by default. When On, the Contour/Criteria Legend will be displayed. When Off, no Legend will be visible.

Contour Arrow

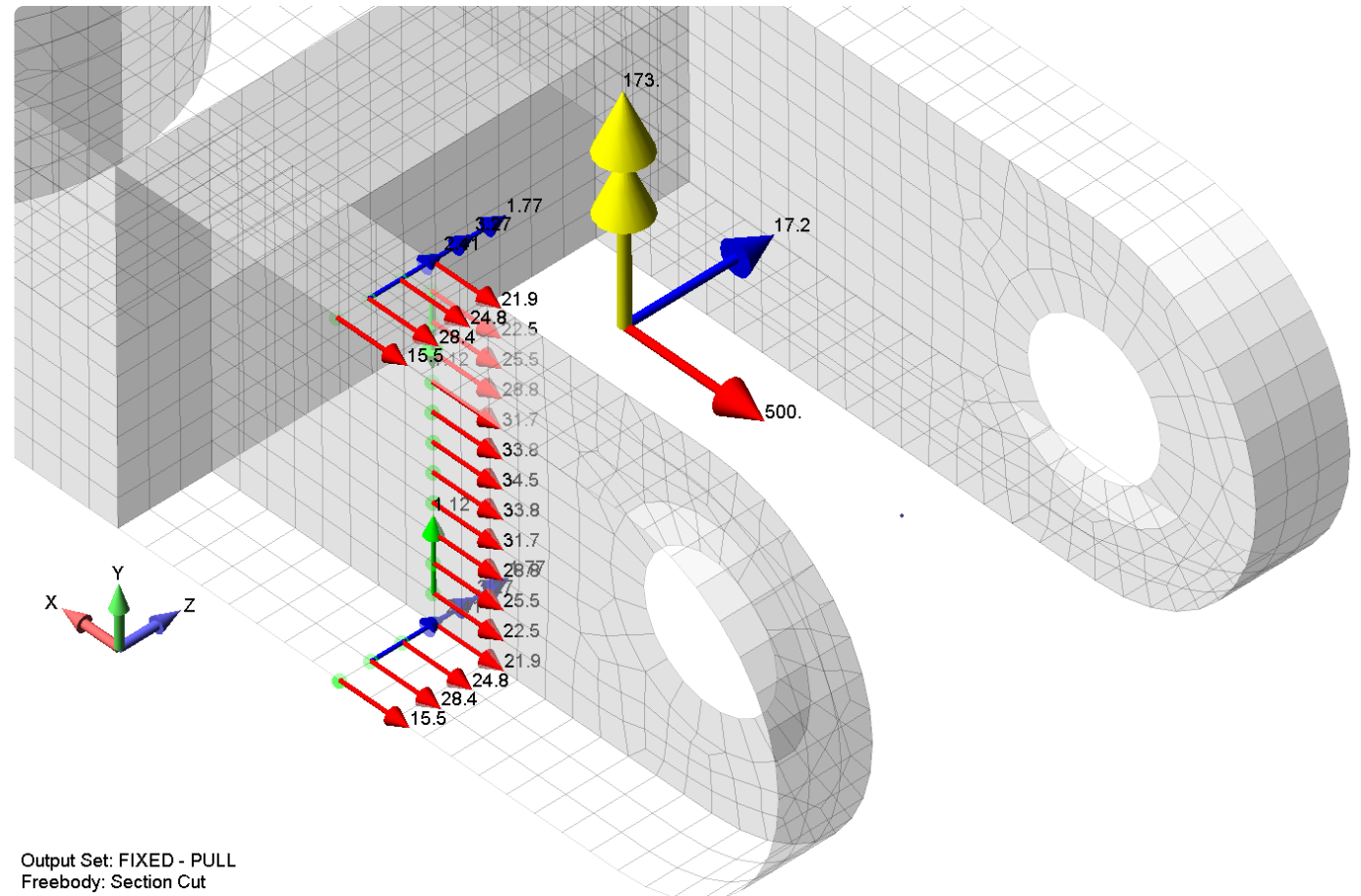
This view style is most useful for looking at the directional flow of stress with the principal stress output vectors. The Vector view style places vectors on the elements to show the direction and magnitude of stresses and forces.



