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

Applied CAx

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

We Do This Everyday

AppliedCAx

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

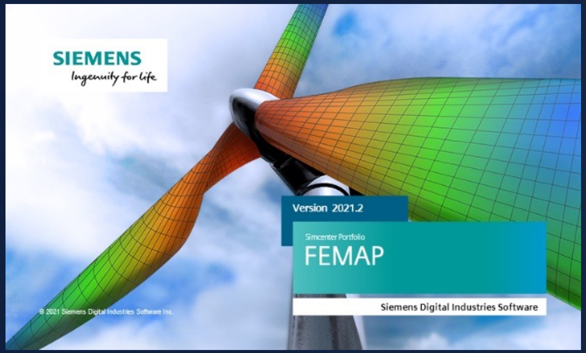
Our Next Femap Training Opportunity
October 31st – November 10th, 2022
Live, Online
AppliedCAx.com/Training

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

AppliedCAx

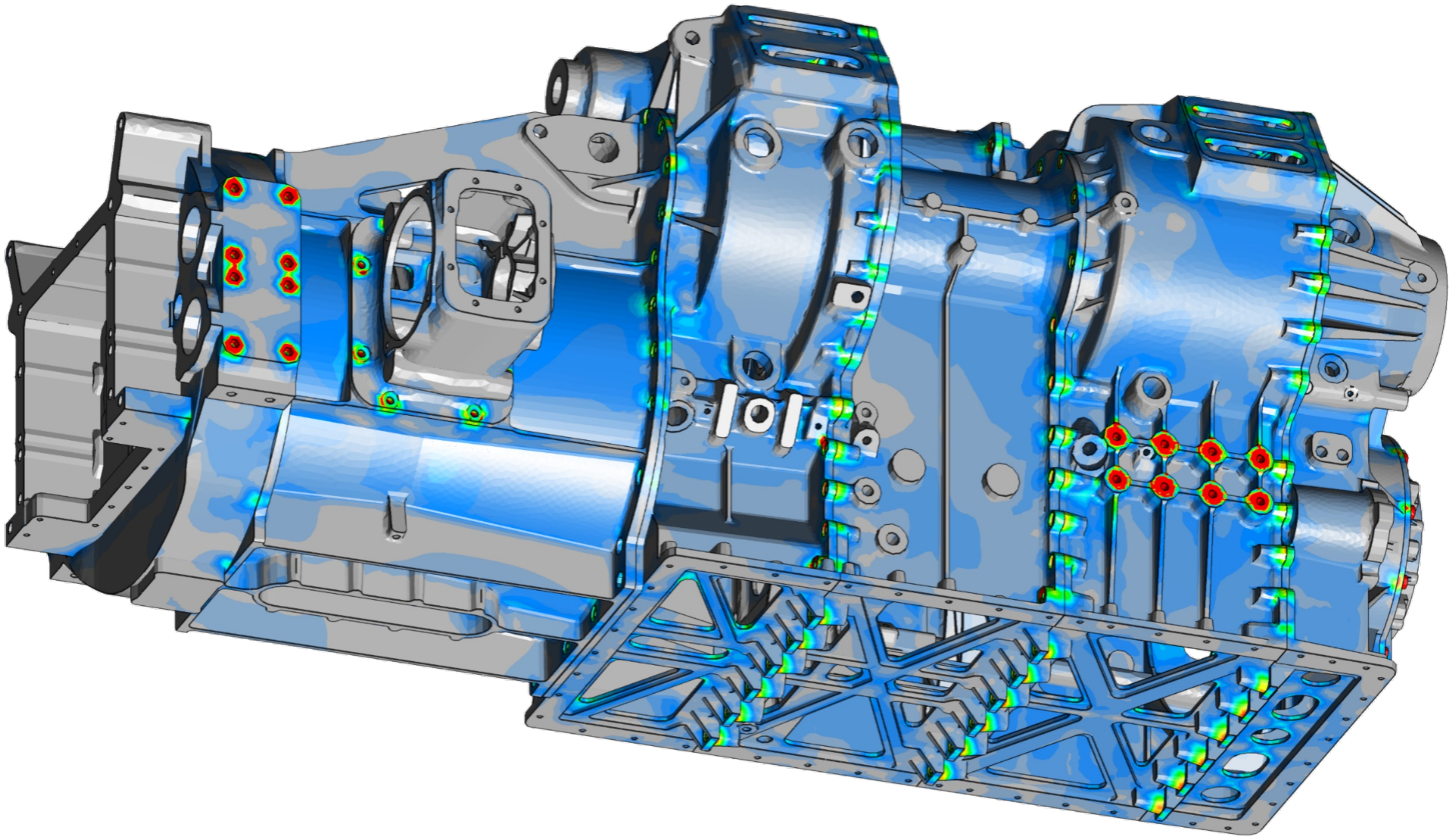
Seminar Outline

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

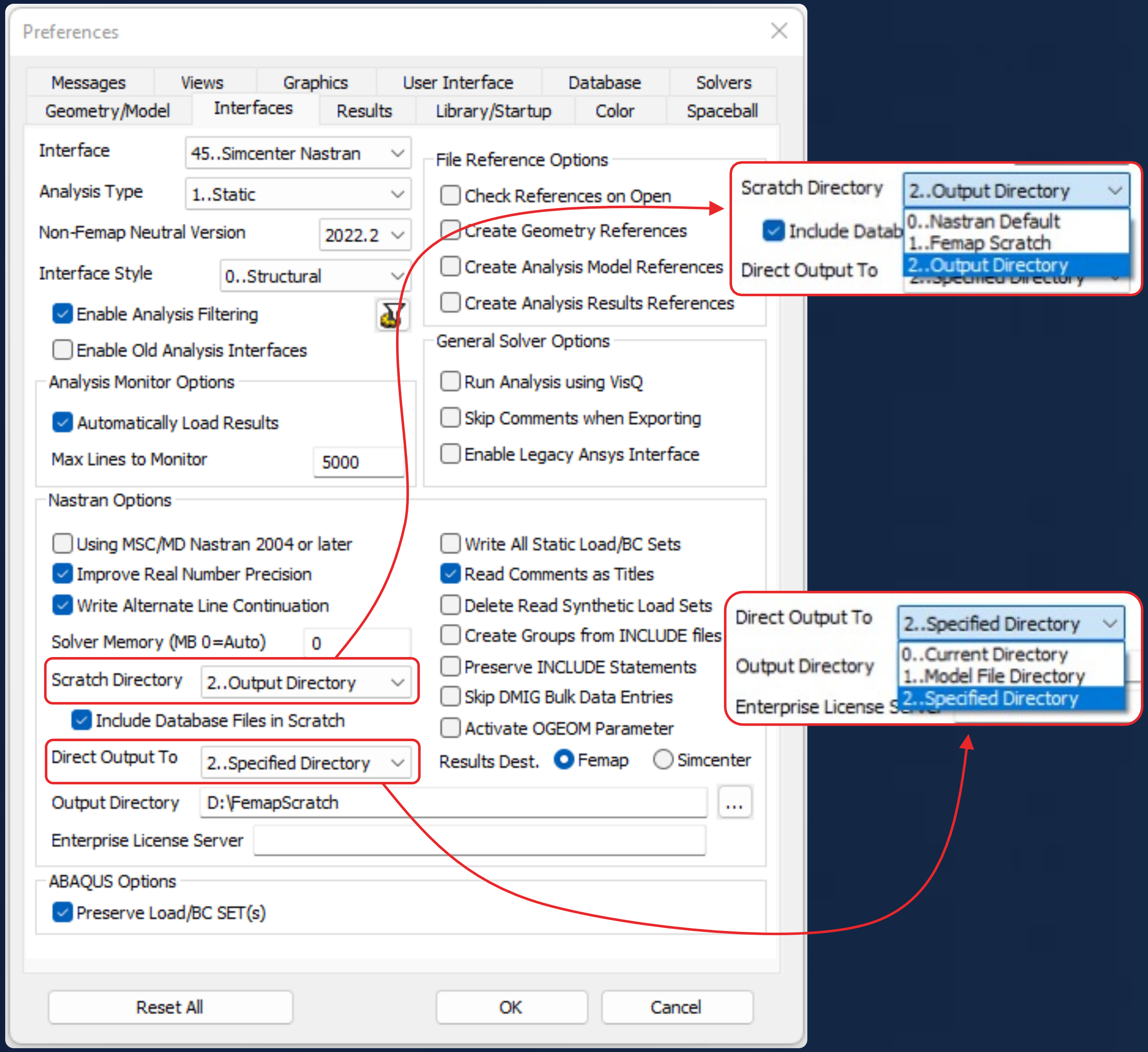
Charting
- Expanding Complex Results
- Charting Complex Results

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

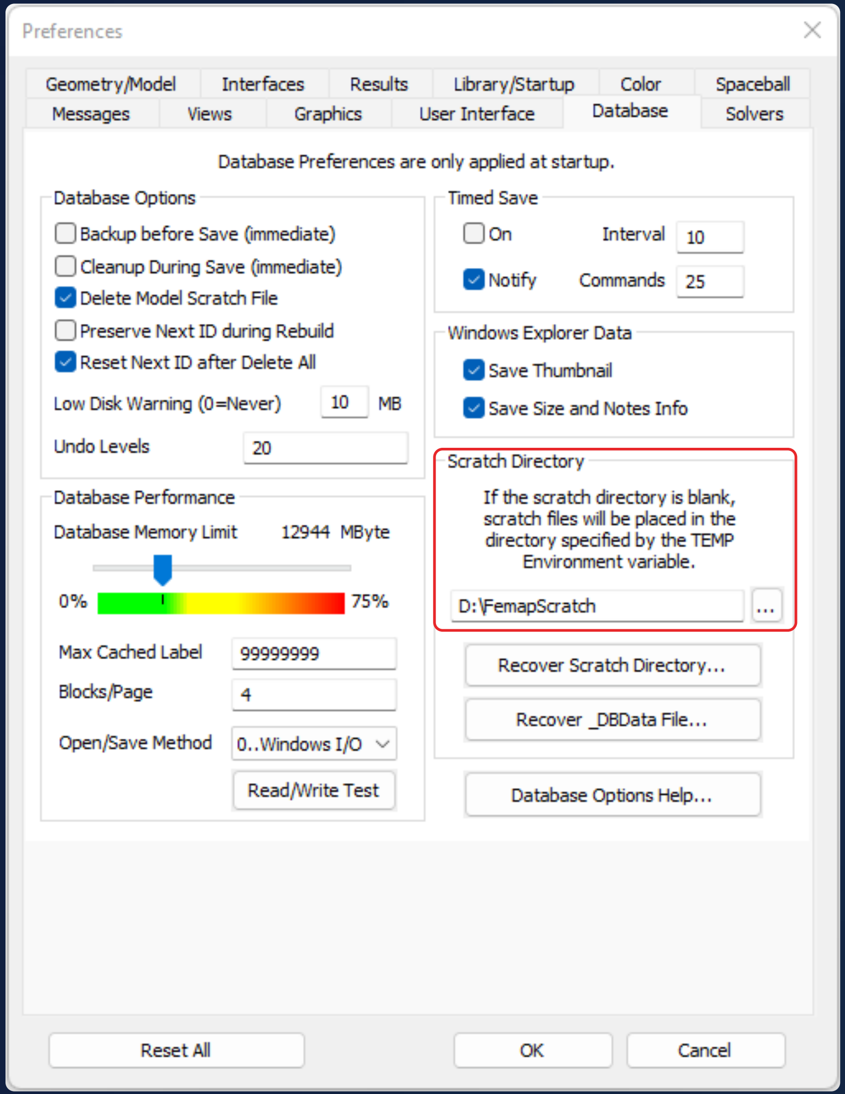
0..Current Directory: Last used directory by FEMAP. If a model has been saved to a directory, the output will be directed to that directory when this option is on.

