Workshop - Introduction to Nastran Results and Post-processor Web App

A SOL 200 WEB APP TUTORIAL

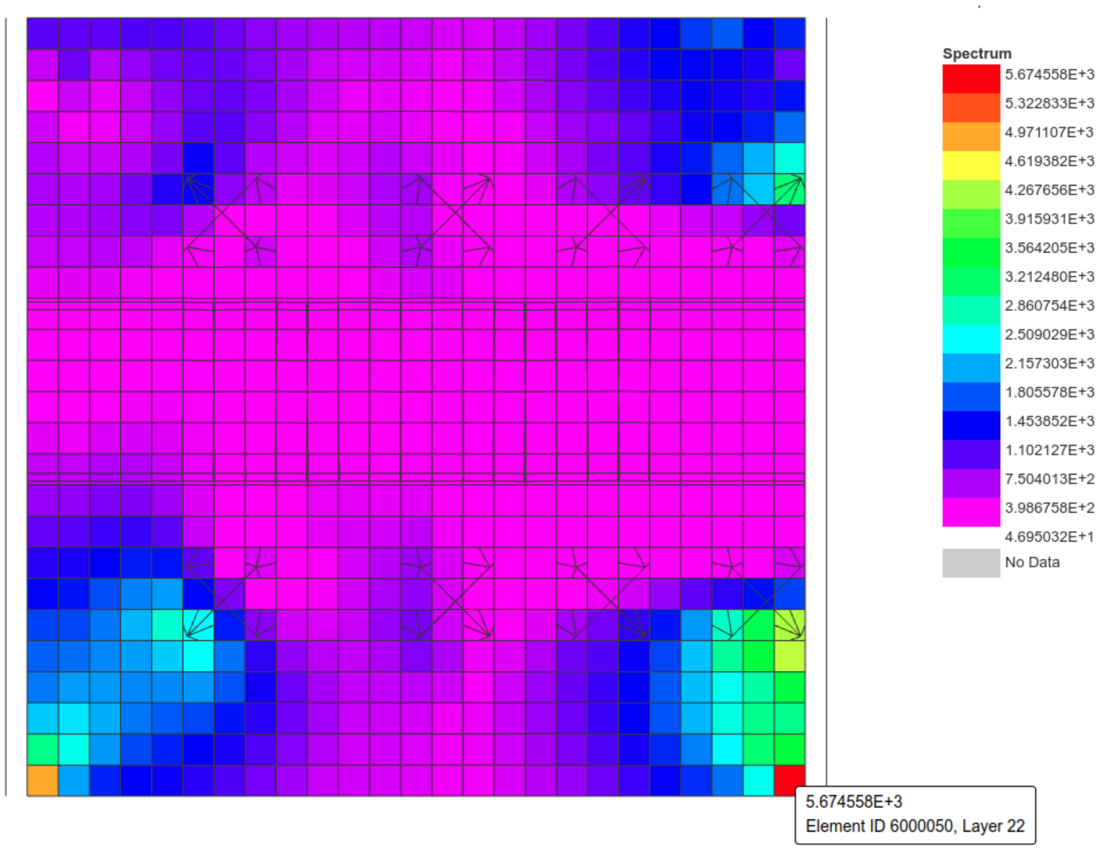
Before Starting

The Post-processor Web App is only compatible with MSC Nastran

Goal: Open the Post-processor web app and Inspect Nastran Results

- Open the Post-processor web app and inspect Nastran results
  - Nastran Displacement Results
  - Nastran Stress Results - CQUAD4 and PSHELL
  - Nastran Composite Stress Results - CQUAD8 and PCOMP
  - Nastran Beam Force and Stress Results - CBEAM and PBMSECT
  - Nastran CBUSH Force Results - CBUSH and PBUSH
  - Nastran CFAST Force Results - CFAST and PFAST

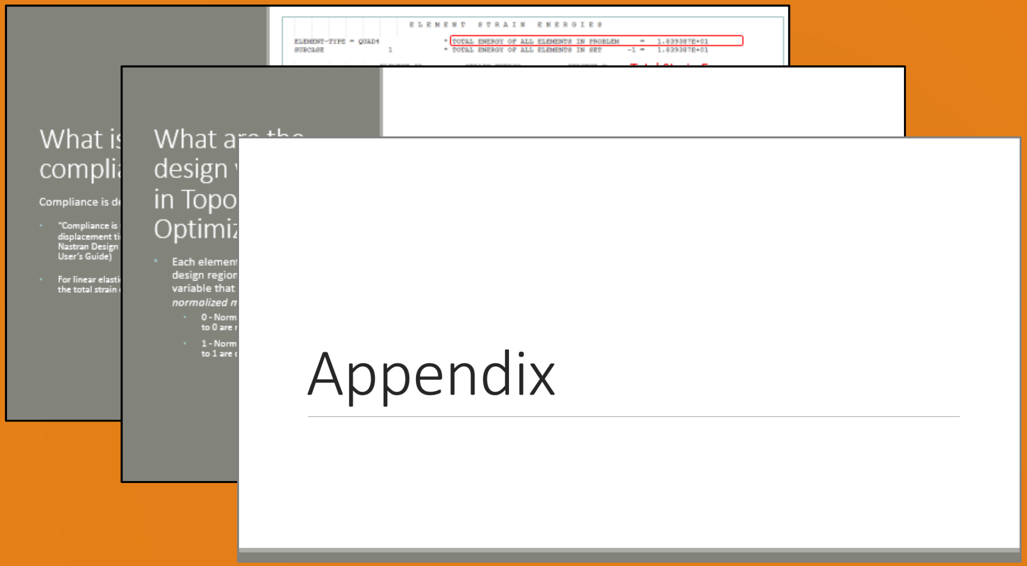
- Refer to the SOL 200 Web App Datasheet for a full list of supported results and supported element types.



More Information Available in the Appendix

The Appendix includes information regarding the following:

- Frequently Asked Questions
  - Why is the PCH file uploaded to the Viewer?
  - What are layers in a PCOMP and PCOMPG entry?
  - What are displacement coordinate systems?



Contact me

- Nastran SOL 200 training
- Nastran SOL 200 questions
- Structural or mechanical optimization questions
- Access to the SOL 200 Web App

christian@ the-engineering-lab.com

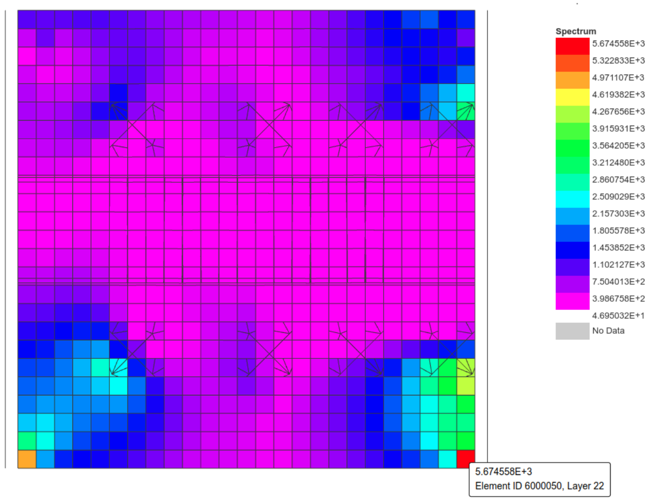
Tutorial

Tutorial Overview

1. Start with a .bdf or .dat file
2. Use the SOL 200 Web App
   - Open the Post-processor web app and inspect Nastran results
     - Nastran Displacement Results
     - Nastran Stress Results - CQUAD4 and PSHELL
     - Nastran Composite Stress Results - CQUAD8 and PCOMP
     - Nastran Beam Force and Stress Results - CBEAM and PBMSECT
     - Nastran CBUSH Force Results - CBUSH and PBUSH
     - Nastran CFAST Force Results - CFAST and PFAST

Special Topics Covered

Nastran Results - The Post-processor web app is used to inspect Nastran results



SOL 200 Web App Capabilities

The Post-processor Web App and HDF5 Explorer are free to MSC Nastran users.

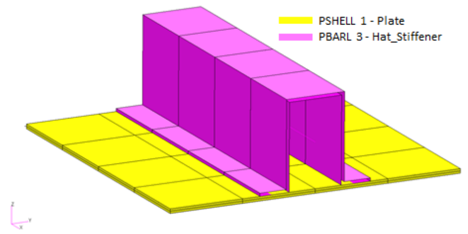
Compatibility

- Google Chrome, Mozilla Firefox or Microsoft Edge
- Windows and Red Hat Linux
- Installable on a company laptop, workstation or server. All data remains within your company.

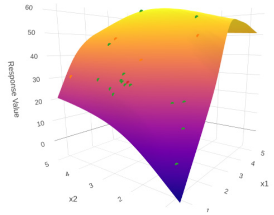
Benefits

- REAL TIME error detection. 200+ error validations.
- REALT TIME creation of bulk data entries.
- Web browser accessible
- Free Post-processor web apps
- +80 tutorials

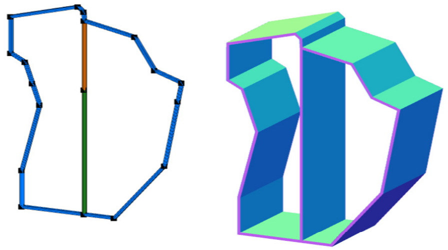
Web Apps



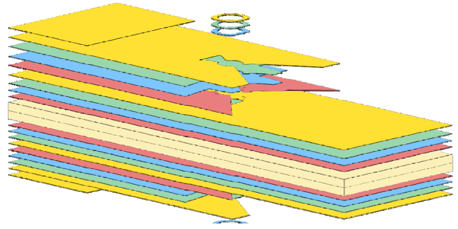
Web Apps for MSC Nastran SOL 200
Pre/post for MSC Nastran SOL 200. Support for size, topology, topometry, topography, multi-model optimization.



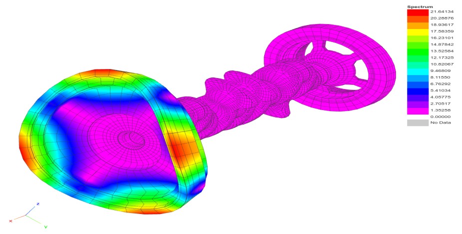
Machine Learning Web App
Bayesian Optimization for nonlinear response optimization (SOL 400)



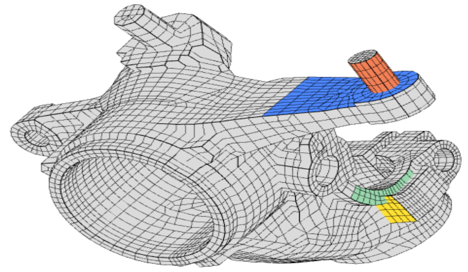
PBMSECT Web App
Generate PBMSECT and PBRSECT entries graphically



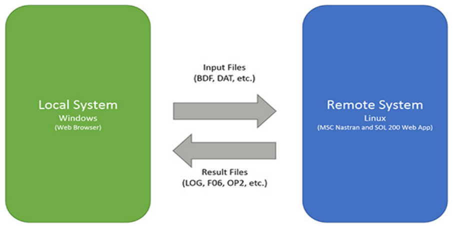
Ply Shape Optimization Web App
Optimize composite ply drop-off locations, and generate new PCOMPG entries



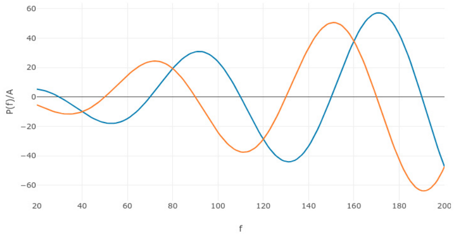
Post-processor Web App
View MSC Nastran results in a web browser on Windows and Linux



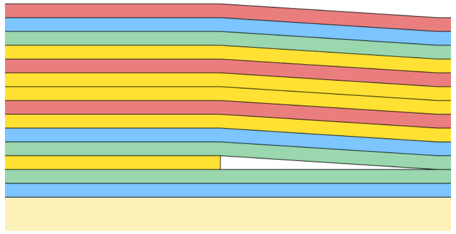
Shape Optimization Web App
Use a web application to configure and perform shape optimization.



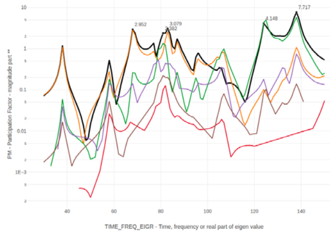
Remote Execution Web App
Run MSC Nastran jobs on remote Linux or Windows systems available on the local network



Dynamic Loads Web App
Generate RLOAD1, RLOAD2 and DLOAD entries graphically



Stacking Sequence Web App
Optimize the stacking sequence of composite laminate plies

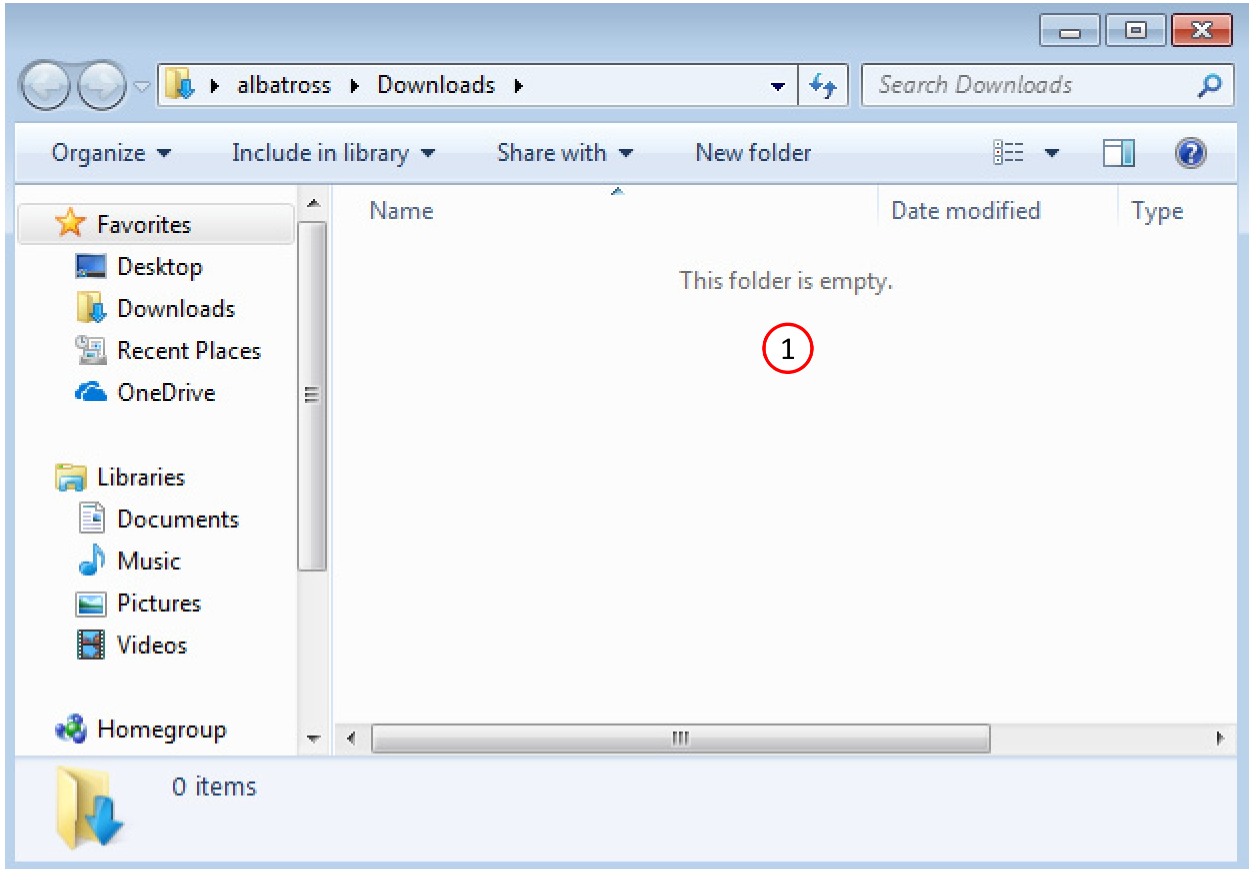


HDF5 Explorer Web App
Create graphs (XY plots) using data from the H5 file

Before Starting

1. Ensure the Downloads directory is empty in order to prevent confusion with other files

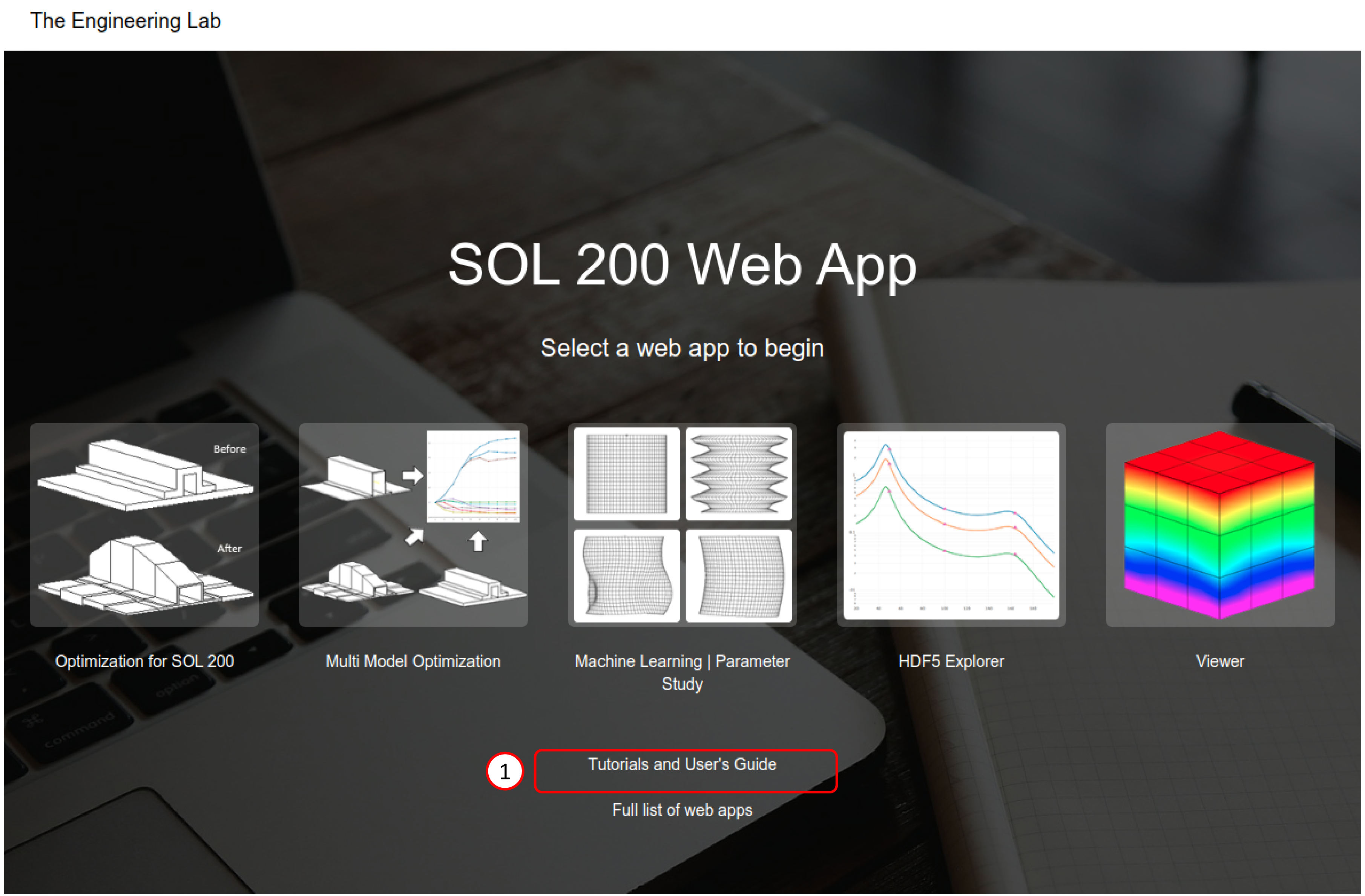
- Throughout this workshop, you will be working with multiple file types and directories such as:
  - .bdf/.dat
  - nastran_working_directory
  - .f06, .log, .pch, .h5, etc.
- To minimize confusion with files and folders, it is encouraged to start with a clean directory.



Go to the User's Guide

1. Click on the indicated link

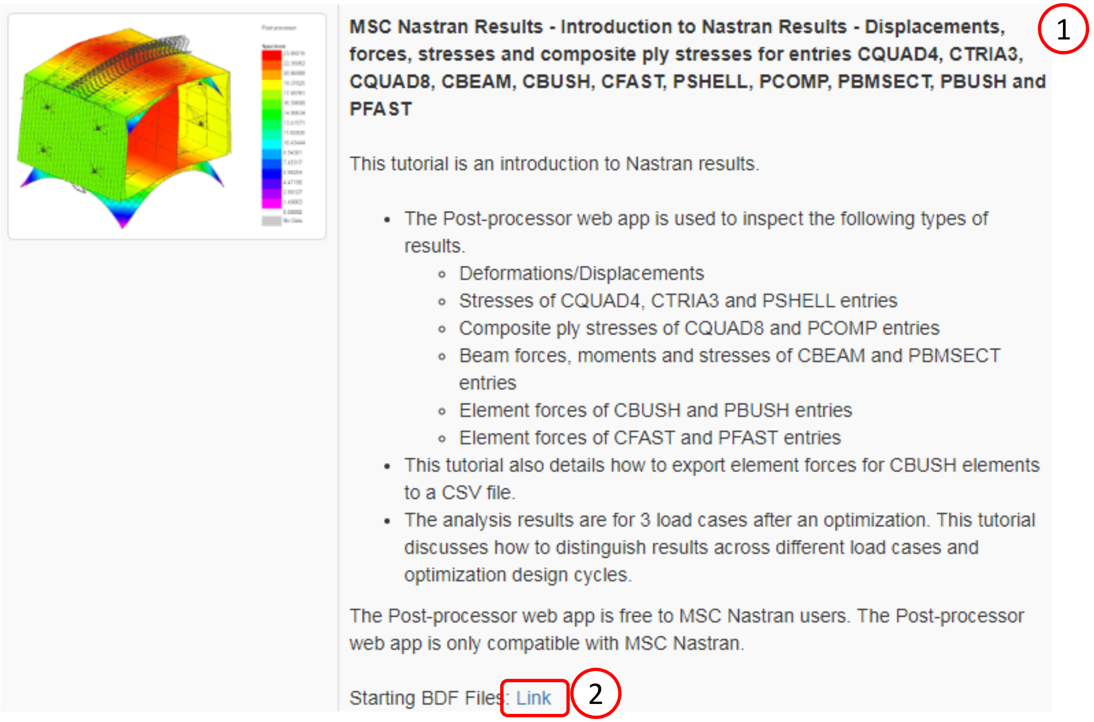
- The necessary BDF files for this tutorial are available in the Tutorials section of the User's Guide.



Obtain Starting Files

1. Find the indicated example
2. Click Link
3. The starting file has been downloaded

- When starting the procedure, all the necessary BDF files must be collected together.



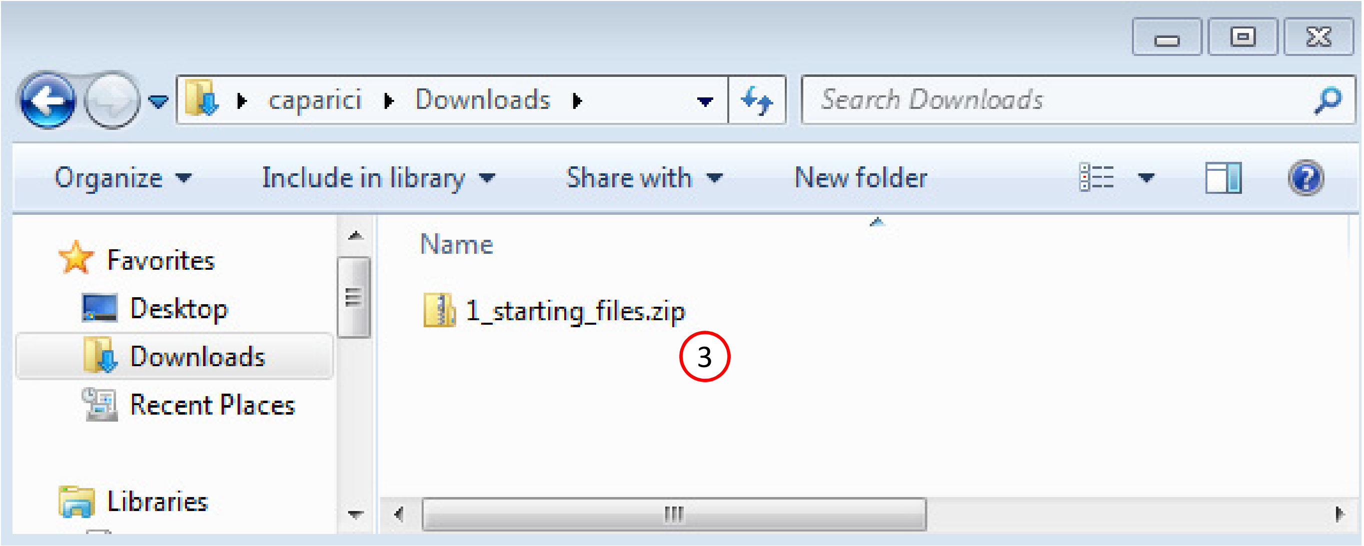
MSC Nastran Results - Introduction to Nastran Results - Displacements, forces, stresses and composite ply stresses for entries CQUAD4, CTRIA3, CQUAD8, CBEAM, CBUSH, CFAST, PSHELL, PCOMP, PBMSECT, PBUSH and PFAST

This tutorial is an introduction to Nastran results.

- The Post-processor web app is used to inspect the following types of results.
  - Deformations/Displacements
  - Stresses of CQUAD4, CTRIA3 and PSHELL entries
  - Composite ply stresses of CQUAD8 and PCOMP entries
  - Beam forces, moments and stresses of CBEAM and PBMSECT entries
  - Element forces of CBUSH and PBUSH entries
  - Element forces of CFAST and PFAST entries
- This tutorial also details how to export element forces for CBUSH elements to a CSV file.
- The analysis results are for 3 load cases after an optimization. This tutorial discusses how to distinguish results across different load cases and optimization design cycles.

The Post-processor web app is free to MSC Nastran users. The Post-processor web app is only compatible with MSC Nastran.

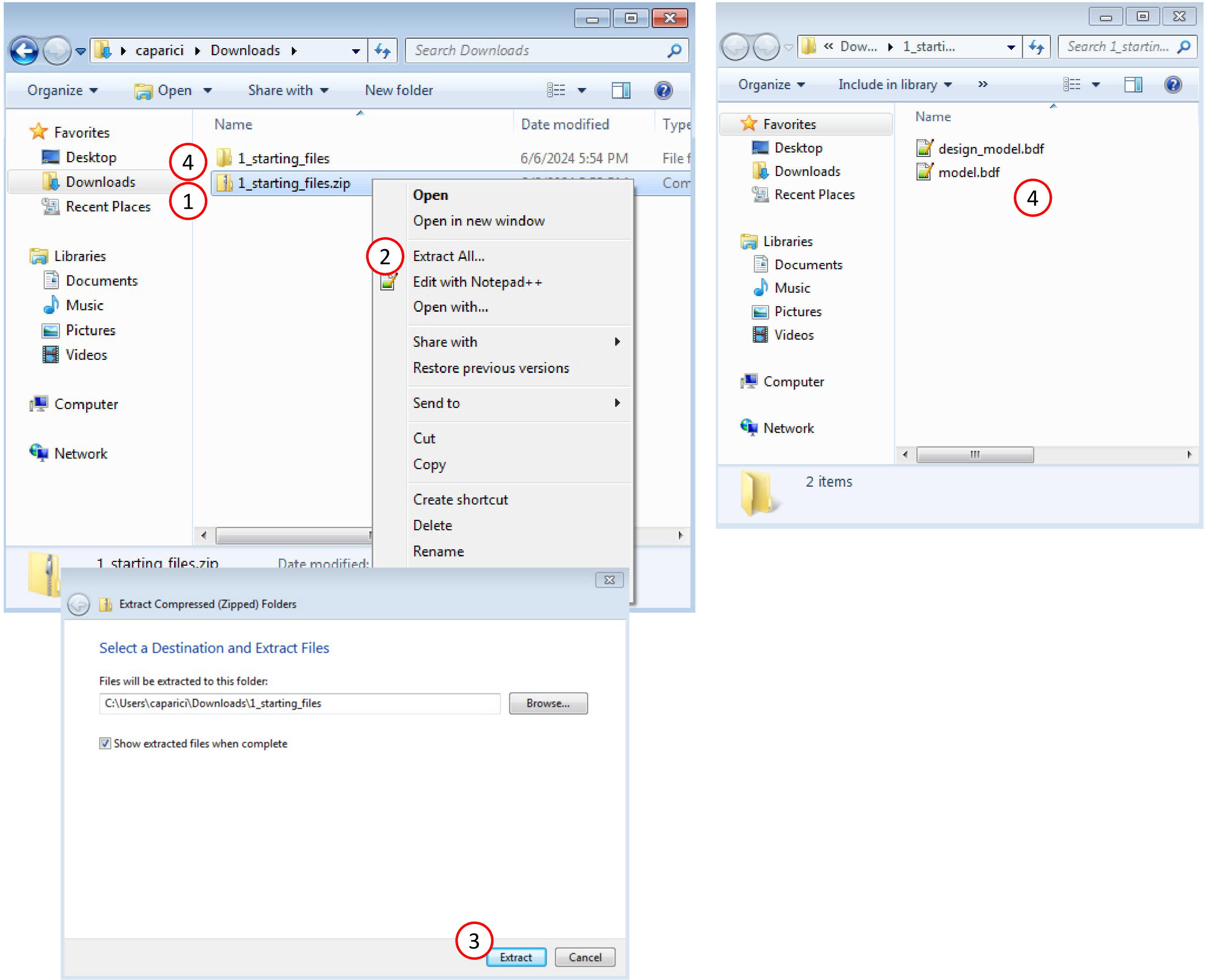
Starting BDF Files: Link



Obtain Starting Files

1. Right click on the zip file
2. Select Extract All…
3. Click Extract
4. The starting files are now available in a folder

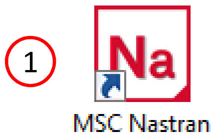
- This example is using a previously created design model. The design model is a model that has been converted to SOL 200 and contains bulk data entries describing the optimization problem statement, e.g. variables, objective and constraints.

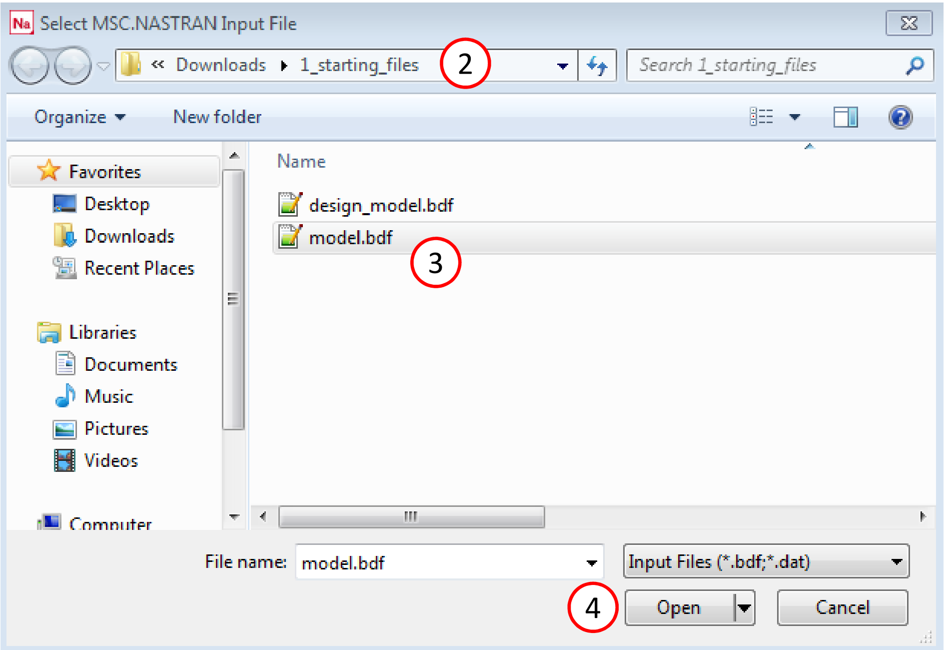


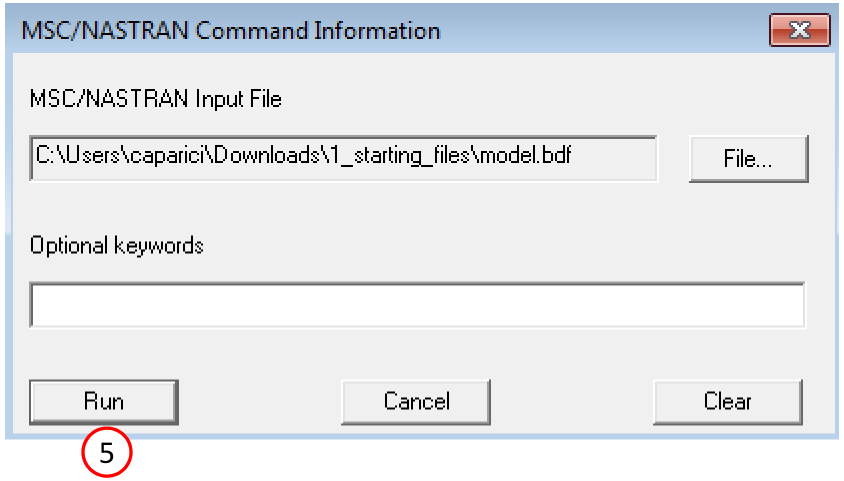
Create the Starting H5 File

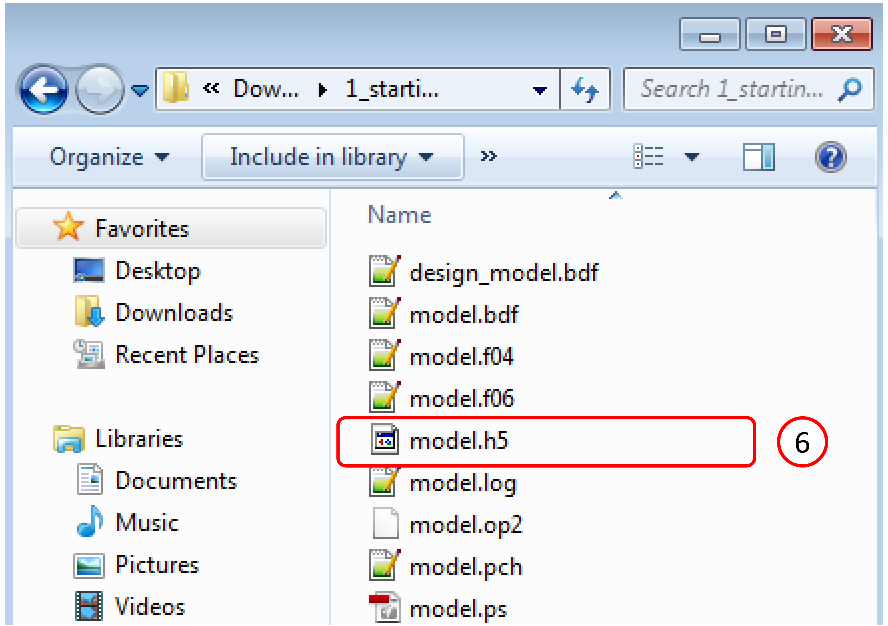
A starting H5 file must be created. This H5 file will be used to configure the responses later on.