Freebody View Style – 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.



Freebody View Style – Section Cut

PostProcessing Toolbox

Deform

Contour

Freebody

Show Visible Freebodies

Output Set 1..FIXED - PULL

Sum Data on Nodes

Reverse Freebody Values

Freebody Properties

Freebody 1..Section Cut

Freebody Tools

Is Visible

Coordinate System 0..Global Rectangular

Display Mode

Freebody

Interface Load

Section Cut

Entities

Entity Selection Mode Plane / Normal

Plane [0.887498,0.,0.]
[1.,0.,0.]

Location Slider 43

Summation Location Section Cut Path

Align Sums to Path

Section Cut Tools

Plane Offset Percentage 10.

Total Summation Vector

Nodal Vector(s)

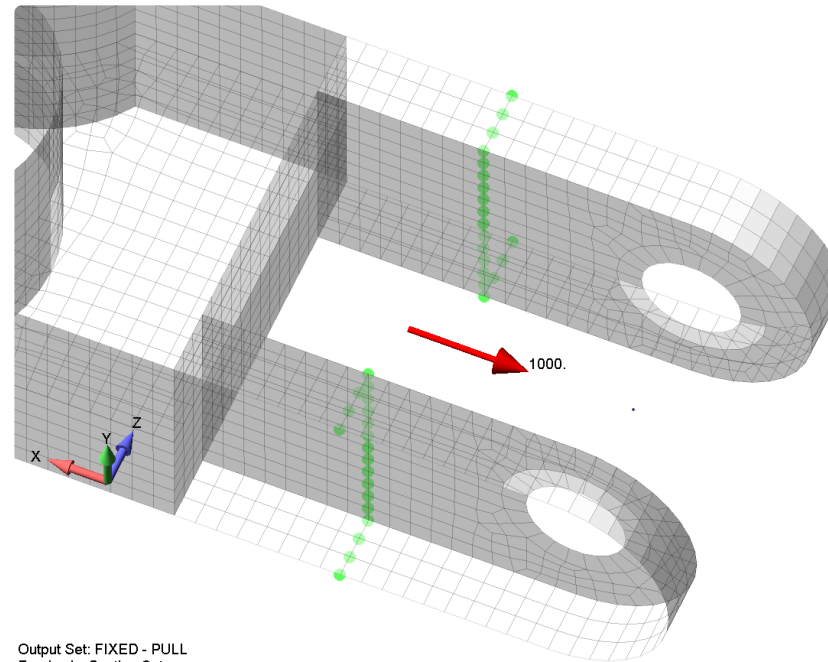
Freebody Contributions From

Freebody Entity Colors

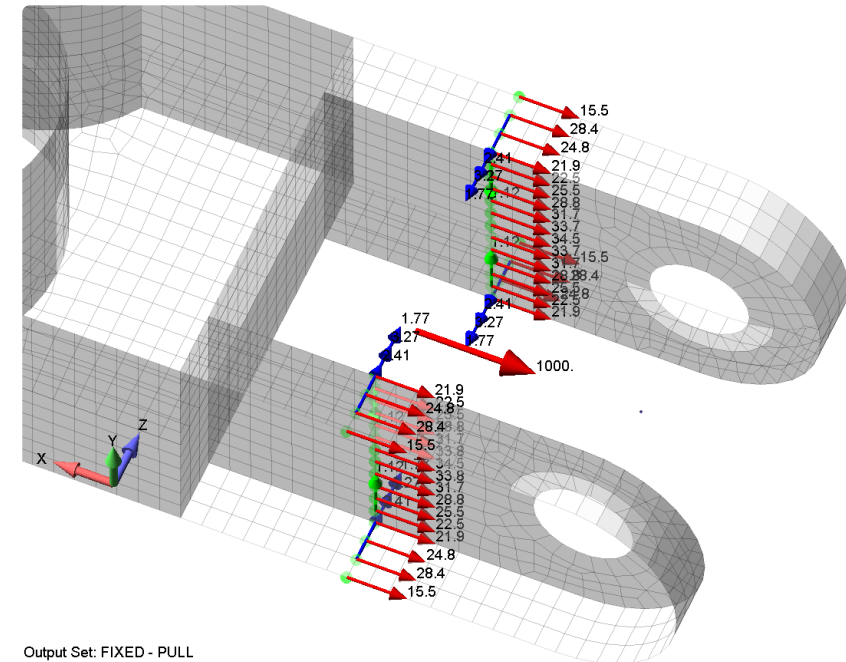
View Properties

Entities

“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 - PULL
Freebody: Section Cut