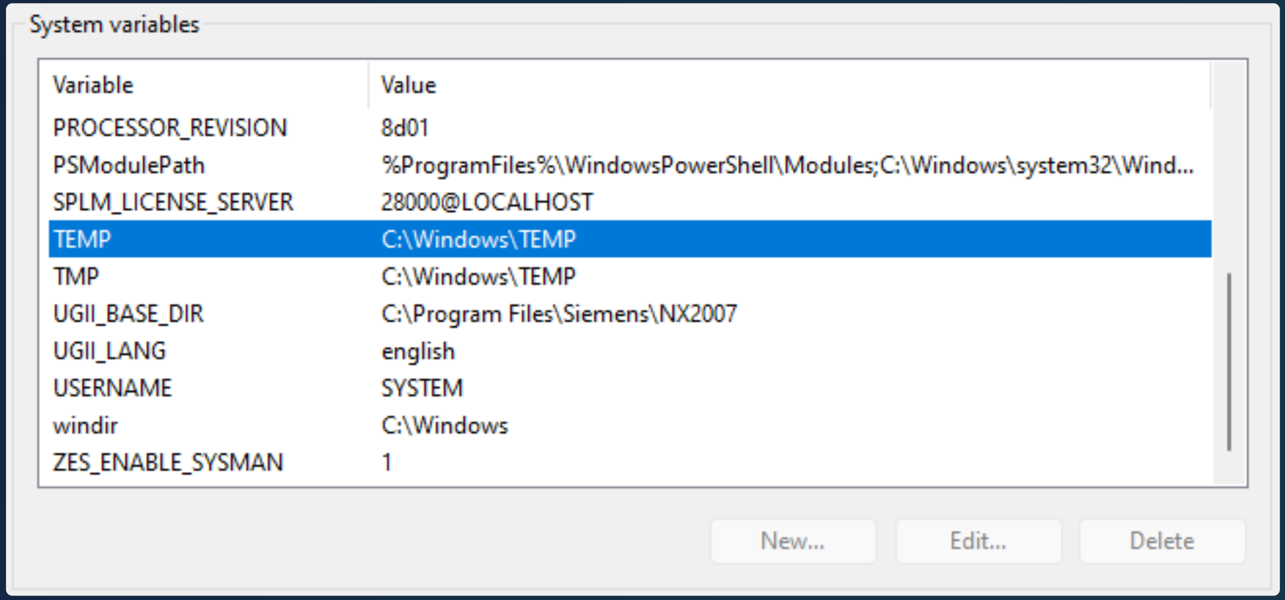
1..Model File Directory (default): The directory where the model file is located. All output will go into this directory until the model is saved somewhere else.

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

Output Management

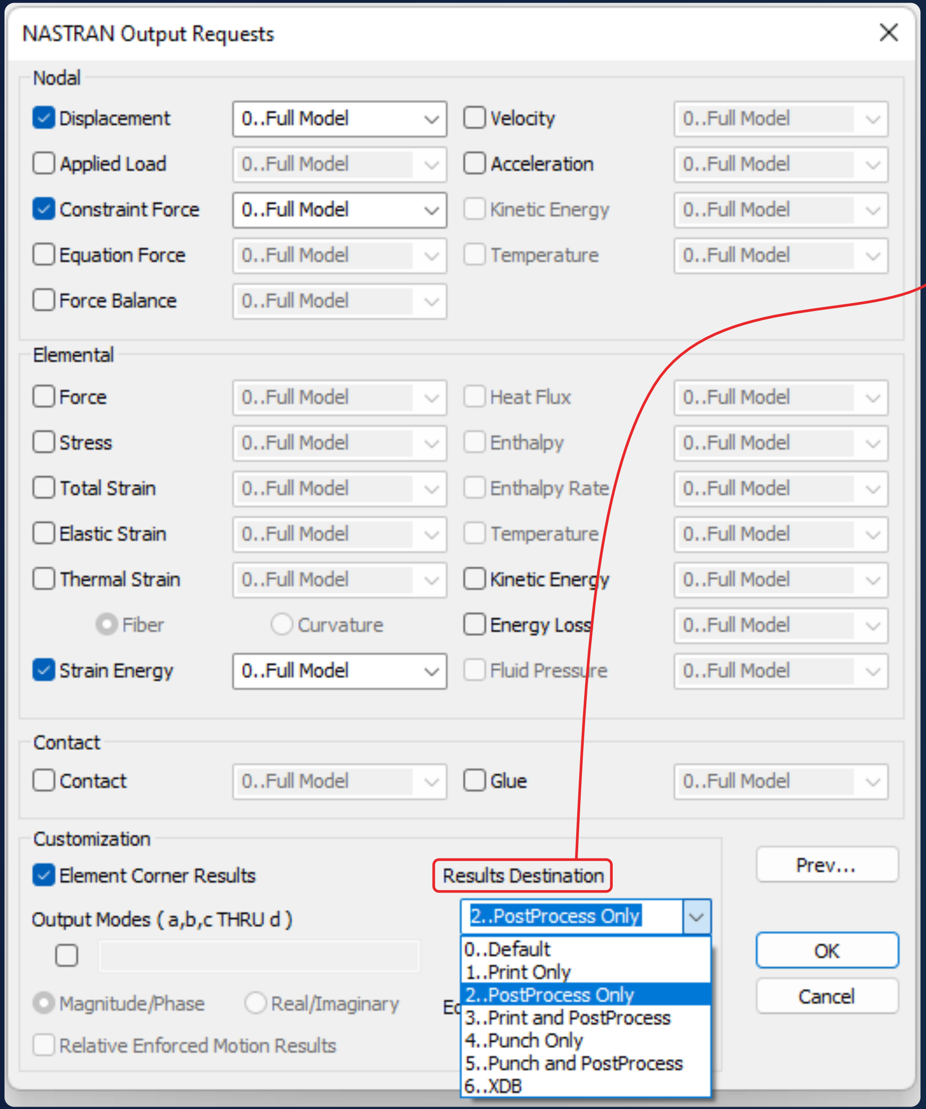
Femap Scratch Files





Output Management

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

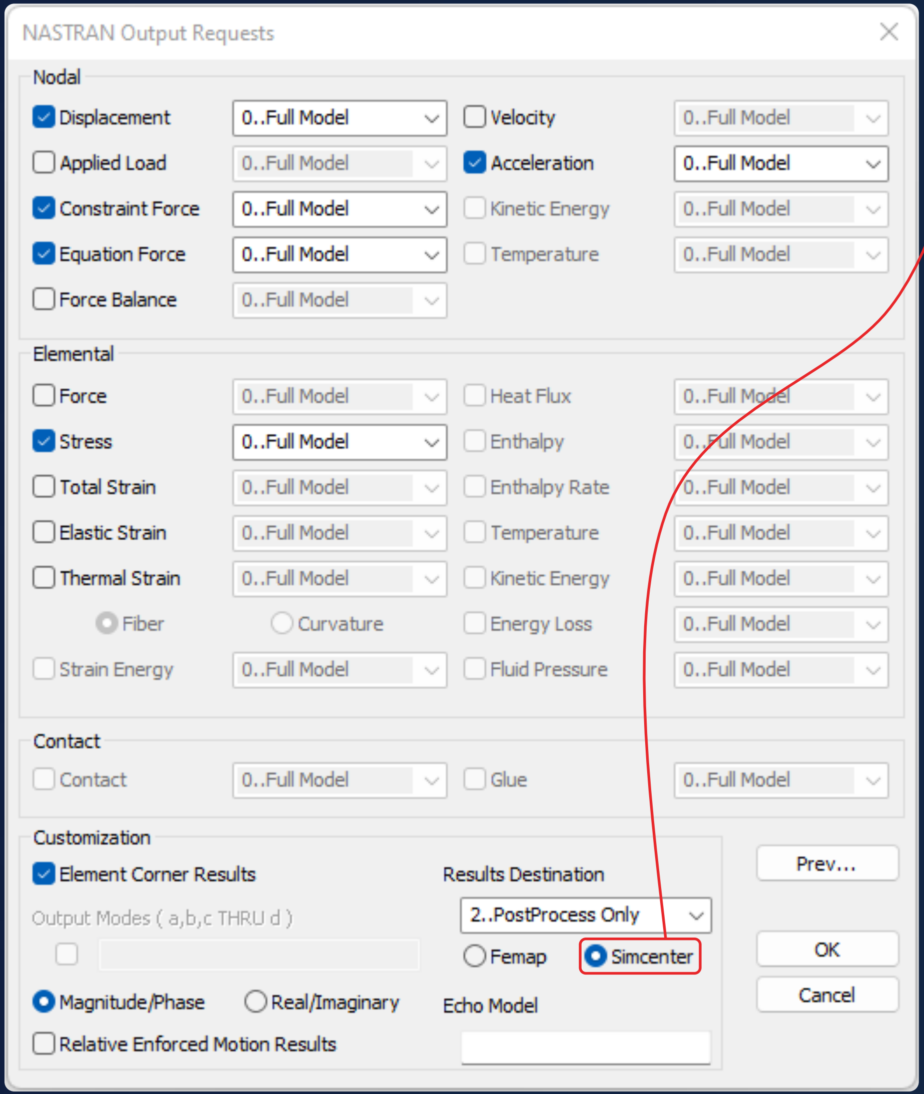
6..XDB = *.xdb – A binary file known as a “results database” file

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

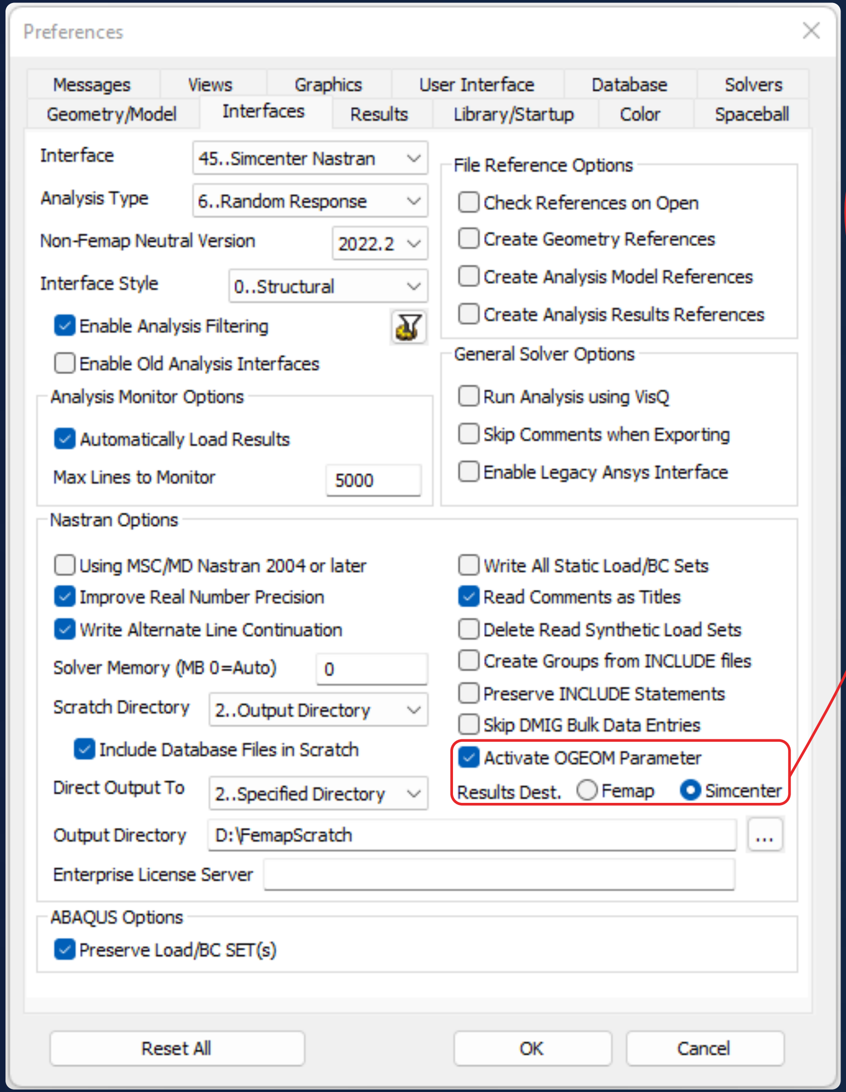


As of version 2022.1 it is now possible to instruct Nastran to create an output file, in particular an .op2 file, which can be used by other applications in the Siemens Software Portfolio, including Simcenter 3D.