1. Double click the MSC Nastran desktop shortcut
2. Navigate to the directory named 1_starting_files
3. Select the indicated file
4. Click Open
5. Click Run
6. The starting H5 file is created









Open the Correct Page

1. Click on the indicated link

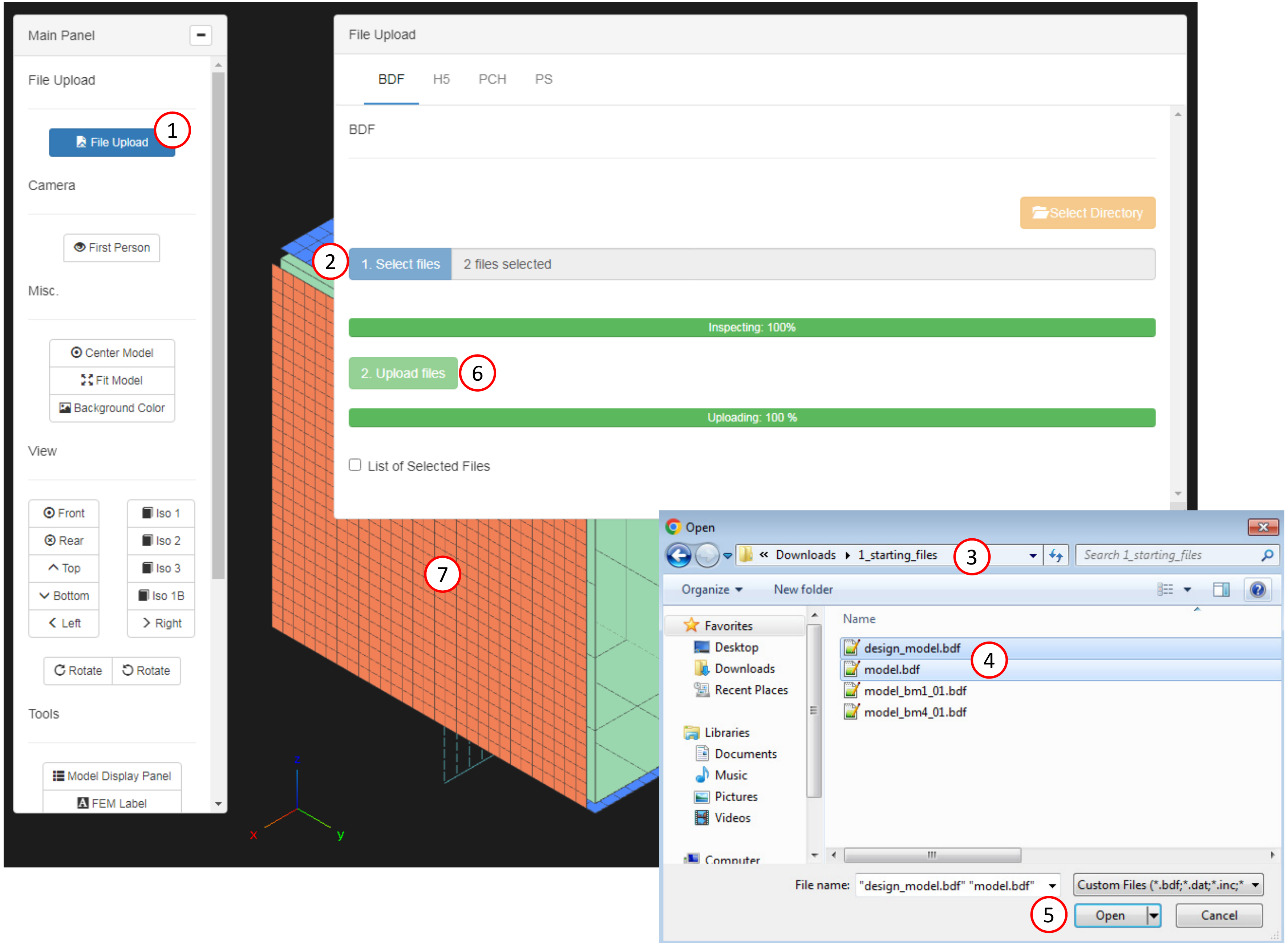
- MSC Nastran can perform many optimization types. The SOL 200 Web App includes dedicated web apps for the following:
  - Optimization for SOL 200 (Size, Topology, Topometry, Topography, Local Optimization, Sensitivity Analysis and Global Optimization)
  - Multi Model Optimization
  - Machine Learning
- The web app also features the HDF5 Explorer, a web application to extract results from the H5 file type.

The Engineering Lab

Upload Files

1. Click File Upload
2. Click Select files
3. Navigate to the directory 1_starting_files
4. Select the following files:
    1. model.bdf
    2. design_model.bdf
5. Click Open
6. Click Upload files
7. The MSC Nastran model has been uploaded to the Viewer

- The selected BDF file was created by a separate pre processor.

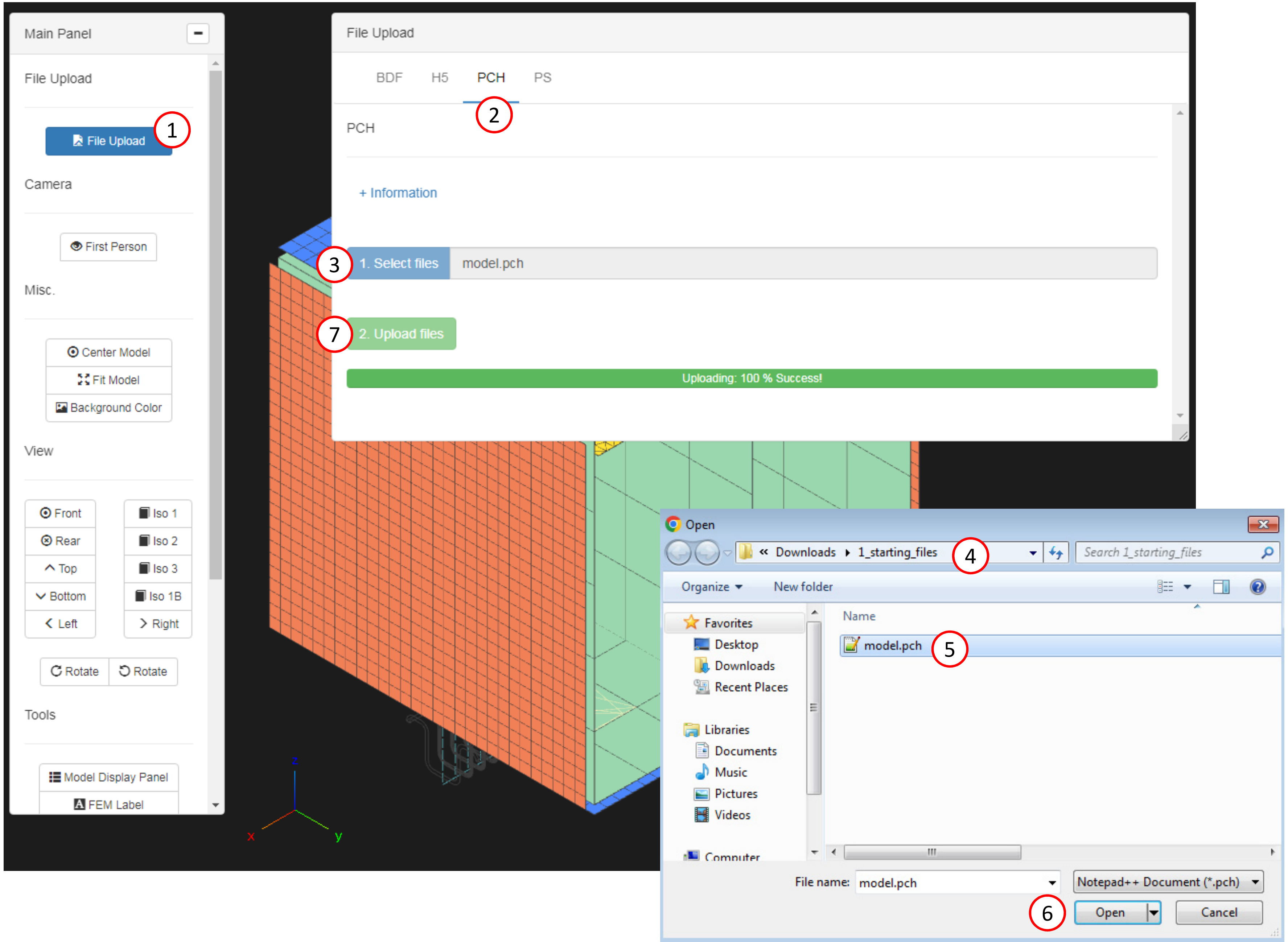


Upload Files

The following is required only if a connector, e.g. CFAST, CSEAM or CWELD is used.

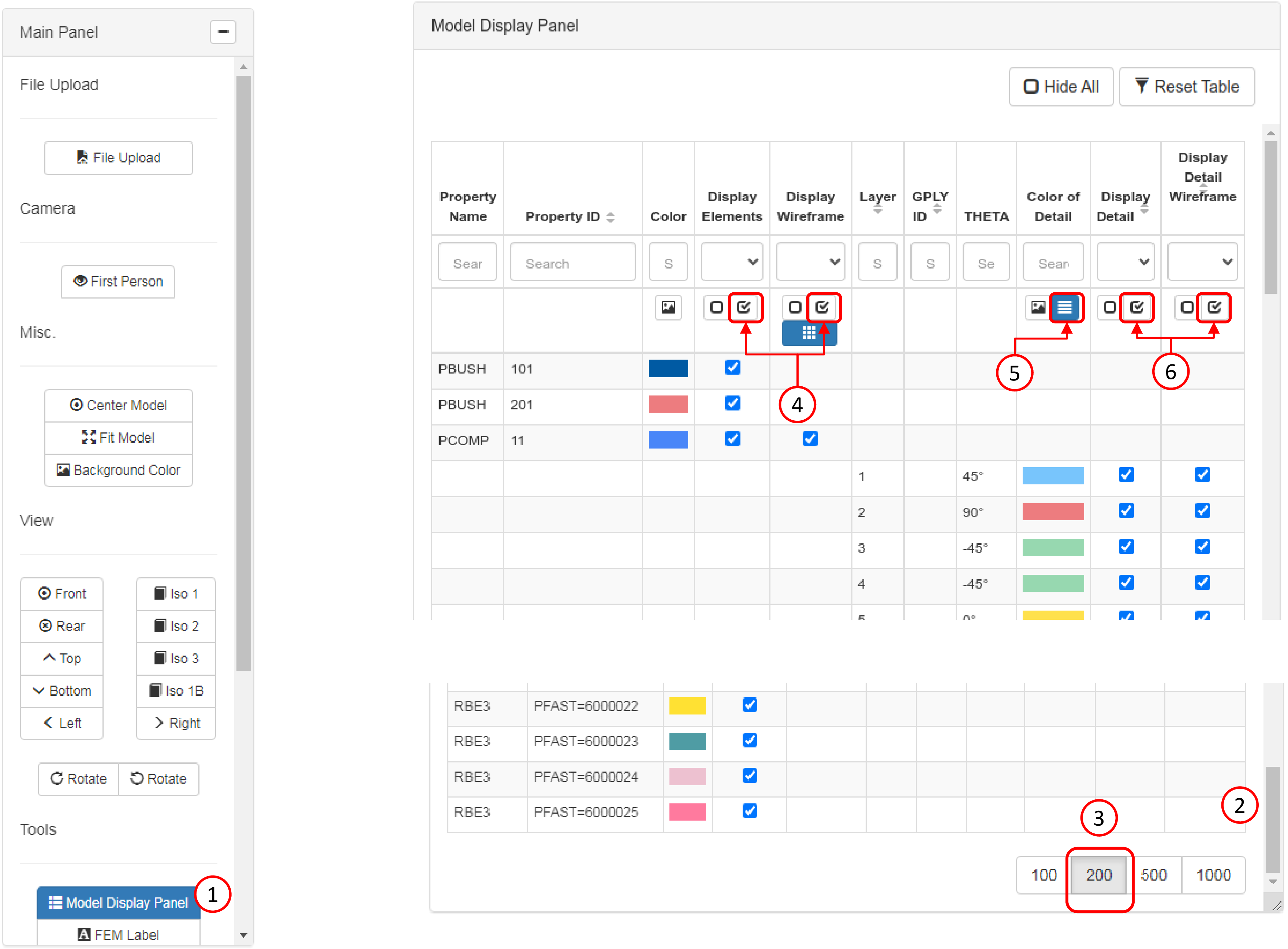
1. Click File Upload
2. Click PCH
3. Click Select files
4. Navigate to the directory 1_starting_files
5. Select the following files:
    1. model.pch
6. Click Open
7. Click Upload files

- Why is the PCH file uploaded to the Viewer? Refer to the appendix for an explanation.



Model Display Panel

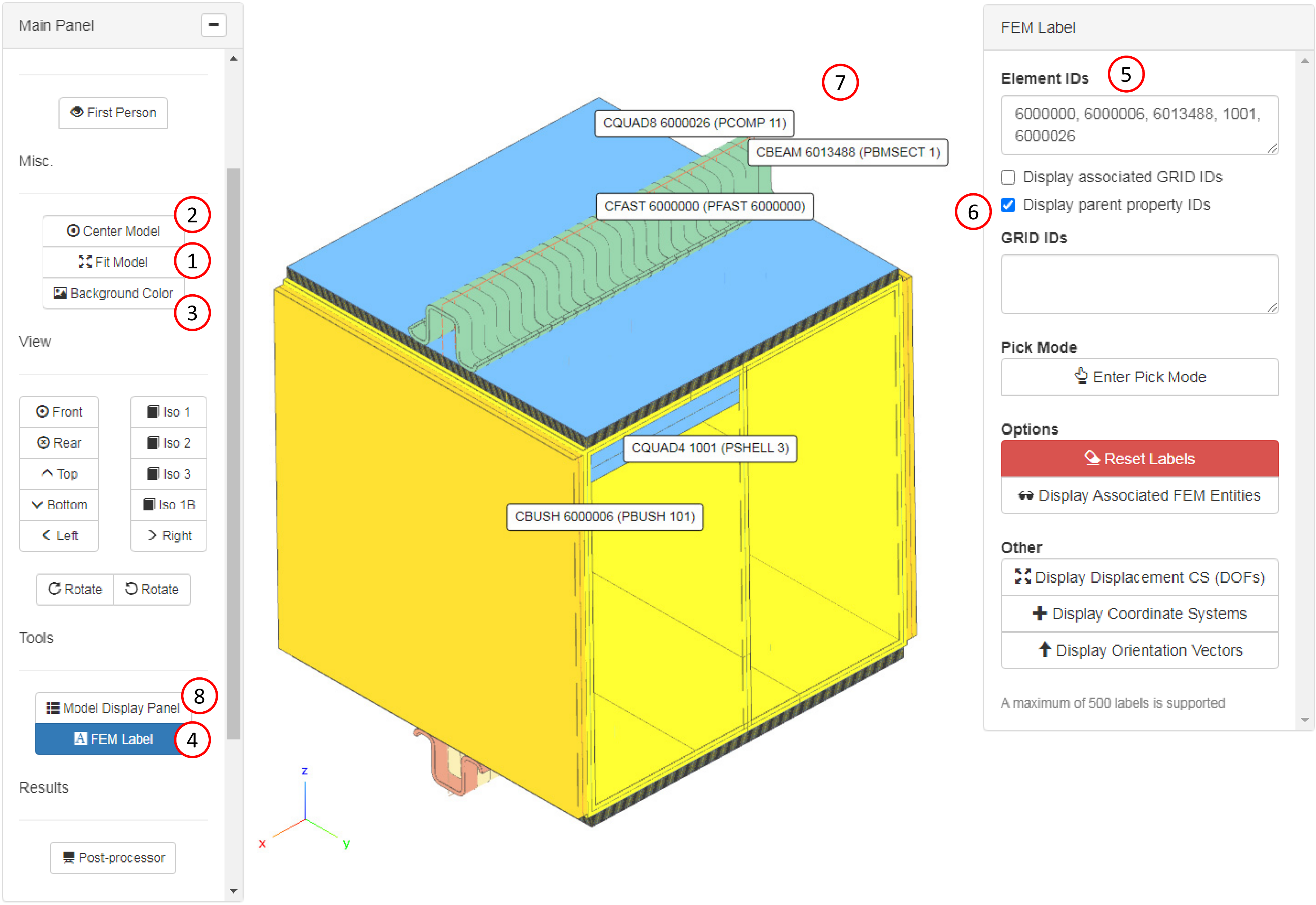
1. Click Model Display Panel
2. Use the scroll bar to scroll the bottom of the table
3. Click 200 to display at most 200 rows in the table
4. Click the indicated button to mark all the visible checkboxes
   - These columns control the display of elements and element edges
5. Click the indicated button to color coordinate the layers of the PCOMP entry. For example, all 45-degree layers are colored blue.
6. Click the indicated button to mark all the visible checkboxes.
   - These columns control the display of shell thickness, composite layer thickness and beam cross section.



FEM Label

1. Click Fit Model
2. Click Center Model
3. Click Background Color
4. Click FEM Label
5. Set Element IDs to: 6000000, 6000006, 6013488, 1001, 6000026
6. Mark the checkbox titled Display parent property IDs
7. Element ID labels are displayed on the FE model
8. Click Model Display Panel

This model consists of various types of elements, including CBUSH, CFAST, CBEAM, CQUAD4 and CQUAD8. . The results of each element type will be inspected in this tutorial.

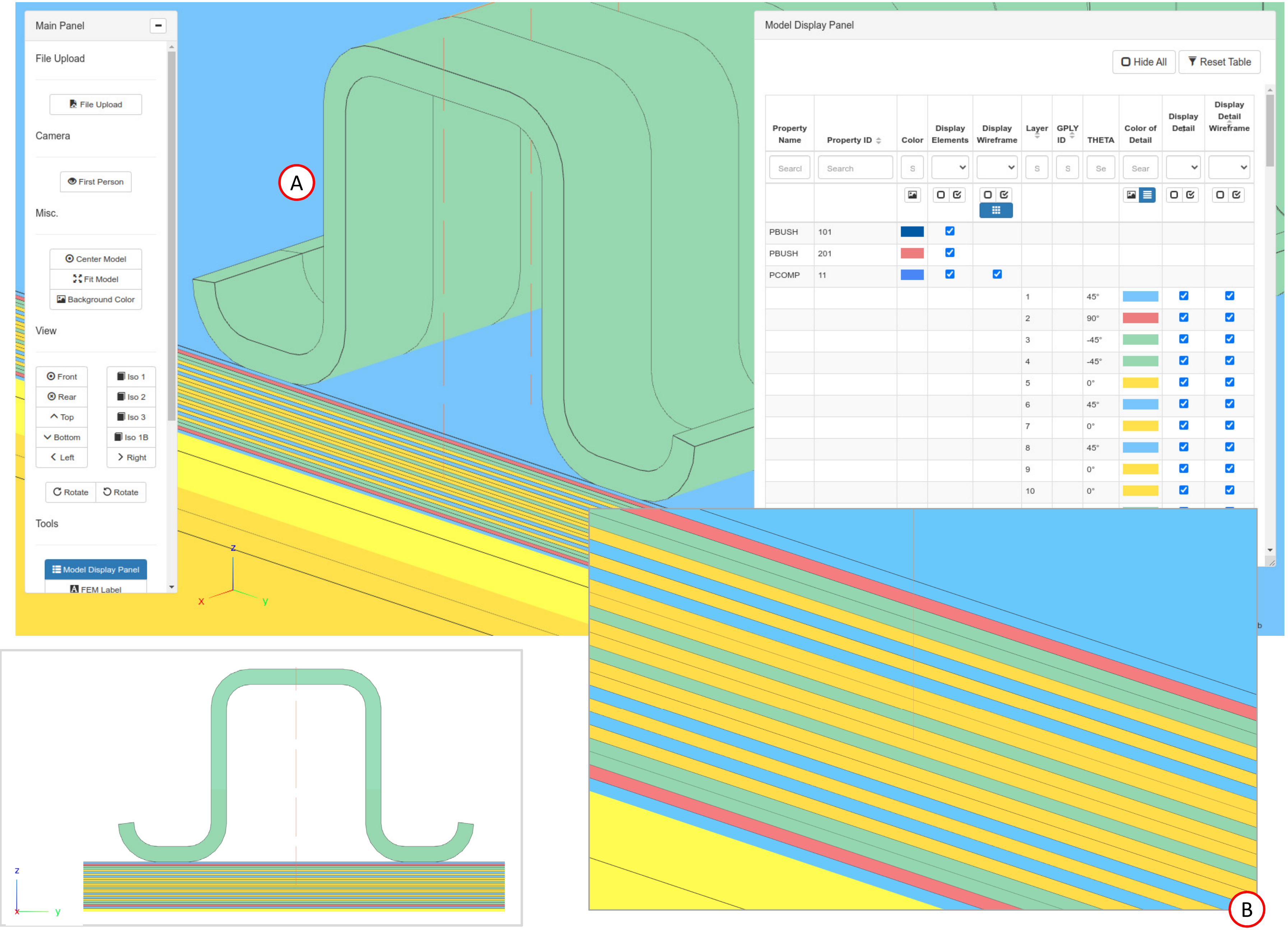


Beam Cross Section and Composite Laminate Layers

Close inspection of the model reveals some interesting details.

A. The CBEAM entries have an arbitrary beam cross section defined via PBMSECT entries.

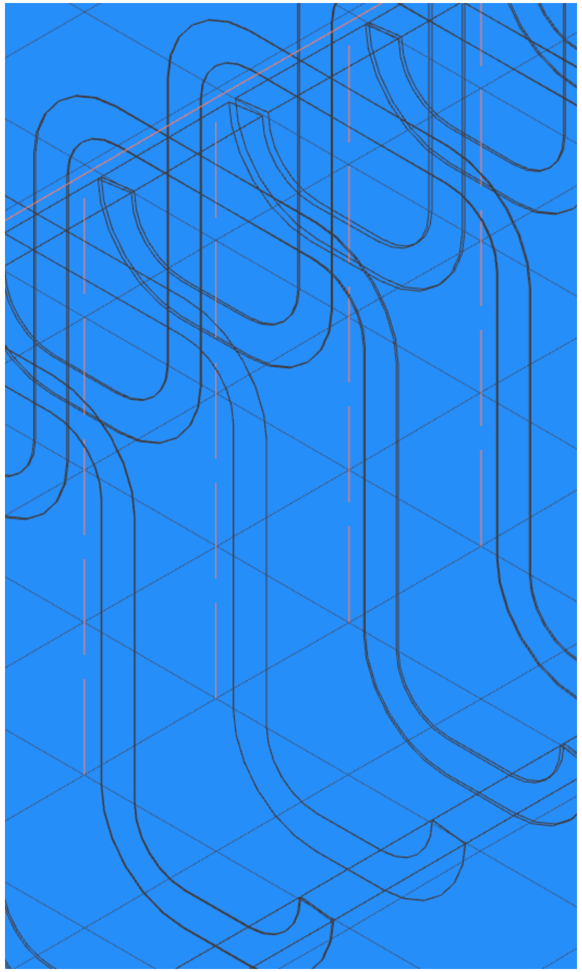
B. The composite defined via a PCOMP entry is composed of multiple layers.

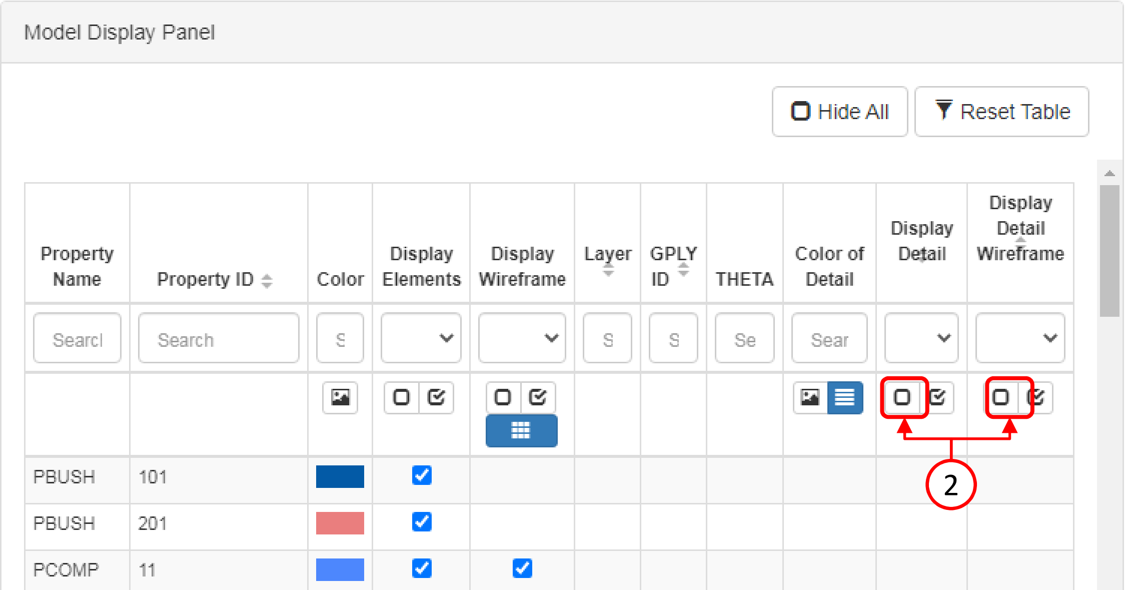


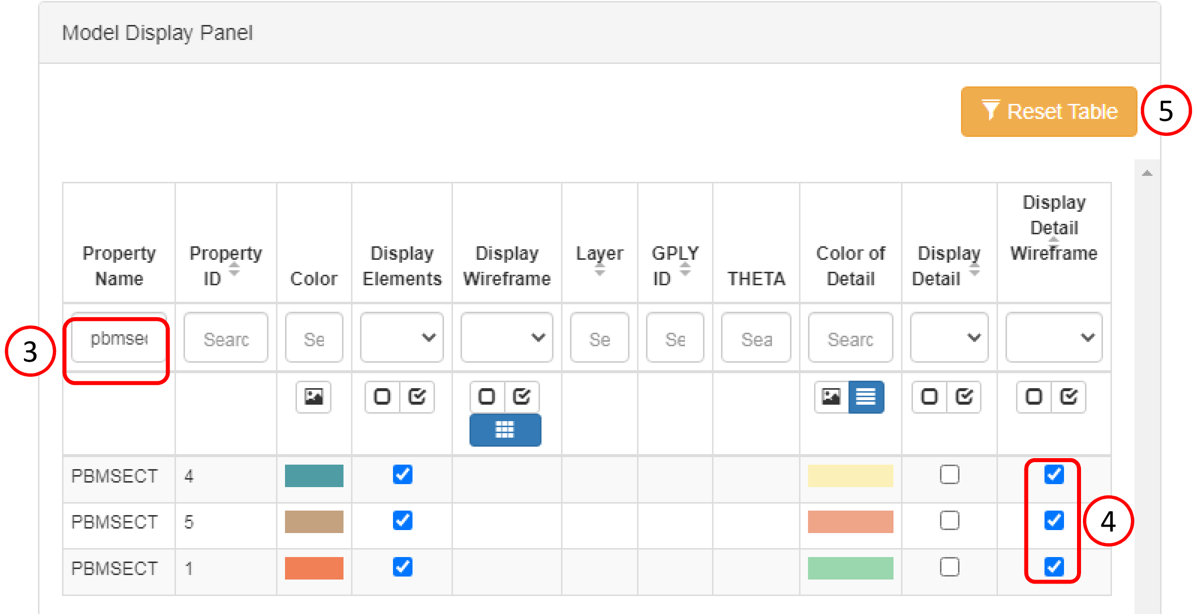
Model Display Panel

1. Click Model Display Panel (not shown)
2. Click the indicated button to unmark all the visible checkboxes
3. Use the search bar to search for: pbmsect
4. Mark the indicated checkboxes
    - This will display the cross section of the arbitrary beam cross section defined with CBEAM and PBMSECT entries
5. Click Reset Table
    - This will reset the table search
6. The beam elements are displayed with only the wireframe of the beam cross sections (black lines)

6



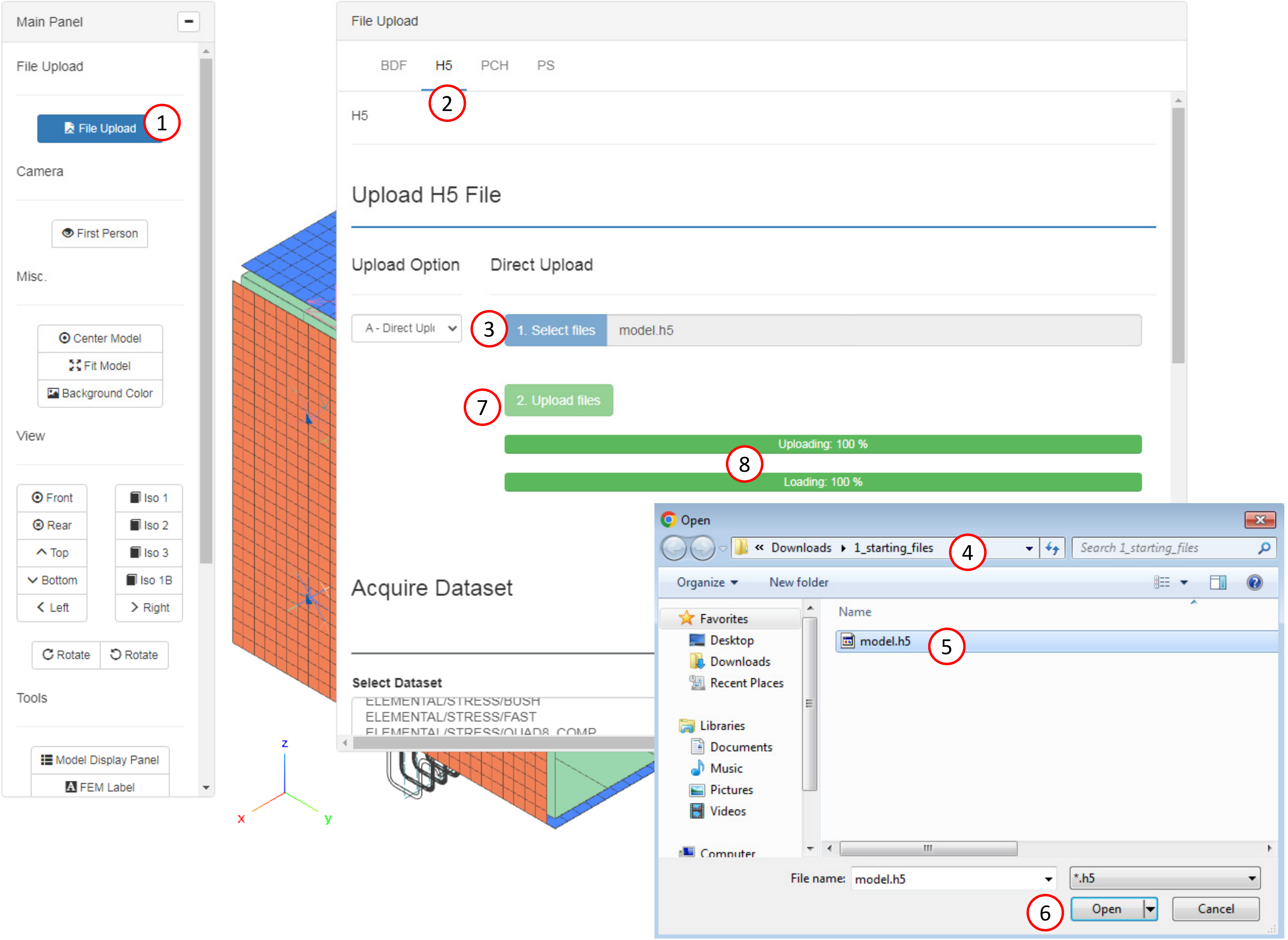




Upload Files

1. Click File Upload
2. Click H5
3. Click Select files
4. Navigate to the directory 1_starting_files
5. Select the following files:
   - model.h5
6. Click Open
7. Click Upload files
8. Wait for both progress bars to reach 100% completion, then continue

- The Post-processor web app extracts results from the H5 file. Other output files, such as OP2 and XDB, are not support. Only the H5 file is supported.

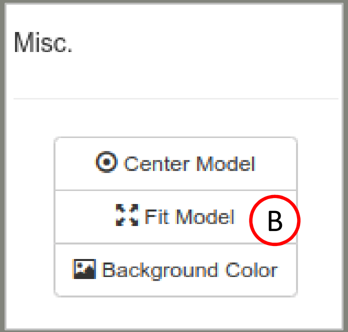


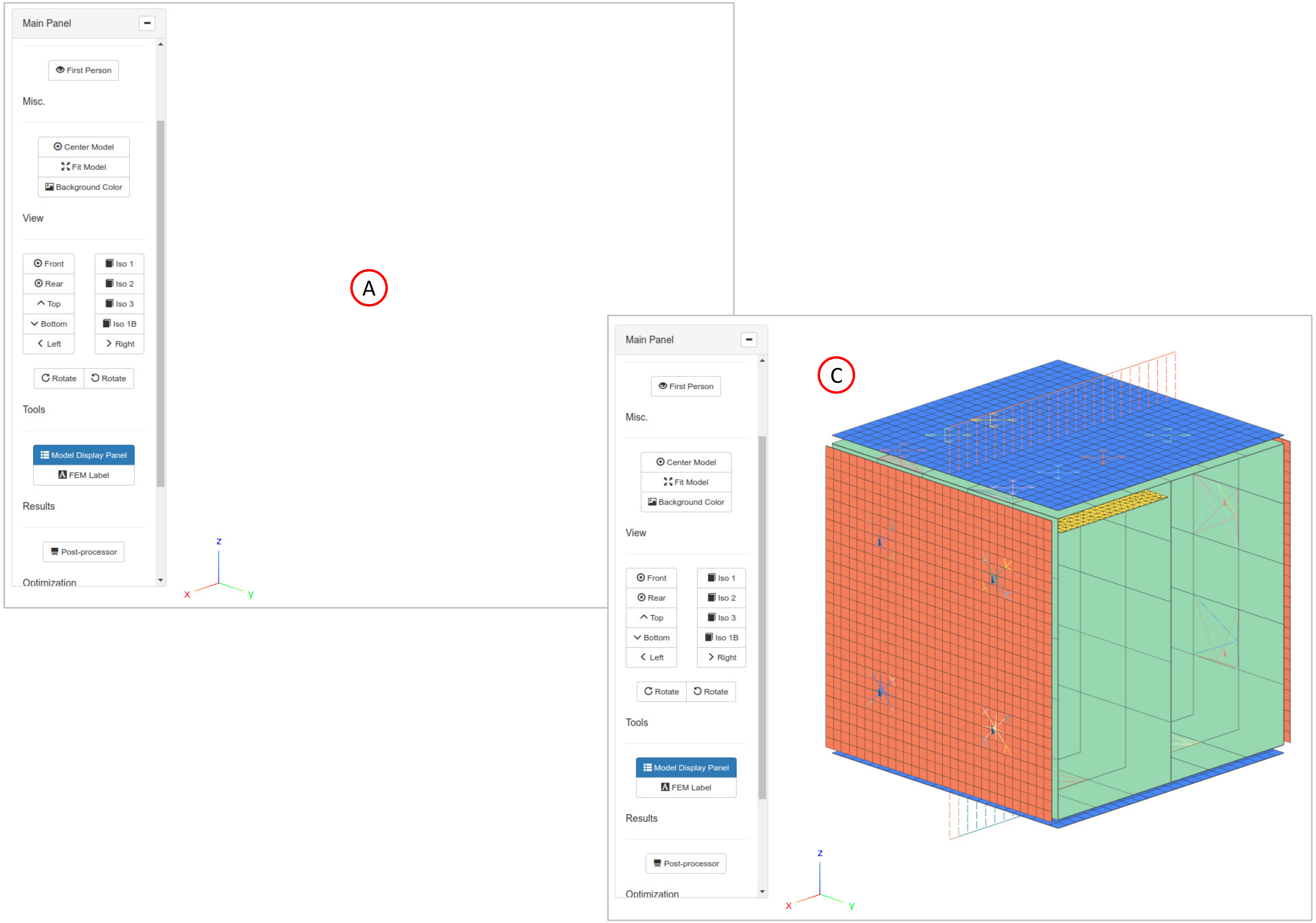
Before Continuing

A. As you hide and show elements and details, and as you navigate the model, you might reach a point where disorientation occurs.