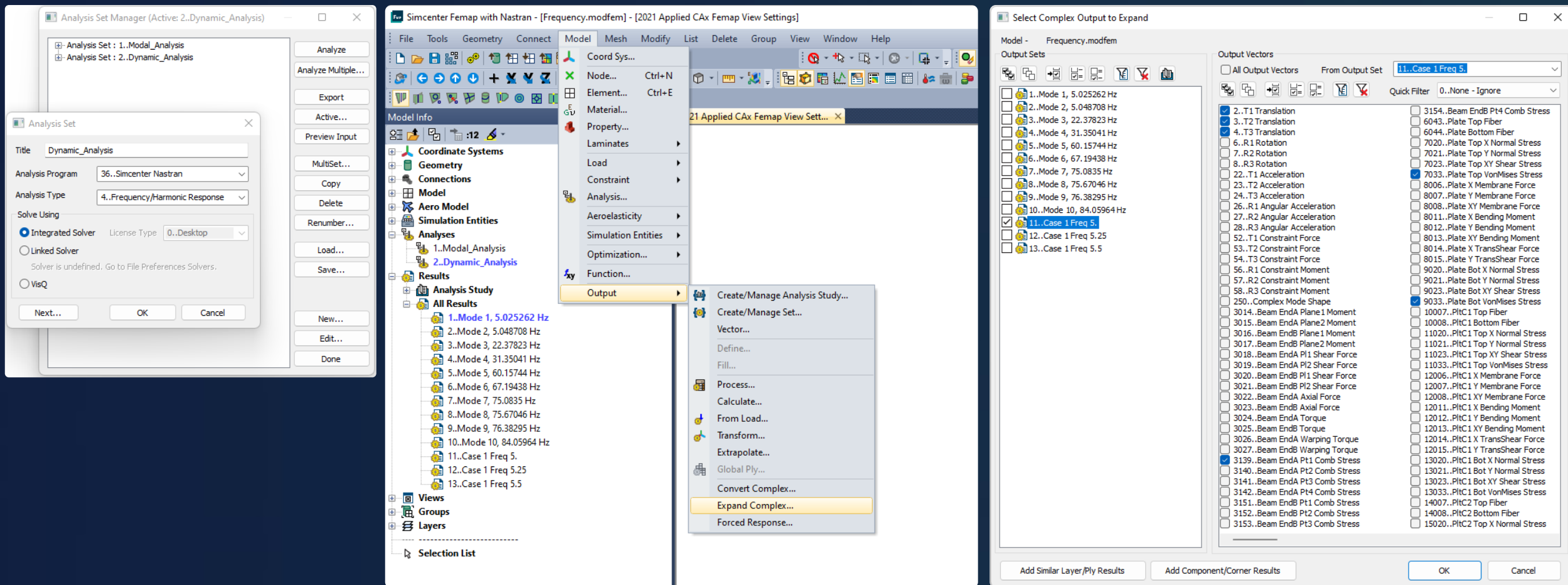


Output Set: FIXED - PULL
Freebody: Section Cut

Charting

Charting – Expanding Complex Results

After running a Frequency/Harmonic Response analysis it is often useful to expand the complex results. Charting is a great way to visualize these expanded results.