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

Output Management

Nastran Output Requests



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

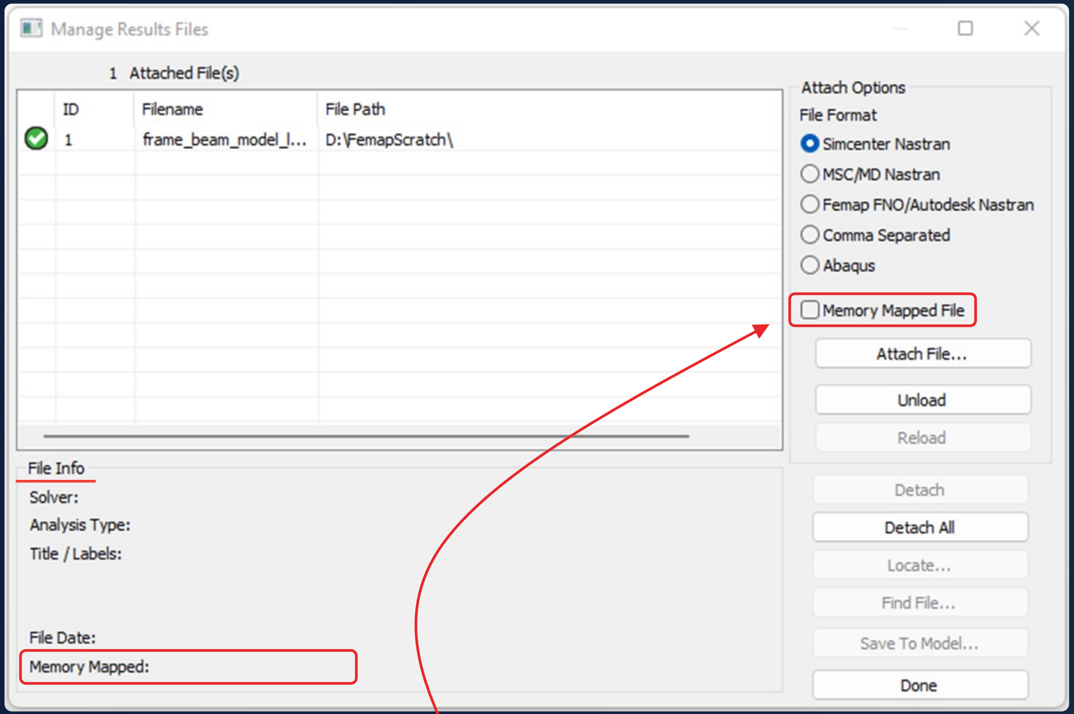
To do this: File → Preferences. In the Preferences dialog box, choose the Interfaces tab, and set the Results Destination option to Simcenter.

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.

Output Management

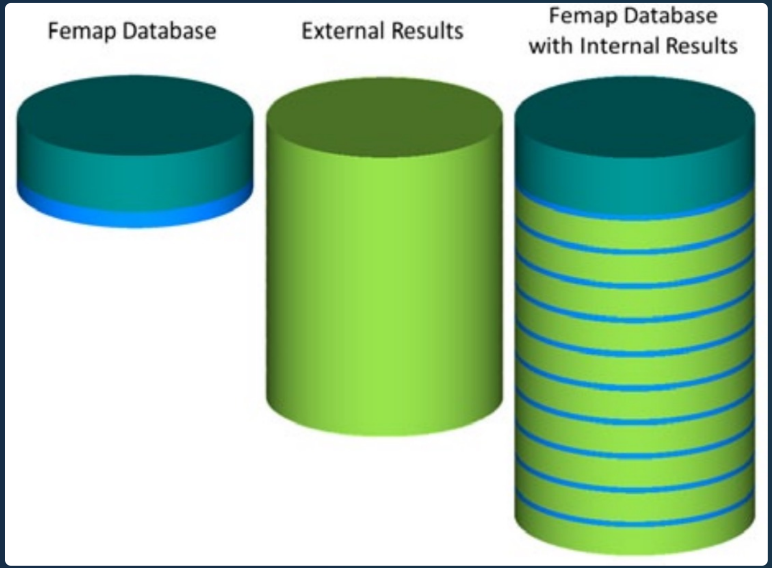
File Options - Attach to Results



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 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.

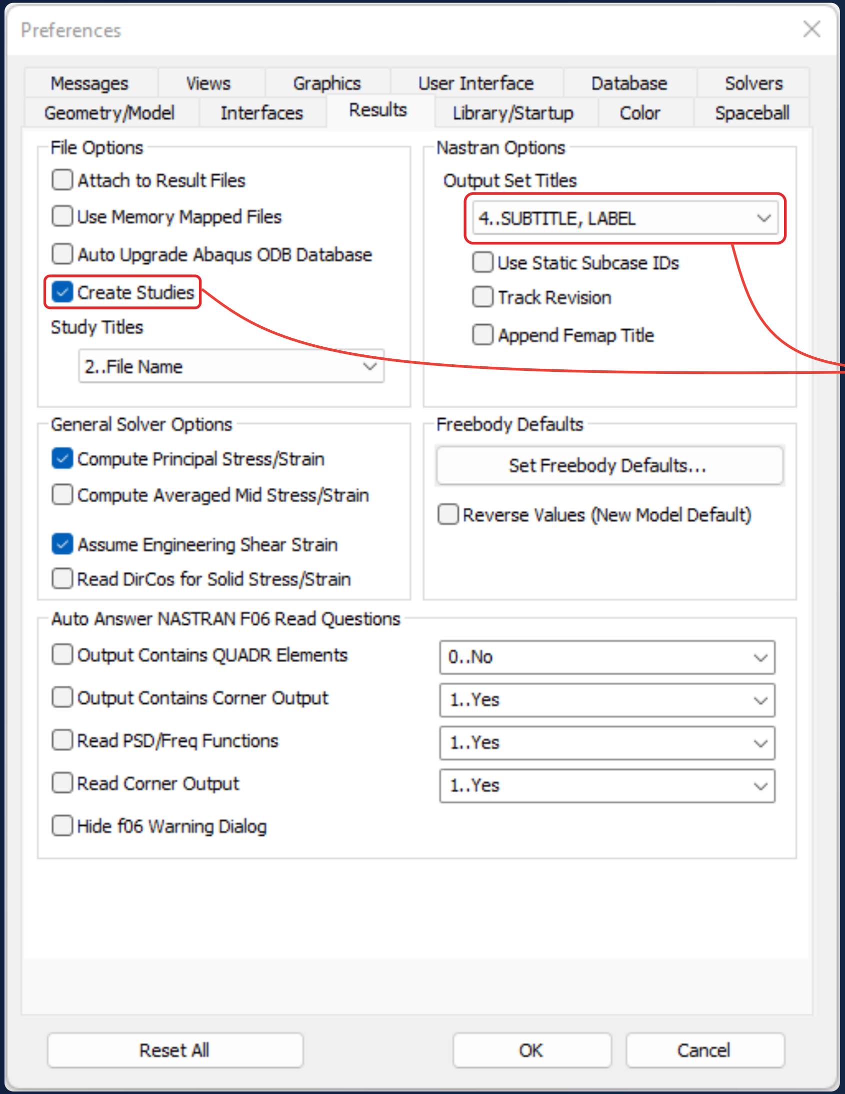
Maintaining external results data in this way minimizes the Femap database size and makes results data handing much more manageable.



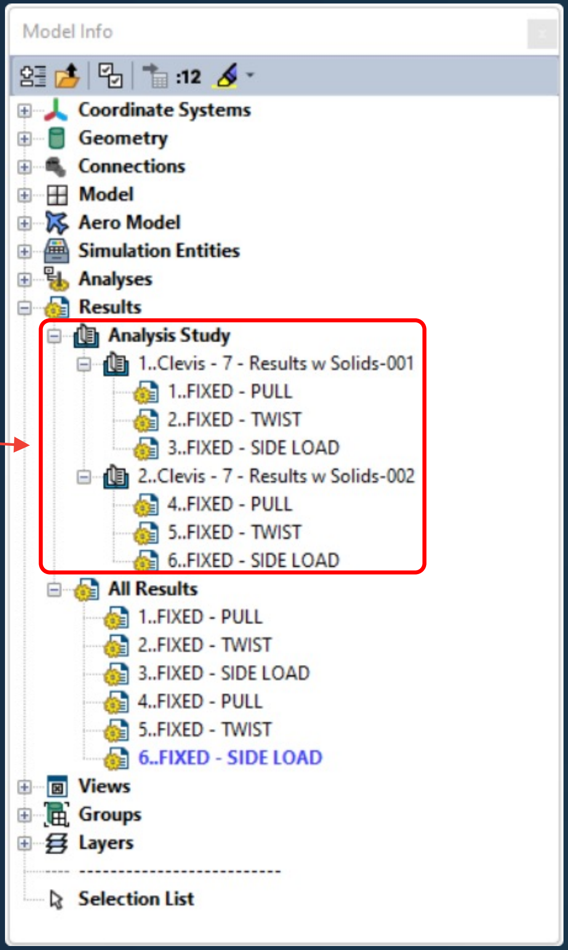
Note: Within File > Preferences you can choose to automatically attach to results files and use “Memory Mapping” for a potential increase in speed. When Memory Mapping is turned on, FEMAP will attempt to attach to the file using RAM. In order for an attached output file to be properly “Memory Mapped” 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

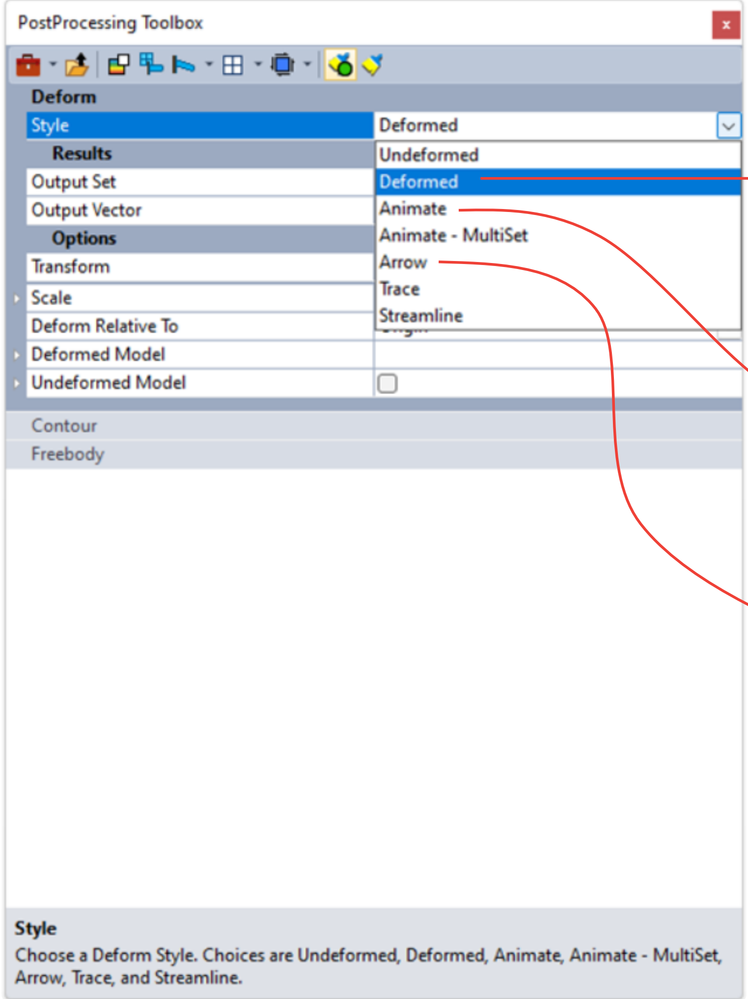


You can use Analysis Studies to group your output sets for better organization and data processing.