B. Click Fit Model.

C. This will restore the model to a visible state.





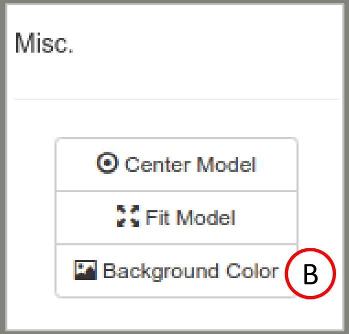
Before Continuing

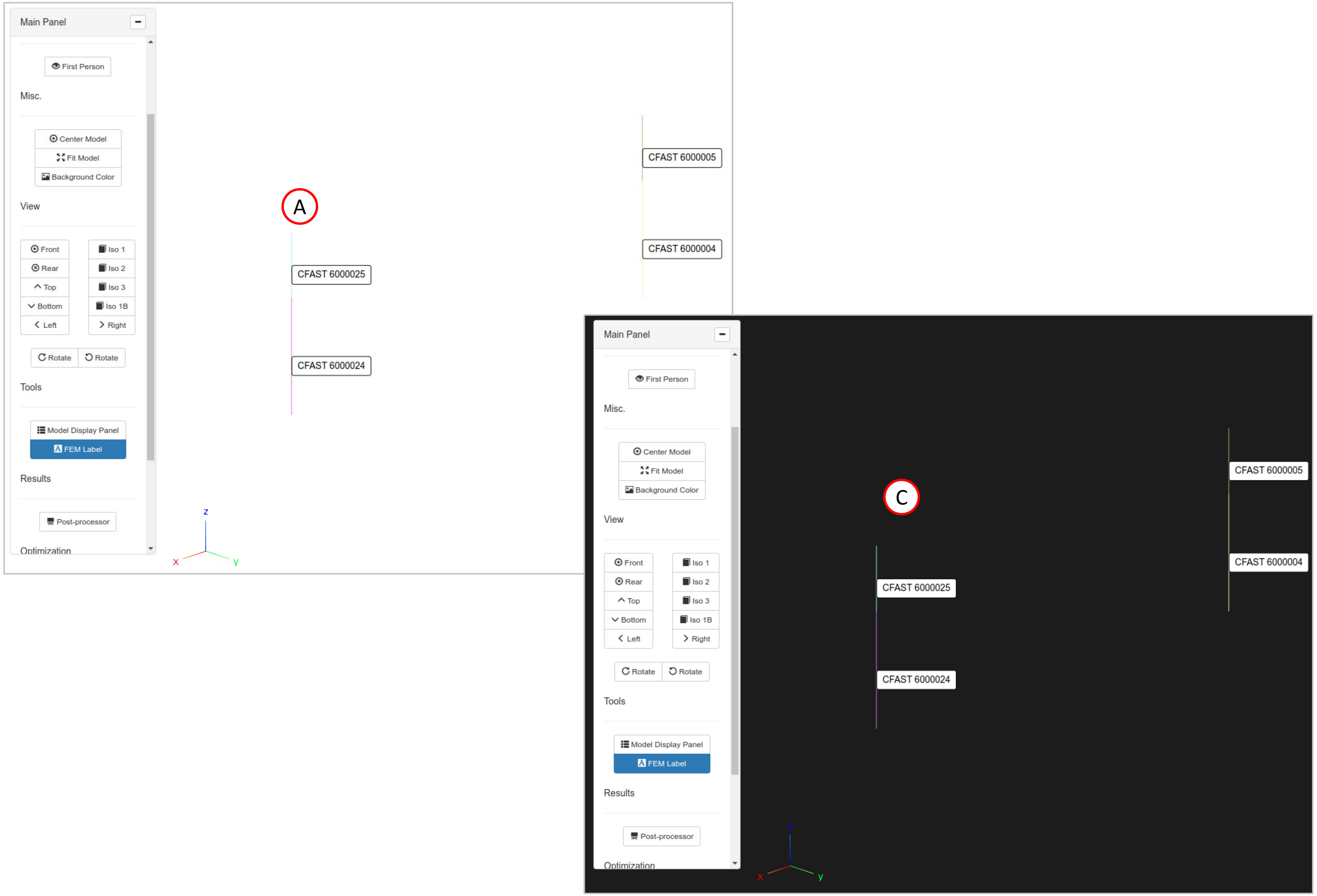
Later on in this tutorial, you will be working with CFAST connector elements. Consider the following when working with 1D elements and connectors.

A. Certain elements, such as CFAST elements, are represented as lines. The color of the lines sometimes does not contrast well against a white background and makes it difficult to distinguish the lines.

B. Click Background Color.

C. This will change the background color to black and will help distinguish the elements represented with lines.



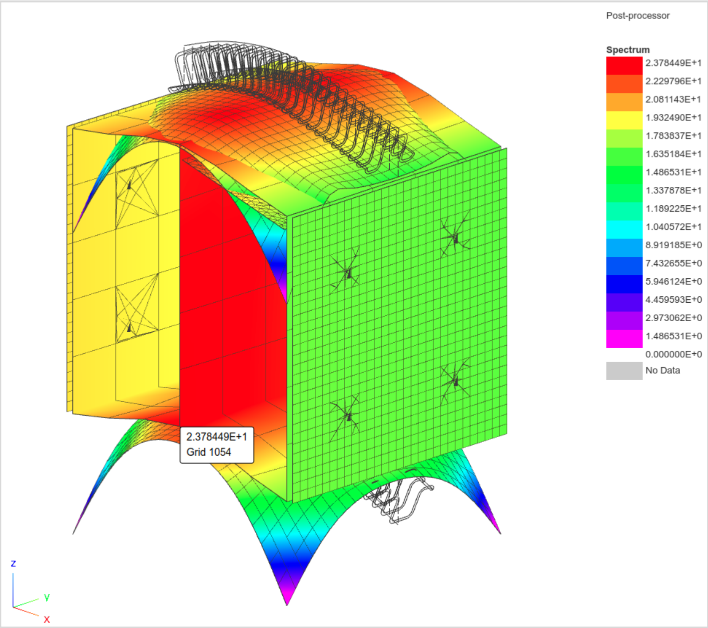


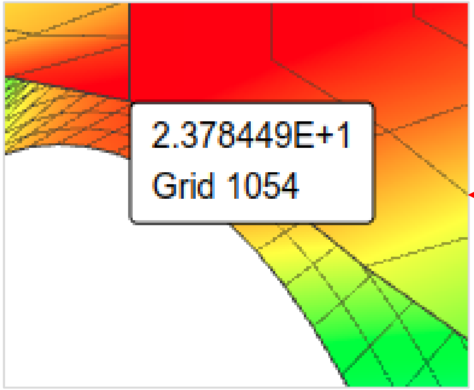
Nastran Displacement Results

Goal: Display CFAST element forces

F06 File

```
0                                                                                                   SUBCASE 1
                                          D I S P L A C E M E N T   V E C T O R

      POINT ID.    TYPE          T1             T2             T3             R1             R2             R3
         1054       G      4.533402E-02   1.694495E+00   2.378449E+01  -2.665313E-02   1.776151E-02   2.036122E-03
```

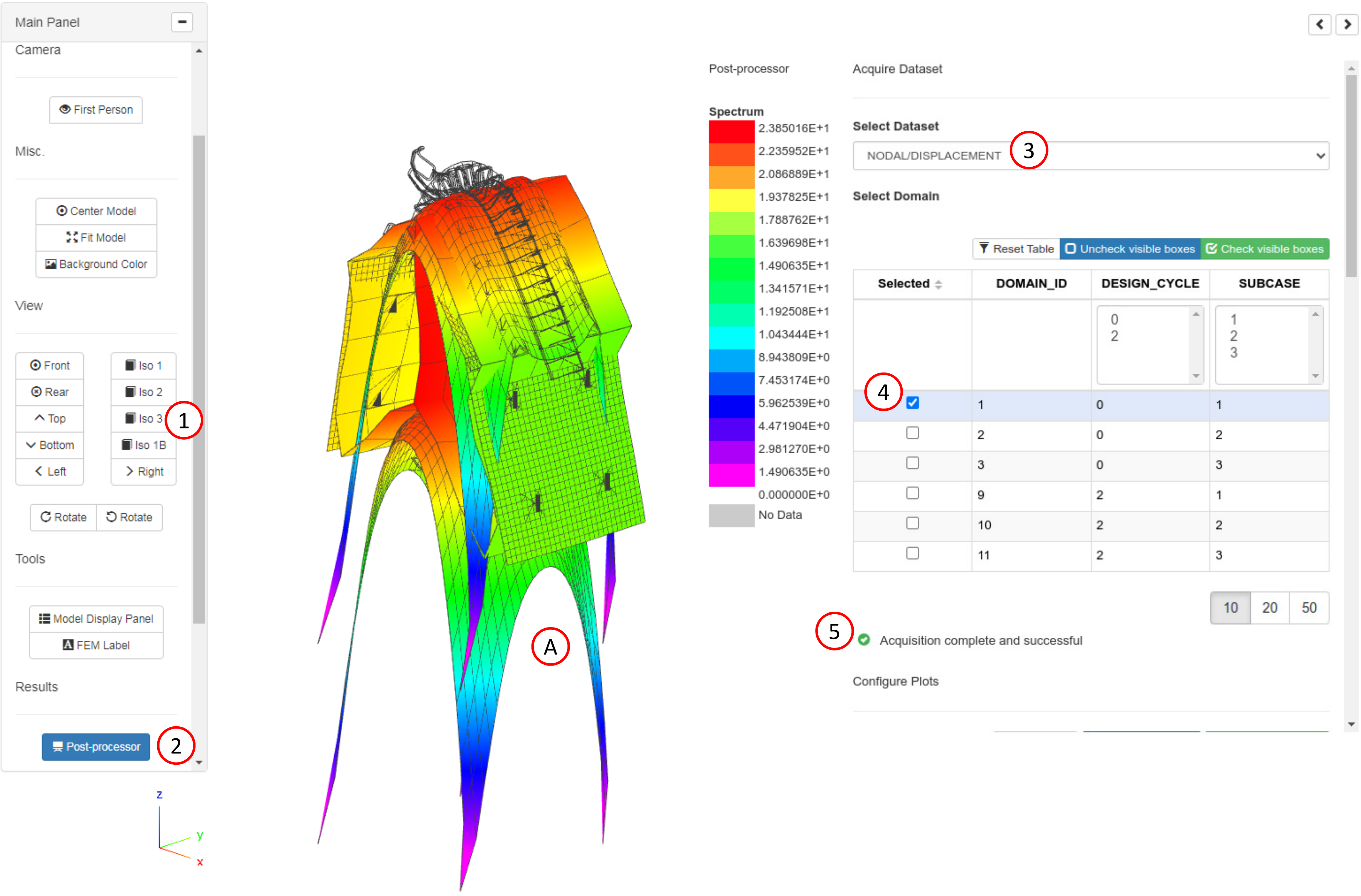




Nastran Displacement Results

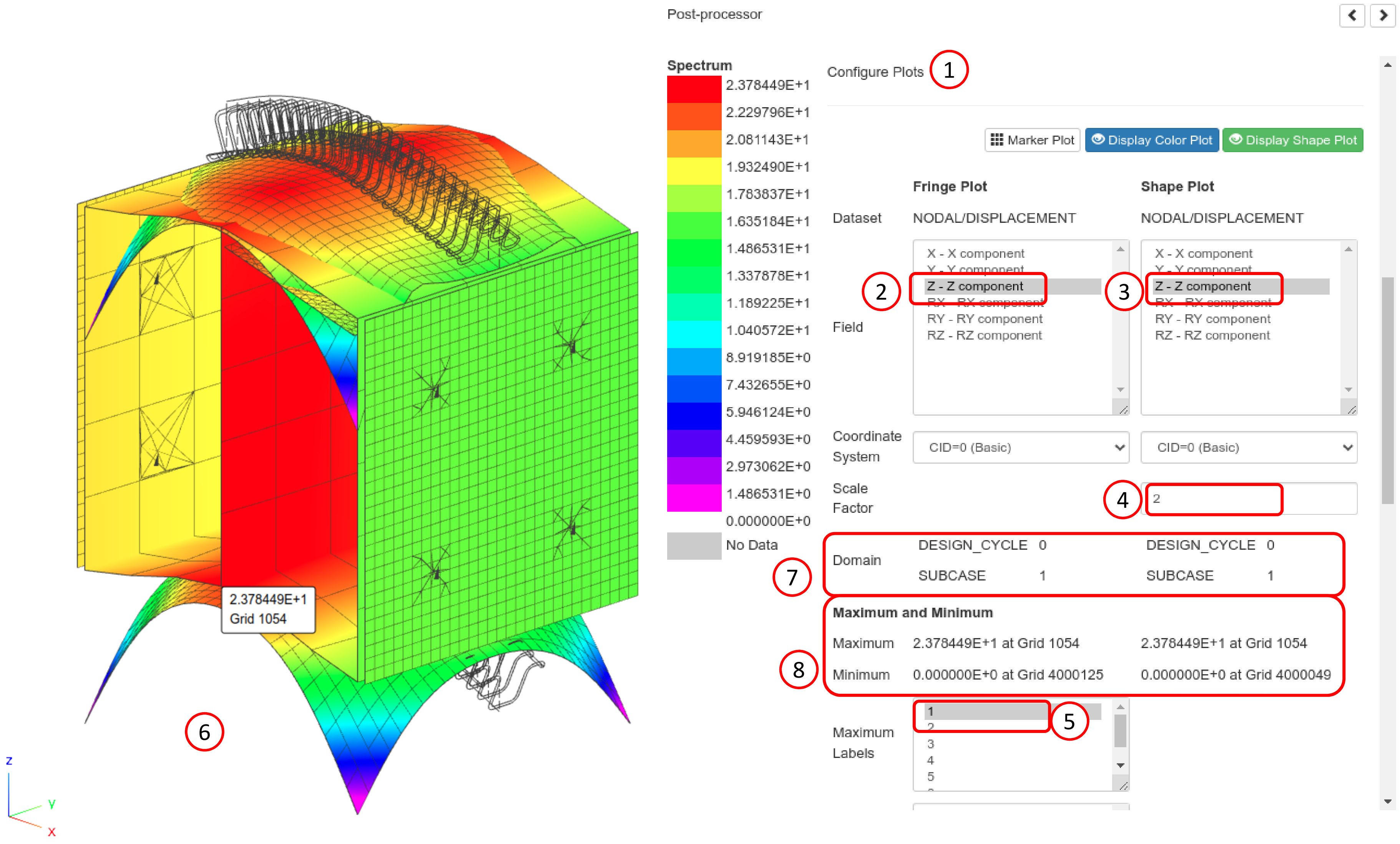
1. Click Iso 3
2. Click Post-processor if not done already
3. Set Select Dataset to NODAL/DISPLACEMENT
4. Mark the checkbox for SUBCASE 1 for DESIGN_CYCLE 0
5. Wait until the status says Acquisition complete and successful

A. The scale factor is used to make the displacements more pronounced and noticeable. The default scale factor is 10 and in this case leads to a significantly deformed shape. On the next page, the scale factor is reduced.



Nastran Displacement Results

1. Use the vertical scroll bar to navigate to section Configure Plots
2. For the fringe plot, for field, select Z - Z component
3. For the shape plot, for field, select Z - Z component
4. Set the scale factor to 2
5. Select the first maximum label
6. The deformation and fringe plot are displayed. Also, a label for the maximum z-component of displacement is displayed.
7. The corresponding domain is displayed
8. The a summary of the maximum and minimum values is displayed



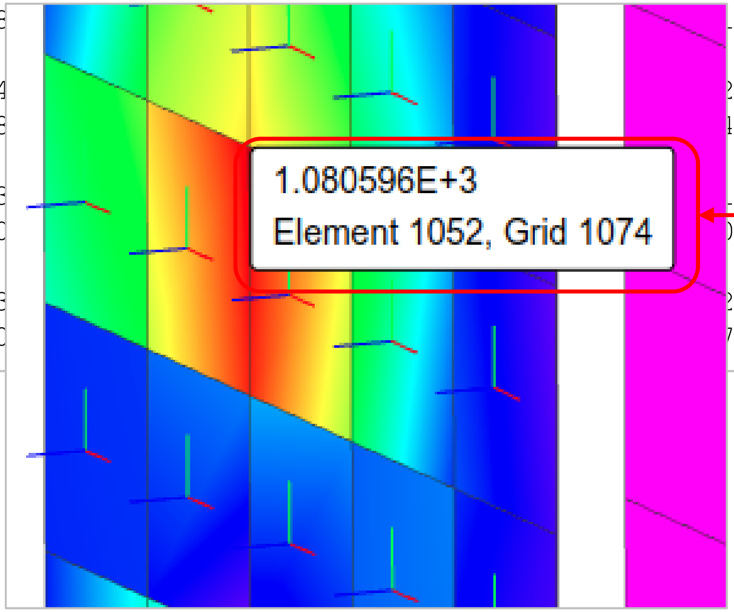
Nastran Stress Results - CQUAD4 and PSHELL

Goal: Display PSHELL stresses

F06 File

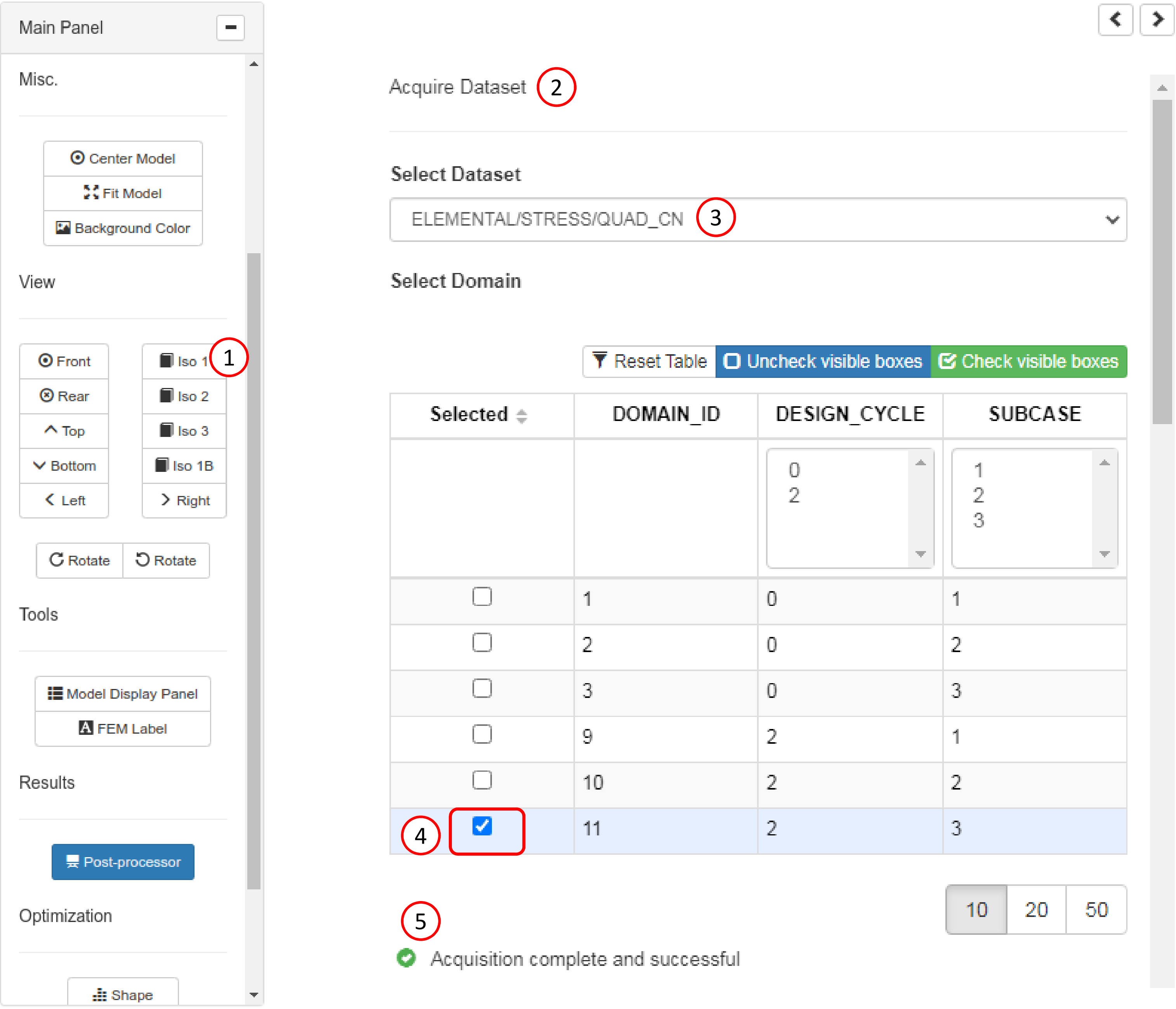
S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   (Q U A D 4)   OPTION = BILIN

	ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
0	1052	CEN/4	-1.000000E+00	-7.439354E+02	-1.063480E+03	8.355292E+01	13.8037	-7.234072E+02	-1.084008E+03	9.561447E+02
			1.000000E+00	7.451064E+02	1.060078E+03	-8.420532E+01	-75.9336	1.081176E+03	7.240080E+02	9.541222E+02
		1067	-1.000000E+00	-6.255414E+02	-9.283089E+02	8.355292E+01	14.4478	-6.040144E+02	-9.498359E+02	8.326506E+02
			1.000000E+00	6.263418			.94	9.477564E+02	6.042814E+02	8.310756E+02
		1068	-1.000000E+00	-6.255414			277	-6.136088E+02	-1.210584E+03	1.048430E+03
			1.000000E+00	6.263418			441	1.206677E+03	6.141238E+02	1.045069E+03
		1074	-1.000000E+00	-8.623293			.05	-8.427159E+02	-1.218265E+03	1.080596E+03
			1.000000E+00	8.638710			)22	1.214672E+03	8.436585E+02	1.078155E+03
		1073	-1.000000E+00	-8.623293			270	-8.054891E+02	-9.851490E+02	9.087379E+02
			1.000000E+00	8.638710			793	9.844836E+02	8.050833E+02	9.081717E+02



Nastran Stress Results - CQUAD4 and PSHELL

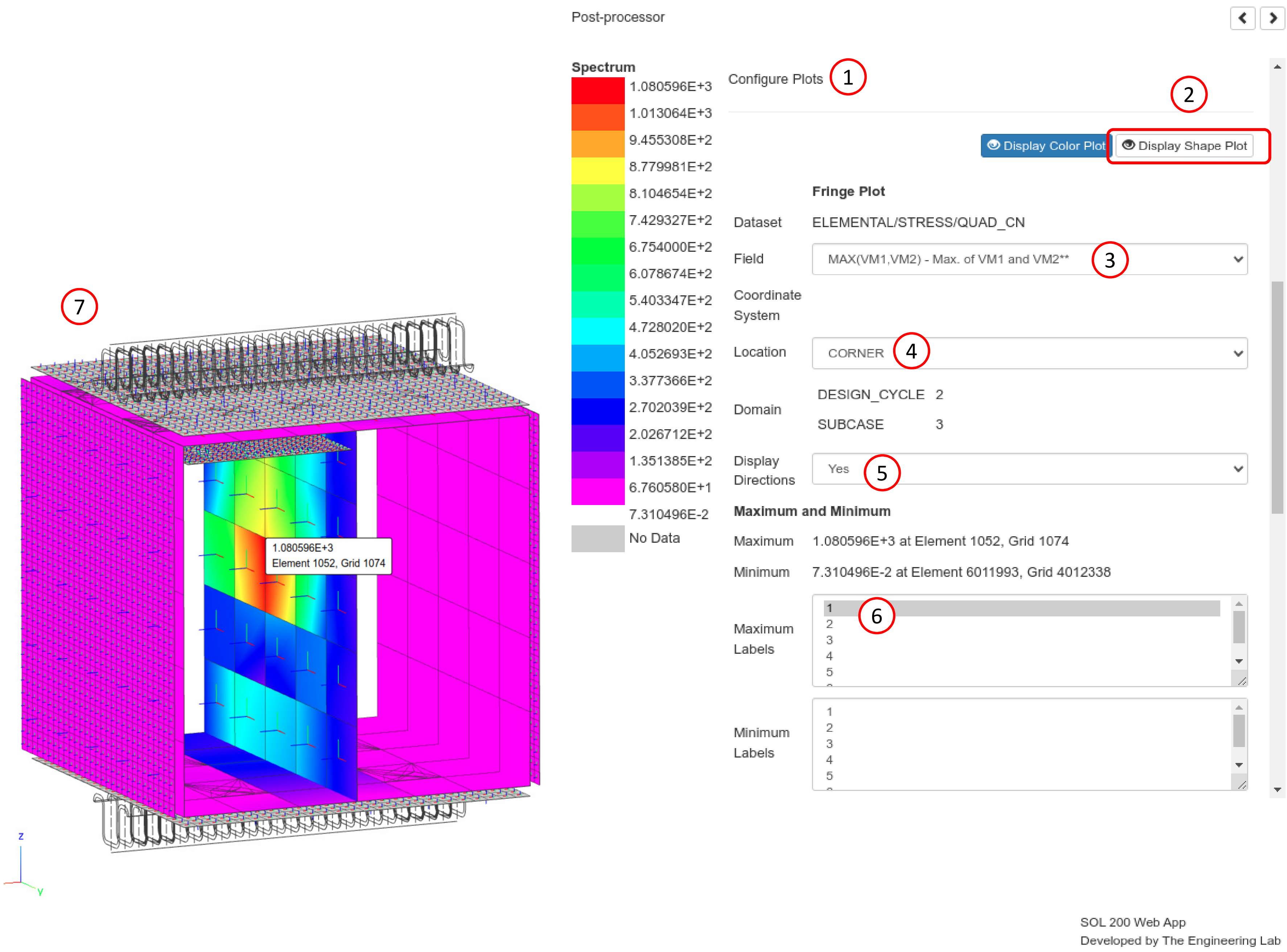
1. Click Iso 1
2. Use the vertical scroll bar to navigate to section Acquire Dataset
3. Set Select Dataset to ELEMENTAL/STRESS/QUAD_CN
4. Mark the checkbox for SUBCASE 3 for DESIGN_CYCLE 2
5. Wait until the status says Acquisition complete and successful



Nastran Stress Results - CQUAD4 and PSHELL

1. Use the vertical scroll bar to navigate to section Configure Plots
2. Click Display Shape Plot to hide the deformation
3. Set Field to MAX(VM1, VM2)
   - This option displays the maximum von Mises stress of the top and bottom of the thickness
4. Set Location to CORNER
5. Set Display Direction to Yes
6. Select the first maximum label
7. The fringe plot is displayed. Also, a label for the maximum von Mises stress is displayed.

- The Display Directions option is used to display the orientation of the forces or stress/strain tensor.



Nastran Stress Results - CQUAD4 Corner and Center Output

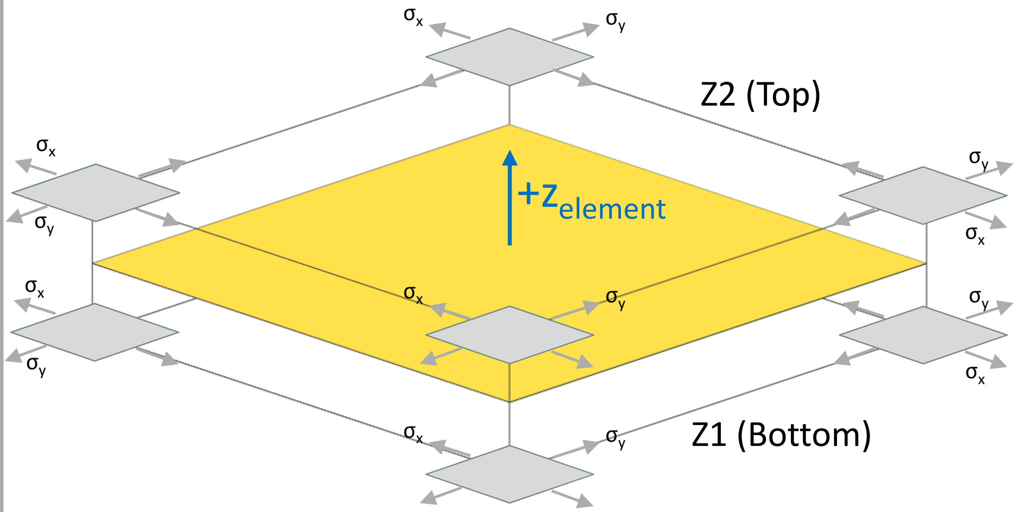
1. Refer to the red box.

When inspecting the output in the F06 file, each row in the red box represents a tensor at either the corners or the center of the bottom or top of the thickness.

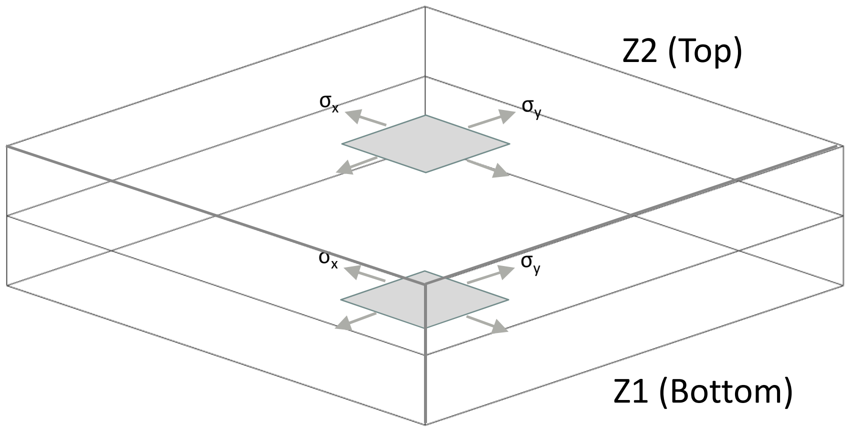
- NORMAL-X: σ_x
- NORMAL-Y: σ_x
- SHEAR-XY: τ_{xy}

F06 File

```
S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )        OPTION = BIL

    ELEMENT                    FIBER          1   STRESSES IN ELEMENT COORD SYSTEM         PRINCIPAL STRESSES (Z
      ID       GRID-ID        DISTANCE         NORMAL-X       NORMAL-Y       SHEAR-XY      ANGLE         MAJOR
0       1052    CEN/4    -1.000000E+00  -7.439354E+02 -1.063480E+03  8.355292E+01   13.8037  -7.234072E+02
                          1.000000E+00   7.451064E+02  1.060078E+03 -8.420532E+01  -75.9336   1.081176E+03

                  1067   -1.000000E+00  -6.255414E+02 -9.283089E+02  8.355292E+01   14.4478  -6.040144E+02
                          1.000000E+00   6.263418E+02  9.256959E+02 -8.420532E+01  -75.3194   9.477564E+02

                  1068   -1.000000E+00  -6.255414E+02 -1.198651E+03  8.355292E+01    8.1277  -6.136088E+02
                          1.000000E+00   6.263418E+02  1.194459E+03 -8.420532E+01  -81.7441   1.206677E+03

                  1074   -1.000000E+00  -8.623293E+02 -1.198651E+03  8.355292E+01   13.2105  -8.427159E+02
                          1.000000E+00   8.638710E+02  1.194459E+03 -8.420532E+01  -76.5022   1.214672E+03

                  1073   -1.000000E+00  -8.623293E+02 -9.283089E+02  8.355292E+01   34.2270  -8.054891E+02
                          1.000000E+00   8.638710E+02  9.256959E+02 -8.420532E+01  -55.0793   9.844836E+02
```



CQUAD4 Corner Output



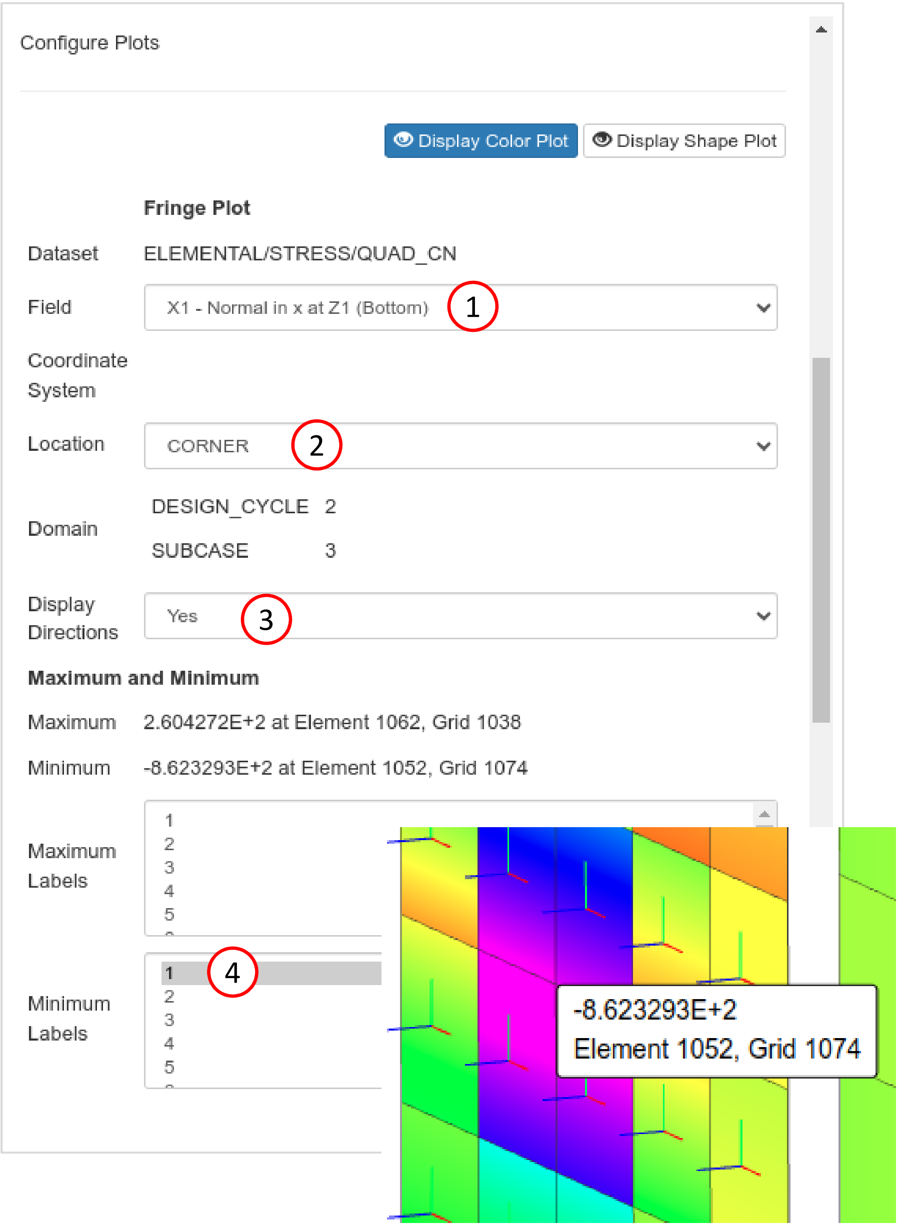
CQUAD4 Center Output

Nastran Stress Results - CQUAD4 and PSHELL

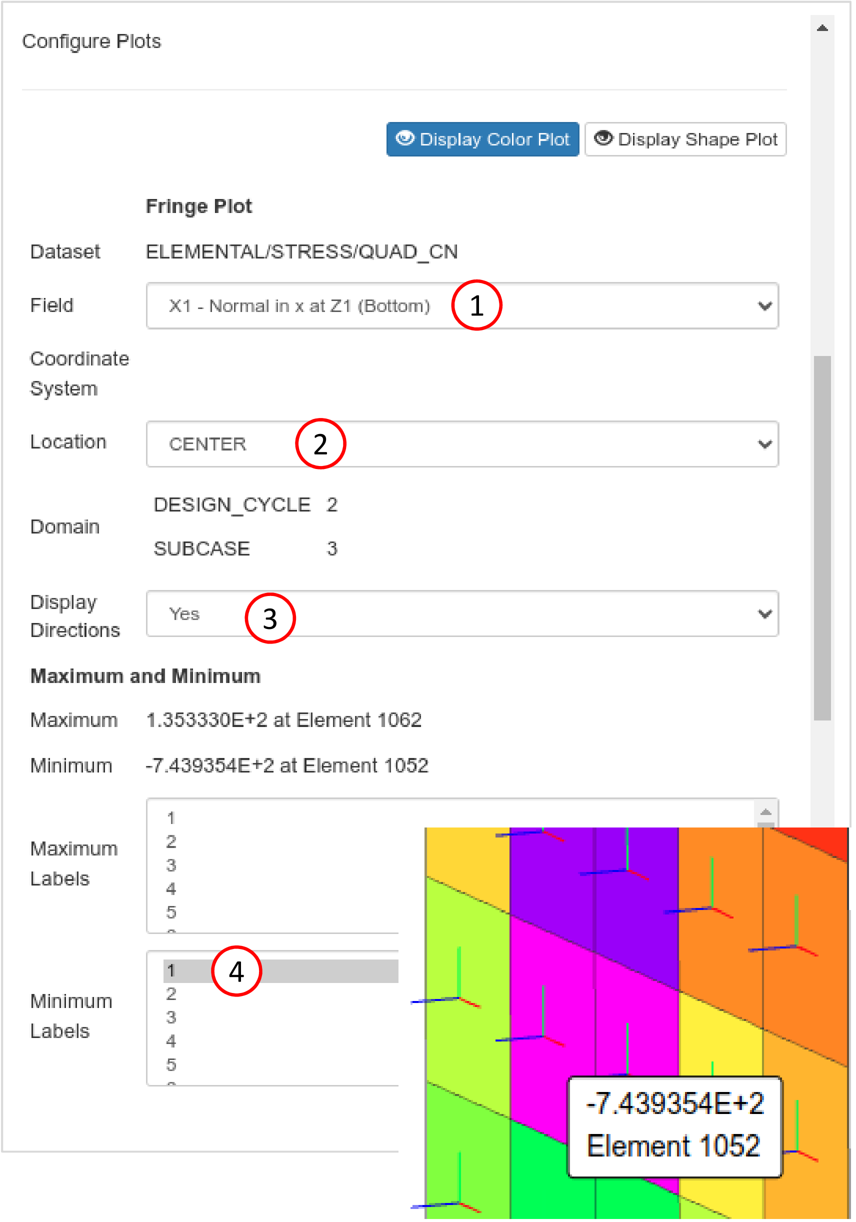
On the next page, 2 stress plots for σ_x (Normal in x) at the bottom of the thickness are created. One plot is for the corner values and the second plot is for the center values. The plots are created as follows.