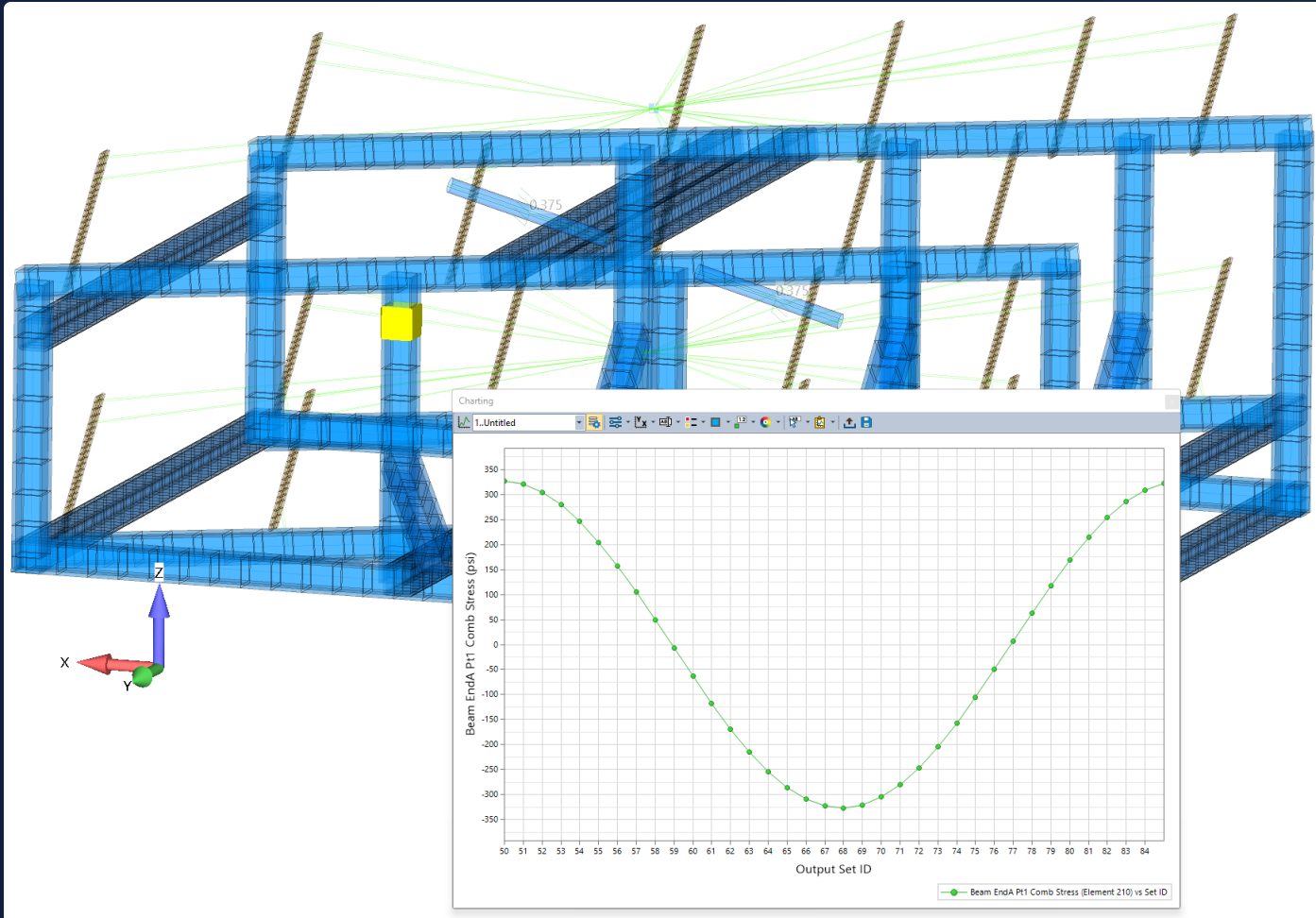
The image displays three screenshots from the Simcenter Femap software interface, illustrating the steps to expand complex results for charting.

Left Screenshot: Analysis Set Manager
This window shows the configuration for the analysis set. The "Analysis Set" dialog is open, showing the title "Dynamic_Analysis", the analysis program "36..Simcenter Nastran", and the analysis type "4..Frequency/Harmonic Response". The "Solve Using" section is set to "Integrated Solver".

Middle Screenshot: Simcenter Femap with Nastran
The main application window shows the "Model" menu open, with the "Output" option selected. The "Output" submenu is visible, showing options like "Create/Manage Analysis Study...", "Vector...", "Define...", "Fill...", "Process...", "Calculate...", "From Load...", "Transform...", "Extrapolate...", "Global Ply...", "Convert Complex...", "Expand Complex...", and "Forced Response...". The "Expand Complex..." option is highlighted.