PostProcessing Toolbox

Deform 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 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 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.

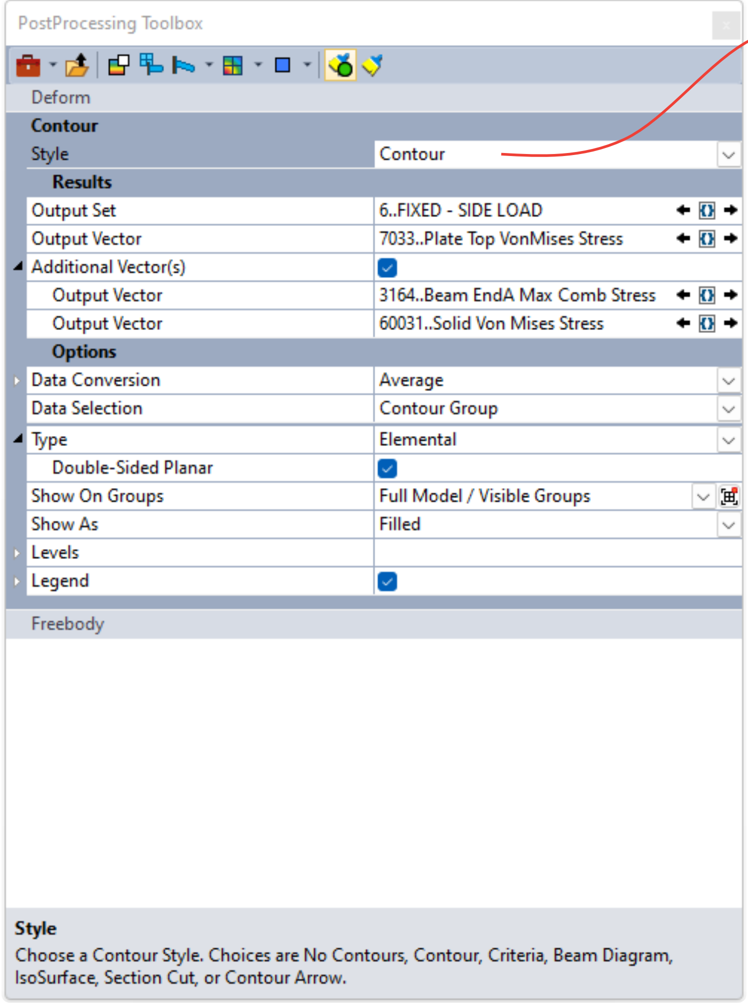
Arrow

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.