1. Set Field to X1 – Normal in x at Z1 (Bottom)
2. Set Location to CORNER or CENTER
3. Set Display Directions to Yes
4. Select the first option for Minimum Labels

For both plots, notice the only difference is the Location. For one plot the location is set to CORNER and for the second plot the location is set to CENTER.



CQUAD4 Corner Output



CQUAD4 Center Output

Nastran Stress Results - CQUAD4 and PSHELL

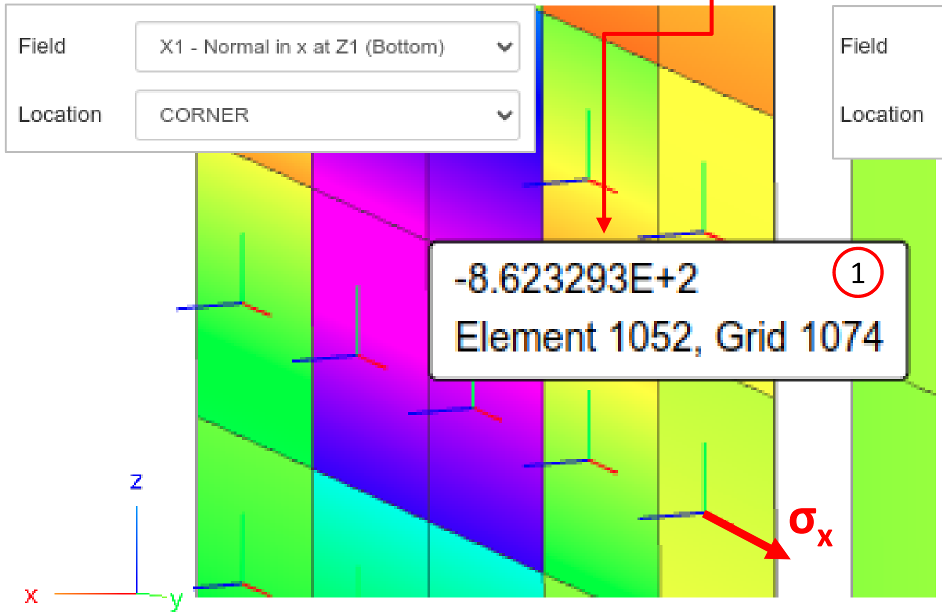
Fringe plotters normally allow the display of one component/field at a single time.

As shown to the right, the X1 field (NORMAL-X/σ_x) is displayed for the bottom (Z1) of the thickness for both the center and corner locations.

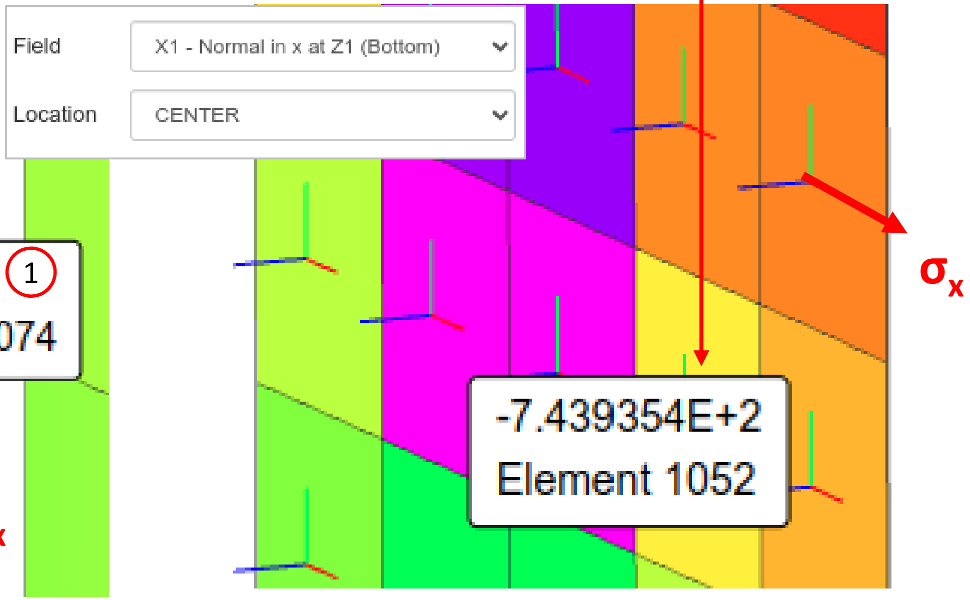
1. When the Display Directions option is set to Yes, an axes helper is displayed for each element and indicates the general orientation of the stress tensor. The red axis represents the direction of the NORMAL-X (σ_x) stress.

F06 File

```
S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )        OPTION = BIL

    ELEMENT              FIBER            STRESSES IN ELEMENT COORD SYSTEM              PRINCIPAL STRESSES (Z
      ID      GRID-ID   DISTANCE        NORMAL-X       NORMAL-Y      SHEAR-XY        ANGLE         MAJOR
0       1052   CEN/4  -1.000000E+00  -7.439354E+02  -1.063480E+03  8.355292E+01    13.8037   -7.234072E+02
                       1.000000E+00   7.451064E+02   1.060078E+03 -8.420532E+01   -75.9336    1.081176E+03

              1067    -1.000000E+00  -6.255414E+02  -9.283089E+02  8.355292E+01    14.4478   -6.040144E+02
                       1.000000E+00   6.263418E+02   9.256959E+02 -8.420532E+01   -75.3194    9.477564E+02

              1068    -1.000000E+00  -6.255414E+02  -1.198651E+03  8.355292E+01     8.1277   -6.136088E+02
                       1.000000E+00   6.263418E+02   1.194459E+03 -8.420532E+01   -81.7441    1.206677E+03

              1074    -1.000000E+00  -8.623293E+02  -1.198651E+03  8.355292E+01    13.2105   -8.427159E+02
                       1.000000E+00   8.638710E+02   1.194459E+03 -8.420532E+01   -76.5022    1.214672E+03

              1073    -1.000000E+00  -8.623293E+02  -9.283089E+02  8.355292E+01    34.2270   -8.054891E+02
                       1.000000E+00   8.638710E+02   9.256959E+02 -8.420532E+01   -55.0793    9.844836E+02
```



CQUAD4 Corner Output



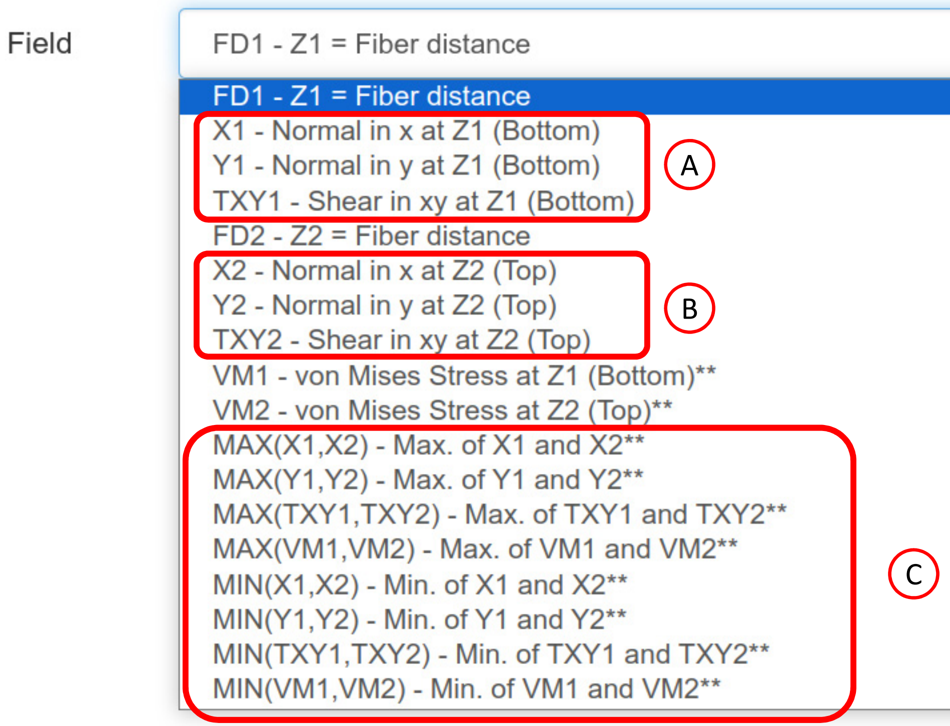
CQUAD4 Center Output

Nastran Stress Results - CQUAD4 and PSHELL

A. Field names associated with Z1 correspond to values at the bottom of the thickness

B. Field names associated with Z2 correspond to values at the top of the thickness

C. Fields are available that consider the maximum or minimum values of the bottom and top values

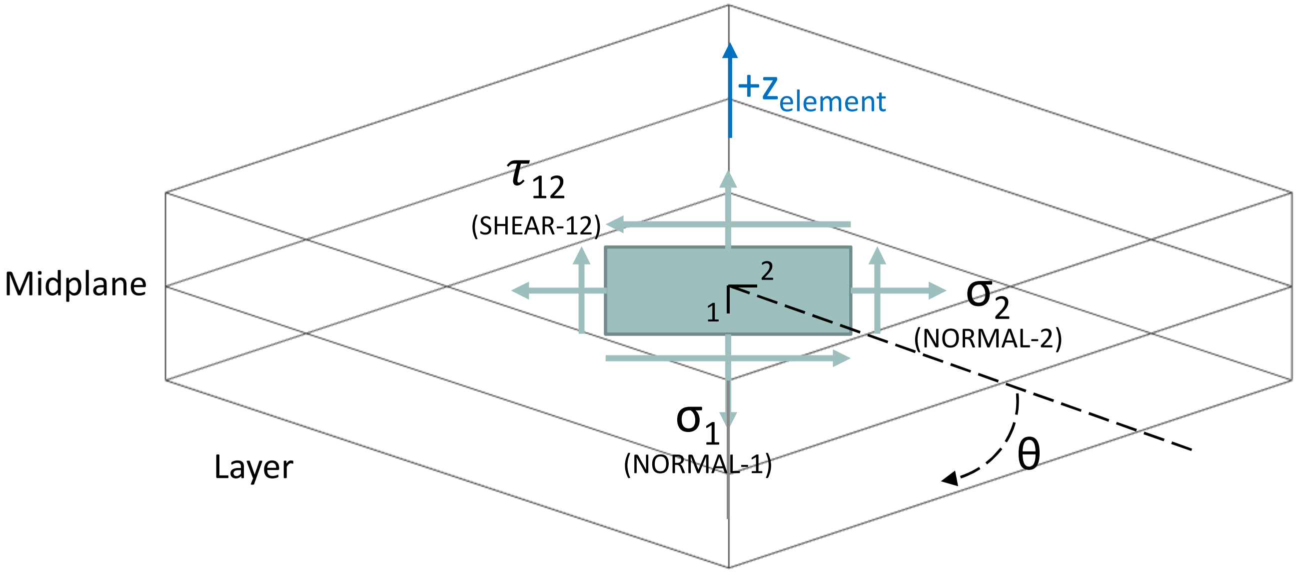


Nastran Composite Stress Results - CQUAD8 and PCOMP

Goal: Display PCOMP layer stresses

F06 File

```
0                                                                                                                  SUBCASE 1
                  S T R E S S E S   I N   L A Y E R E D   C O M P O S I T E   E L E M E N T S   ( Q U A D 8 )
   ELEMENT  PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS    INTER-LAMINAR  STRESSES  PRINCIPAL STRESSES (ZERO SHEAR)      MAX
     ID      ID    NORMAL-1      NORMAL-2      SHEAR-12    SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE     MAJOR         MINOR        SHEAR
0   6000050   1   2.10704E+03  2.89577E+02  2.34311E+02   -6.89357E+00  9.02331E+00    7.23  2.13676E+03  2.59856E+02  9.38451E+02
0   6000050   2   2.59868E+03  2.18754E+02 -8.73254E+01   -9.97859E+00  2.78711E+01   -2.10  2.60188E+03  2.15554E+02  1.19316E+03
[...]
0   6000050  22   5.67456E+03 -6.32410E+01 -2.60880E+02   -9.97859E+00  2.78711E+01   -2.60  5.68640E+03 -7.50779E+01  2.88074E+03
```



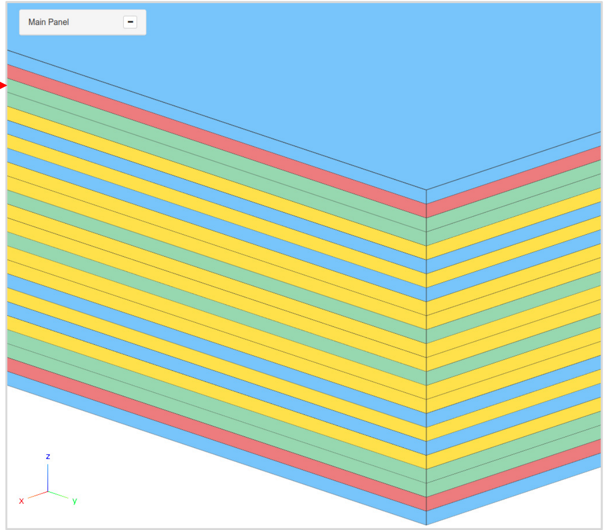
- Layer responses are reported in the 1-2 coordinate system
- Responses are reported at the midplane of each layer

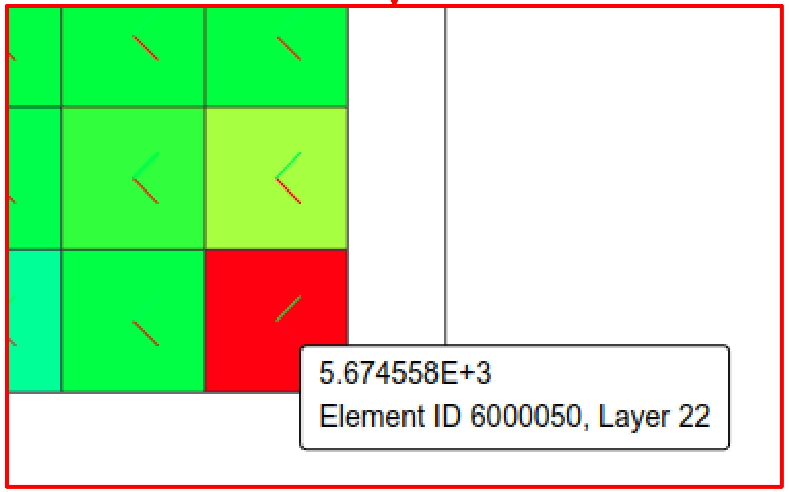
Goal: Display PCOMP layer stresses

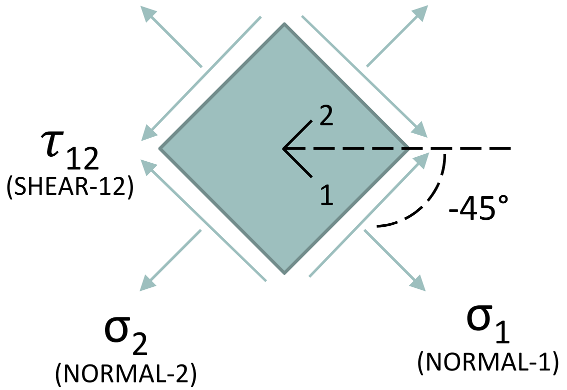
F06 File

```
0                                                                                                                  SUBCASE 1
                     S T R E S S E S   I N   L A Y E R E D   C O M P O S I T E   E L E M E N T S   ( Q U A D 8 )
   ELEMENT   PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS    INTER-LAMINAR  STRESSES  PRINCIPAL STRESSES (ZERO SHEAR)      MAX
     ID      ID    NORMAL-1     NORMAL-2     SHEAR-12      SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE    MAJOR        MINOR        SHEAR
0   6000050    1    2.10704E+03  2.89577E+02  2.34311E+02   -6.89357E+00  9.02331E+00    7.23  2.13676E+03  2.59856E+02  9.38451E+02
0   6000050    2    2.59868E+03  2.18754E+02 -8.73254E+01   -9.97859E+00  2.78711E+01   -2.10  2.60188E+03  2.15554E+02  1.19316E+03
[...]
0   6000050   22    5.67456E+03 -6.32410E+01 -2.60880E+02   -9.97859E+00  2.78711E+01   -2.60  5.68640E+03 -7.50779E+01  2.88074E+03
```

Layer 22

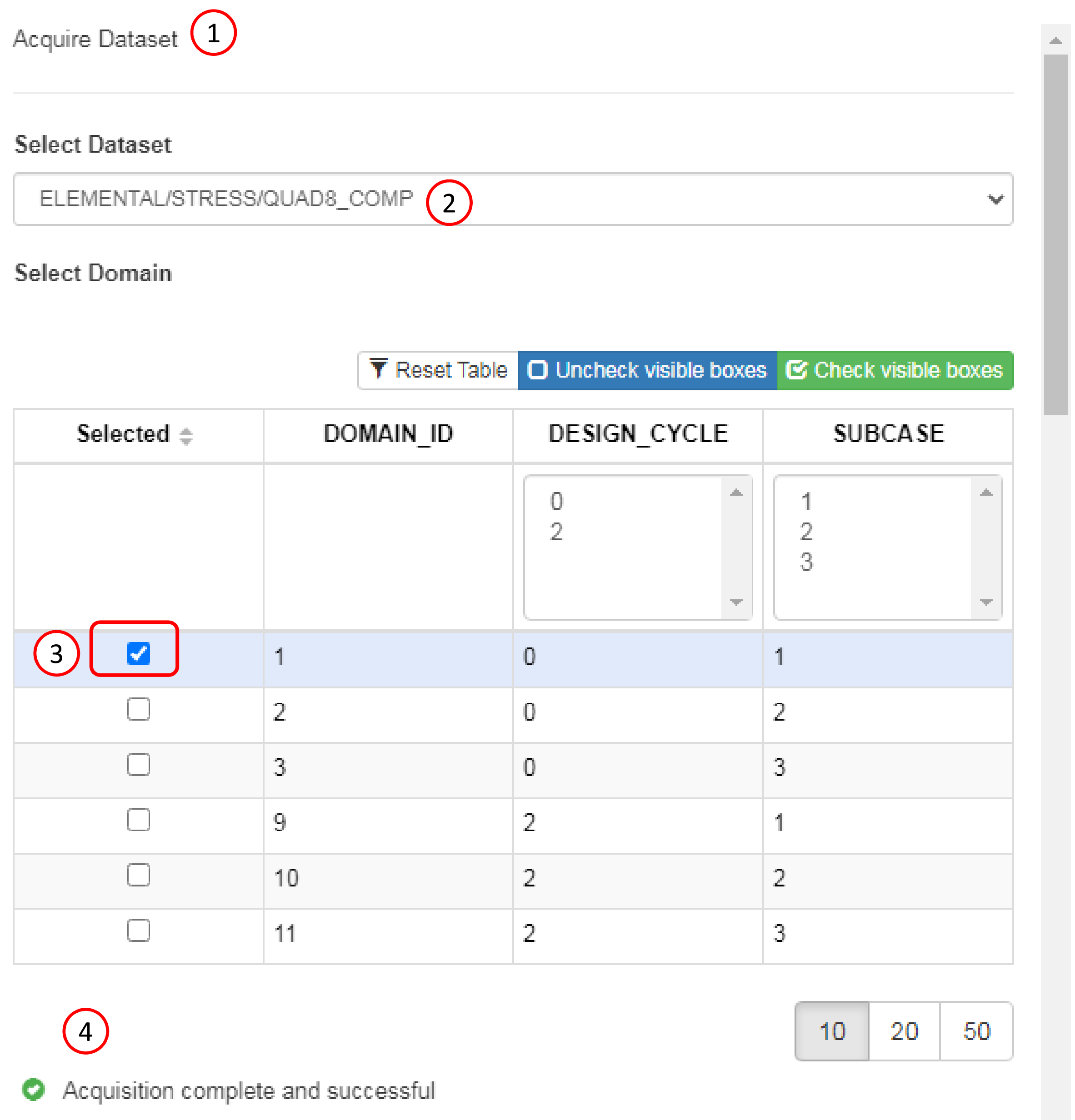






Nastran Composite Stress Results - CQUAD8 and PCOMP

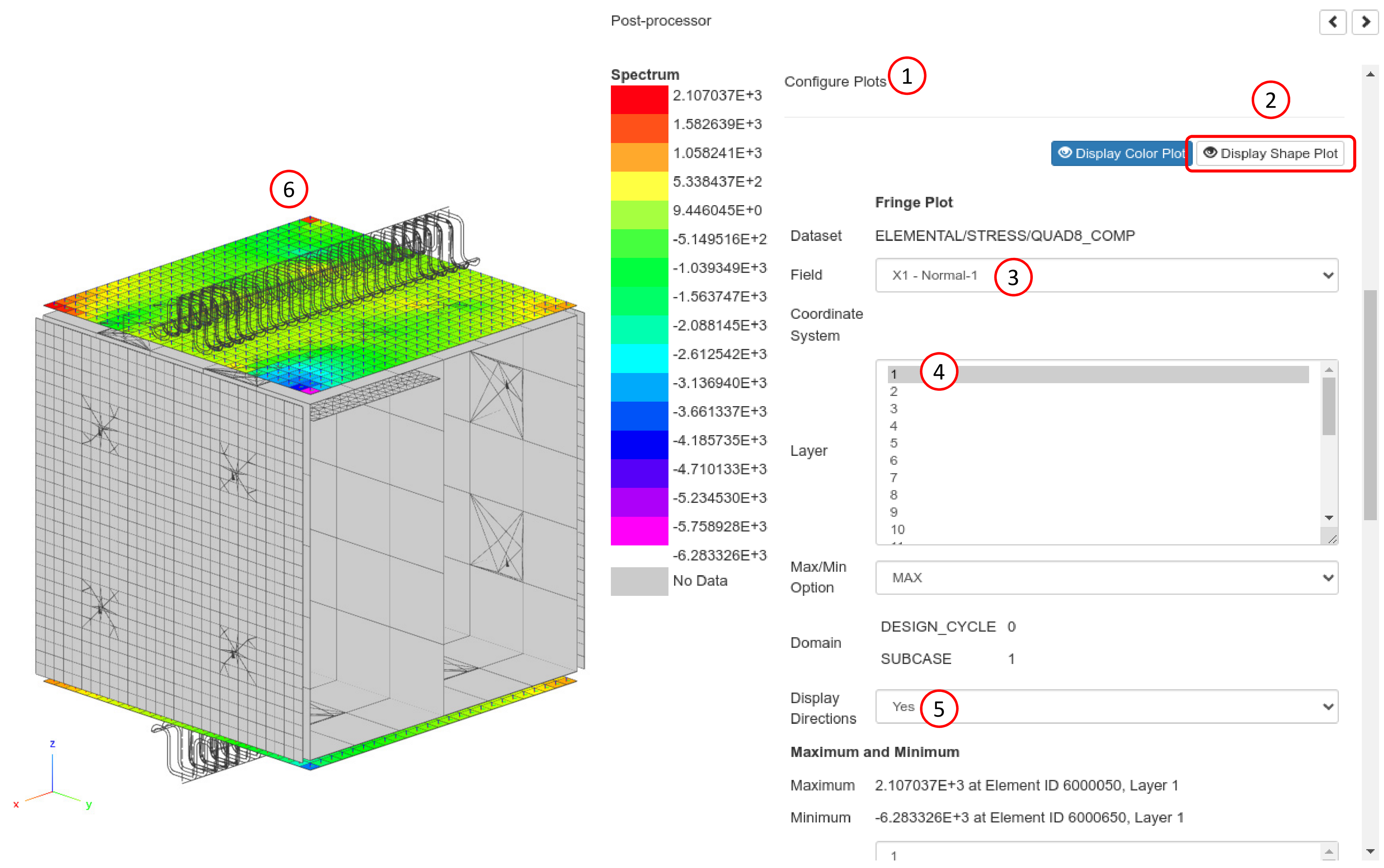
1. Use the vertical scroll bar to navigate to section Acquire Dataset
2. Set Select Dataset to ELEMENTAL/STRESS/QUAD8_COMP
3. Mark the checkbox for SUBCASE 1 for DESIGN_CYCLE 0
4. Wait until the status says Acquisition complete and successful



Nastran Composite Stress Results - CQUAD8 and PCOMP

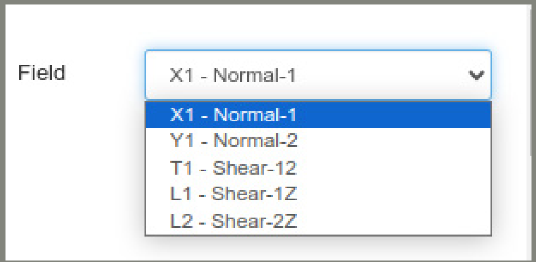
1. Use the vertical scroll bar to navigate to section Configure Plots
2. Click Display Shape Plot to hide the deformation
3. Set Field to X1 - Normal 1
   - This option displays the layer stress in the 1 direction
4. Set Layer to 1
   - This select the layer stress of layer 1 of the composite laminate
5. Set Display Direction to Yes
6. The fringe plot is displayed

- The Display Directions option is used to display the 1-2 coordinate system of each layer.

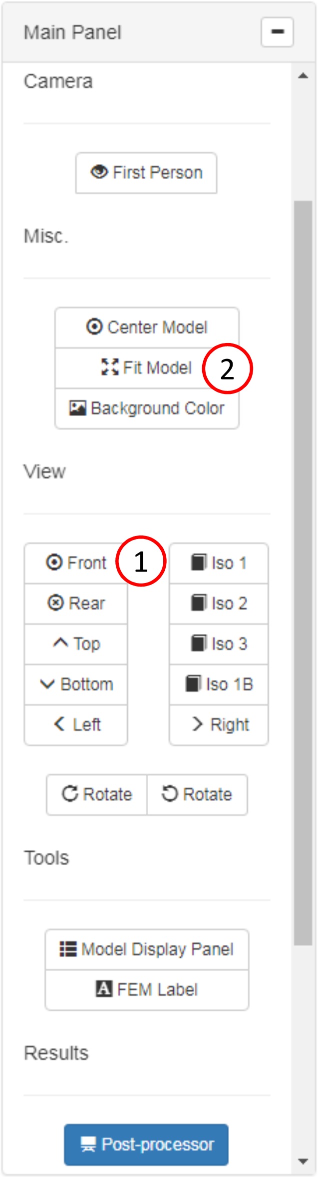


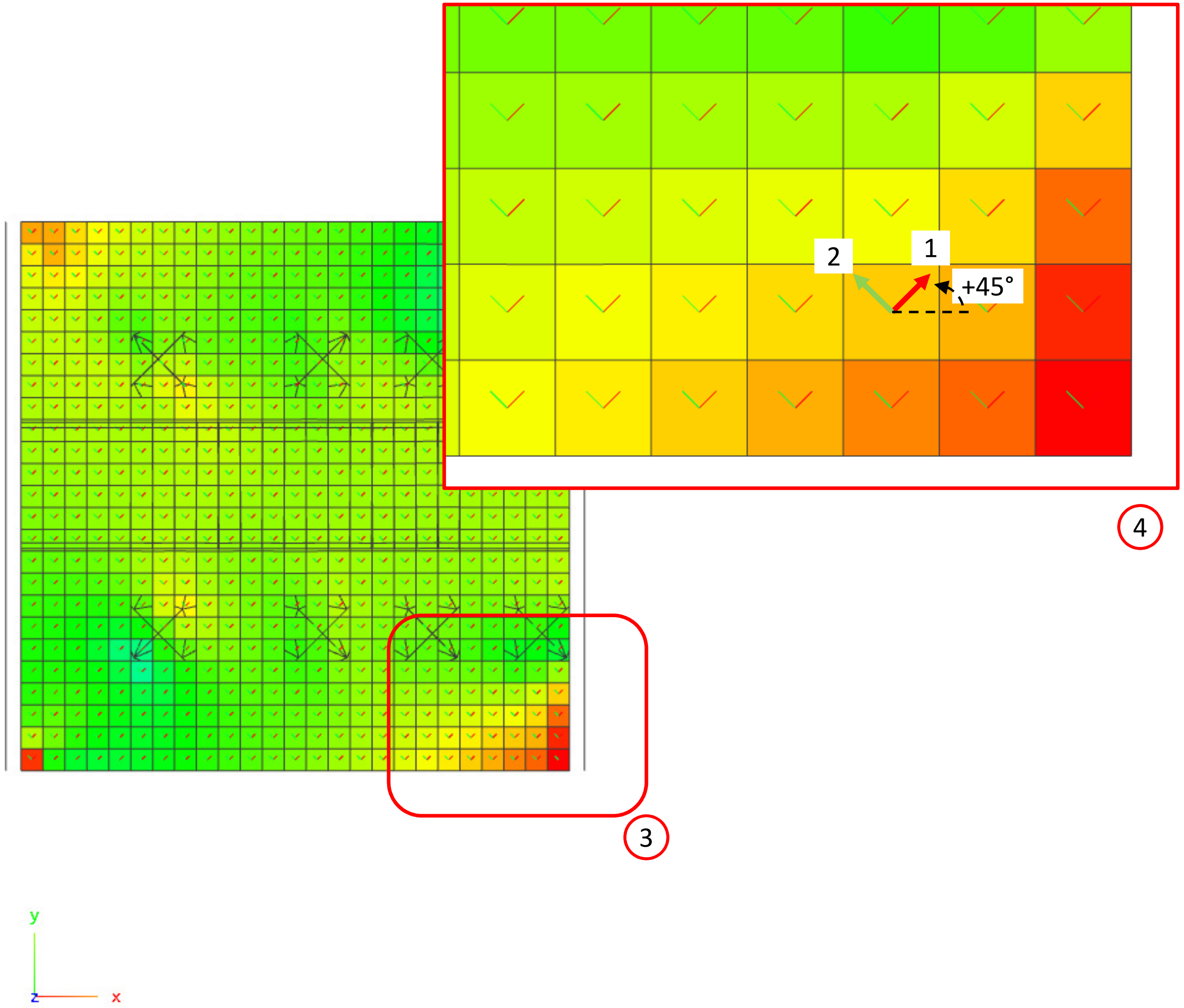
Nastran Composite Stress Results - CQUAD8 and PCOMP

1. Click Front
2. Click Fit Model
3. Zoom in to the bottom right corner
4. The Display Directions option displays the 1 and 2 direction of the composite layer
    - The fields available include the layer stress in the 1 and 2 directions (X1, Y1) and shear stresses (T1, L1, L2)



Field	Component
X1 – Normal-1	σ_1
Y1 – Normal-2	σ_2
T1 – Shear-12	τ_{12}



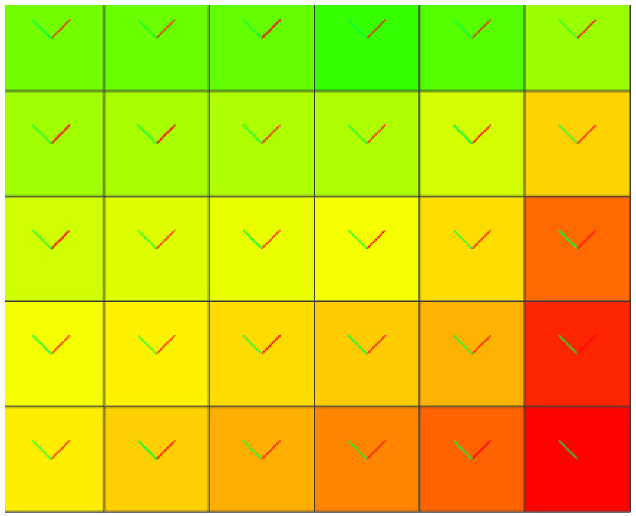


Nastran Composite Stress Results - CQUAD8 and PCOMP

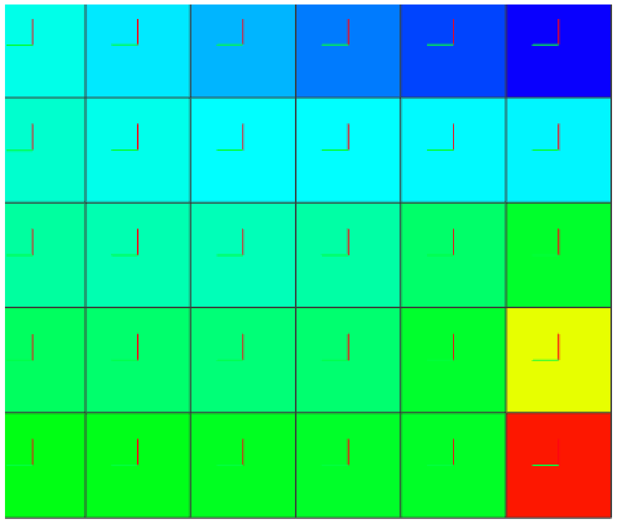
- The X1 (σ_1) layer stresses for layers 1, 2, 3 and 5 are displayed

- Layer stresses, strains and failure indices are calculated at the midplane of the layers.