Right Screenshot: Select Complex Output to Expand
This dialog box shows the "Output Vectors" section. The "From Output Set" dropdown is set to "11..Case 1 Freq 5". The "Quick Filter" is set to "0..None - Ignore". The list of output vectors includes various stress and force components, with several items checked, such as "2..T1 Translation", "3..T2 Translation", "4..T3 Translation", "6..R1 Rotation", "7..R2 Rotation", "8..R3 Rotation", "22..T1 Acceleration", "23..T2 Acceleration", "24..T3 Acceleration", "26..R1 Angular Acceleration", "27..R2 Angular Acceleration", "28..R3 Angular Acceleration", "52..T1 Constraint Force", "53..T2 Constraint Force", "54..T3 Constraint Force", "56..R1 Constraint Moment", "57..R2 Constraint Moment", "58..R3 Constraint Moment", "250..Complex Mode Shape", "3014..Beam EndA Plane1 Moment", "3015..Beam EndA Plane2 Moment", "3016..Beam EndB Plane1 Moment", "3017..Beam EndB Plane2 Moment", "3018..Beam EndA Pl1 Shear Force", "3019..Beam EndA Pl2 Shear Force", "3020..Beam EndB Pl1 Shear Force", "3021..Beam EndB Pl2 Shear Force", "3022..Beam EndA Axial Force", "3023..Beam EndB Axial Force", "3024..Beam EndA Torque", "3025..Beam EndB Torque", "3026..Beam EndA Warping Torque", "3027..Beam EndB Warping Torque", "3139..Beam EndA Pt1 Comb Stress", "3140..Beam EndA Pt2 Comb Stress", "3141..Beam EndA Pt3 Comb Stress", "3142..Beam EndA Pt4 Comb Stress", "3151..Beam EndB Pt1 Comb Stress", "3152..Beam EndB Pt2 Comb Stress", "3153..Beam EndB Pt3 Comb Stress", "3154..Beam EndB Pt4 Comb Stress", "6043..Plate Top Fiber", "6044..Plate Bottom Fiber", "7020..Plate Top X Normal Stress", "7021..Plate Top Y Normal Stress", "7023..Plate Top XY Shear Stress", "7033..Plate Top VonMises Stress", "8006..Plate X Membrane Force", "8007..Plate Y Membrane Force", "8008..Plate XY Membrane Force", "8011..Plate X Bending Moment", "8012..Plate Y Bending Moment", "8013..Plate XY Bending Moment", "8014..Plate X TransShear Force", "8015..Plate Y TransShear Force", "9020..Plate Bot X Normal Stress", "9021..Plate Bot Y Normal Stress", "9023..Plate Bot XY Shear Stress", "9033..Plate Bot VonMises Stress", "10007..PltC1 Top Fiber", "10008..PltC1 Bottom Fiber", "11020..PltC1 Top X Normal Stress", "11021..PltC1 Top Y Normal Stress", "11023..PltC1 Top XY Shear Stress", "11033..PltC1 Top VonMises Stress", "12006..PltC1 X Membrane Force", "12007..PltC1 Y Membrane Force", "12008..PltC1 XY Membrane Force", "12011..PltC1 X Bending Moment", "12012..PltC1 Y Bending Moment", "12013..PltC1 XY Bending Moment", "12014..PltC1 X TransShear Force", "12015..PltC1 Y TransShear Force", "13020..PltC1 Bot X Normal Stress", "13021..PltC1 Bot Y Normal Stress", "13023..PltC1 Bot XY Shear Stress", "13033..PltC1 Bot VonMises Stress", "14007..PltC2 Top Fiber", "14008..PltC2 Bottom Fiber", "15020..PltC2 Top X Normal Stress".