PostProcessing Toolbox

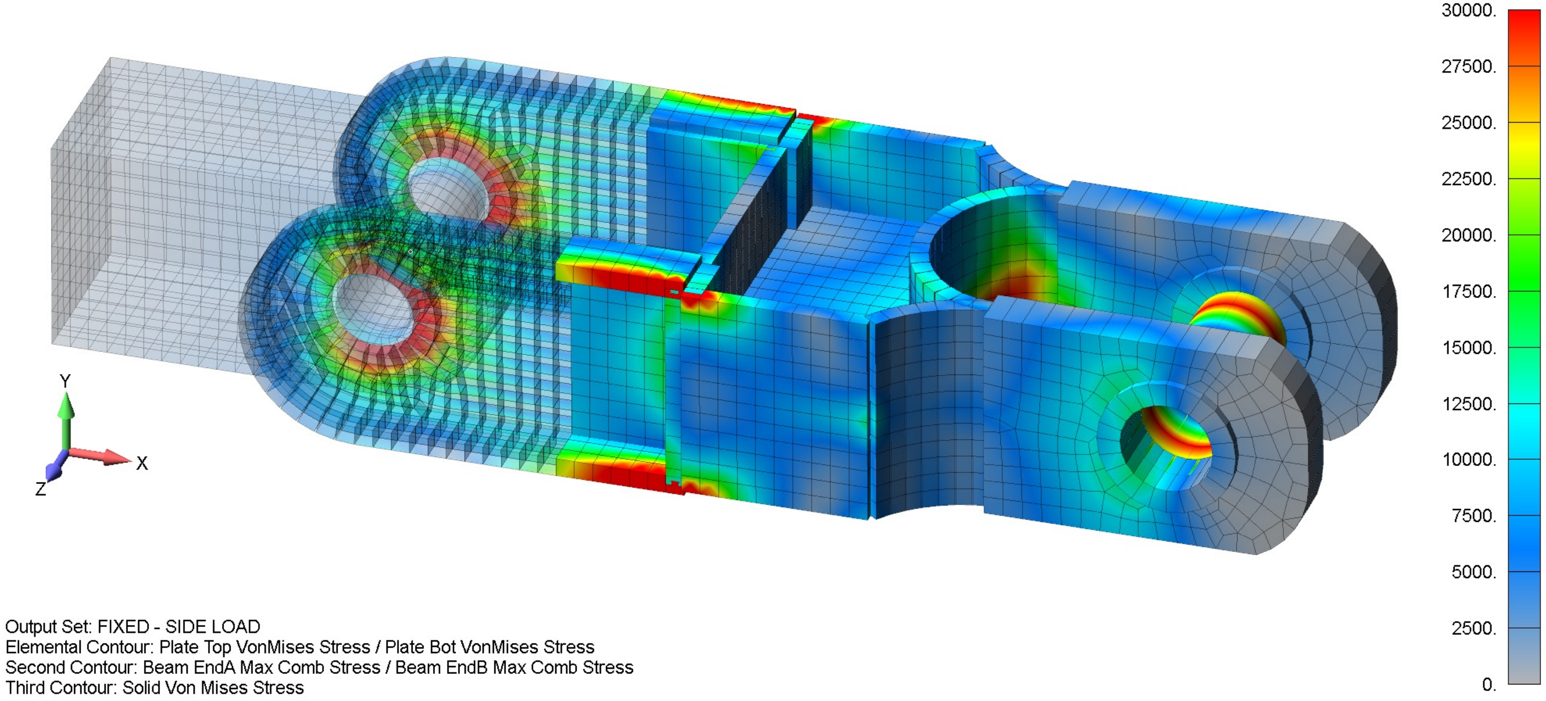
AppliedCAx

Contour Style – Contour



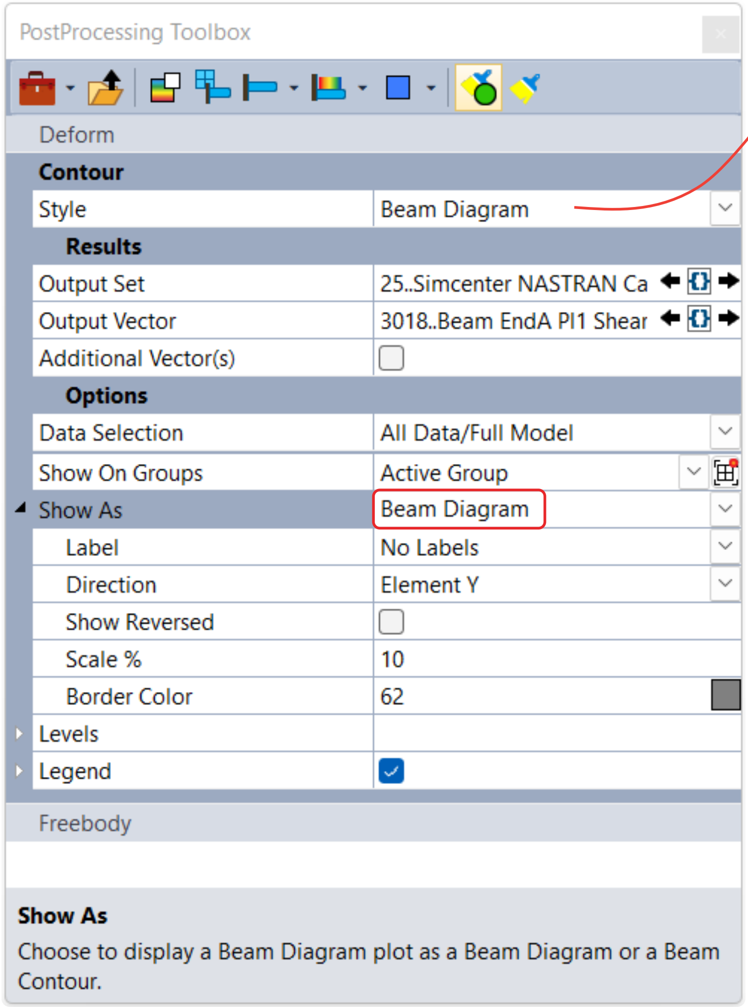
Contour

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



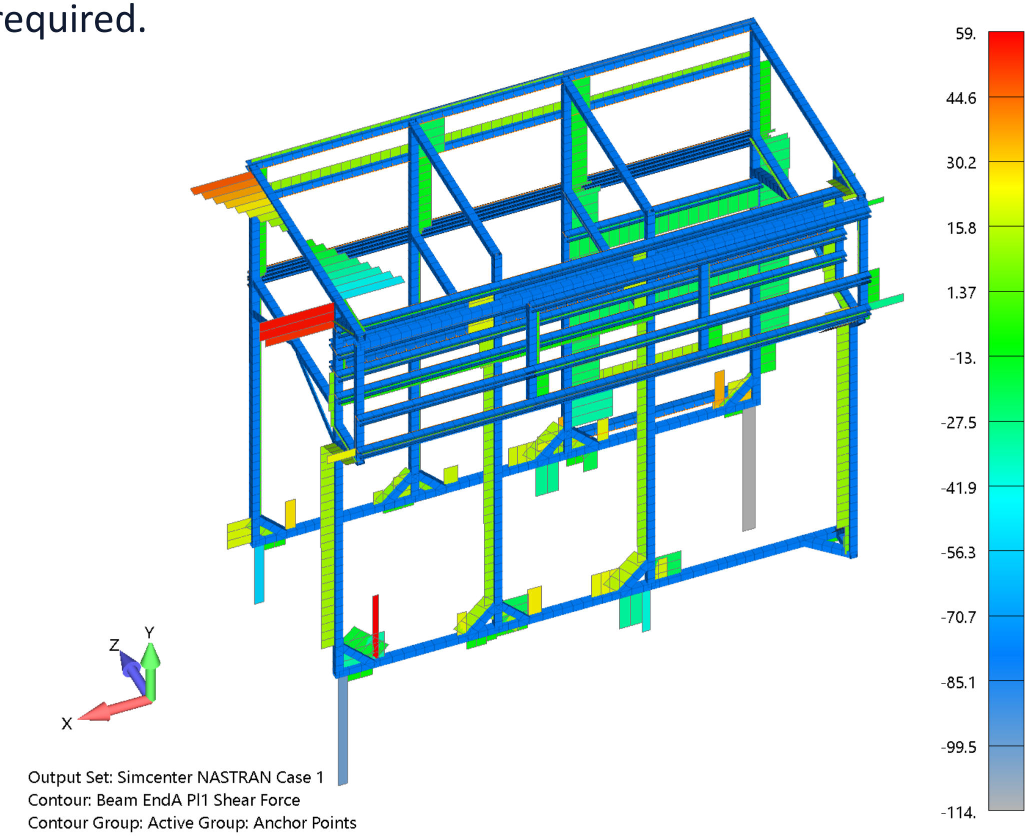
PostProcessing Toolbox

Contour Style – Beam Diagram



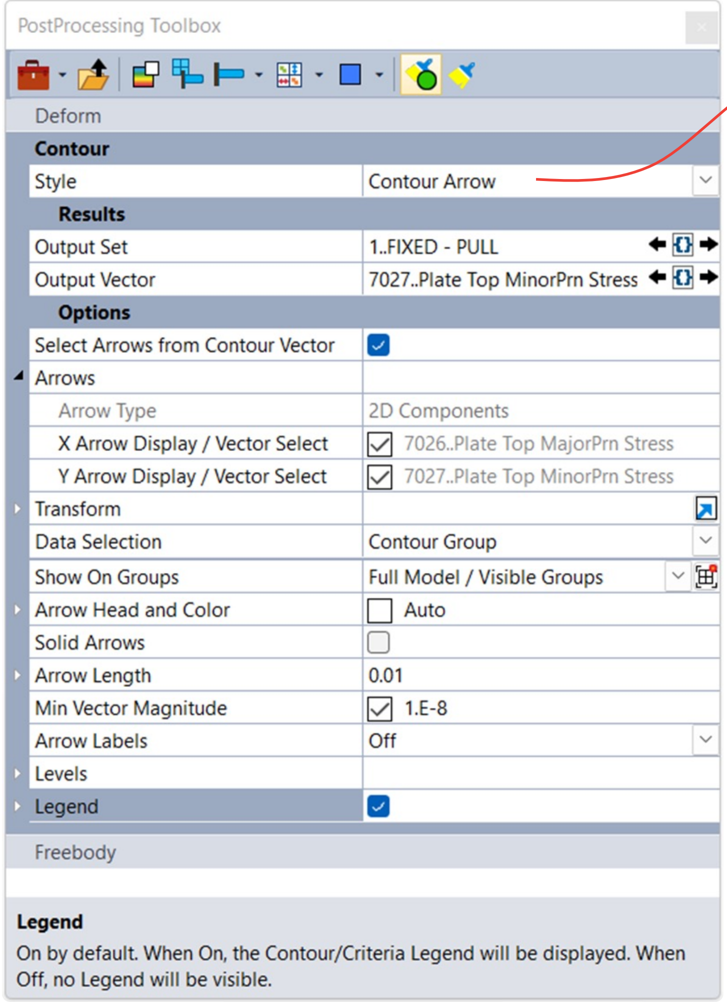
Beam Diagram

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.



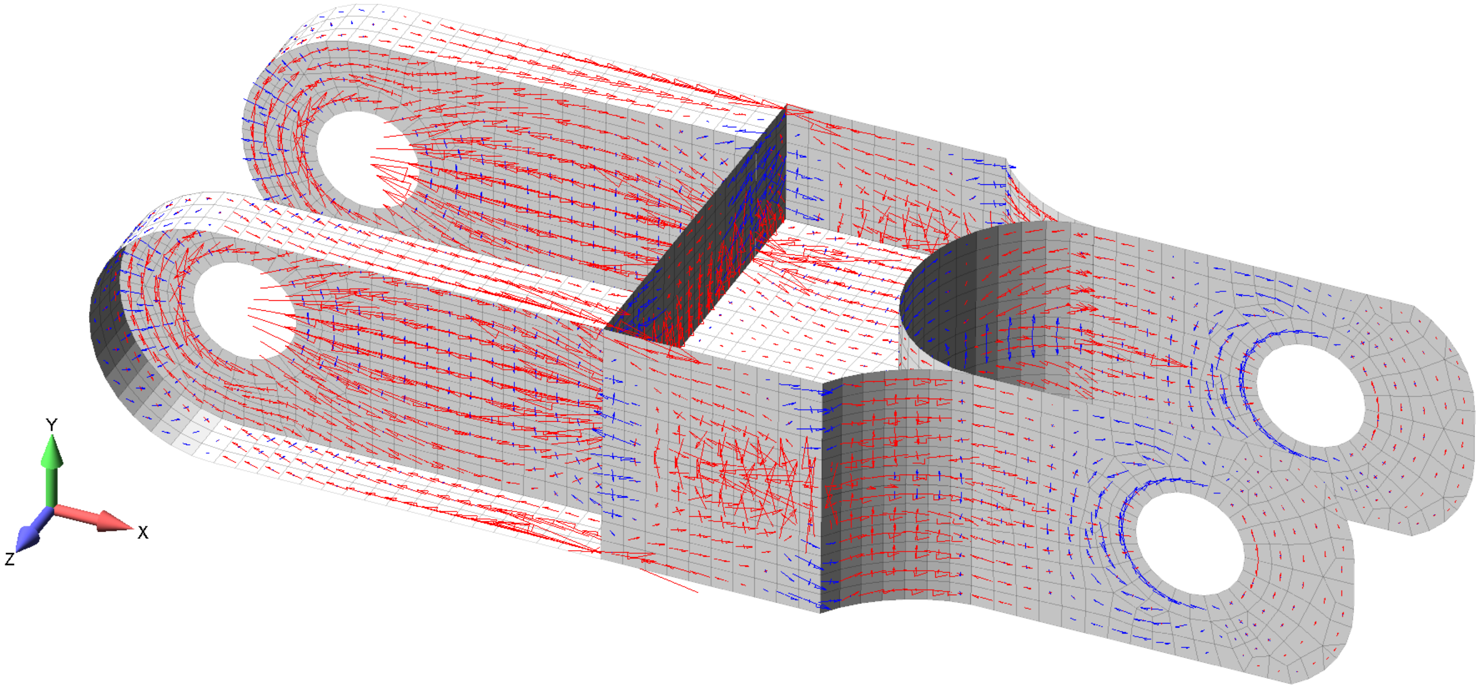
PostProcessing Toolbox

Contour Style – Contour Arrow



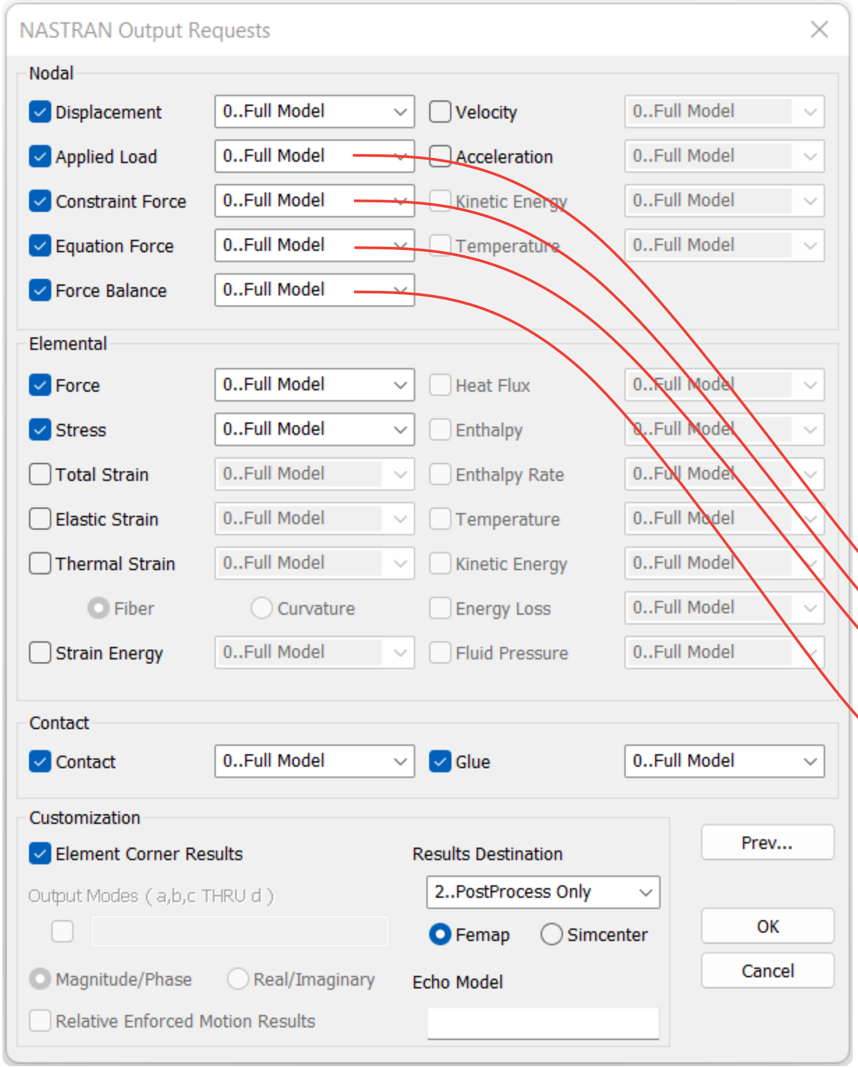
Contour Arrow

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.



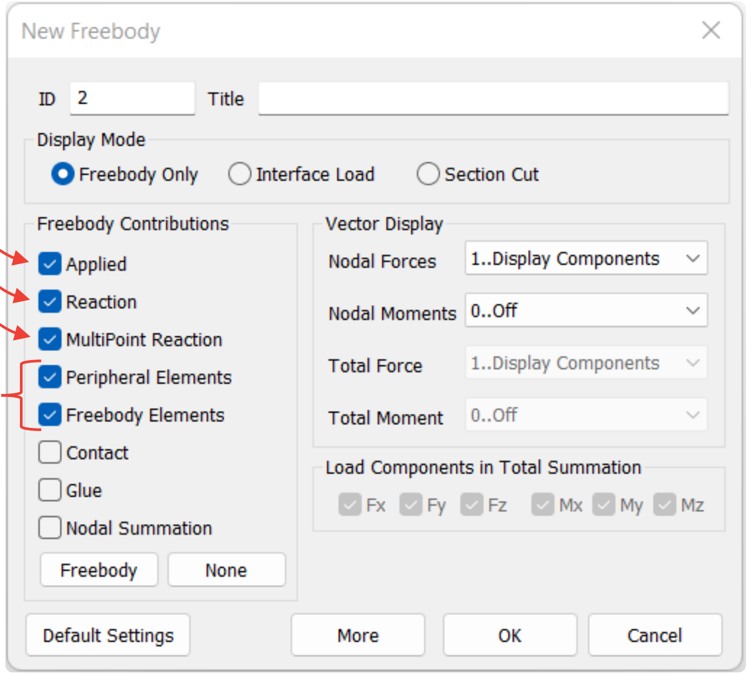
PostProcessing Toolbox

Freebody View Style – New Freebody



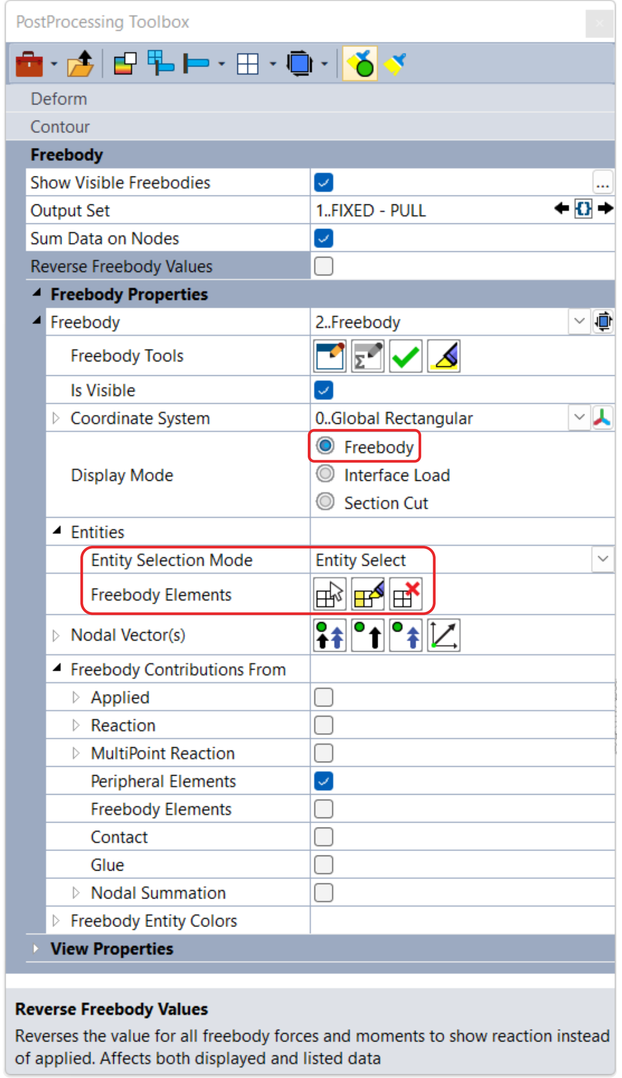
Freebody Diagrams (or FBDs) are a somewhat advanced post technique, yet they are beautifully simple at the same time. It’s all about summing the forces and moments for a selection of elements and nodes. The trick is carefully selecting the entities and making sure that the “Freebody Contributions” are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