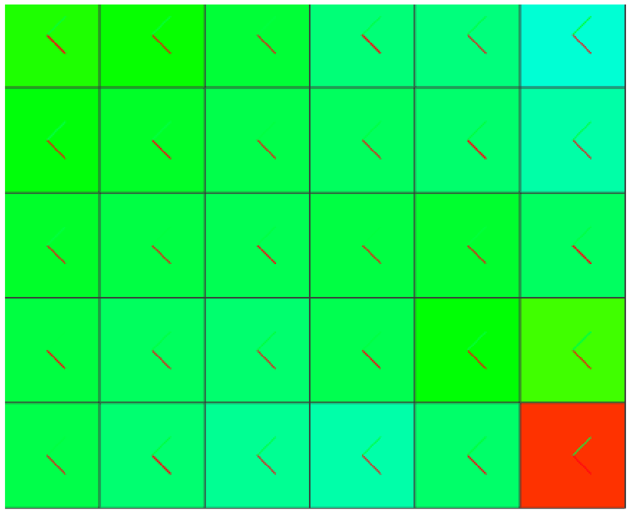
Layer 1 (45°)



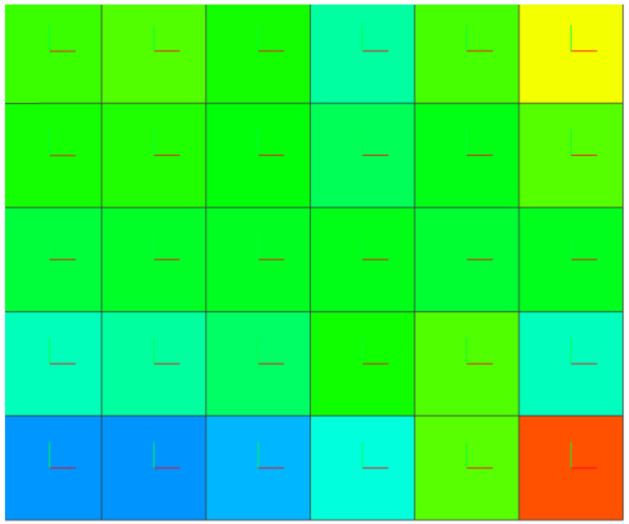
Layer 2 (90°)



Layer 3 (-45°)



Layer 5 (0°)

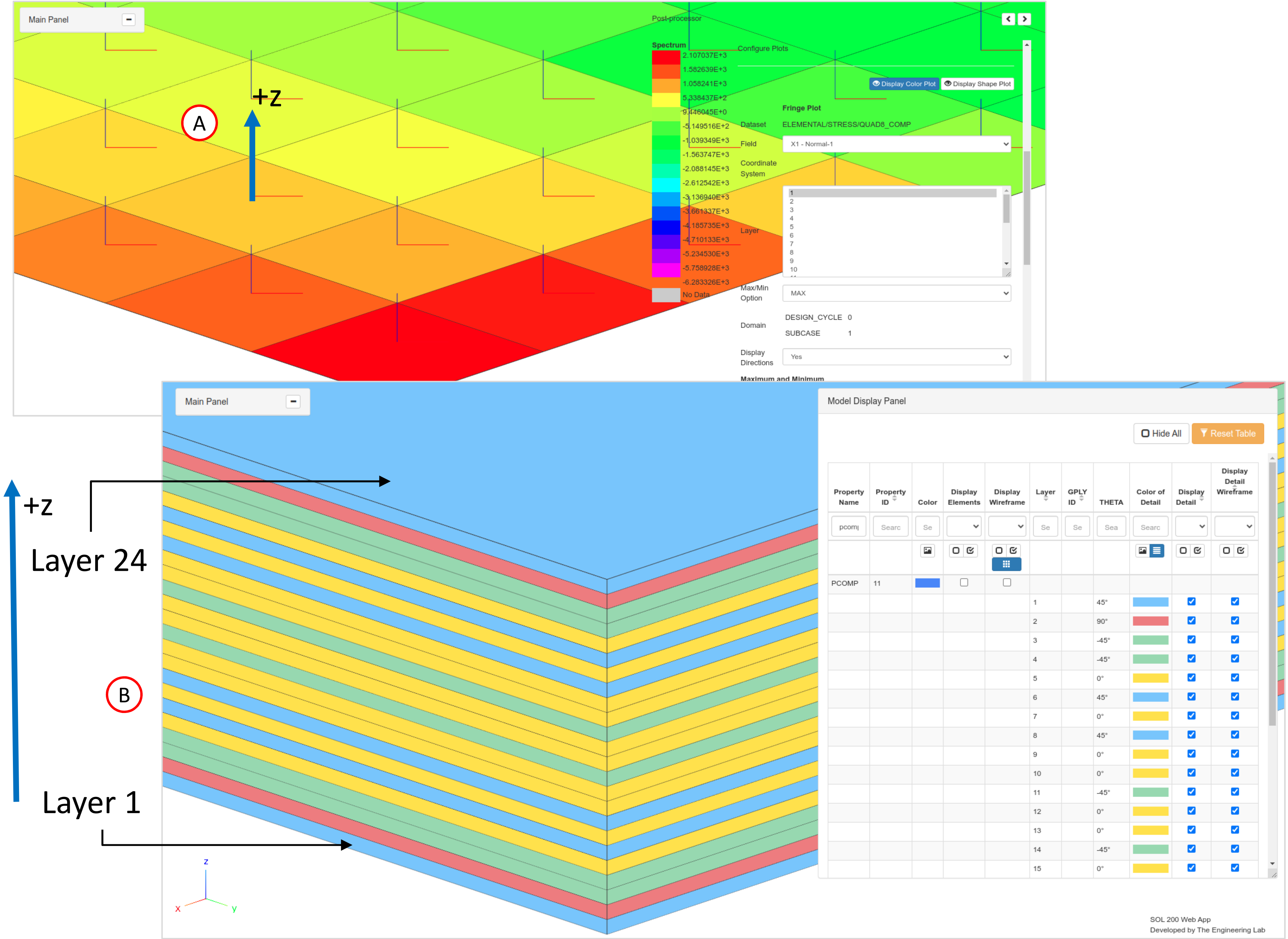


Nastran Composite Stress Results - CQUAD8 and PCOMP

A. The Display Directions option is useful for determining the order of the layers. Layers are numbered in ascending order along the +z direction.

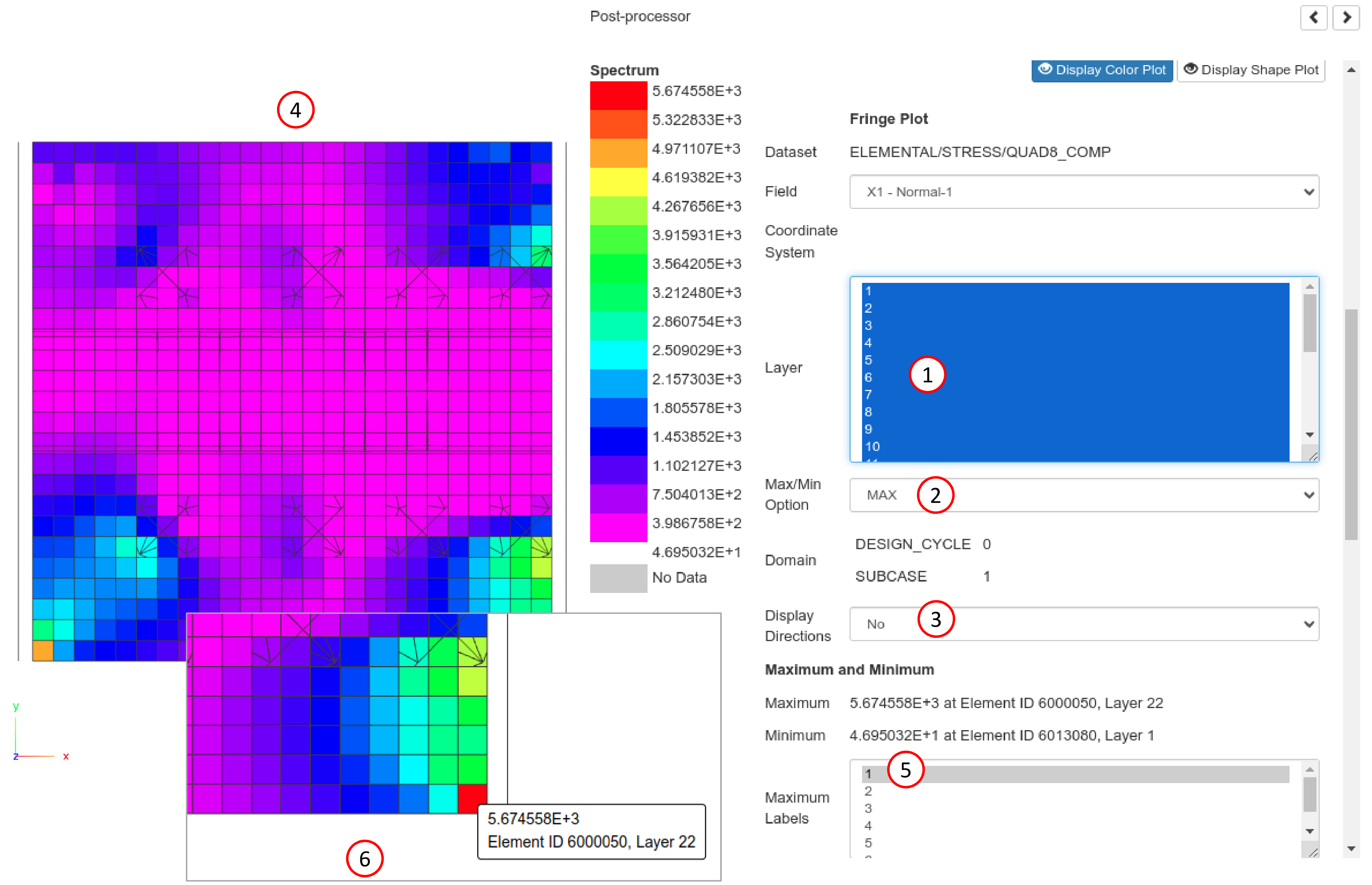
B. For comparison, the actual layer thicknesses are displayed via the Model Display Panel.

- Layer stresses, strains and failure indices are calculated at the midplane of the layers.



Nastran Composite Stress Results - CQUAD8 and PCOMP

1. For Layer, select all layers. This can be done with one of the following ways.
   - Select multiple rows with the mouse.
   - Or select one layer then press CTRL+A on the keyboard.
2. Set Max/Min Option to MAX
3. Set Display Directions to No
4. The fringe plot is updated to show the maximum X1 layer stress for all layers
5. Select the first maximum label
6. A label is displayed indicating the maximum response and corresponding element ID and layer number

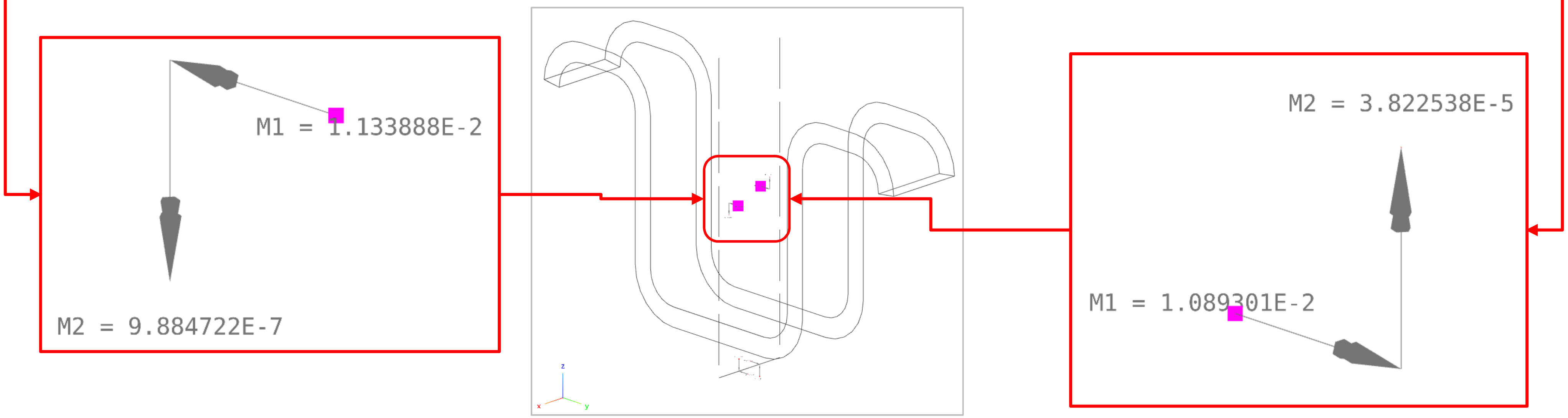


Nastran Beam Force and Stress Results - CBEAM and PBMSECT

Goal: Display CBEAM element forces

F06 File

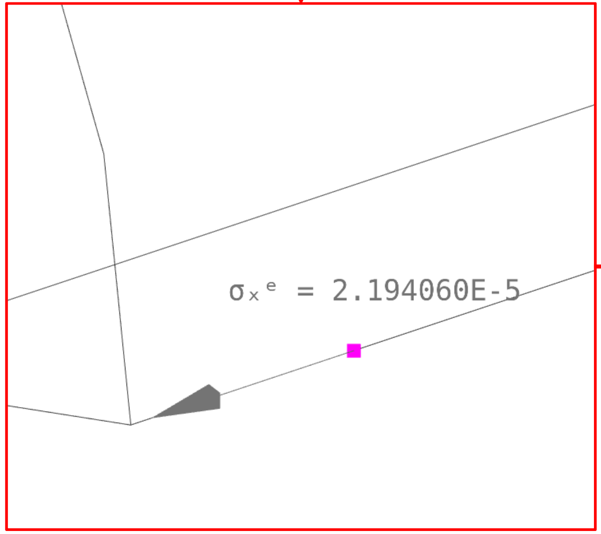
```
0                                                                                                         SUBCASE 1

                                   F O R C E S   I N   B E A M   E L E M E N T S        ( C B E A M )
                    STAT DIST/   - BENDING MOMENTS -            - WEB  SHEARS -           AXIAL          TOTAL          WARPING
   ELEMENT-ID  GRID   LENGTH    PLANE 1       PLANE 2        PLANE 1       PLANE 2        FORCE          TORQUE         TORQUE
0   6013710
             4014176   0.000   -1.133888E-02  9.884721E-07  -1.114667E-04 -9.309227E-06   4.047441E-03  -2.465291E-05   0.0
             4014547   1.000   -1.089301E-02  3.822538E-05   1.114667E-04  9.309227E-06   4.047441E-03   2.465291E-05   0.0
```

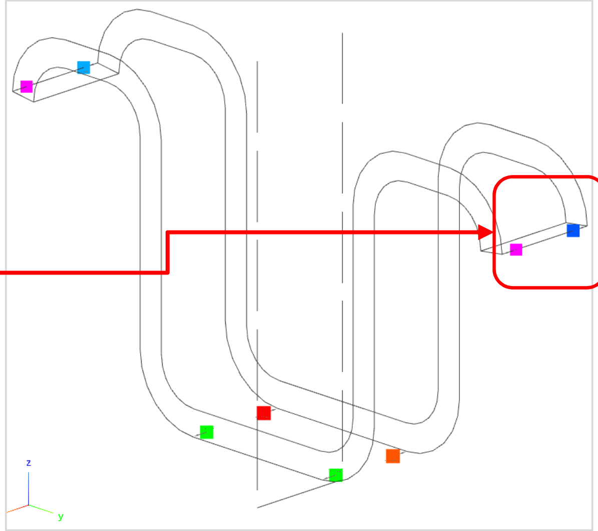


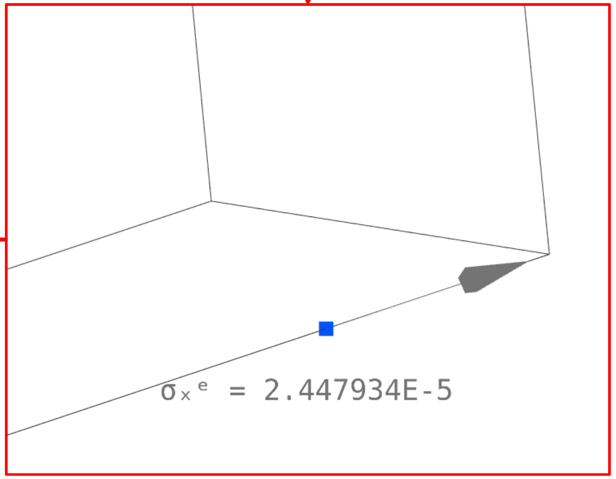
Goal: Display CBEAM element stresses

F06 File

```
0                                                                                                                 SUBCASE 1

                                   S T R E S S E S   I N   B E A M   E L E M E N T S        ( C B E A M )
                    STAT DIST/
   ELEMENT-ID  GRID   LENGTH    SXC           SXD           SXE           SXF           S-MAX         S-MIN         M.S.-T   M.S.-C
0   6013710
            4014176   0.000    2.709380E-05  2.195210E-05  2.194060E-05  2.708385E-05  2.709380E-05  2.194060E-05
            4014547   1.000    2.990706E-05  2.495574E-05  2.447934E-05  2.940855E-05  2.990706E-05  2.447934E-05
```

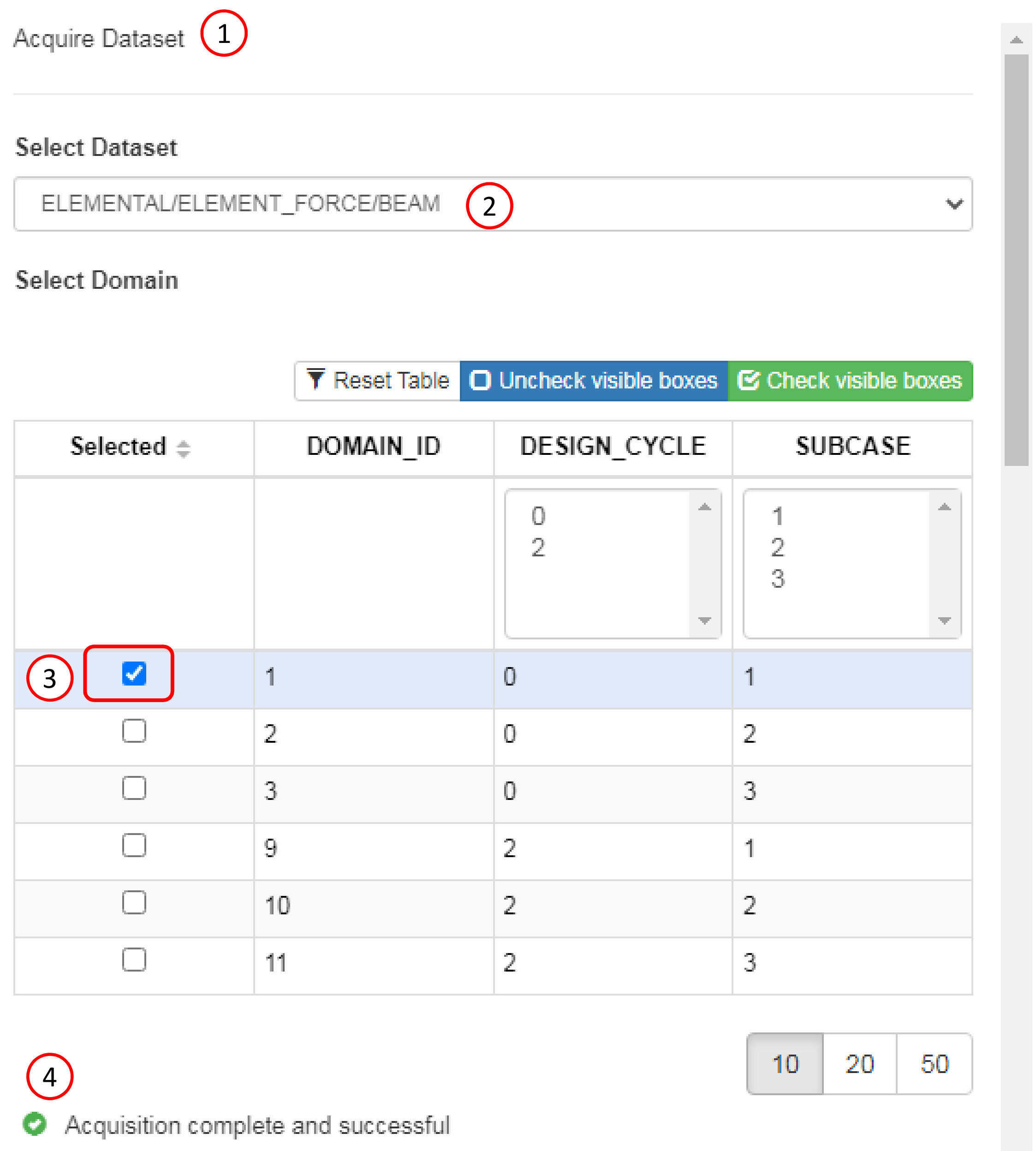






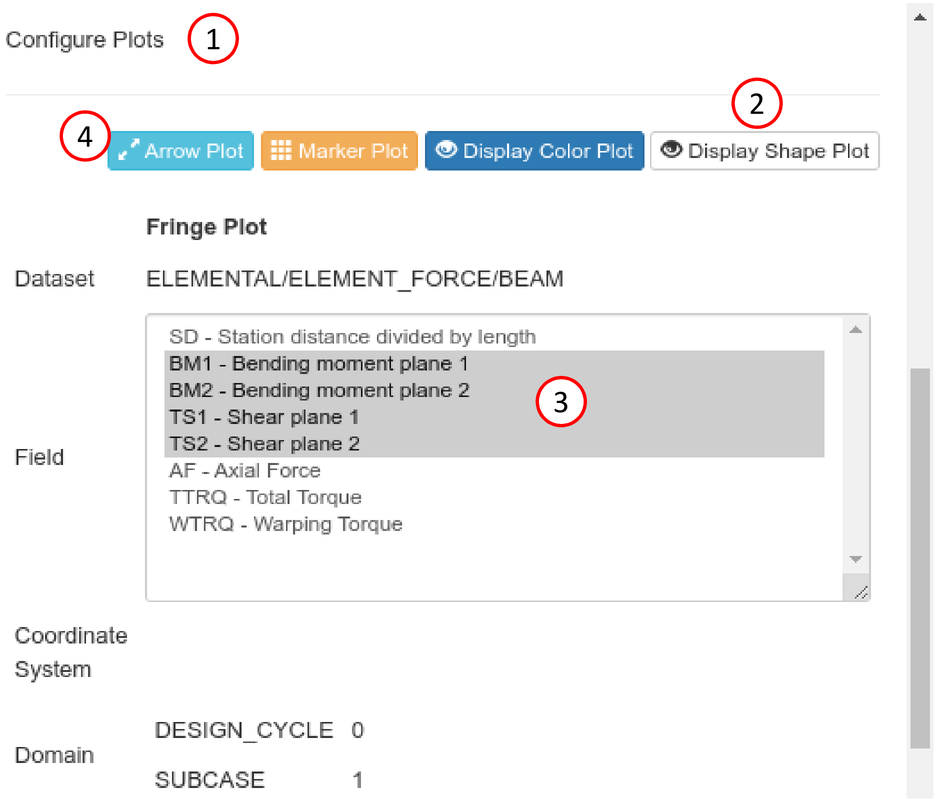
Nastran Beam Force and Stress Results - CBEAM and PBMSECT

1. Use the vertical scroll bar to navigate to section Acquire Dataset
2. Set Select Dataset to ELEMENTAL/ELEMENT_FORCE/BEAM
3. Mark the checkbox for SUBCASE 1 for DESIGN_CYCLE 0
4. Wait until the status says Acquisition complete and successful



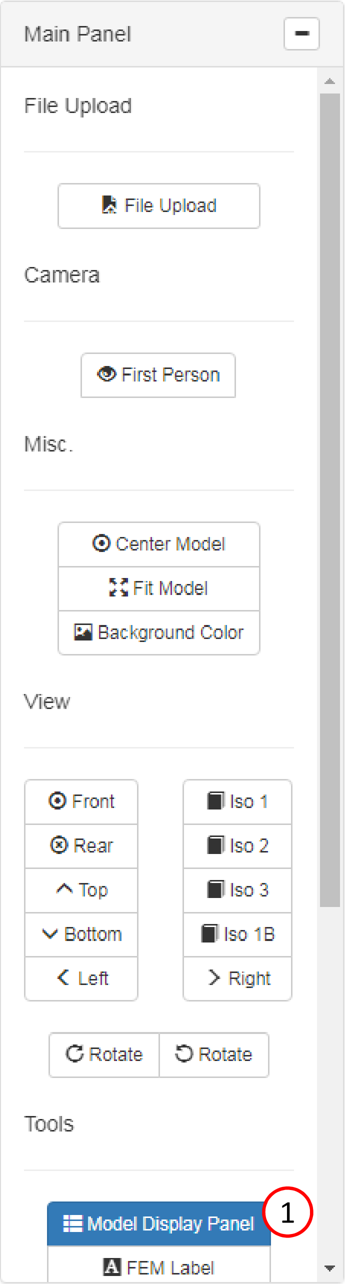
Nastran Beam Force and Stress Results - CBEAM and PBMSECT

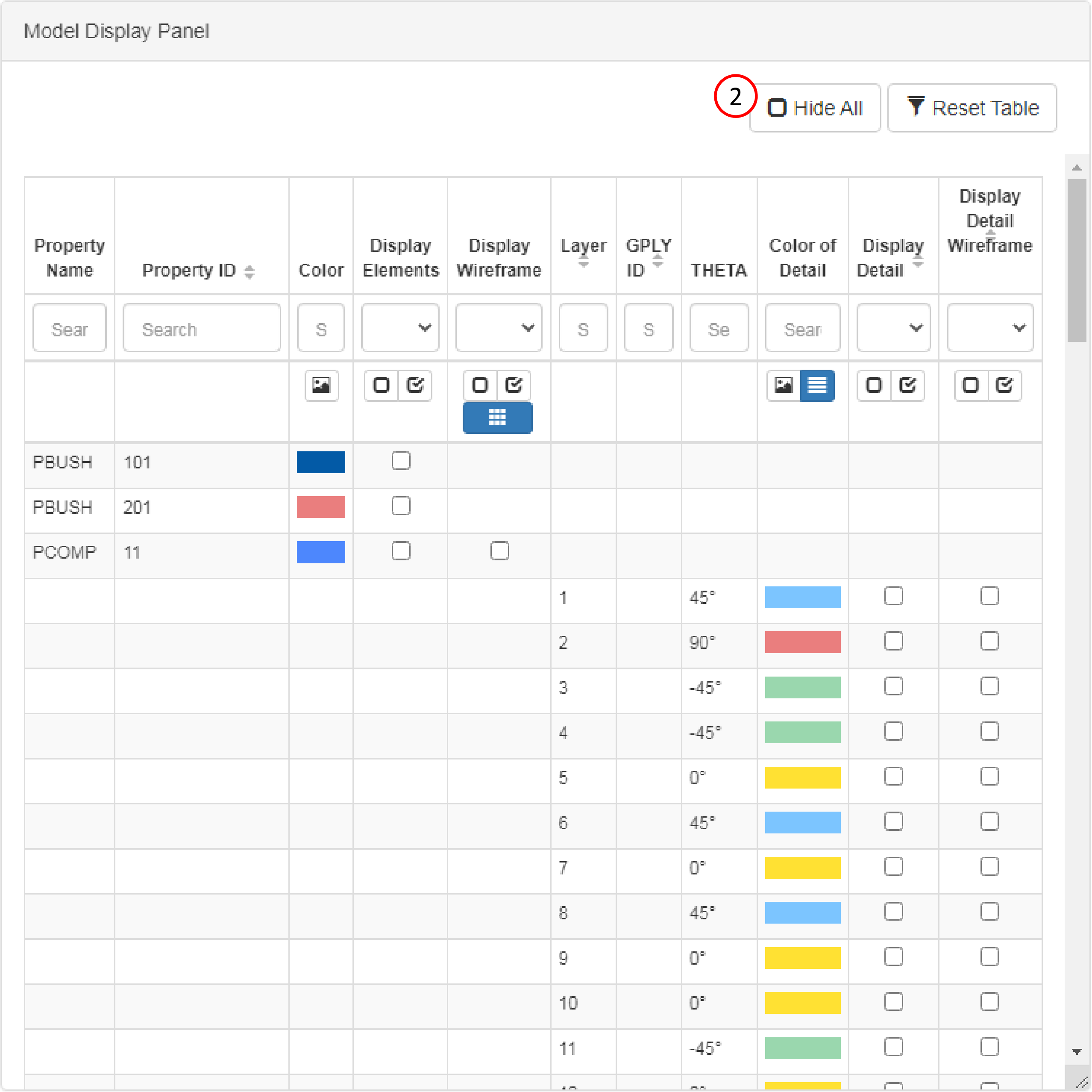
1. Use the vertical scroll bar to navigate to section Configure Plots
2. Click Display Shape Plot to hide the deformation
3. Set the following fields:
    - BM1
    - BM2
    - TS1
    - TS2
4. Click Arrow Plot



Nastran Beam Force and Stress Results - CBEAM and PBMSECT

1. Click Model Display Panel
2. Click Hide All

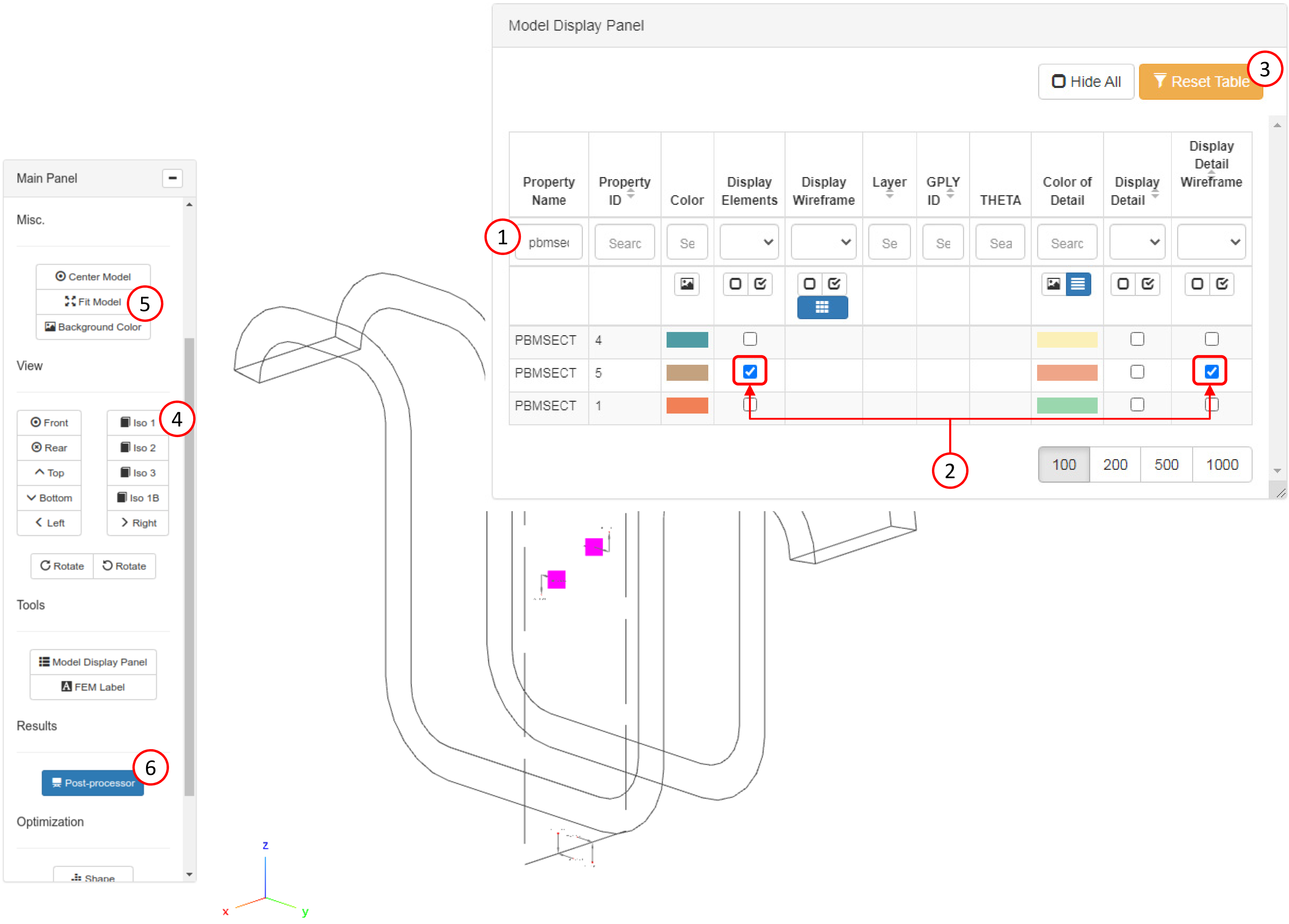




Nastran Beam Force and Stress Results - CBEAM and PBMSECT

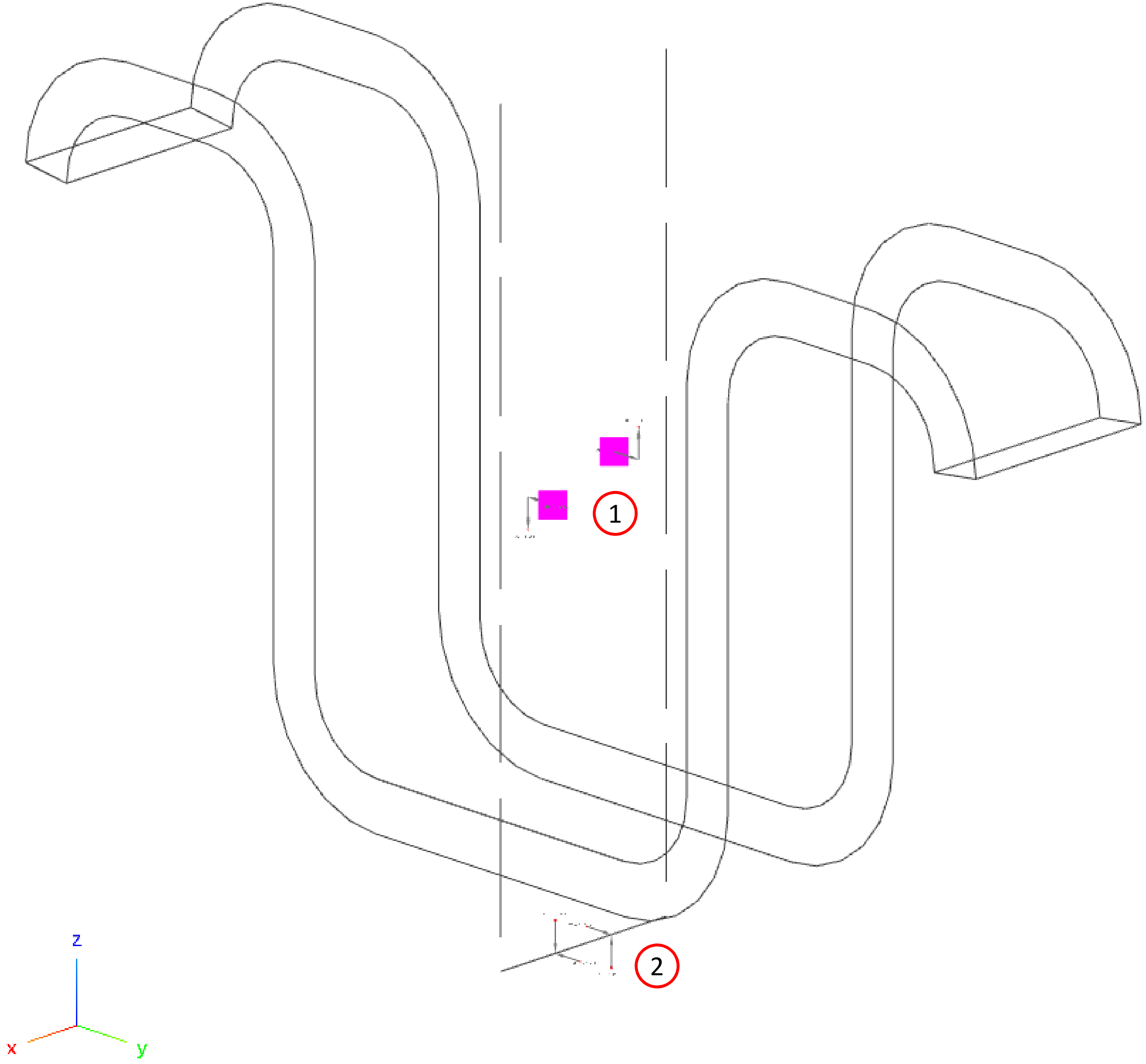
1. Use the search bar to search for: pbmsect
2. Mark the indicated checkboxes for PBMSECT 5
   - This will display the line element and the arbitrary beam cross section defined with CBEAM and PBMSECT entries
3. Click Reset Table
   - This will reset the table search
4. Click Iso 1 (Isometric 1 View)
5. Click Fit Model
6. Click Post-processor

The PBMSECT 5 entry is used by only one CBEAM element, so only one element is displayed.