Charting – Charting Complex Results



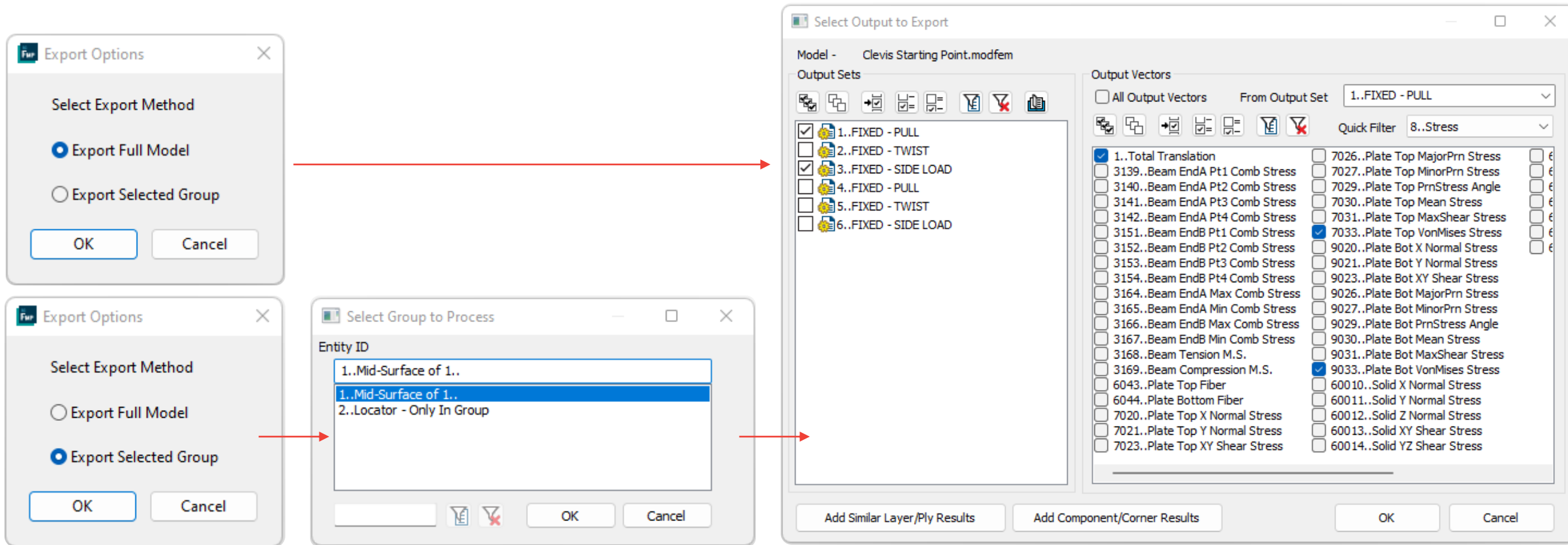
The Charting Pane provides a simple interface for creating plots of output data as well as function data without relying on an external program, such as Excel.

It's a quick and easy way to get data into Excel as well if that's ultimately where you want the information.

Here, we can see the P1 Comb Stress in the square tube (highlighted) for one full cycle, or complete load reversal.

Group_Post_FNO

“Group_Post_FNO” is the API for you if your model generates extremely large output files. This tool can be used to export a portion of the results from an analysis. Working with smaller results subsets will improve performance and may be the only practical way of sending/sharing results via the internet if the output file is enormous.