Step one is ensuring you have requested the appropriate output for the FBD you intend to create prior to running the model

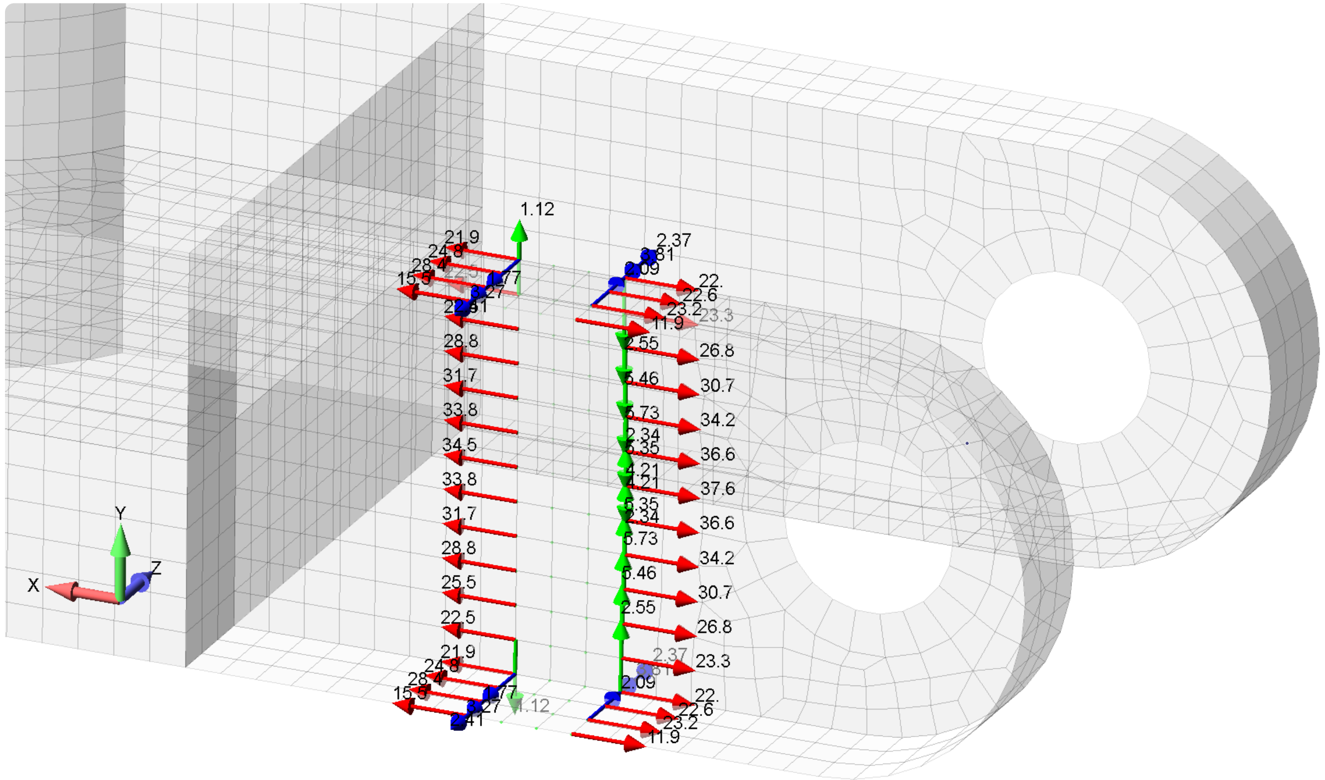


PostProcessing Toolbox

Freebody View Style – Freebody



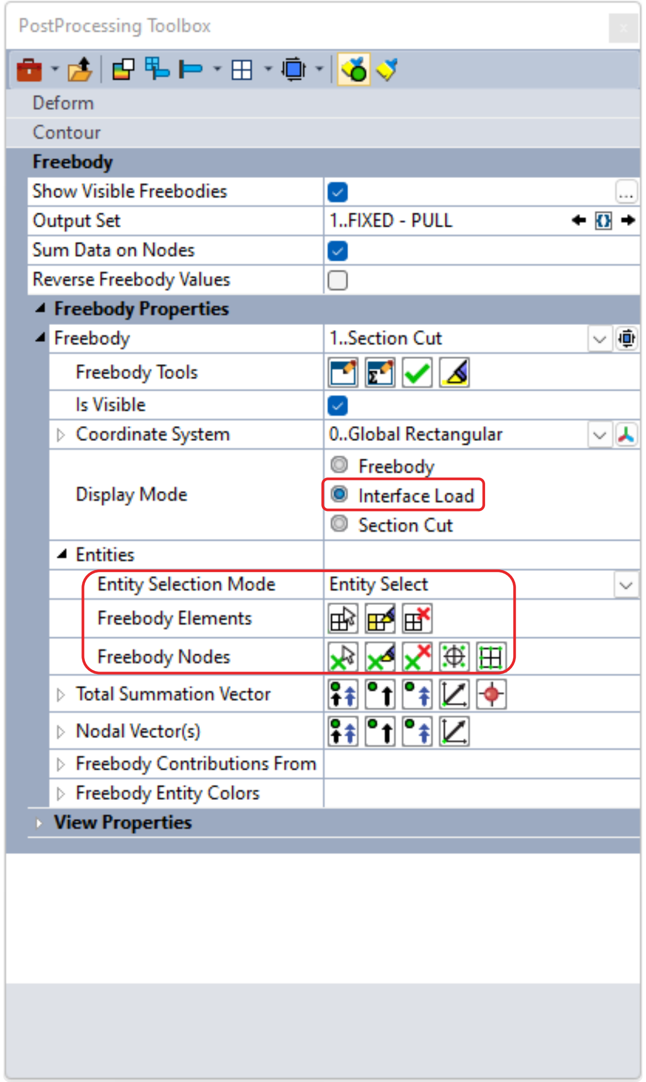
The default display mode for the FBD toolbox is simply called "Freebody". This display mode only requires the user to select elements. FBDs will be generated on any on the nodes of the selected elements that are connected to loads, constraints, RBEs, or other elements.



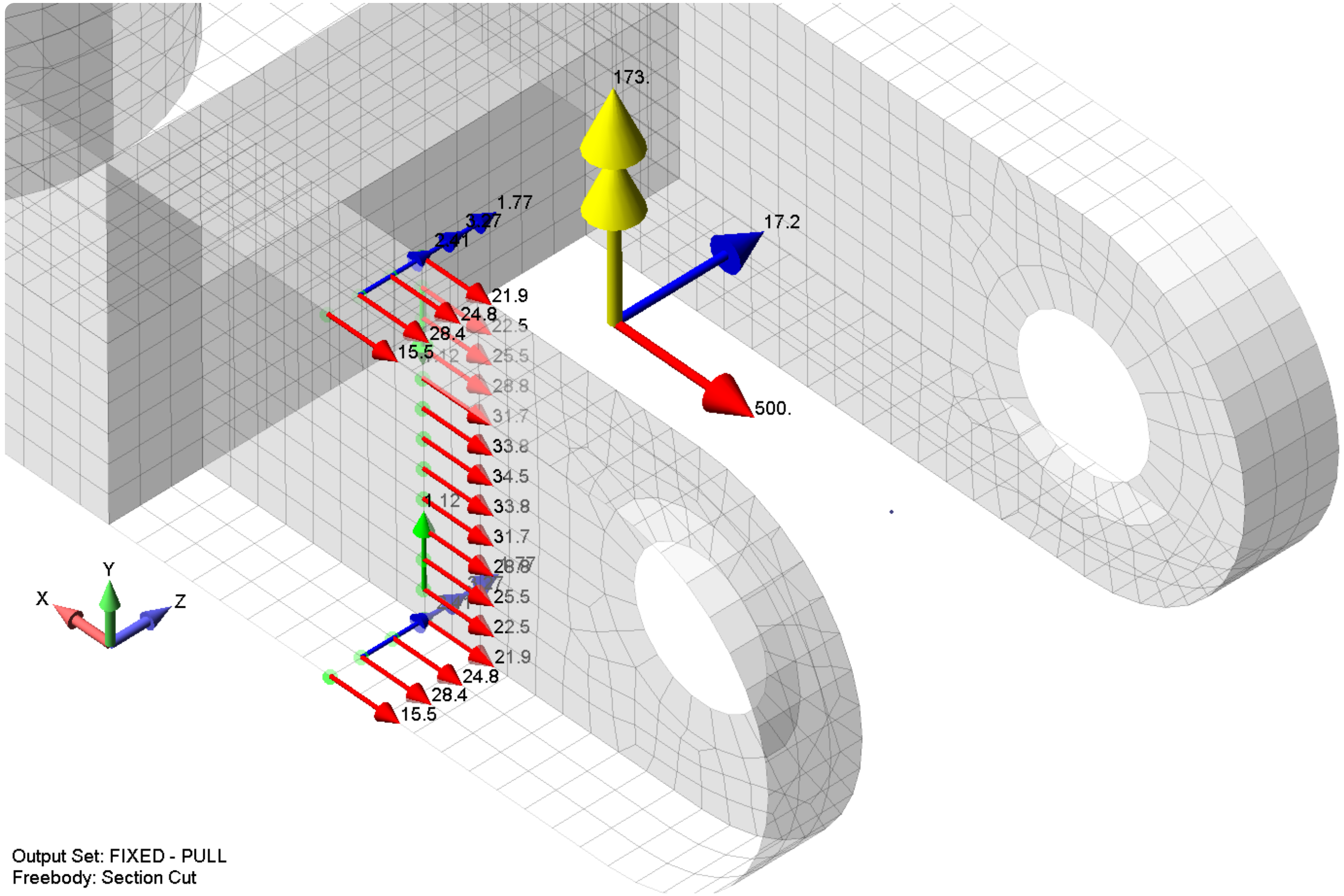
Output Set: FIXED - PULL
Freebody: Freebody