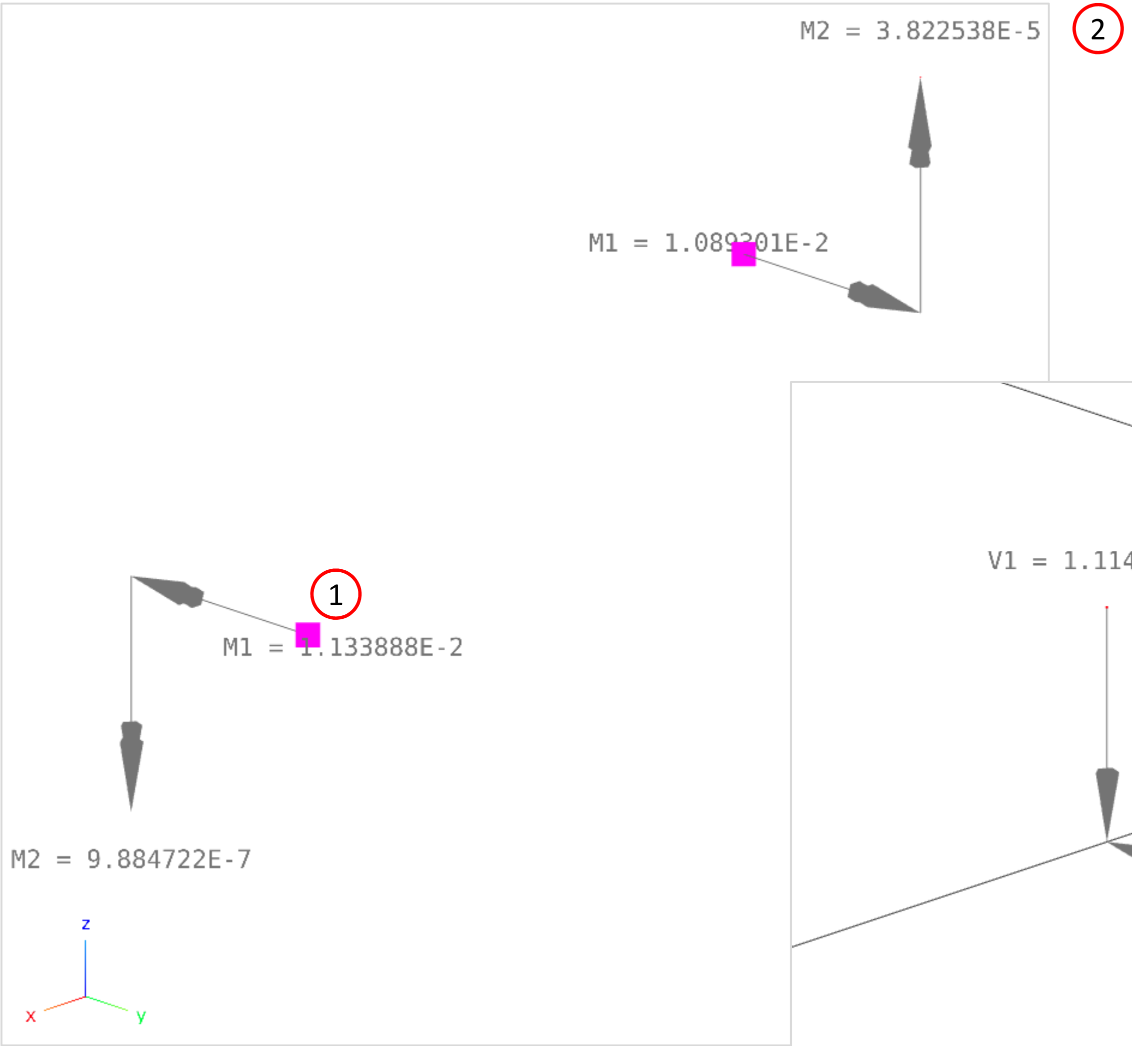
Nastran Beam Force and Stress Results - CBEAM and PBMSECT

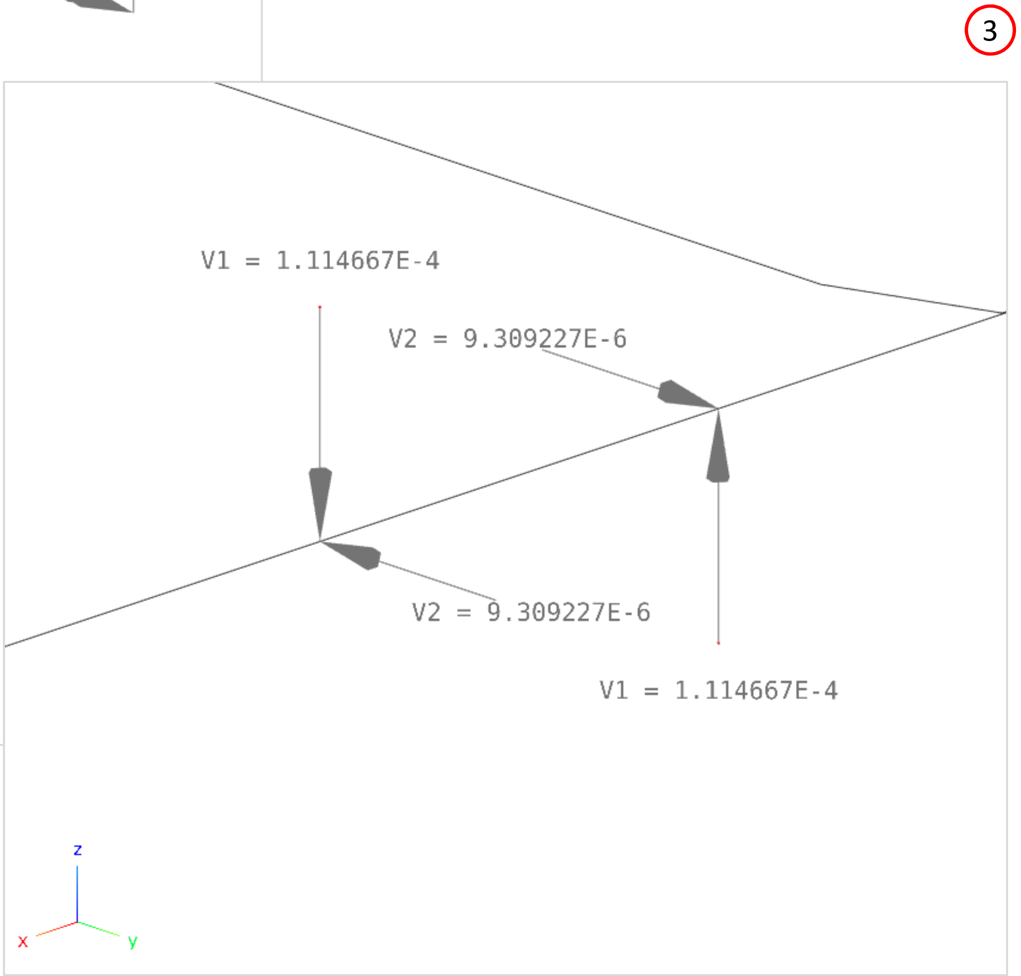
1. The moments are displayed at the neutral axis of the beam cross section.
2. The shear forces are displayed at the shear center of the beam cross section.



Nastran Beam Force and Stress Results - CBEAM and PBMSECT

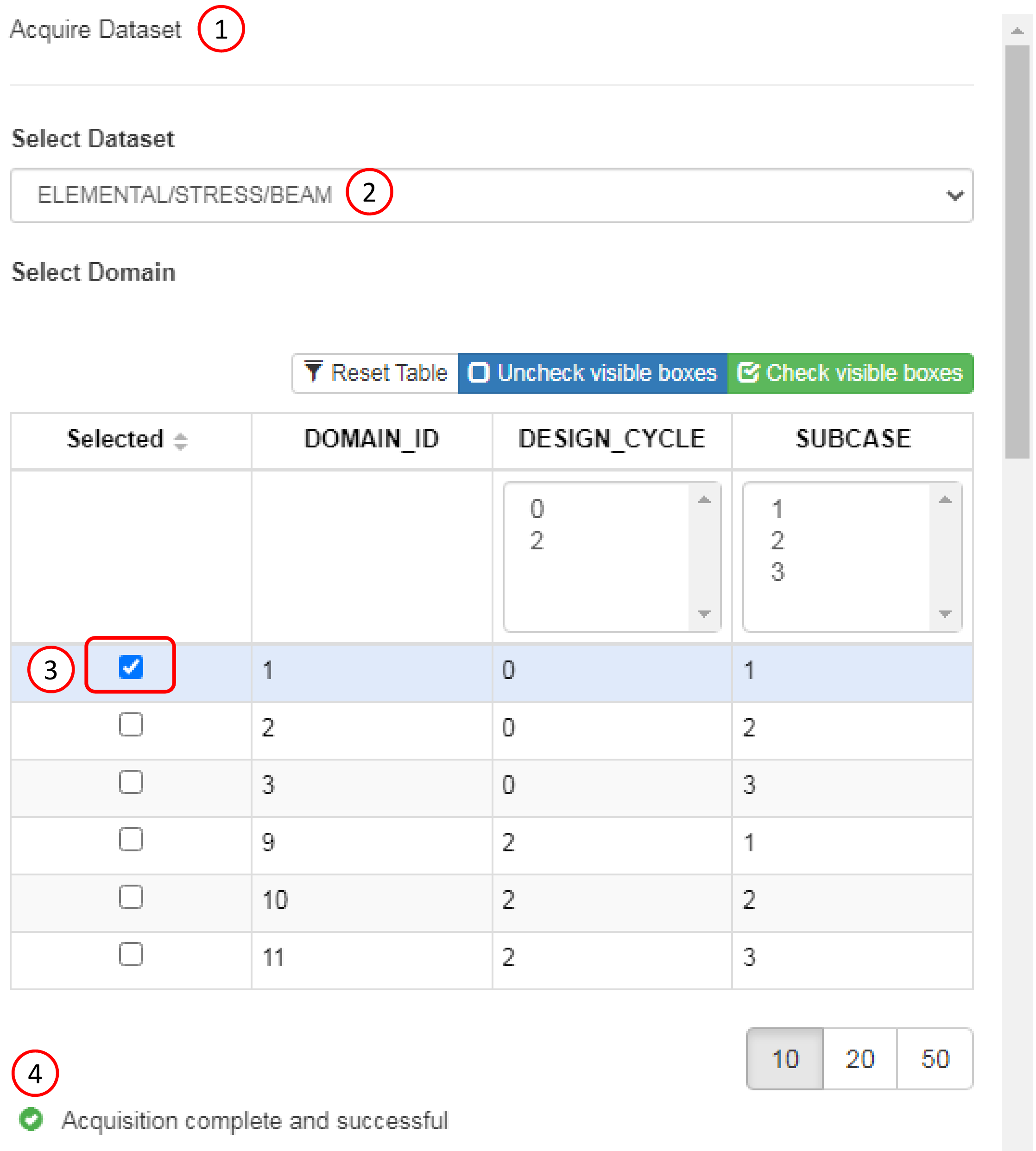
1. Zoom in very closely to the arrows.
   - The arrows and labels are purposely very small to avoid interference with the arrows and labels of adjacent elements.
2. The moments are displayed at the neutral axis of the beam cross section.
3. The shear forces are displayed at the shear center of the beam cross section.





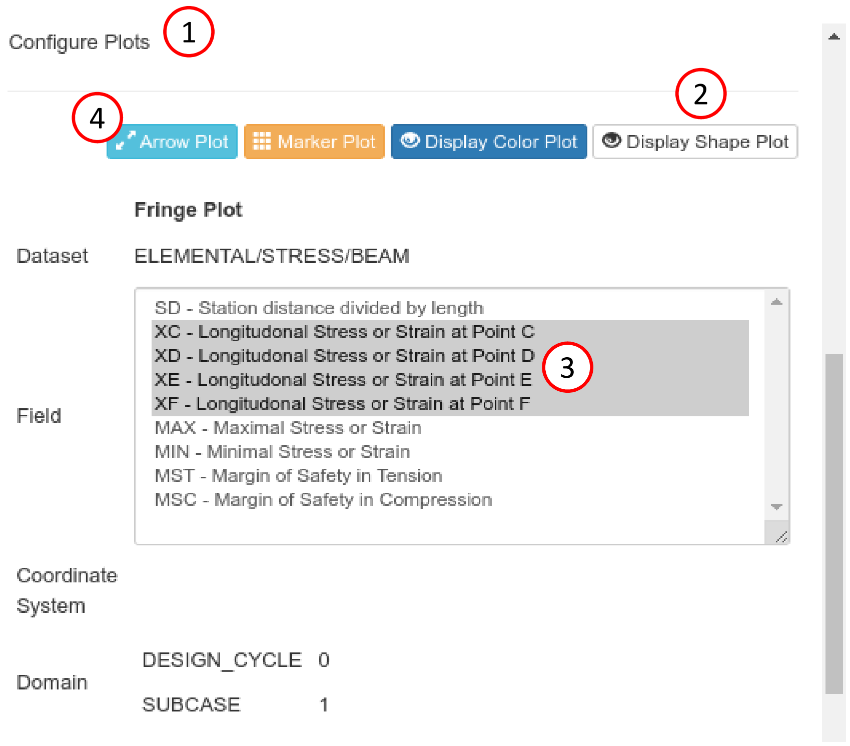
Nastran Beam Force and Stress Results - CBEAM and PBMSECT

1. Use the vertical scroll bar to navigate to section Acquire Dataset
2. Set Select Dataset to ELEMENTAL/STRESS/BEAM
3. Mark the checkbox for SUBCASE 1 for DESIGN_CYCLE 0
4. Wait until the status says Acquisition complete and successful



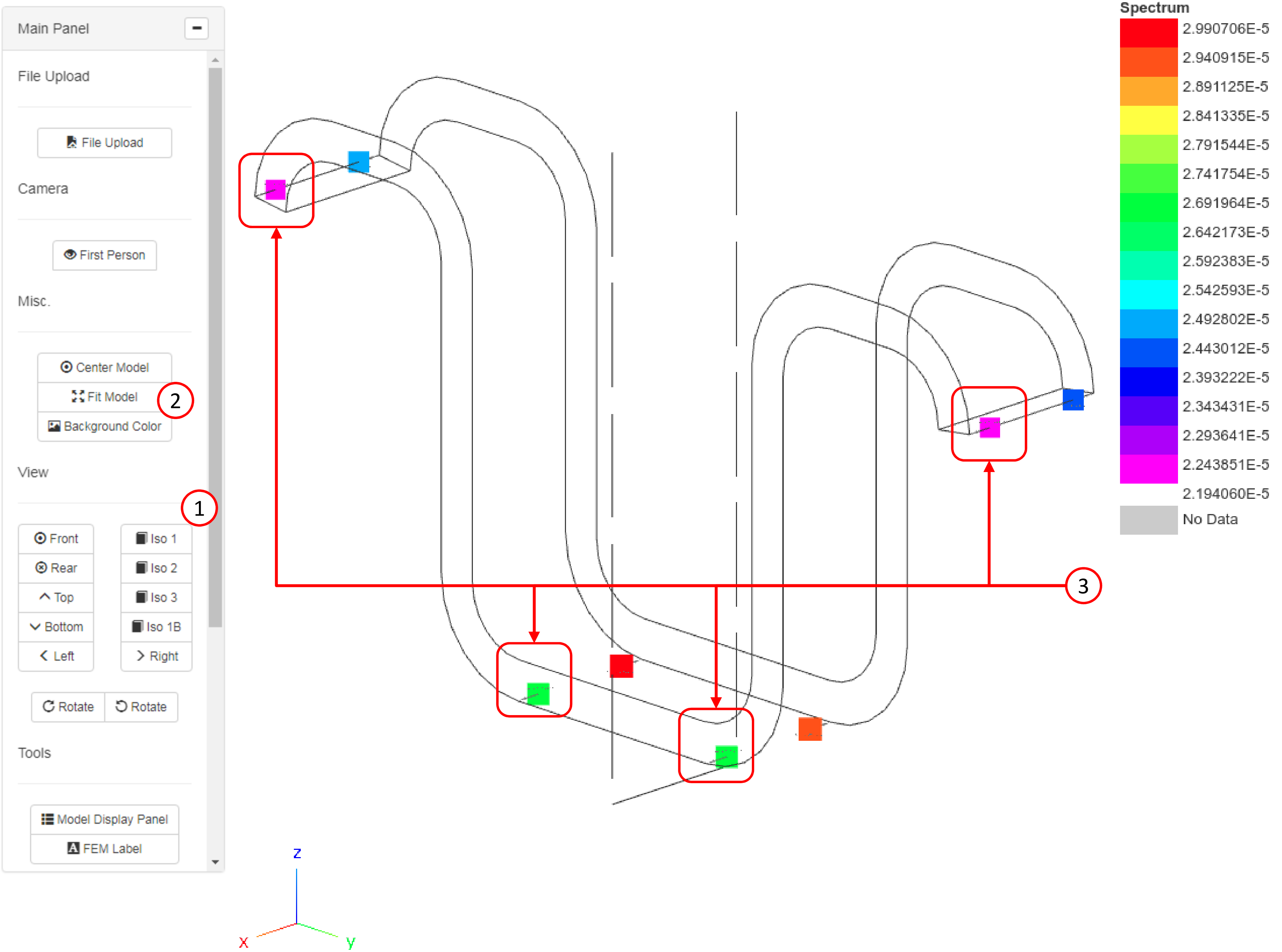
Nastran Beam Force and Stress Results - CBEAM and PBMSECT

1. Use the vertical scroll bar to navigate to section Configure Plots
2. Click Display Shape Plot to hide the deformation
3. Set the following fields:
   - XC
   - XD
   - XE
   - XF
4. Click Arrow Plot



Nastran Beam Force and Stress Results - CBEAM and PBMSECT

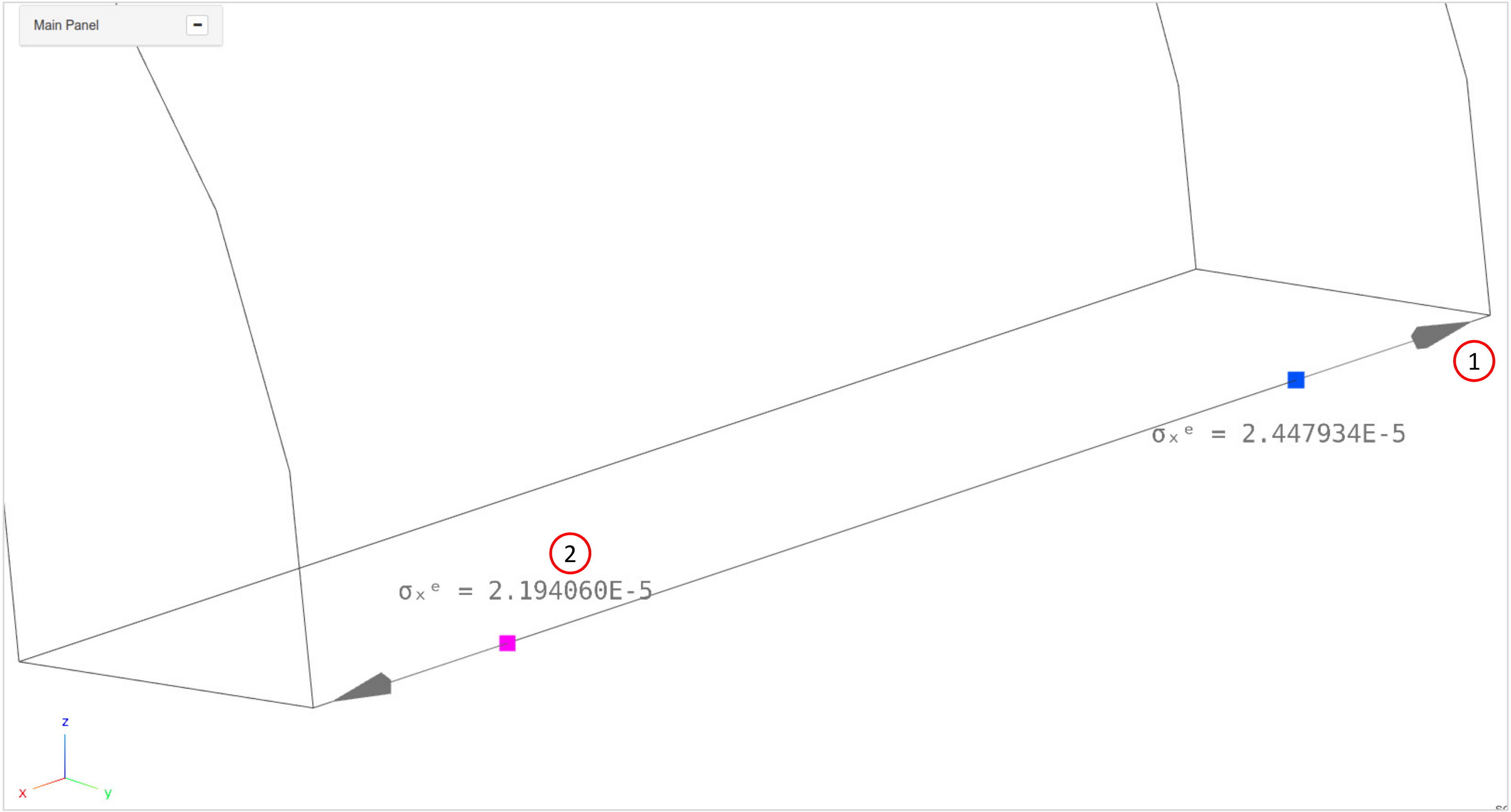
1. Click Iso 1 (Isometric 1 View)
2. Zoom in to the indicated region
3. Beam stresses are output and displayed at the points C, D, E and F



Nastran Beam Force and Stress Results - CBEAM and PBMSECT

1. Zoom in very closely to the arrows.
   - The arrows and labels are purposely very small to avoid interference with the arrows and labels of adjacent elements.

2. The direction of the arrows indicate the stresses are in tension or compression.
   - The stresses are in tension if the arrows point away from each other.
   - The stresses are in compression if the arrows point towards each other.



Nastran CBUSH Force Results - CBUSH and PBUSH

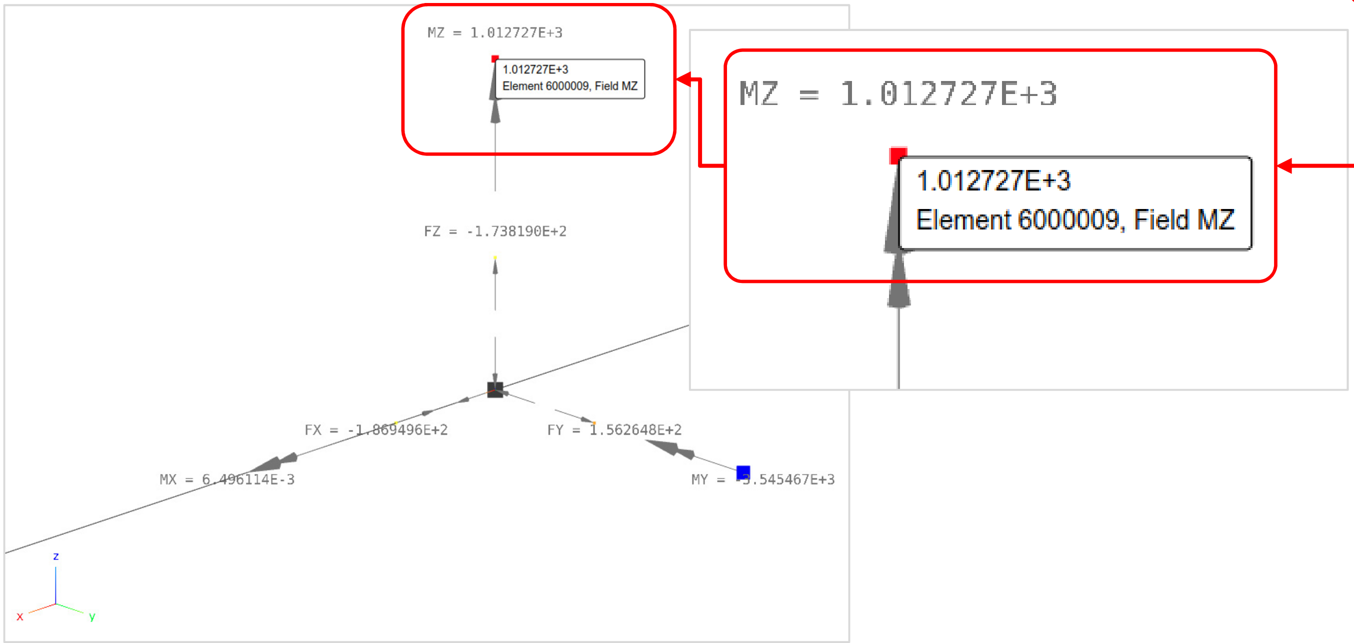
Goal: Display CBUSH element forces

F06 File

0 SUBCASE 1

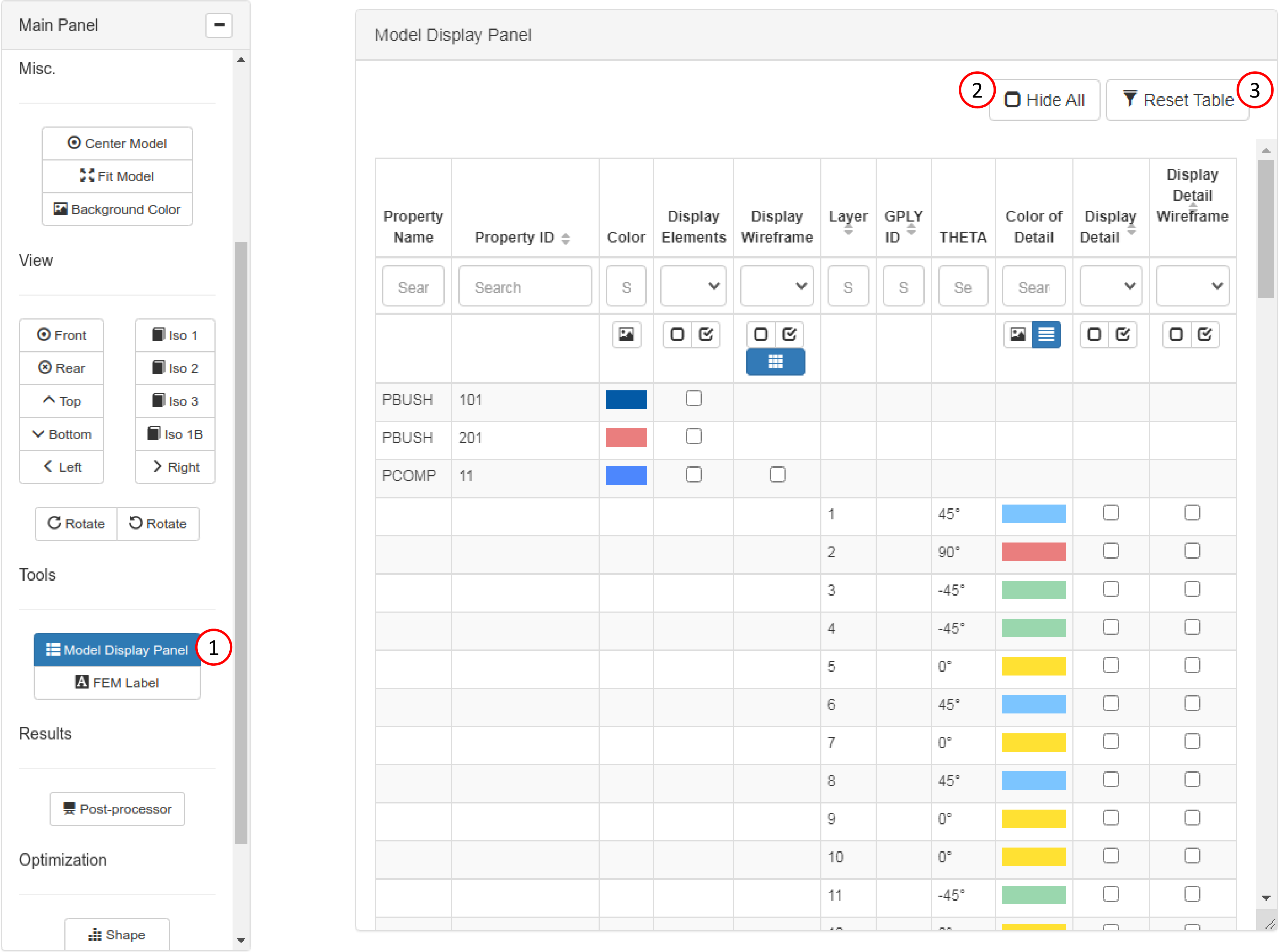
F O R C E S   I N   B U S H   E L E M E N T S   (C B U S H)

ELEMENT-ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
6000009	-1.869496E+02	1.562642E+02	-1.738190E+02	6.496129E-03	-3.545466E+03	1.012724E+03



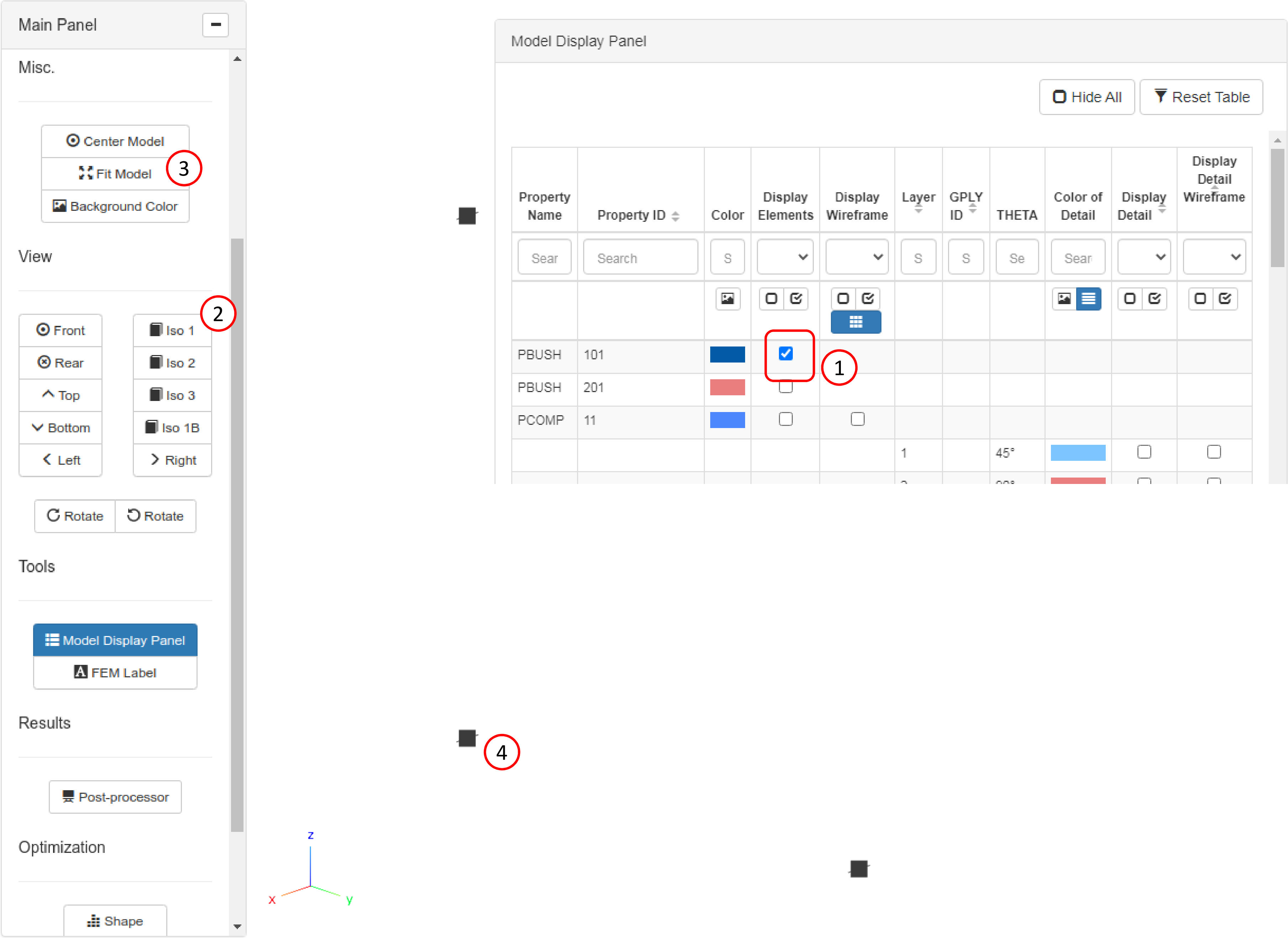
Nastran CBUSH Force Results - CBUSH and PBUSH

1. Click Model Display Panel
2. Click Hide All
3. Click Reset Table



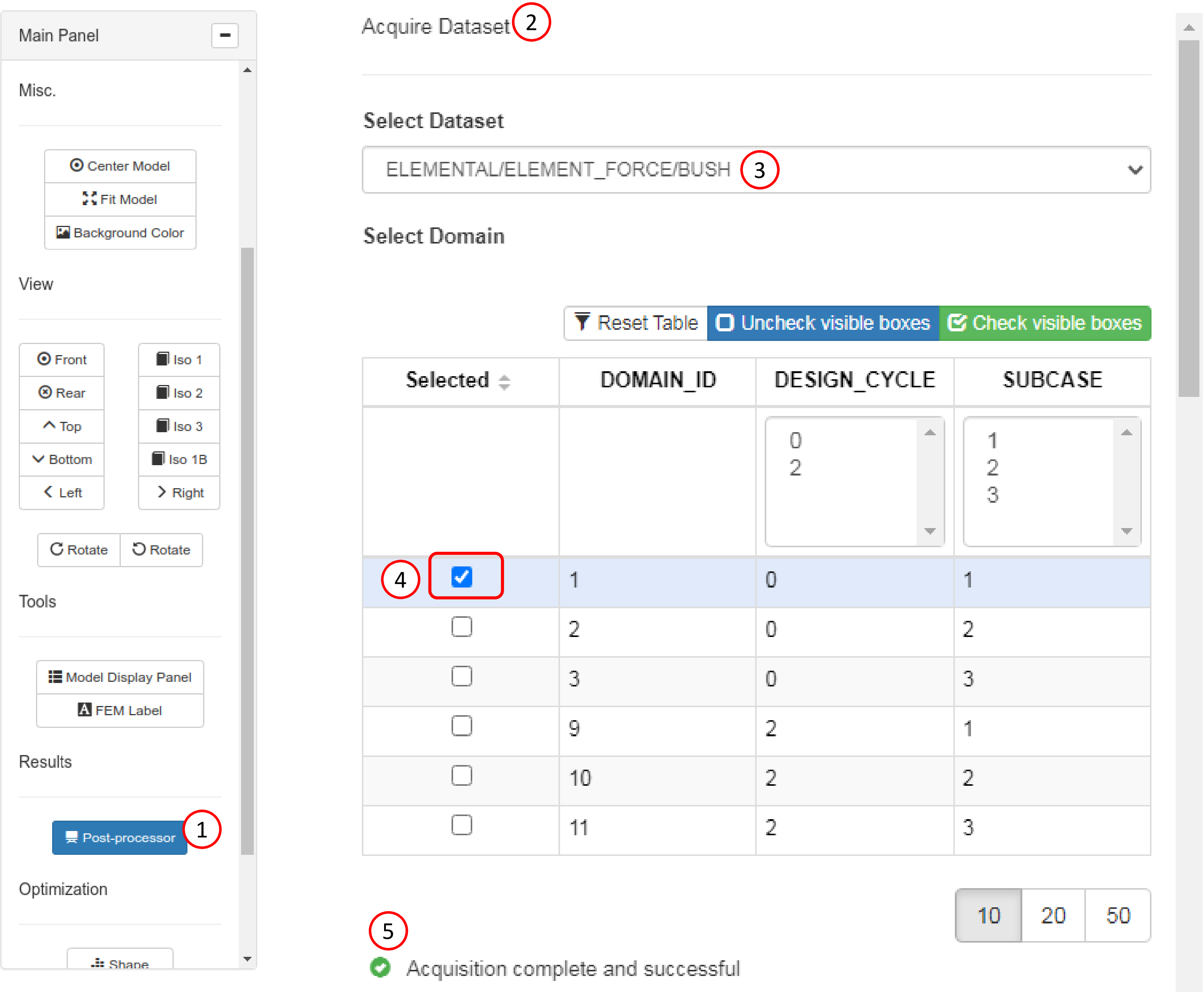
Nastran CBUSH Force Results - CBUSH and PBUSH

1. Mark the indicated checkboxes for PBUSH 101
2. Click Iso 1 (Isometric 1 View)
3. Click Fit Model
4. The elements configured via CBUSH and PBUSH entries are now visible