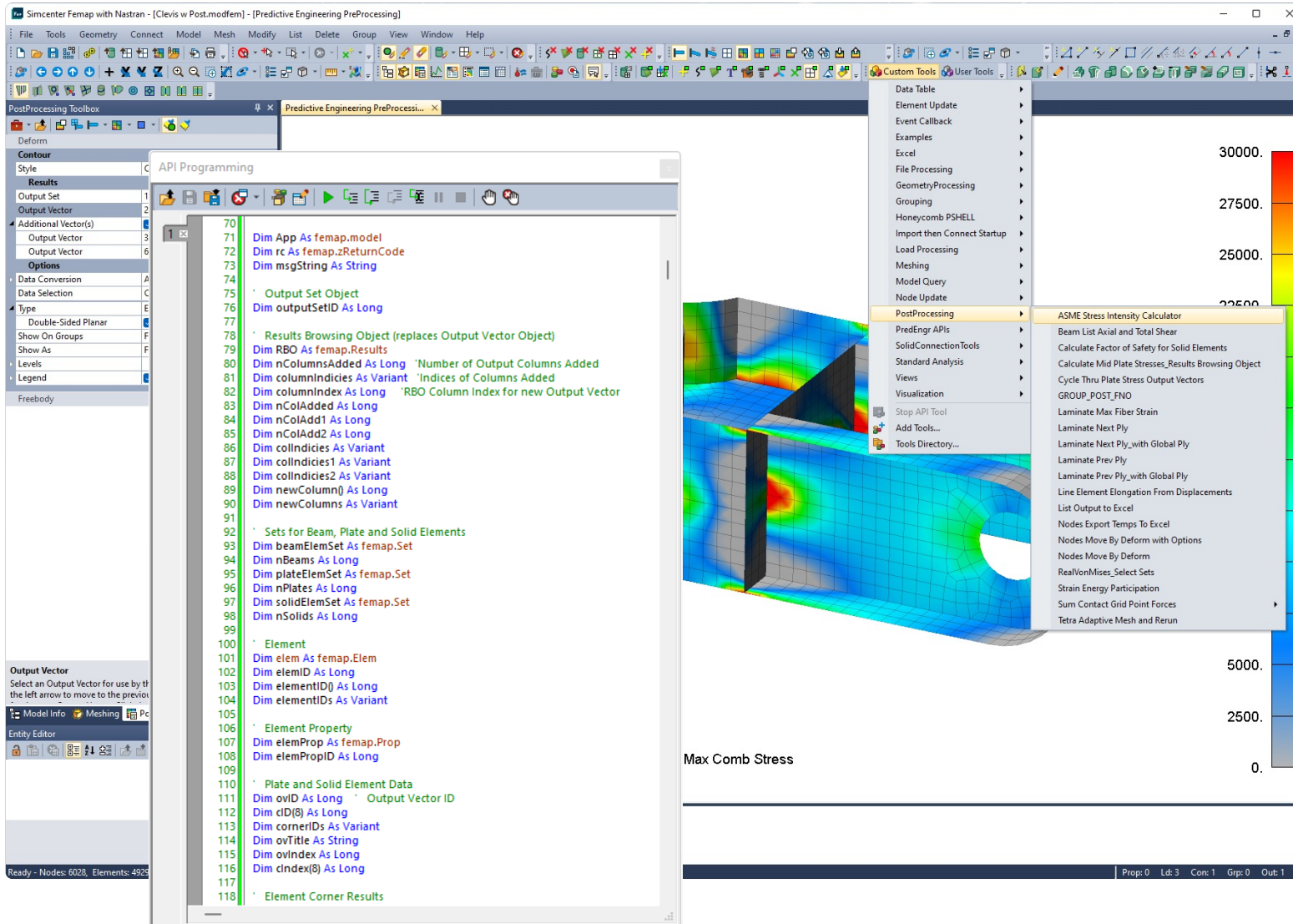


The image illustrates the workflow for using the Group_Post_FNO API in Femap. It consists of three main dialog boxes:

- Export Options (Top Left):** Shows the selection of the export method. The "Export Full Model" radio button is selected.
- Export Options (Bottom Left):** Shows the selection of the export method. The "Export Selected Group" radio button is selected.
- Select Group to Process (Middle):** A list box showing the "Entity ID" for the selected group. The first two items, "1..Mid-Surface of 1.." and "1..Mid-Surface of 1..", are selected.
- Select Output to Export (Right):** A large dialog box showing the "Output Sets" and "Output Vectors" for the selected model. The "Output Sets" list includes "1..FIXED - PULL", "2..FIXED - TWIST", "3..FIXED - SIDE LOAD", "4..FIXED - PULL", "5..FIXED - TWIST", and "6..FIXED - SIDE LOAD". The "Output Vectors" list includes "1..Total Translation", "3139..Beam EndA Pt1 Comb Stress", "3140..Beam EndA Pt2 Comb Stress", "3141..Beam EndA Pt3 Comb Stress", "3142..Beam EndA Pt4 Comb Stress", "3151..Beam EndB Pt1 Comb Stress", "3152..Beam EndB Pt2 Comb Stress", "3153..Beam EndB Pt3 Comb Stress", "3154..Beam EndB Pt4 Comb Stress", "3164..Beam EndA Max Comb Stress", "3165..Beam EndA Min Comb Stress", "3166..Beam EndB Max Comb Stress", "3167..Beam EndB Min Comb Stress", "3168..Beam Tension M.S.", "3169..Beam Compression M.S.", "6043..Plate Top Fiber", "6044..Plate Bottom Fiber", "7020..Plate Top X Normal Stress", "7021..Plate Top Y Normal Stress", "7023..Plate Top XY Shear Stress", "7026..Plate Top MajorPrn Stress", "7027..Plate Top MinorPrn Stress", "7029..Plate Top PrnStress Angle", "7030..Plate Top Mean Stress", "7031..Plate Top MaxShear Stress", "7033..Plate Top VonMises Stress", "9020..Plate Bot X Normal Stress", "9021..Plate Bot Y Normal Stress", "9023..Plate Bot XY Shear Stress", "9026..Plate Bot MajorPrn Stress", "9027..Plate Bot MinorPrn Stress", "9029..Plate Bot PrnStress Angle", "9030..Plate Bot Mean Stress", "9031..Plate Bot MaxShear Stress", "9033..Plate Bot VonMises Stress", "60010..Solid X Normal Stress", "60011..Solid Y Normal Stress", "60012..Solid Z Normal Stress", "60013..Solid XY Shear Stress", and "60014..Solid YZ Shear Stress".

Red arrows indicate the flow of data: from the "Export Full Model" option to the "Select Output to Export" dialog, from the "Export Selected Group" option to the "Select Group to Process" dialog, and from the "Select Group to Process" dialog to the "Select Output to Export" dialog.

ASME Stress Intensity Calculator



The screenshot displays the Simcenter Femap interface with a stress analysis visualization of a mechanical part. A color scale on the right indicates stress intensity, ranging from 0 (blue) to 30,000 (red). The text "Max Comb Stress" is visible near the bottom of the visualization. In the foreground, the "API Programming" window is open, showing a list of API commands for the ASME Stress Intensity Calculator. The "Custom Tools" menu is also open, highlighting the "ASME Stress Intensity Calculator" option.

```
70  
71 Dim App As femap.model  
72 Dim rc As femap.zReturnCode  
73 Dim msgString As String  
74  
75 * Output Set Object  
76 Dim outputSetID As Long  
77  
78 * Results Browsing Object (replaces Output Vector Object)  
79 Dim RBO As femap.Results  
80 Dim nColumnsAdded As Long 'Number of Output Columns Added  
81 Dim columnIndices As Variant 'Indices of Columns Added  
82 Dim columnIndex As Long 'RBO Column Index for new Output Vector  
83 Dim nColAdded1 As Long  
84 Dim nColAdd1 As Long  
85 Dim nColAdd2 As Long  
86 Dim colIndices1 As Variant  
87 Dim colIndices2 As Variant  
88 Dim colIndices3 As Variant  
89 Dim newColumn() As Long  
90 Dim newColumns As Variant  
91  
92 * Sets for Beam, Plate and Solid Elements  
93 Dim beamElemSet As femap.Set  
94 Dim nBeams As Long  
95 Dim plateElemSet As femap.Set  
96 Dim nPlates As Long  
97 Dim solidElemSet As femap.Set  
98 Dim nSolids As Long  
99  
100 * Element  
101 Dim elem As femap.Elem  
102 Dim elemID As Long  
103 Dim elementID() As Long  
104 Dim elementIDs As Variant  
105  
106 * Element Property  
107 Dim elemProp As femap.Prop  
108 Dim elemPropID As Long  
109  
110 * Plate and Solid Element Data  
111 Dim ovID As Long ' Output Vector ID  
112 Dim cID(8) As Long  
113 Dim cornerIDs As Variant  
114 Dim ovTitle As String  
115 Dim ovIndex As Long  
116 Dim cindex(8) As Long  
117  
118 * Element Corner Results
```

“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.

Additionally, if you need to create your own custom output vectors, this API could be a good starting point.