PostProcessing Toolbox

Freebody View Style – Interface Load

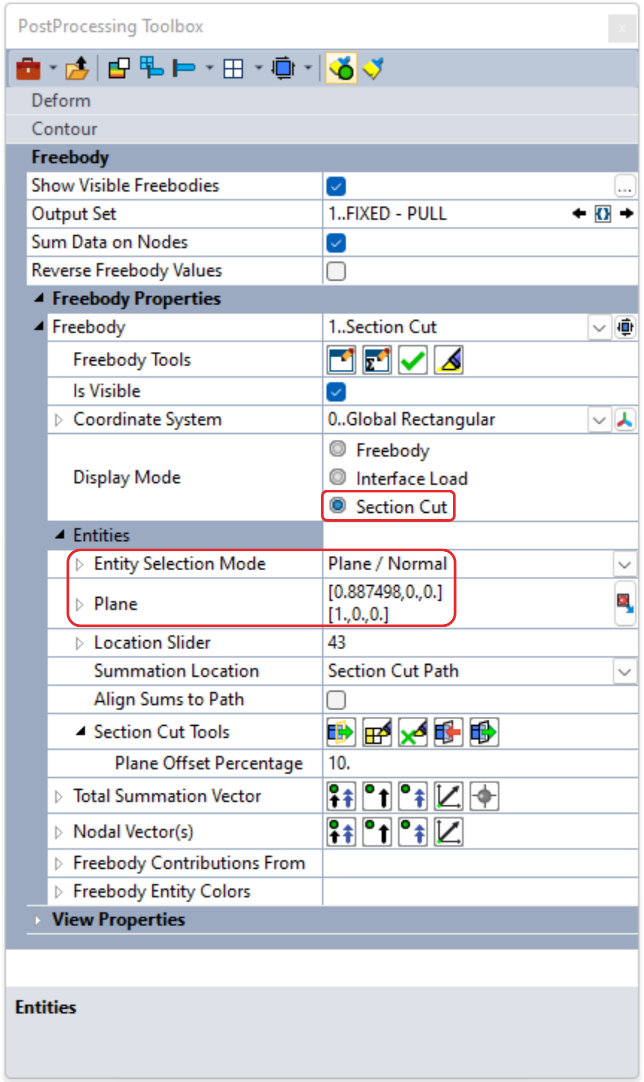


The “Interface Load” display mode will only generate FBDs on selected nodes attached to selected elements. In addition to the nodal FBDs, the Interface Load option allows the user to generate total summation vectors.

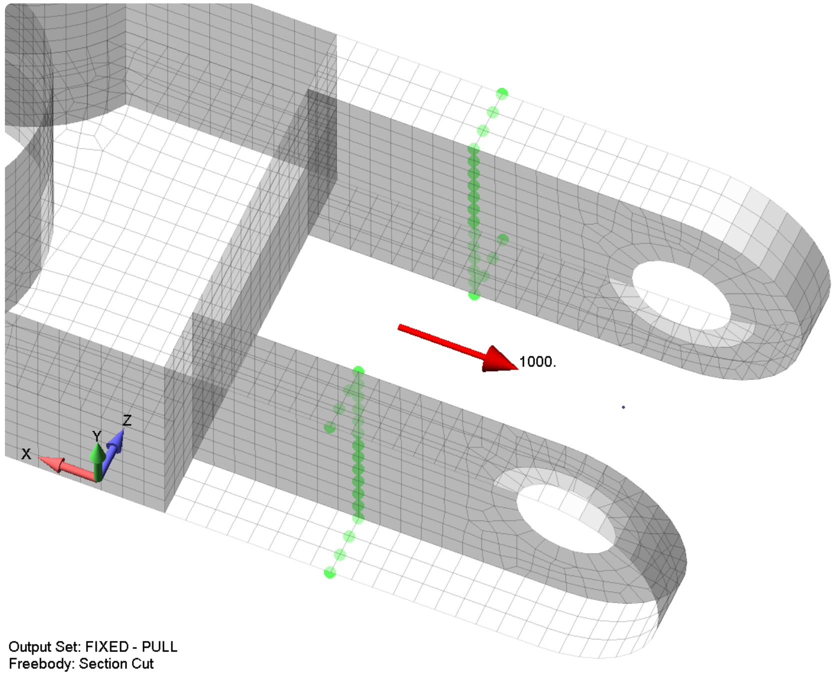


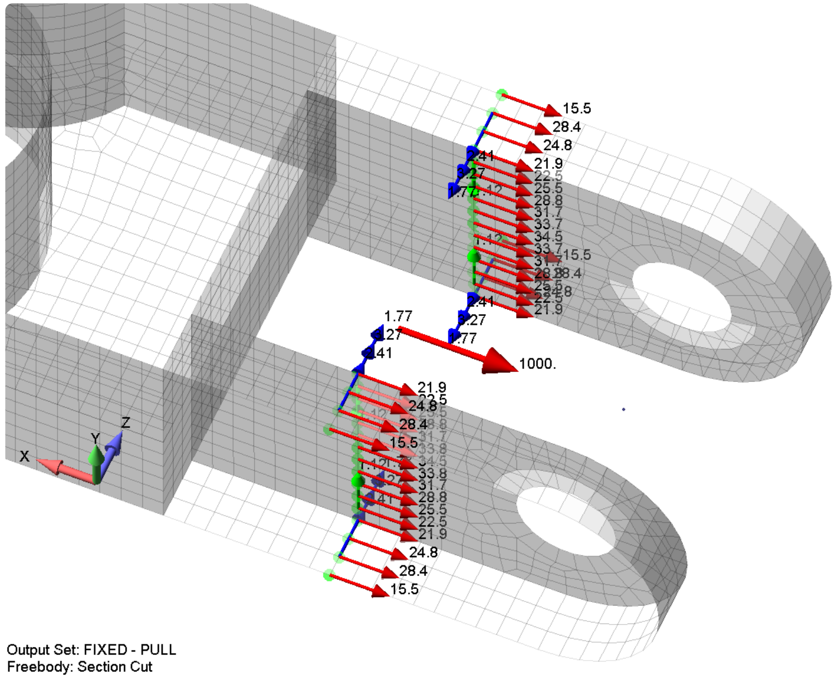
PostProcessing Toolbox

Freebody View Style – 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.

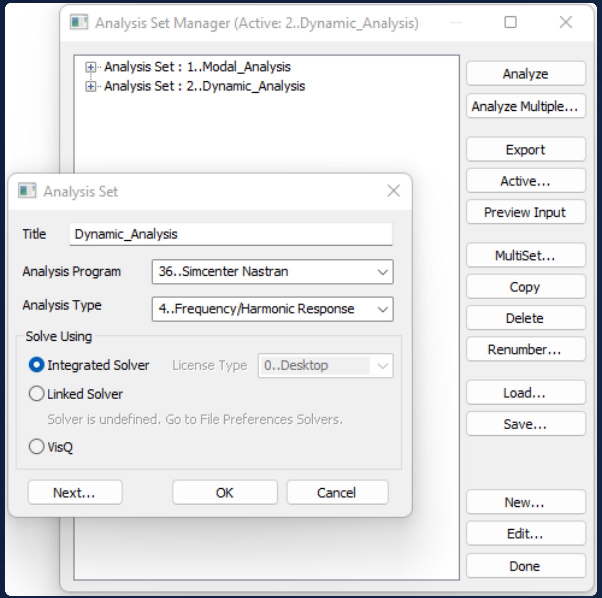


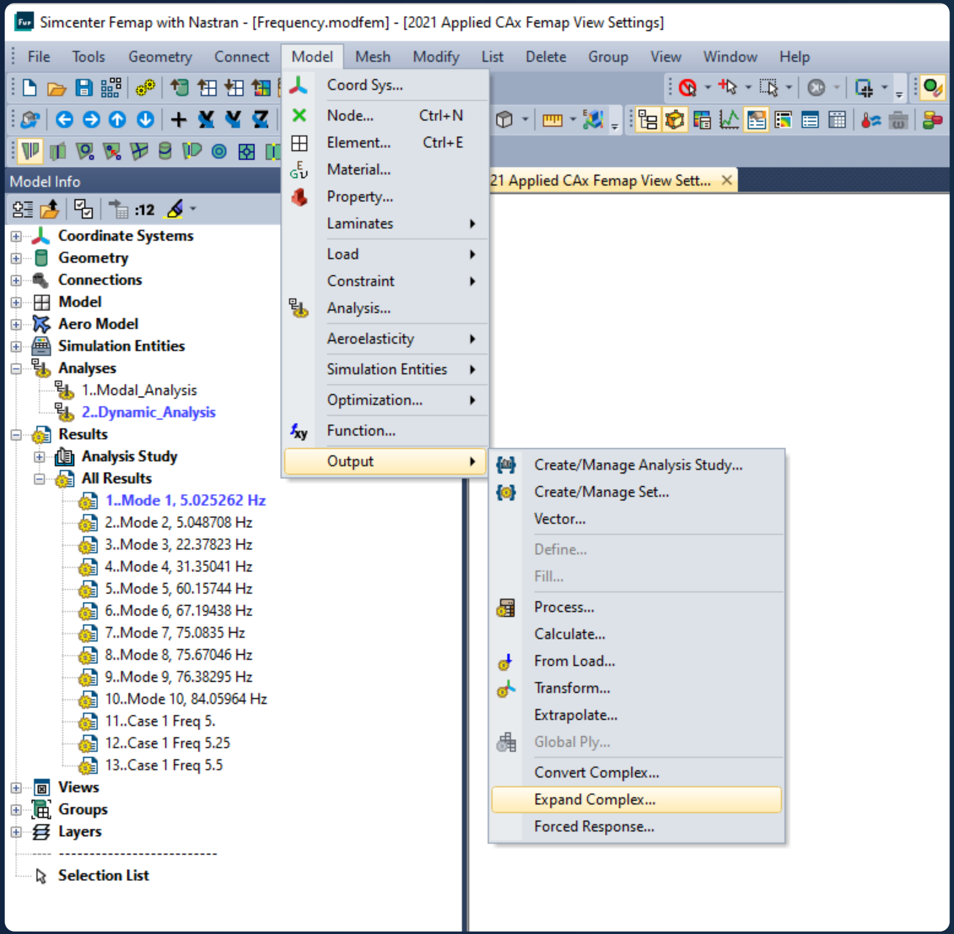


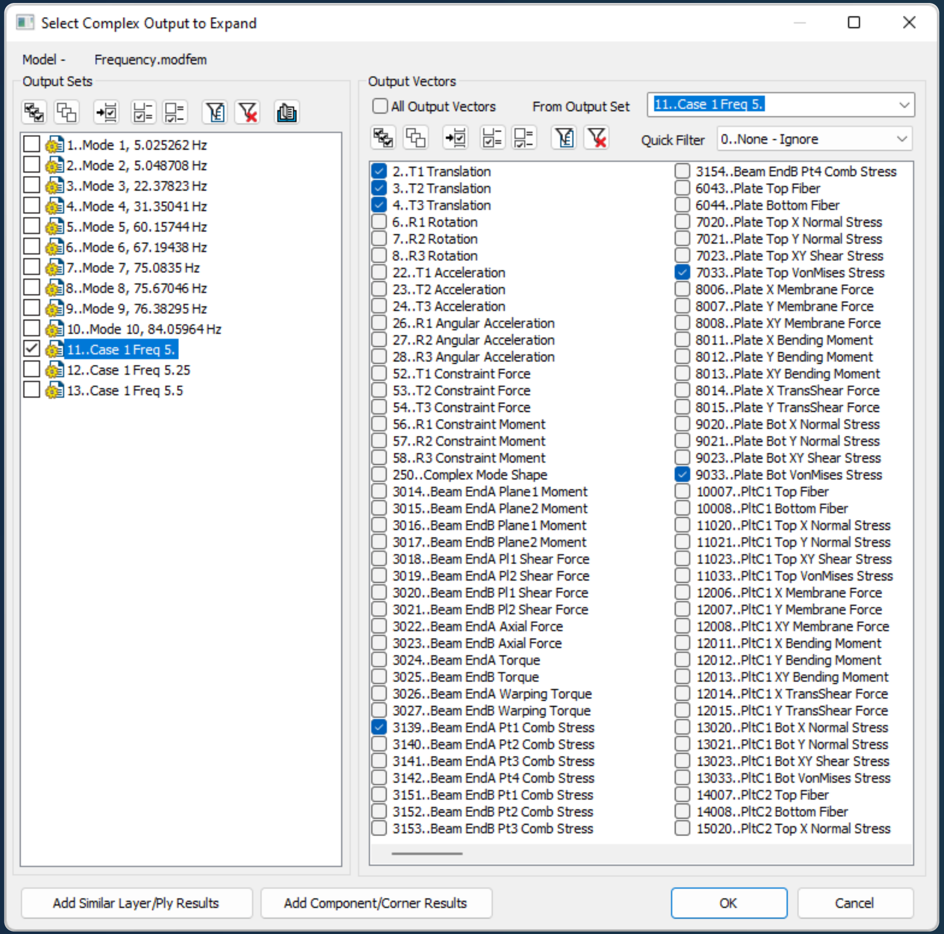
Charting

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.