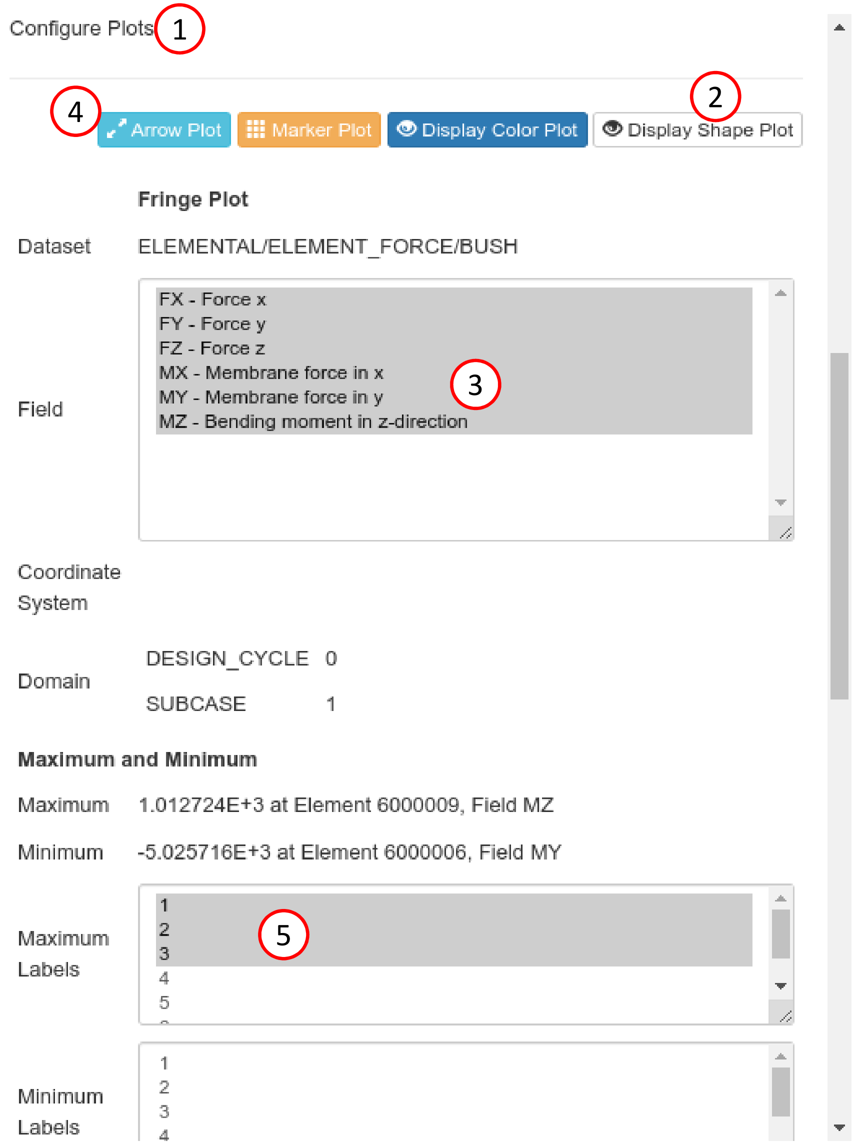
Nastran CBUSH Force Results - CBUSH and PBUSH

1. Click Post-processor
2. Use the vertical scroll bar to navigate to section Acquire Dataset
3. Set Select Dataset to ELEMENTAL/ELEMENT_FORCE/BUSH
4. Mark the checkbox for SUBCASE 1 for DESIGN_CYCLE 0
5. Wait until the status says Acquisition complete and successful



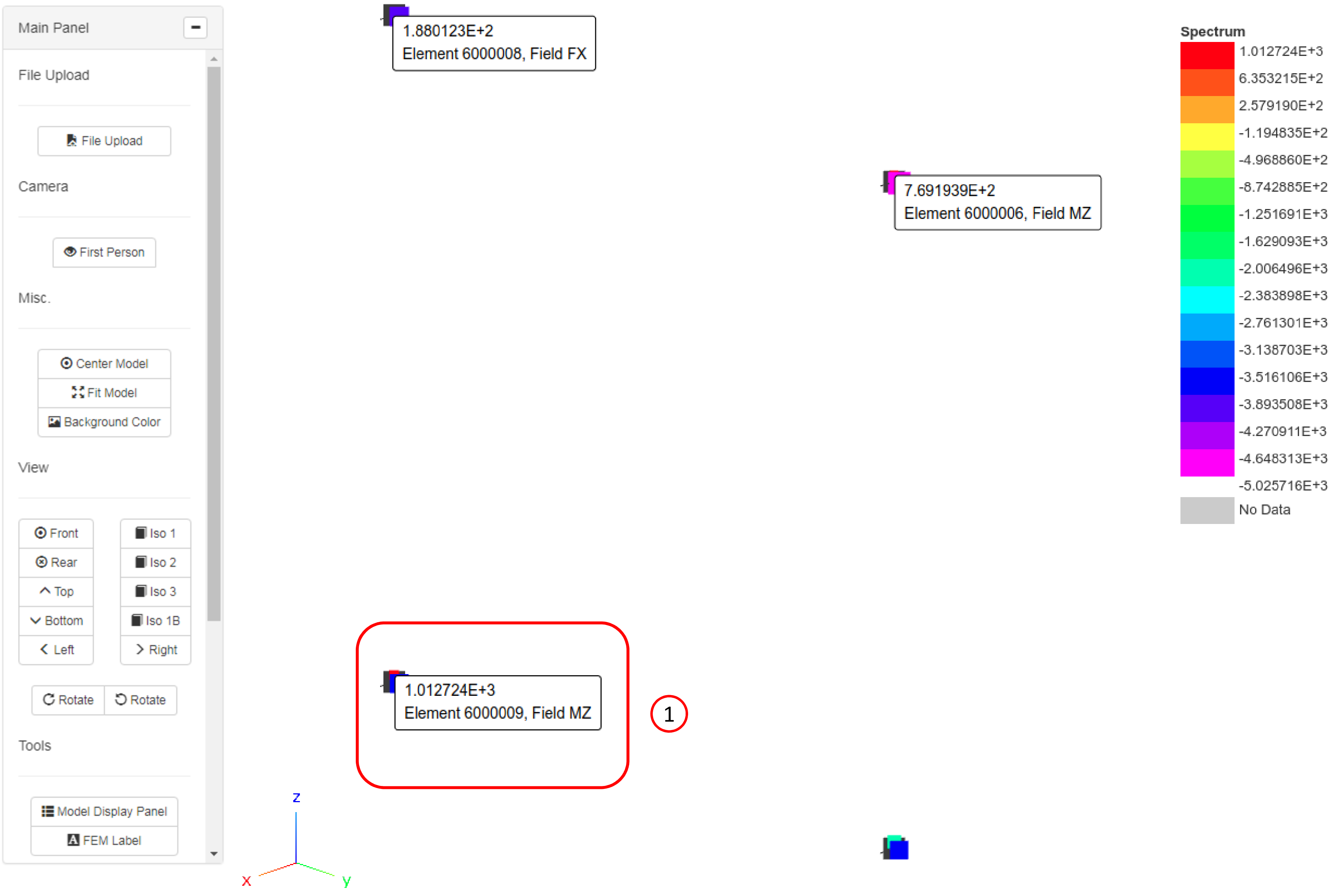
Nastran CBUSH Force Results - CBUSH and PBUSH

1. Use the vertical scroll bar to navigate to section Configure Plots
2. Click Display Shape Plot to hide the deformation
3. Set the following fields:
    - FX
    - FY
    - FZ
    - MX
    - MY
    - MZ
4. Click Arrow Plot
5. Select the first 3 maximum labels



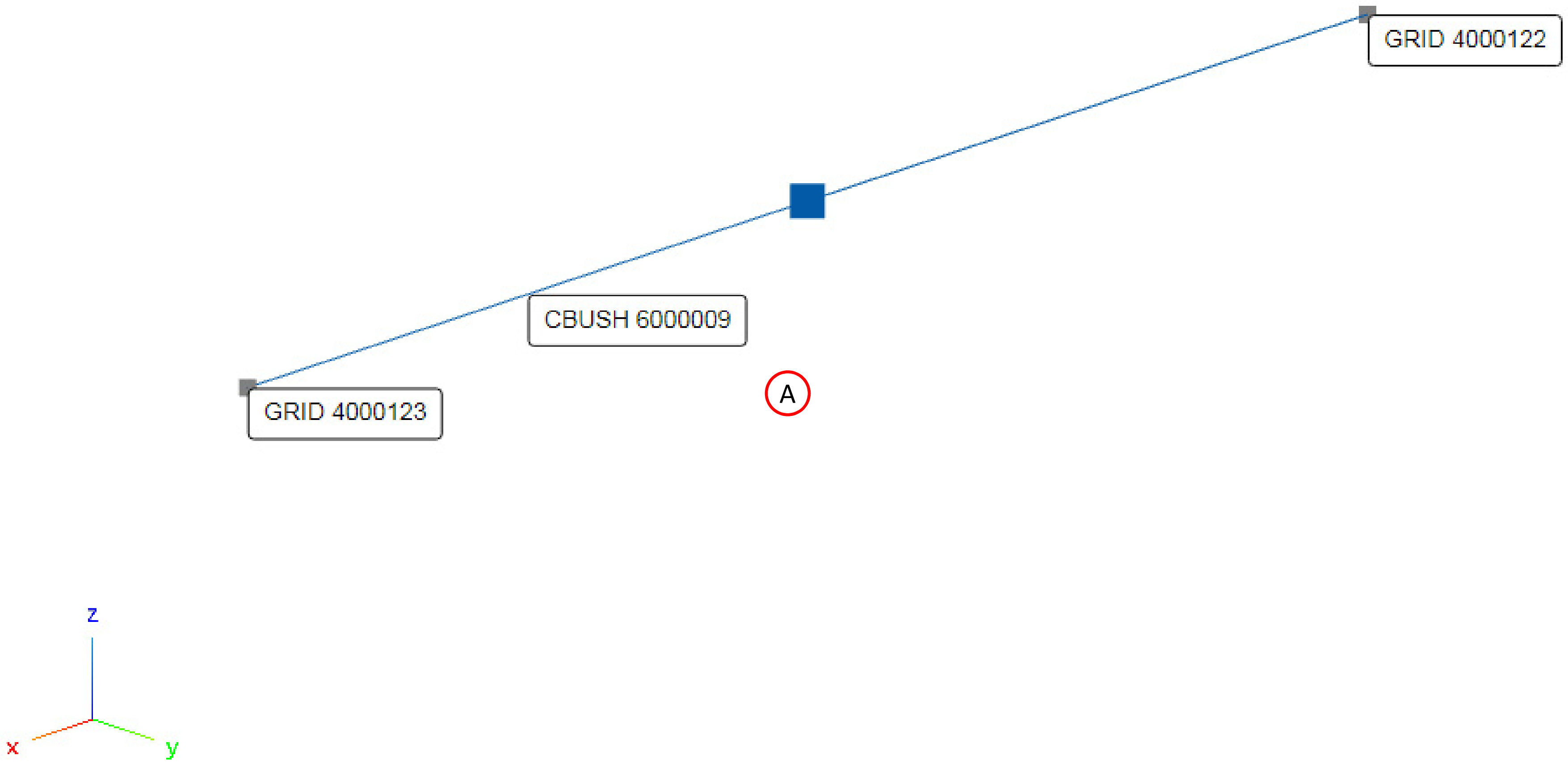
Nastran CBUSH Force Results - CBUSH and PBUSH

1. Zoom in to the indicated region



Nastran CBUSH Force Results - CBUSH and PBUSH

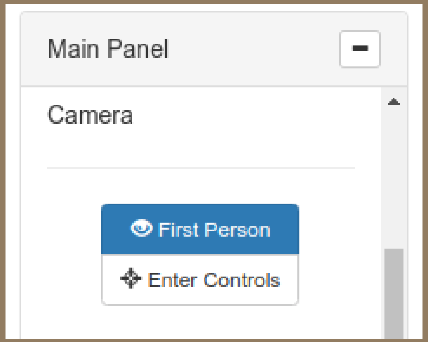
A. As you zoom in, note that this CBUSH element is used to connect two grids (nodes). The FEM Label Tool was used to display the element and grid IDs.

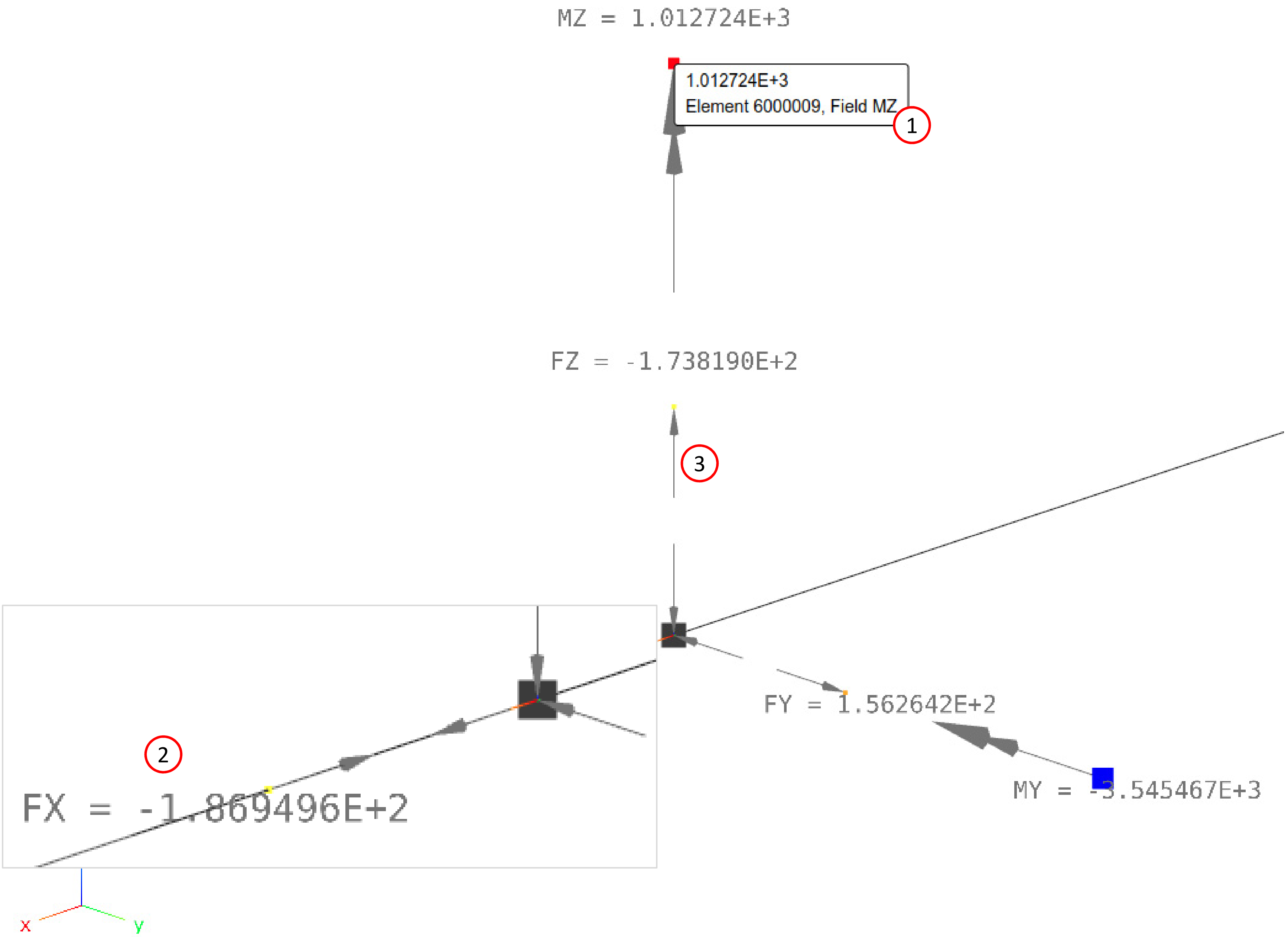


Nastran CBUSH Force Results - CBUSH and PBUSH

1. The label indicates the maximum response value and corresponding element ID and field name
2. For force along the x-axis (FX) of the CBUSH element coordinate system (ECS), the sign is negative. The sign of the force does NOT always indicate tension or compression. Instead, use the displayed arrows to determine if the force is tension or compression. In this case, the arrows point towards each other and indicate the force is compression.
3. For force along the z-axis (FZ) of the CBUSH ECS, the sign is negative, but the arrows point away from each other. The force is tension.

- If the default method of inspecting the model is not suitable for inspecting the CBUSH element forces, consider using the First Person option.

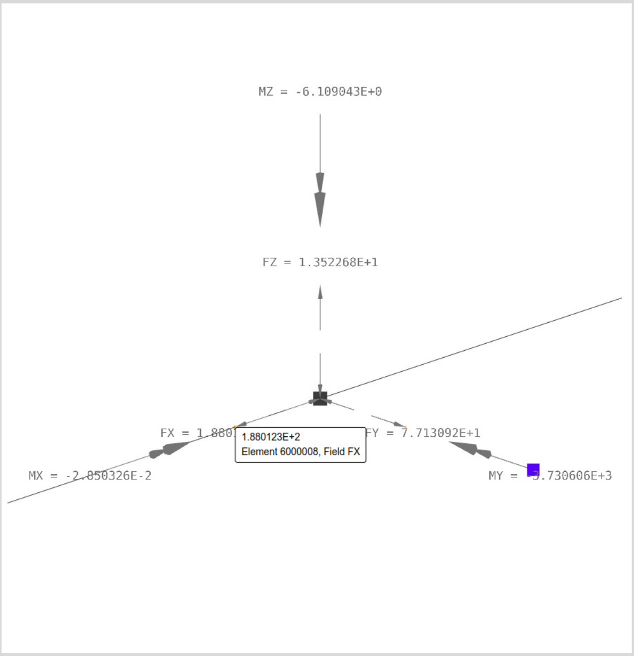




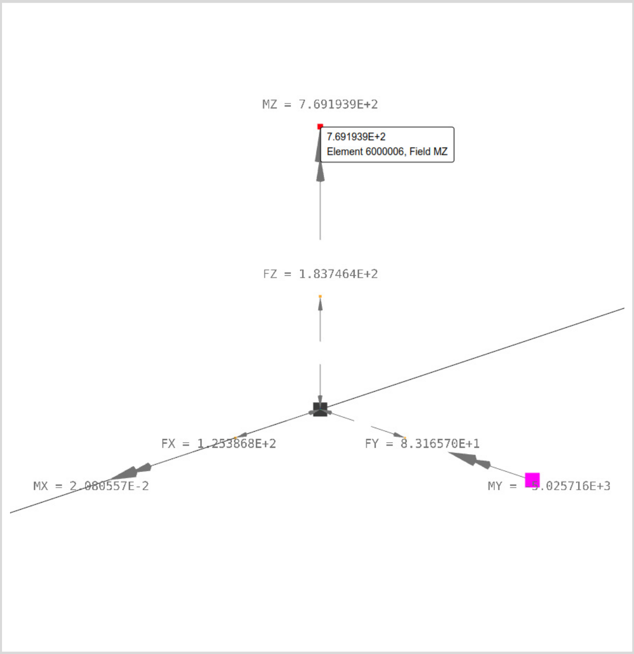
Nastran CBUSH Force Results - CBUSH and PBUSH

- The forces for the other CBUSH elements are displayed.

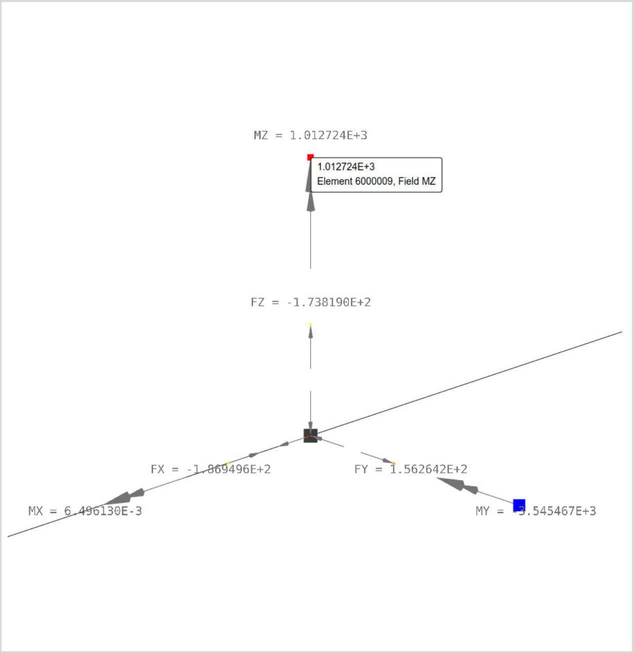
CBUSH 6000008



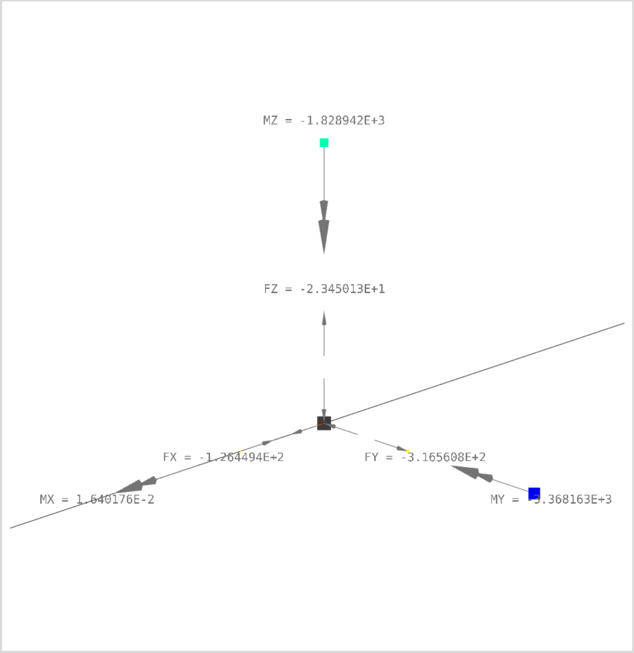
CBUSH 6000006



CBUSH 6000009

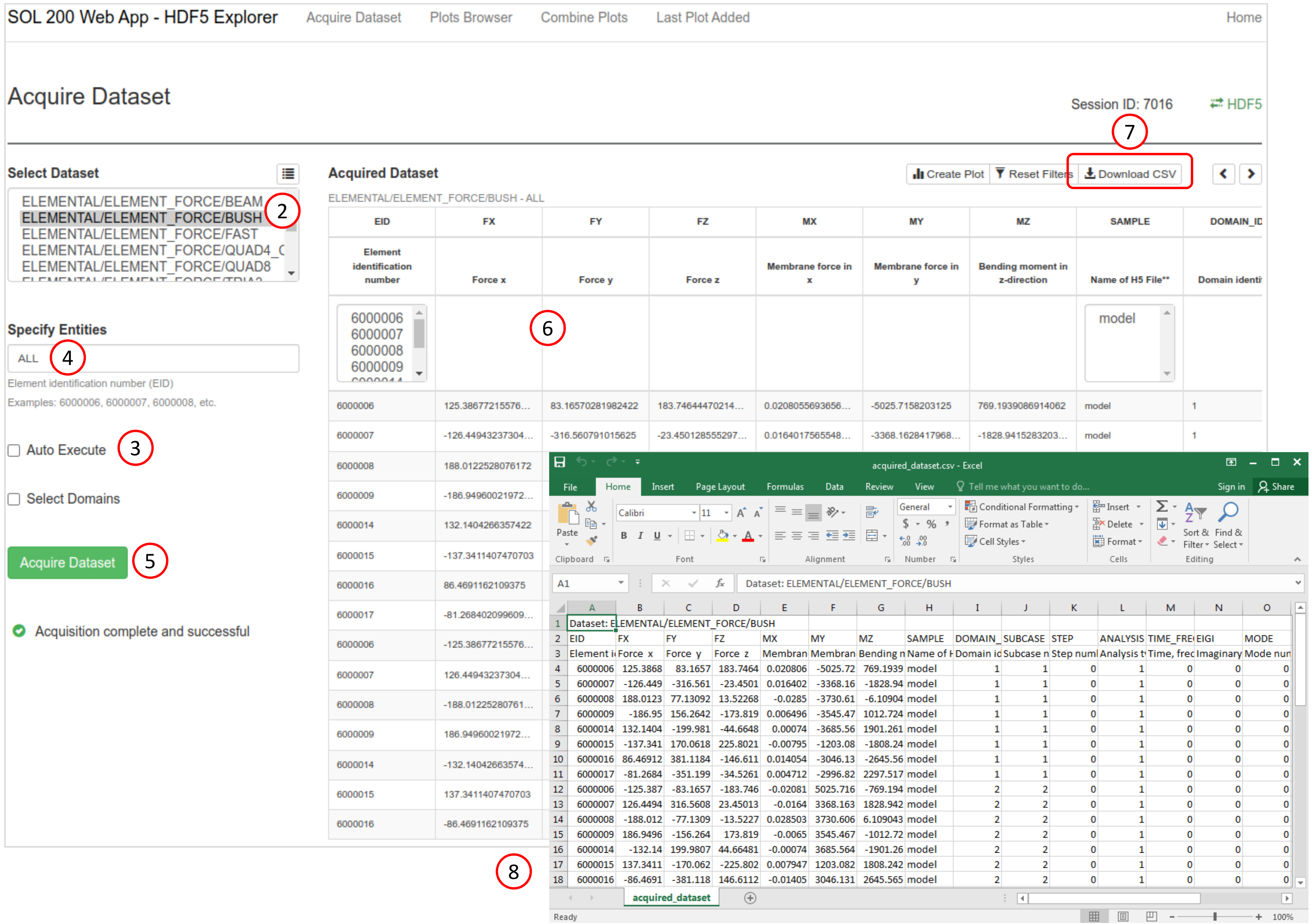


CBUSH 6000007



CBUSH Force Output to CSV

1. Upload the H5 file to the HDF5 Explorer (not shown)
2. Select dataset ELEMENTAL/ELEMENT_FORCE/BUSH
3. Unmark the checkbox Auto Execute
4. Set Specify Entities to All
5. Click Acquire Dataset
6. The web app acquires the forces for all CBUSH elements
7. Click Download CSV
8. The downloaded CSV file maybe opened in Excel or Python for future hand calculations



CBUSH Force Output to CSV with Python

Alternatively, Python may be used to write out the CBUSH element forces to a CSV file. Refer to the Python script shown to the right.

```
import h5py

# Necessary for H5 files generated with MSC Nastran 2021 and newer. MSC
Nastran 2021 and newer supports new compression methods, so the plugin is
necessary to access H5 files in different compression methods
import hdf5plugin


# Comments:
# A list of datasets is available in:
# 1. http://web.mscsoftware.com/doc/nastran/2018/release/DataType.html
# 2. The nastran documentation directory, e.g.
/msc/MSC_Nastran_Documentation/2021.4/doc/relnotes/v20214/DataType_v20214.h
tml
def write_dataset_to_csv_file(path_of_h5_file, dataset_name,
name_of_csv_file):
    file = h5py.File(path_of_h5_file, 'r')

    # Recover the DOMAINS dataset and index it
    # The DOMAINS dataset contains information about the SUBCASE,
TIME_FREQ_EIGR, etc.
    dataset_domains = file['/NASTRAN/RESULT/DOMAINS']
    dataset_original_domains_in_list_form = dataset_domains[...].tolist()
    dataset_domains_index = ['dummy_element_a', 'dummy_element_b']

    for line in dataset_original_domains_in_list_form:
        dataset_domains_index.insert(line[0], line)

    # Recover the dataset of interest
    dataset1 = file[dataset_name]

    dataset_original = dataset1

    # Column names
    # Take the column names from the H5 file (type: tuple), convert to a
python list (type: list), and
    # generate a string to add to the CSV file
    column_names_domains = dataset_domains.dtype.names
    column_names_domains = list(column_names_domains)
    column_names = dataset1.dtype.names
    column_names = list(column_names)
    name_of_last_column = column_names[len(column_names) - 1]
    column_names = ', '.join(column_names)
    column_names = column_names + ', ' + ', '.join(column_names_domains)

    # Determine if there are SUBCASEs (DOMAINS) to add
    attach_domains = False

    if name_of_last_column == 'DOMAIN_ID':
        attach_domains = True

    # Begin adding the data to the CSV file
    text_file = open(name_of_csv_file, 'w', encoding='utf8',
errors='replace')
    text_file.write(column_names + '\n')

    for line in dataset1:
        # Convert tuple to string
        # tuple to list, use list(line)
        # list of floats to list of strings, use str(x) for x in list(line)
        # list to string of comma separate values, use ','.join()
        outgoing_string = ','.join(str(x) for x in list(line))

        # If this dataset has corresponding DOMAINs (SUBCASE,
TIME_FREQ_EIGR, etc.), then associate
        # the information
        if attach_domains is True:
            domain_id = line[len(line) - 1]  # The DOMAIN_ID is in the last
column of the dataset of interest
            line_in_domains = dataset_domains_index[domain_id]  # Recover
the corresponding DOMAIN from the indexed list dataset_domains_index
            outgoing_string_domain = ','.join(str(x) for x in
list(line_in_domains))  # Convert tuple to string
            outgoing_string = outgoing_string + ',' +
outgoing_string_domain  # Create a line to add to the CSV file

        # Replace any brackets
        outgoing_string = str.replace(outgoing_string, ']', '')
        outgoing_string = str.replace(outgoing_string, '[', '')

        # Add a new line character to force a separate line
        outgoing_string = outgoing_string + '\n'

        # Add the line to the CSV file
        text_file.write(outgoing_string)

    # Close the file
    text_file.close()


if __name__ == '__main__':
    # What does this script do?
    #   This script writes out all the CBUSH element forces to a CSV file
    #
    # Instructions
    #   1. Change argument 1 to point to your H5 file
    #   2. Execute this python script
    #   3. The CSV file is written to the python working directory
    #      Use print(os.getcwd()) to display the path of the working
directory
    write_dataset_to_csv_file('/home/usera/Downloads/model.h5',
'/NASTRAN/RESULT/ELEMENTAL/ELEMENT_FORCE/BUSH', 'cbush_element_forces.csv')
```

Nastran CFAST Force Results - CFAST and PFAST

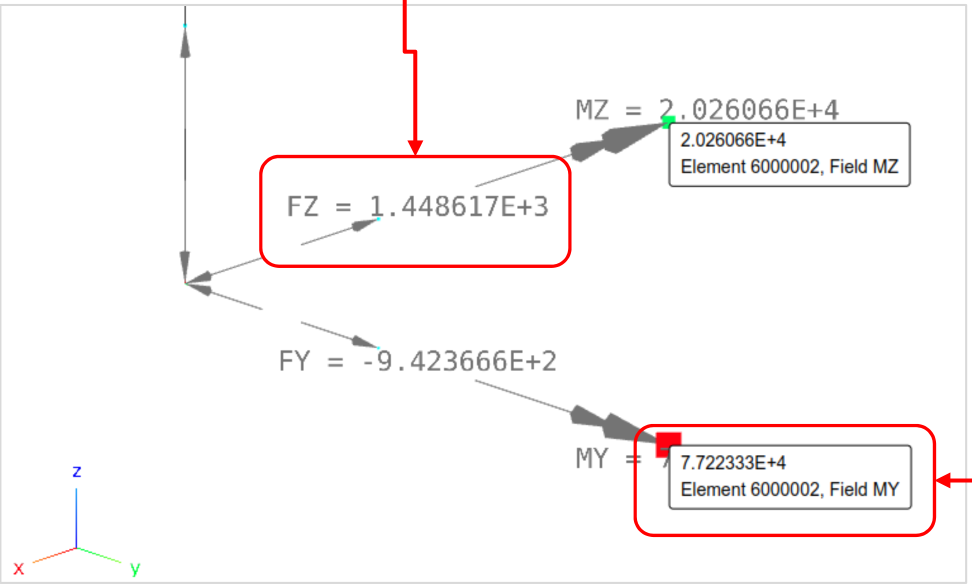
Goal: Display CFAST element forces

F06 File

0 SUBCASE 1

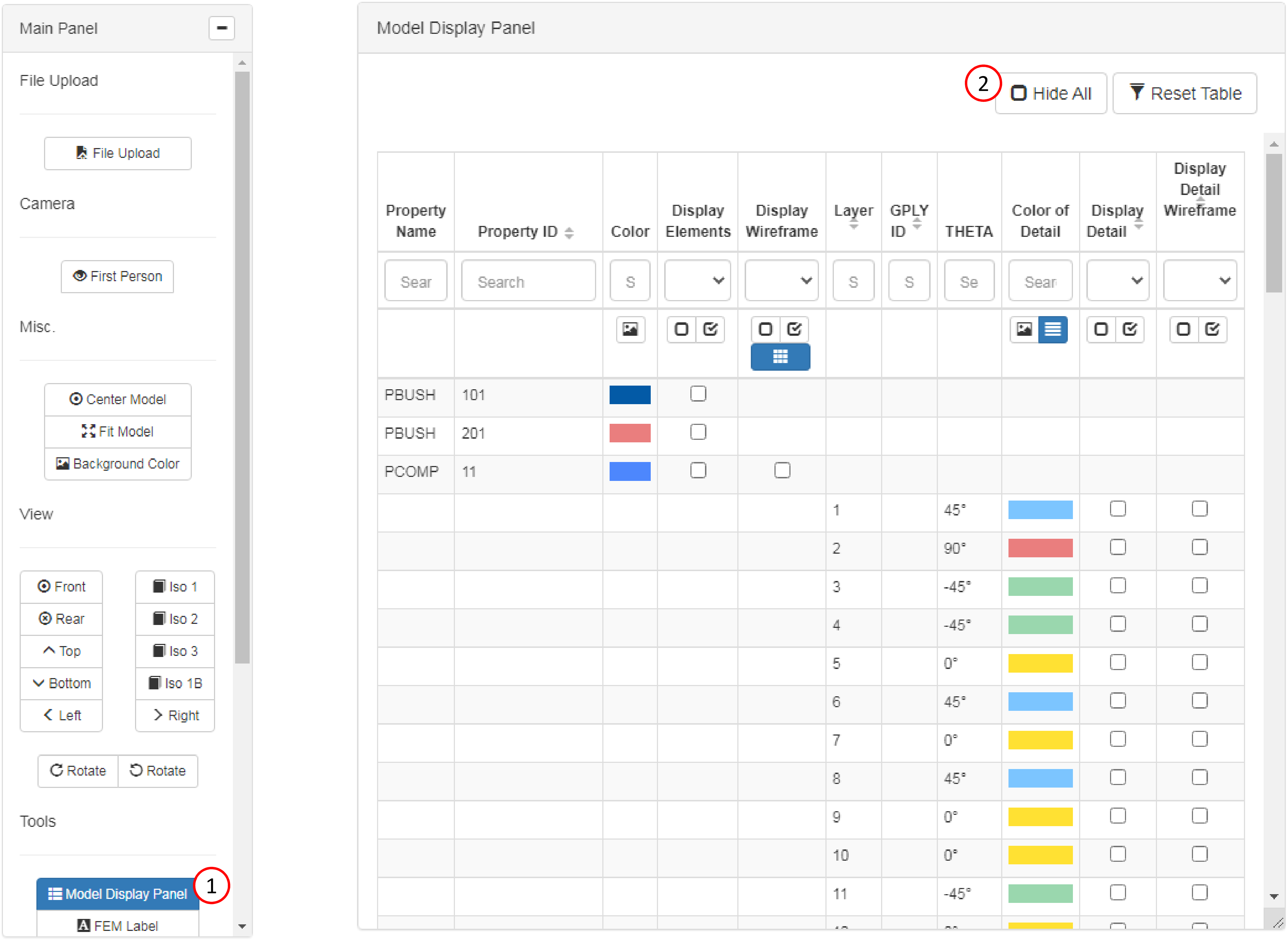
F O R C E S   I N   F A S T E N E R   E L E M E N T S   (C F A S T)

ELEMENT-ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
6000002	1.349132E+03	-9.423671E+02	1.448617E+03	-5.632881E-02	7.722330E+04	2.026070E+04