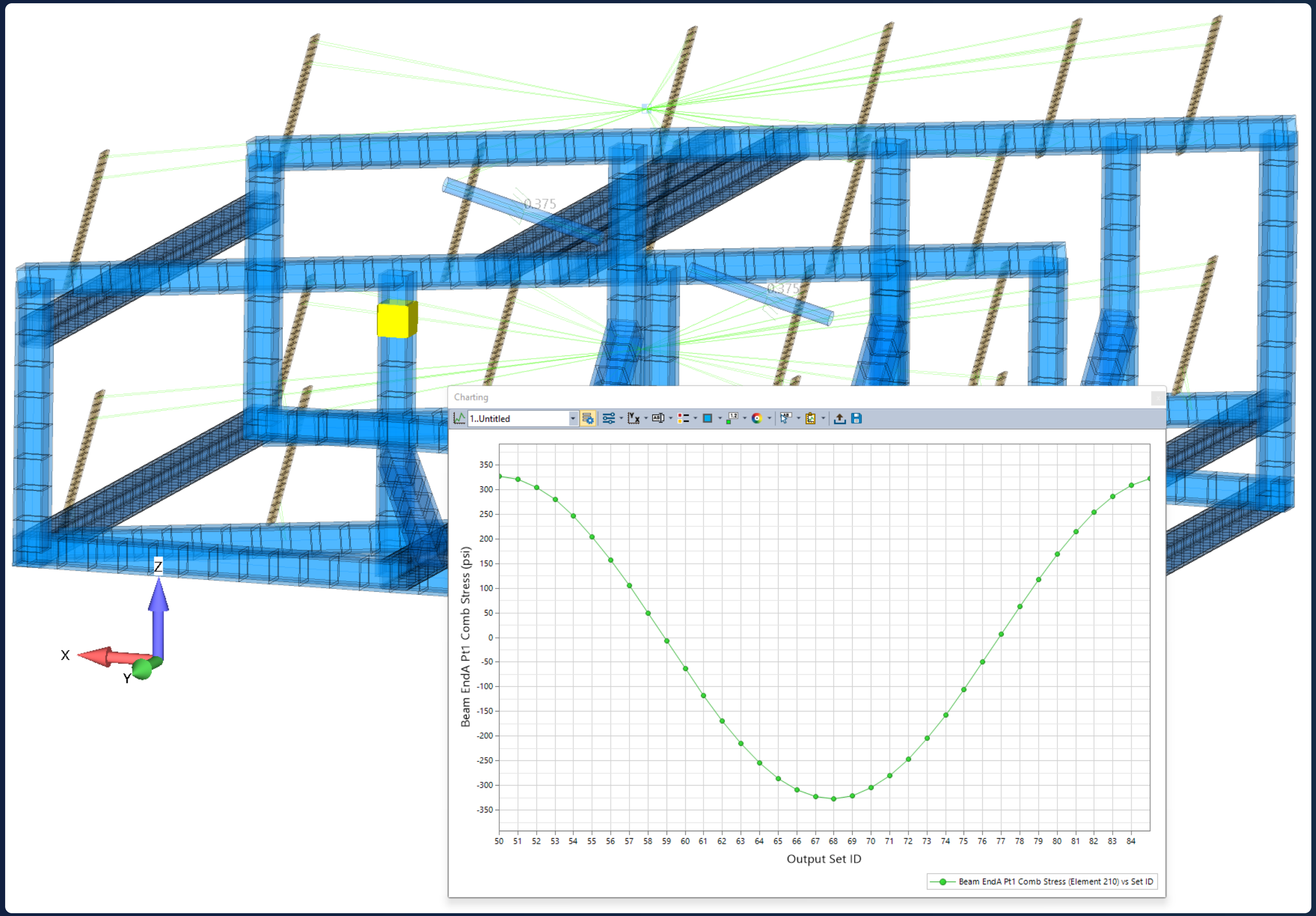






Charting

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.

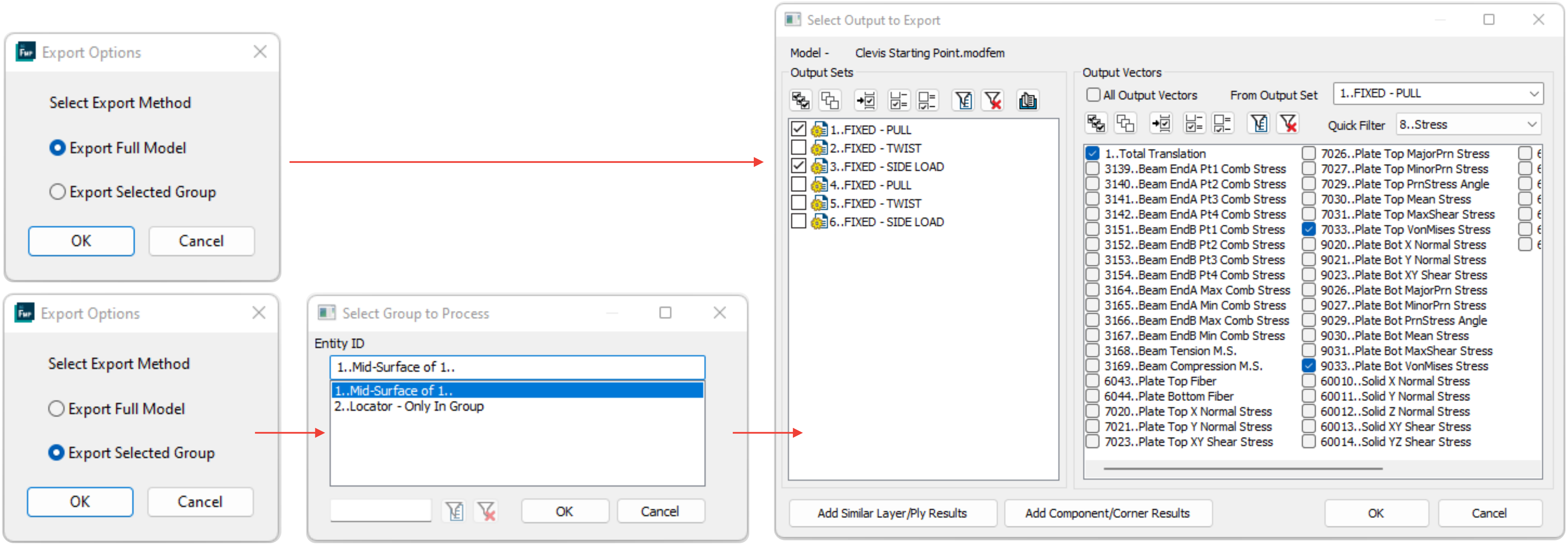
Here, we can see the P1 Comb Stress in the square tube (highlighted) for one full cycle, or complete load reversal.

Femap API

AppliedCAx

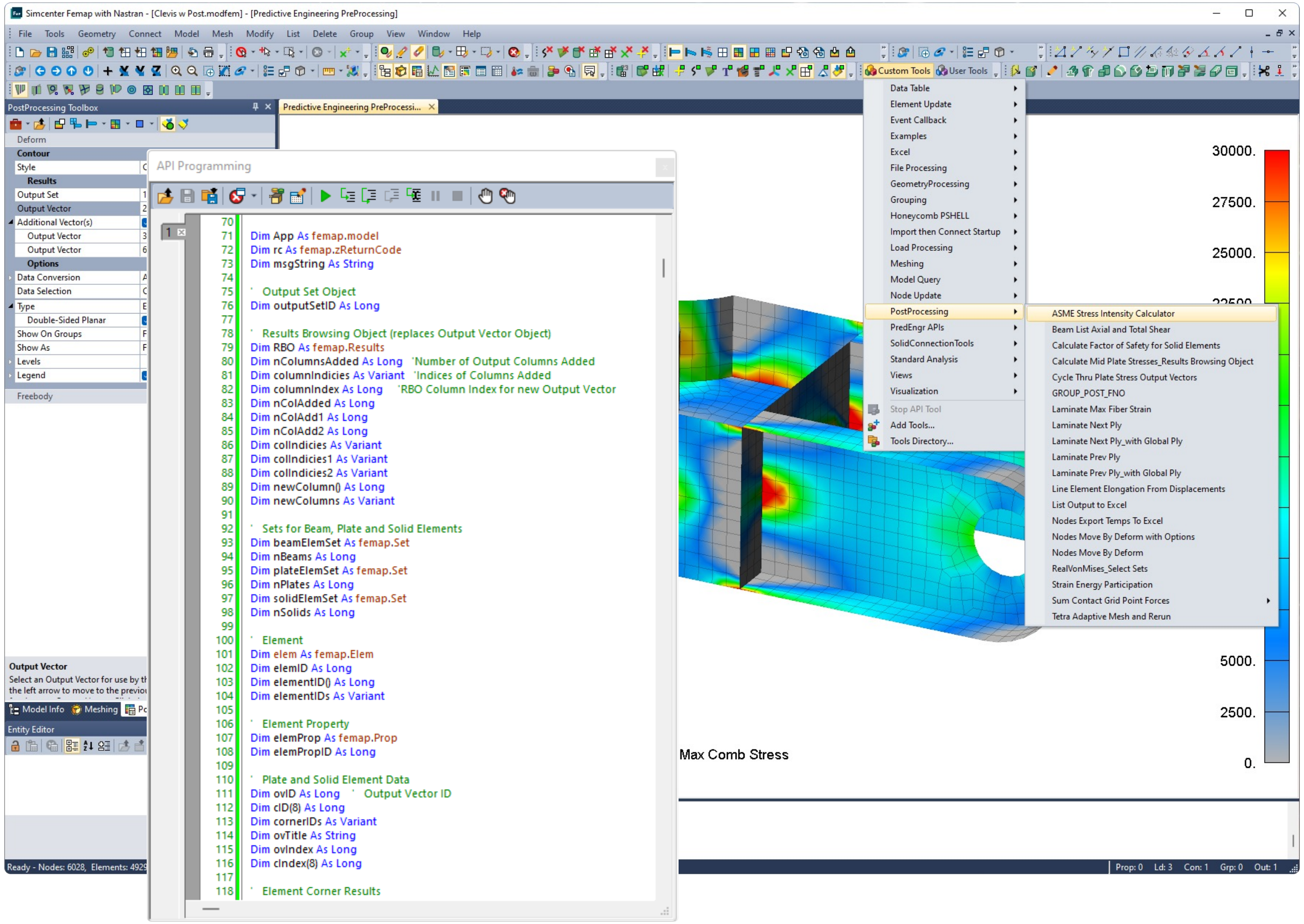
Group_Post_FNO

“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.



AppliedCAx

ASME Stress Intensity Calculator



"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.