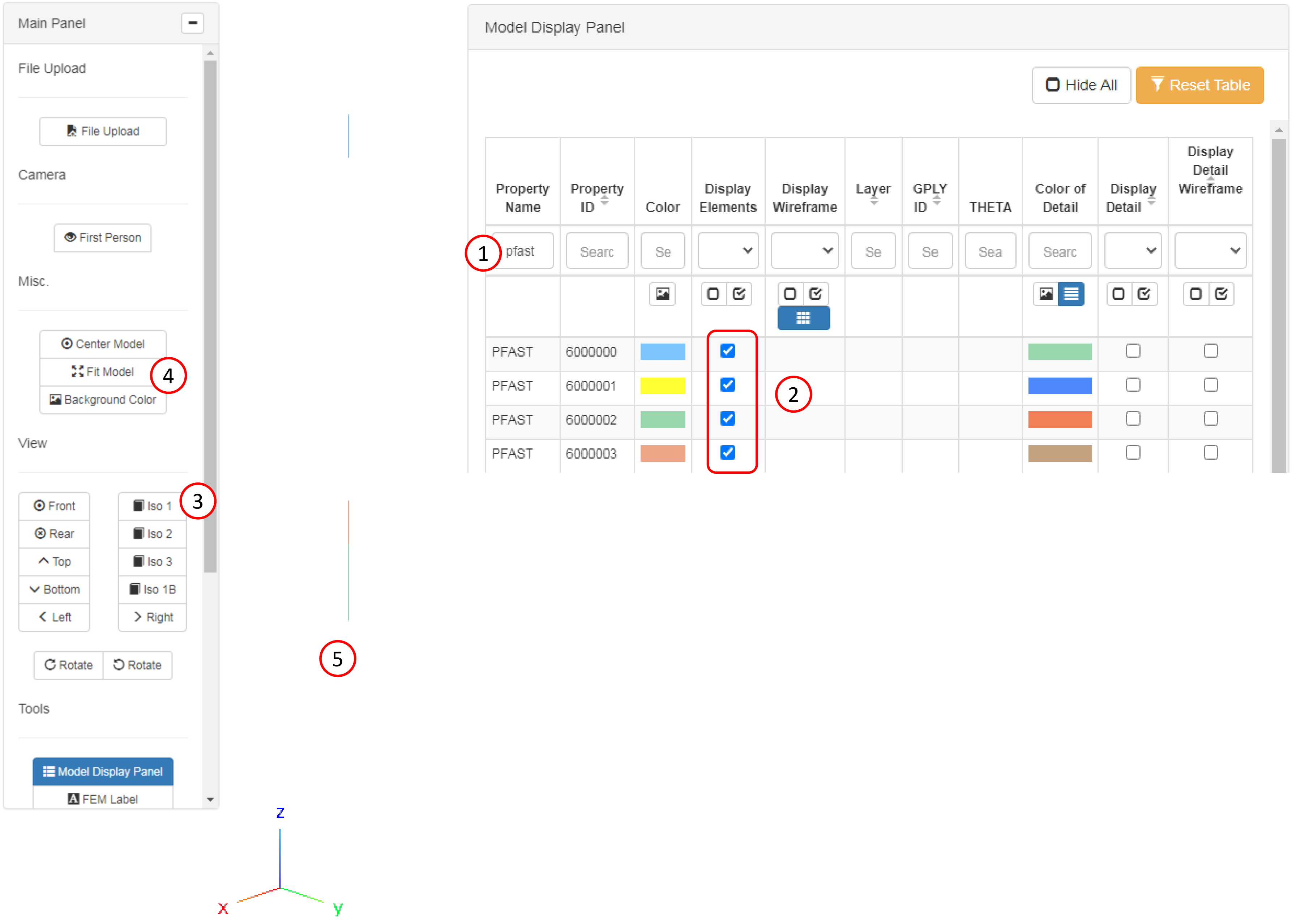
Nastran CFAST Force Results - CFAST and PFAST

1. Click Model Display Panel
2. Click Hide All



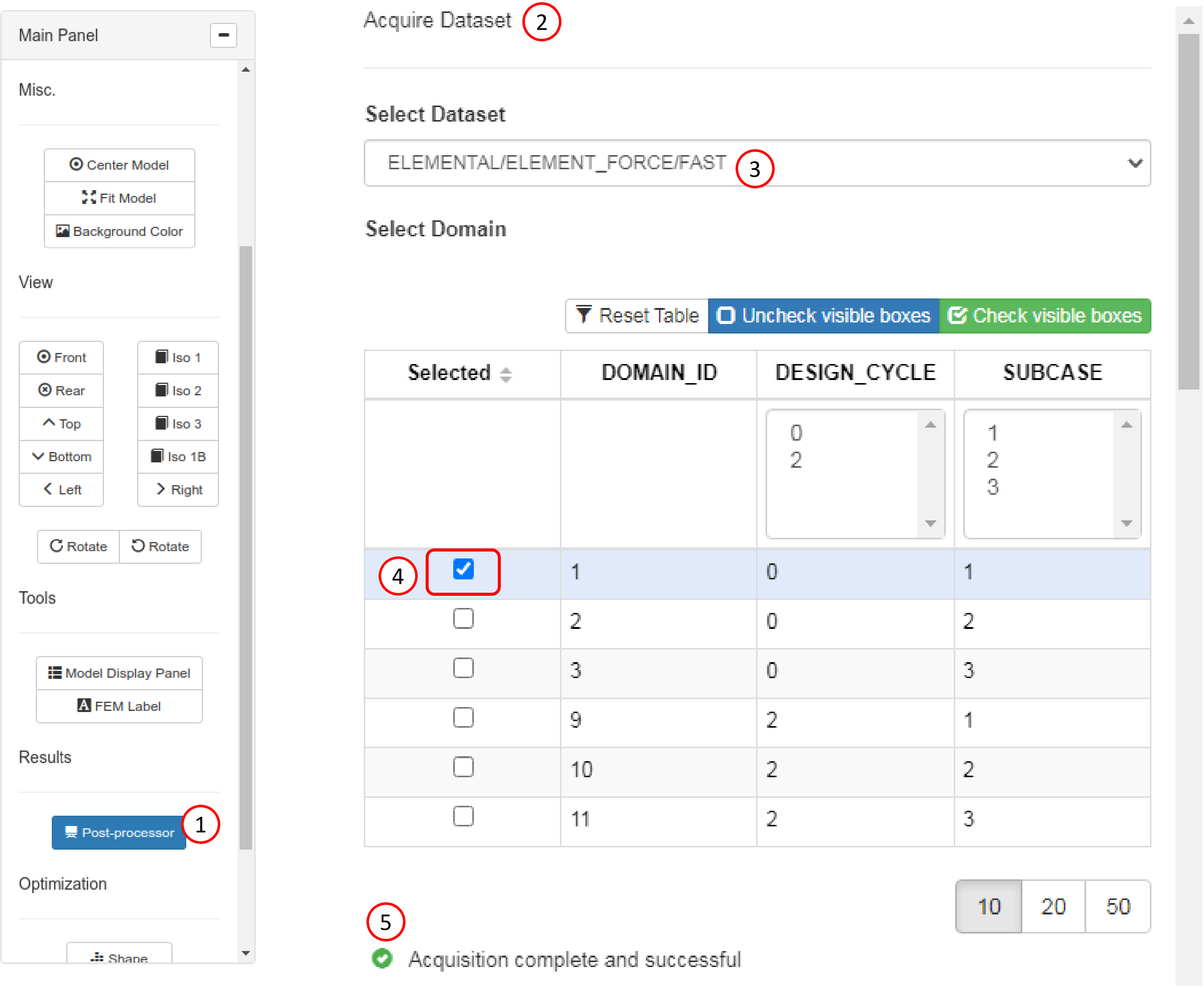
Nastran CFAST Force Results - CFAST and PFAST

1. Use the search bar to search for: pfast
2. Mark the indicated checkboxes
3. Click Iso 1 (Isometric 1 View)
4. Click Fit Model
5. The elements configured via CFAST and PFAST entries are now visible



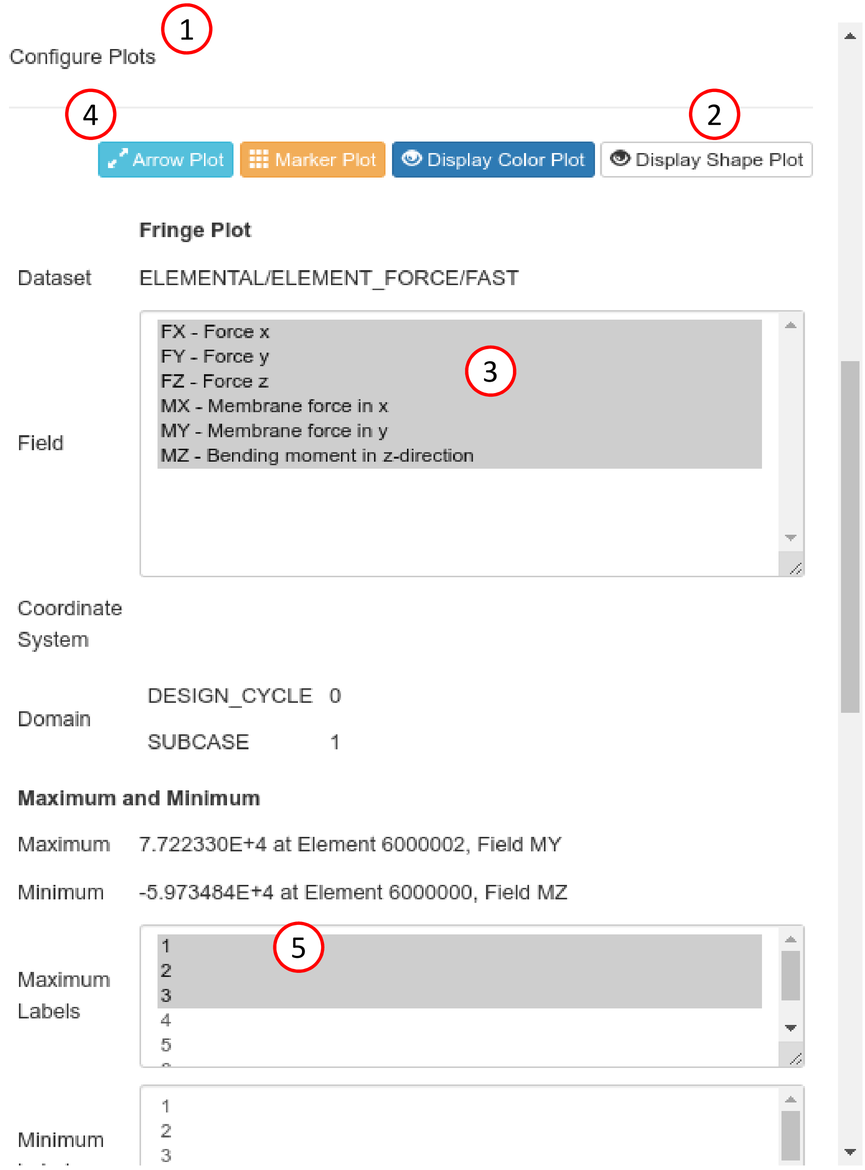
Nastran CFAST Force Results - CFAST and PFAST

1. Click Post-processor
2. Use the vertical scroll bar to navigate to section Acquire Dataset
3. Set Select Dataset to ELEMENTAL/ELEMENT_FORCE/FAST
4. Mark the checkbox for SUBCASE 1 for DESIGN_CYCLE 0
5. Wait until the status says Acquisition complete and successful



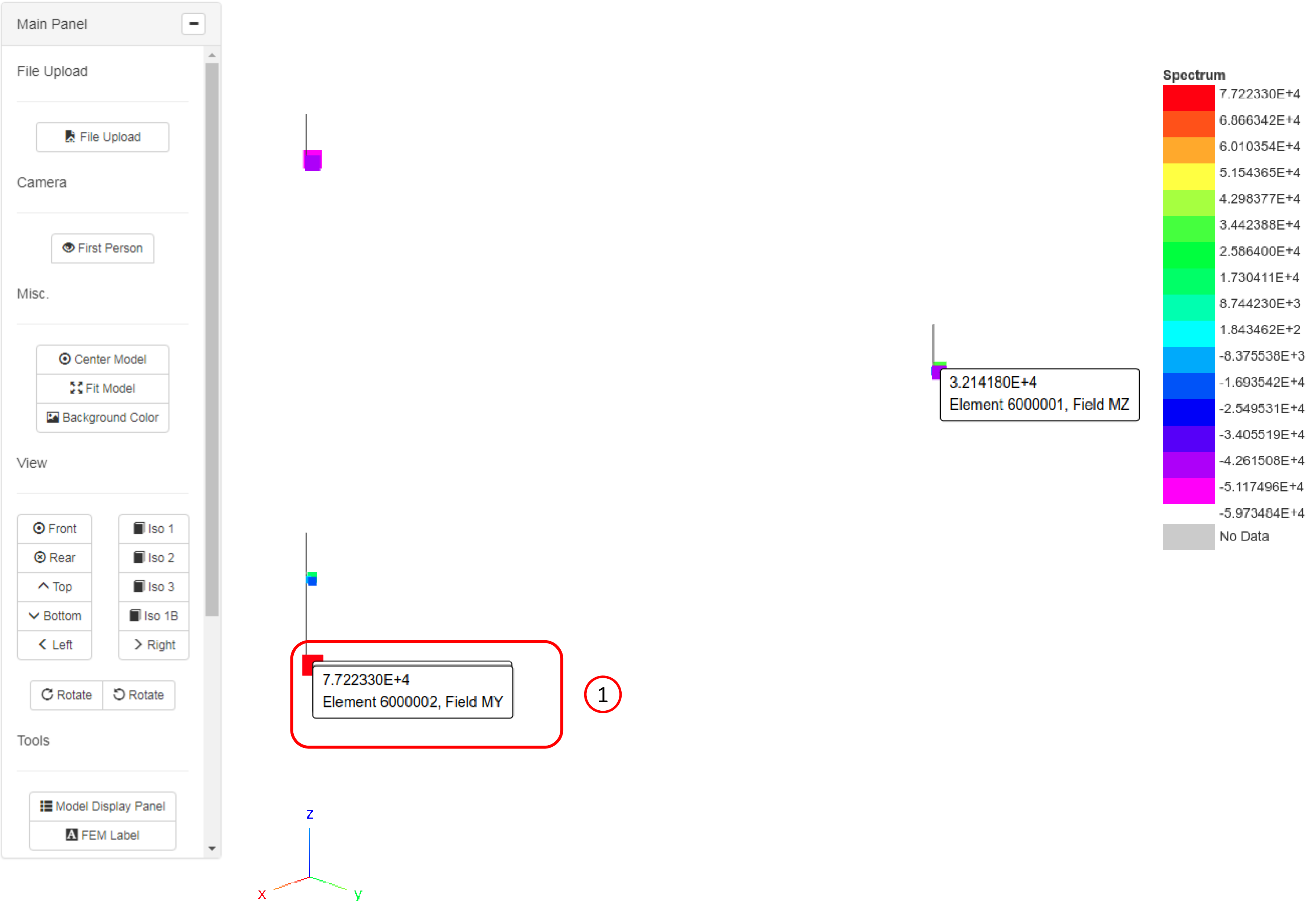
Nastran CFAST Force Results - CFAST and PFAST

1. Use the vertical scroll bar to navigate to section Configure Plots
2. Click Display Shape Plot to hide the deformation
3. Set the following fields:
    - FX
    - FY
    - FZ
    - MX
    - MY
    - MZ
4. Click Arrow Plot
5. Select the first 3 maximum labels



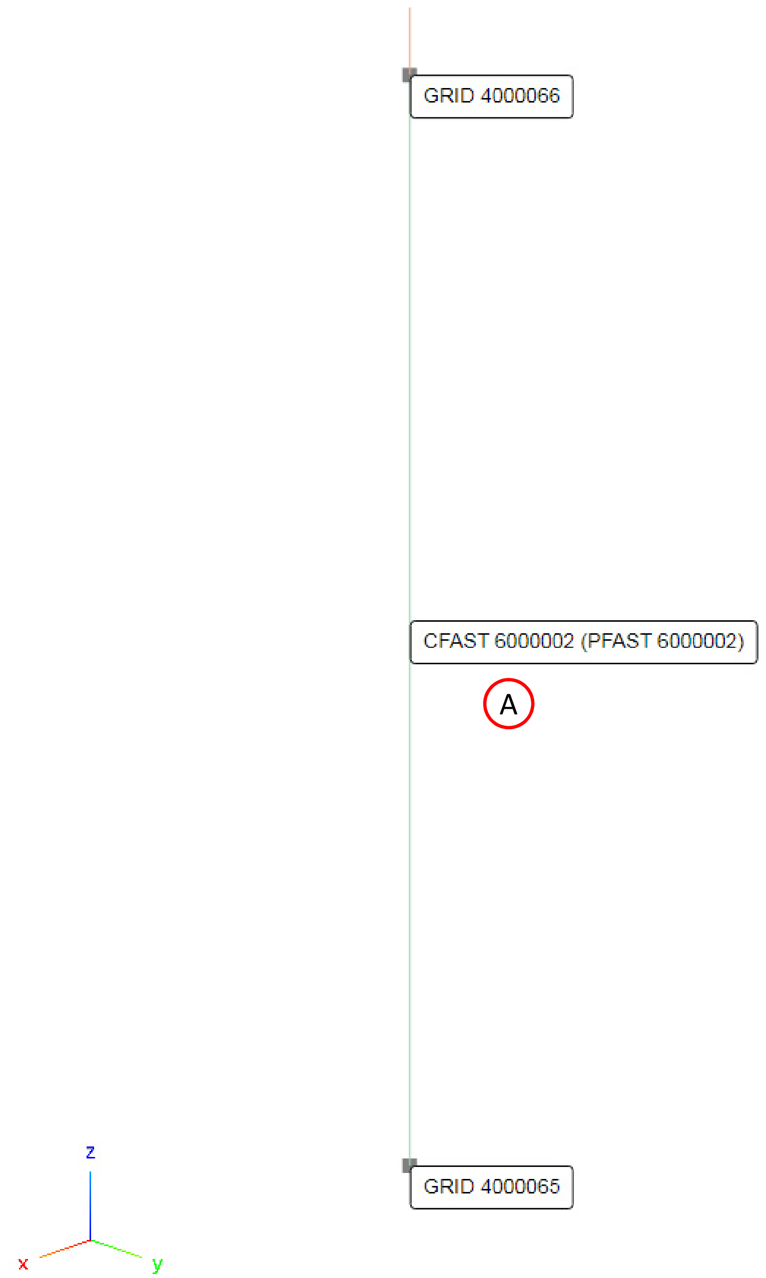
Nastran CFAST Force Results - CFAST and PFAST

1. Zoom in to the indicated region



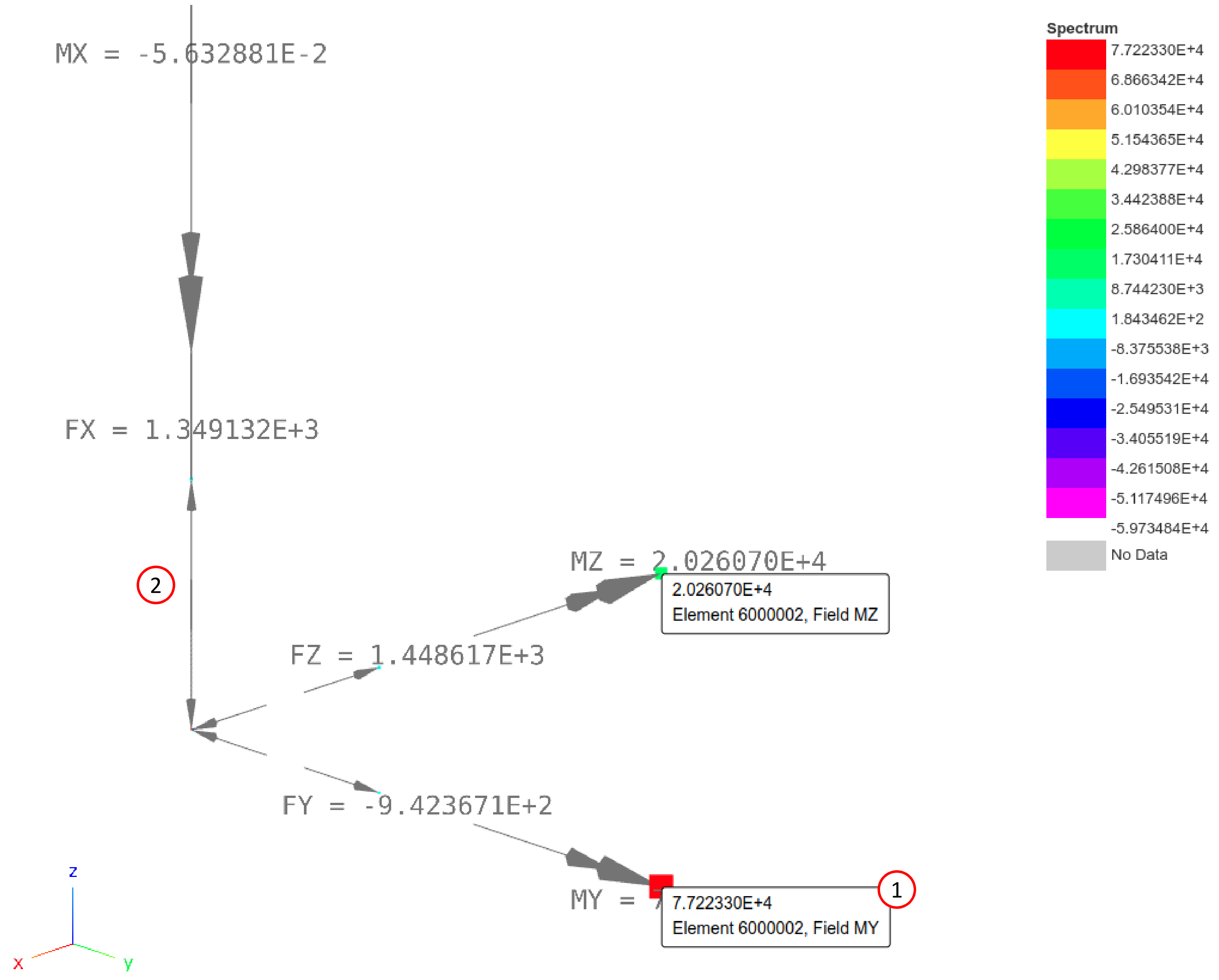
Nastran CFAST Force Results - CFAST and PFAST

A. As you zoom in, note that this CFAST element is used to connect two grids (nodes). The FEM Label Tool was used to display the element and grid IDs.



Nastran CFAST Force Results - CFAST and PFAST

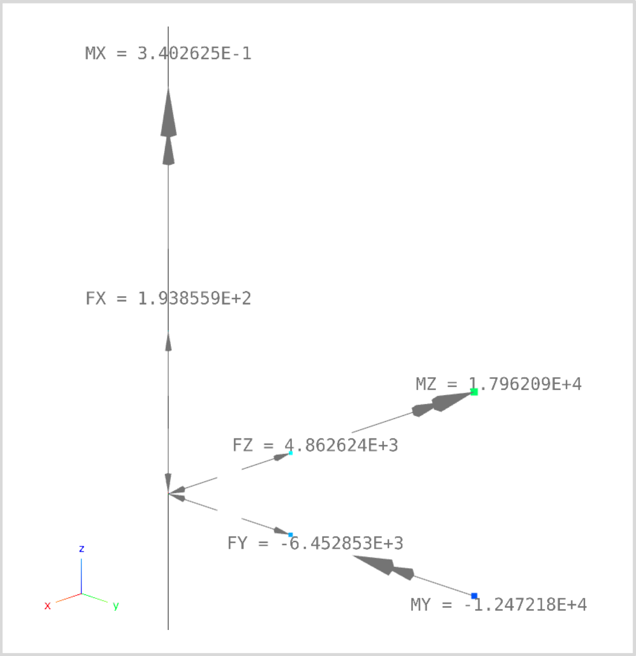
1. The label indicates the maximum response value and corresponding element ID and field name
2. For force along the x-axis (FX) of the CFAST element coordinate system (ECS), the sign is positive. The sign of the force does NOT always indicate tension or compression. Instead, use the displayed arrows to determine if the force is tension or compression. In this case, the arrows point away from each other and indicate the force is tension.



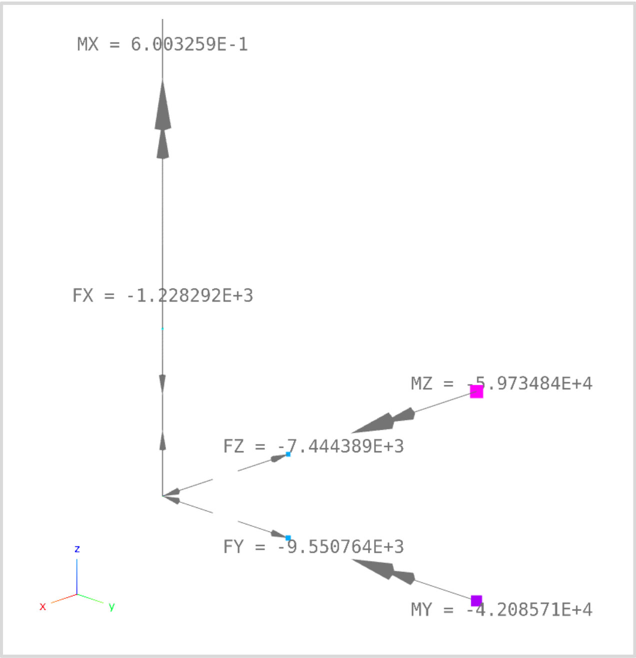
Nastran CFAST Force Results - CFAST and PFAST

- The forces for the other CFAST elements are displayed.

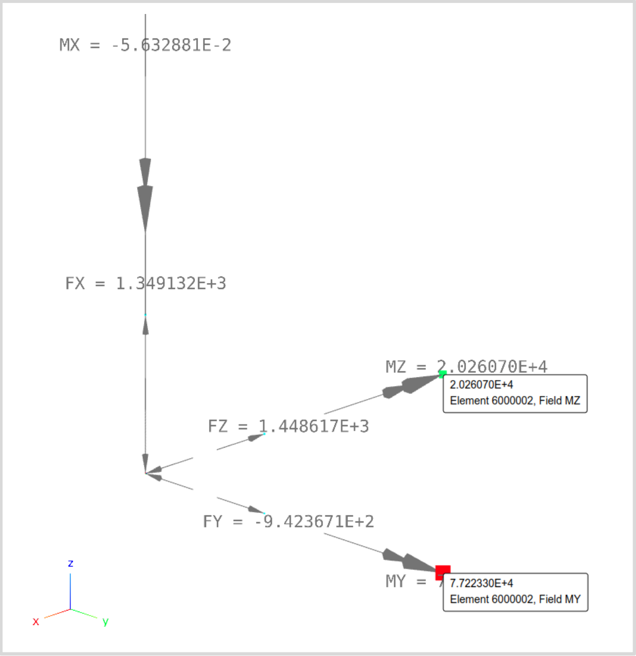
CFAST 6000003



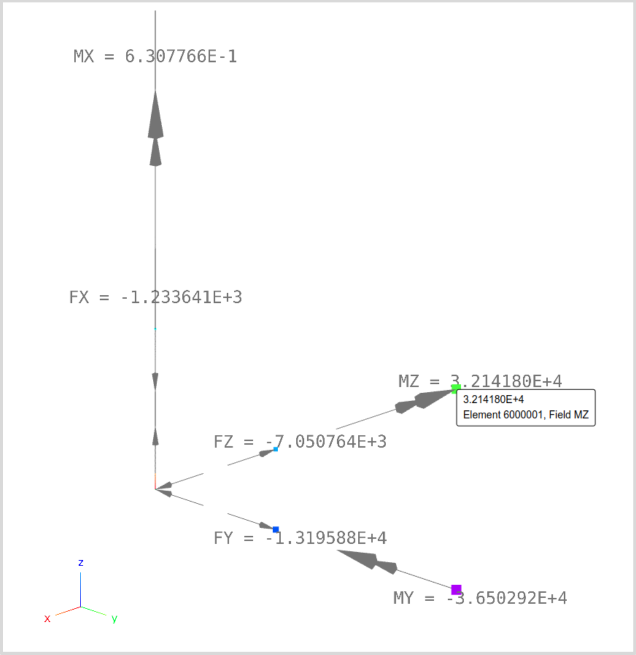
CFAST 6000000



CFAST 6000002

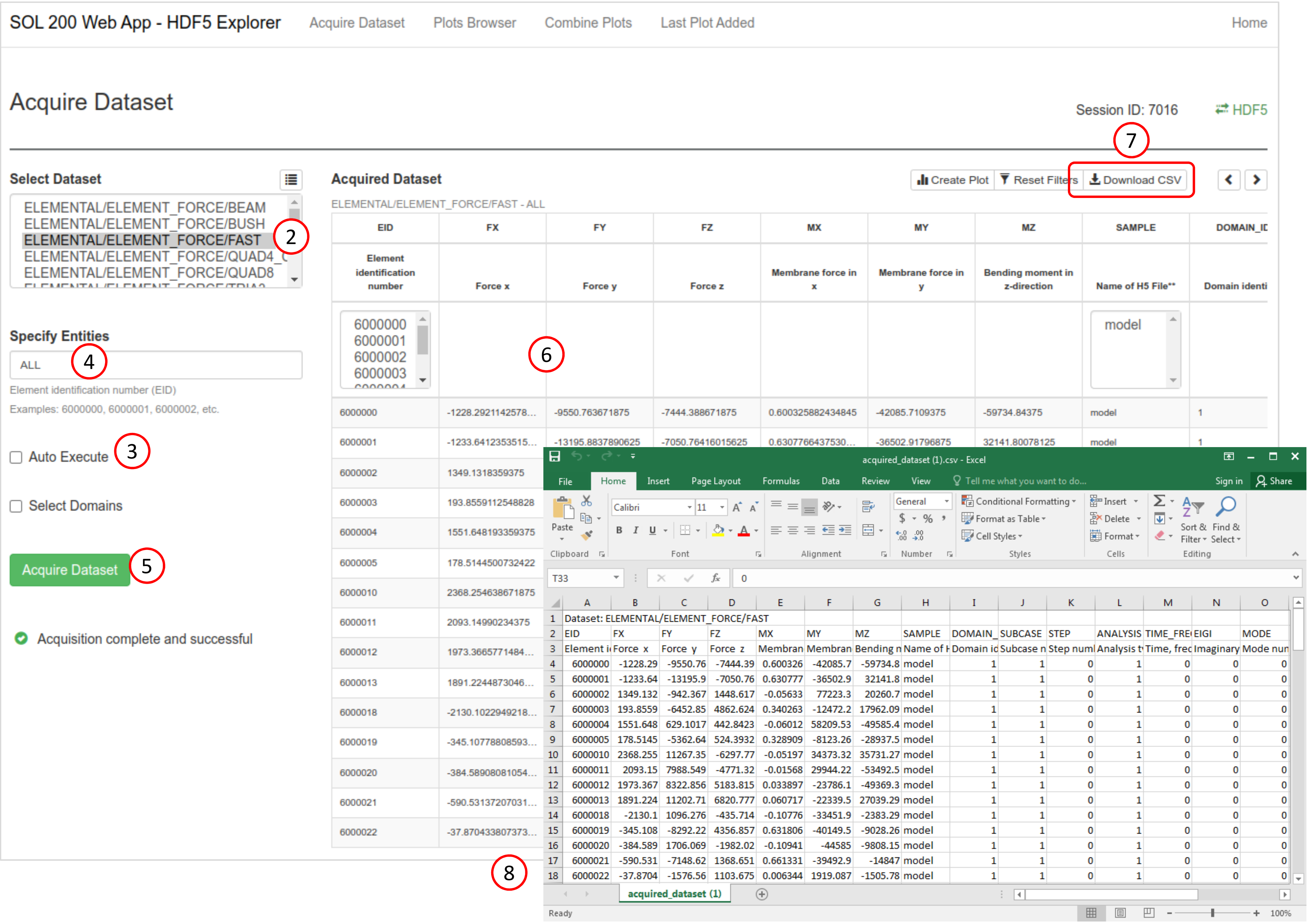


CFAST 6000001



CFAST Force Output to CSV

1. Upload the H5 file to the HDF5 Explorer (not shown)
2. Select dataset ELEMENTAL/ELEMENT_FORCE/FAST
3. Unmark the checkbox Auto Execute
4. Set Specify Entities to All
5. Click Acquire Dataset
6. The web app acquires the forces for all CFAST elements
7. Click Download CSV
8. The downloaded CSV file maybe opened in Excel or Python for future hand calculations



CFAST Force Output to CSV with Python

Alternatively, Python may be used to write out the CFAST element forces to a CSV file. Refer to the Python script shown to the right.

1. The same script as before is used, but the name of the dataset is updated to extract results for CFAST elements. Also, the name of the CSV file is updated.

```
import h5py

# Necessary for H5 files generated with MSC Nastran 2021 and newer. MSC
Nastran 2021 and newer supports new compression methods, so the plugin is
necessary to access H5 files in different compression methods
import hdf5plugin


# Comments:
# A list of datasets is available in:
# 1. http://web.mscsoftware.com/doc/nastran/2018/release/DataType.html
# 2. The nastran documentation directory, e.g.
/msc/MSC_Nastran_Documentation/2021.4/doc/relnotes/v20214/DataType_v20214.h
tml
def write_dataset_to_csv_file(path_of_h5_file, dataset_name,
name_of_csv_file):
    file = h5py.File(path_of_h5_file, 'r')

    # Recover the DOMAINS dataset and index it
    # The DOMAINS dataset contains information about the SUBCASE,
TIME_FREQ_EIGR, etc.
    dataset_domains = file['/NASTRAN/RESULT/DOMAINS']
    dataset_original_domains_in_list_form = dataset_domains[...].tolist()
    dataset_domains_index = ['dummy_element_a', 'dummy_element_b']

    for line in dataset_original_domains_in_list_form:
        dataset_domains_index.insert(line[0], line)

    # Recover the dataset of interest
    dataset1 = file[dataset_name]

    dataset_original = dataset1

    # Column names
    # Take the column names from the H5 file (type: tuple), convert to a
python list (type: list), and
    # generate a string to add to the CSV file
    column_names_domains = dataset_domains.dtype.names
    column_names_domains = list(column_names_domains)
    column_names = dataset1.dtype.names
    column_names = list(column_names)
    name_of_last_column = column_names[len(column_names) - 1]
    column_names = ', '.join(column_names)
    column_names = column_names + ', ' + ', '.join(column_names_domains)

    # Determine if there are SUBCASEs (DOMAINS) to add
    attach_domains = False

    if name_of_last_column == 'DOMAIN_ID':
        attach_domains = True

    # Begin adding the data to the CSV file
    text_file = open(name_of_csv_file, 'w', encoding='utf8',
errors='replace')
    text_file.write(column_names + '\n')

    for line in dataset1:
        # Convert tuple to string
        # tuple to list, use list(line)
        # list of floats to list of strings, use str(x) for x in list(line)
        # list to string of comma separate values, use ','.join()
        outgoing_string = ','.join(str(x) for x in list(line))

        # If this dataset has corresponding DOMAINs (SUBCASE,
TIME_FREQ_EIGR, etc.), then associate
        # the information
        if attach_domains is True:
            domain_id = line[len(line) - 1]  # The DOMAIN_ID is in the last
column of the dataset of interest
            line_in_domains = dataset_domains_index[domain_id]  # Recover
the corresponding DOMAIN from the indexed list dataset_domains_index
            outgoing_string_domain = ','.join(str(x) for x in
list(line_in_domains))  # Convert tuple to string
            outgoing_string = outgoing_string + ',' +
outgoing_string_domain  # Create a line to add to the CSV file

        # Replace any brackets
        outgoing_string = str.replace(outgoing_string, ']', '')
        outgoing_string = str.replace(outgoing_string, '[', '')

        # Add a new line character to force a separate line
        outgoing_string = outgoing_string + '\n'

        # Add the line to the CSV file
        text_file.write(outgoing_string)

    # Close the file
    text_file.close()


if __name__ == '__main__':
    # What does this script do?
    #   This script writes out all the CFAST element forces to a CSV file
    #
    # Instructions
    #   1. Change argument 1 to point to your H5 file
    #   2. Execute this python script
    #   3. The CSV file is written to the python working directory
    #      Use print(os.getcwd()) to display the path of the working
directory
    write_dataset_to_csv_file('/home/usera/Downloads/model.h5',
'/NASTRAN/RESULT/ELEMENTAL/ELEMENT_FORCE/FAST', 'cfast_element_forces.csv')
```

1

End of Tutorial

Appendix

Appendix Contents

- Frequently Asked Questions
  - Why is the PCH file uploaded to the Viewer?
  - What are layers in a PCOMP and PCOMPG entry?
  - What are displacement coordinate systems?

Why is the PCH file uploaded to the Viewer?

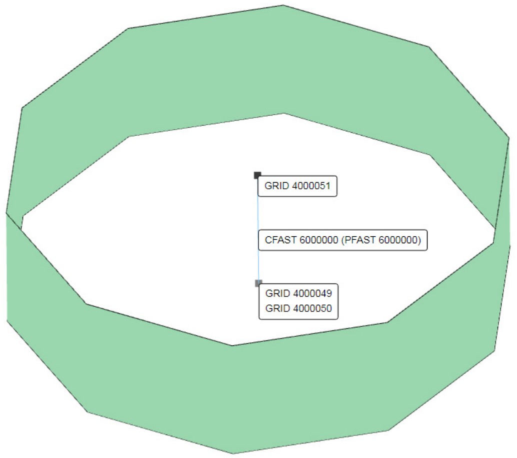
When a connector element, such as a CFAST element, is defined, MSC Nastran internally generates equivalent entries.

Connector Element	Equivalent Entries
CFAST	CBUSH, RBE3, GRID
CSEAM	CBAR, RBE3, GRID
CWELD	CHEXA, RBE3, GRID

The equivalent entries may be output to the punch file (.pch) via the following bulk data entry.

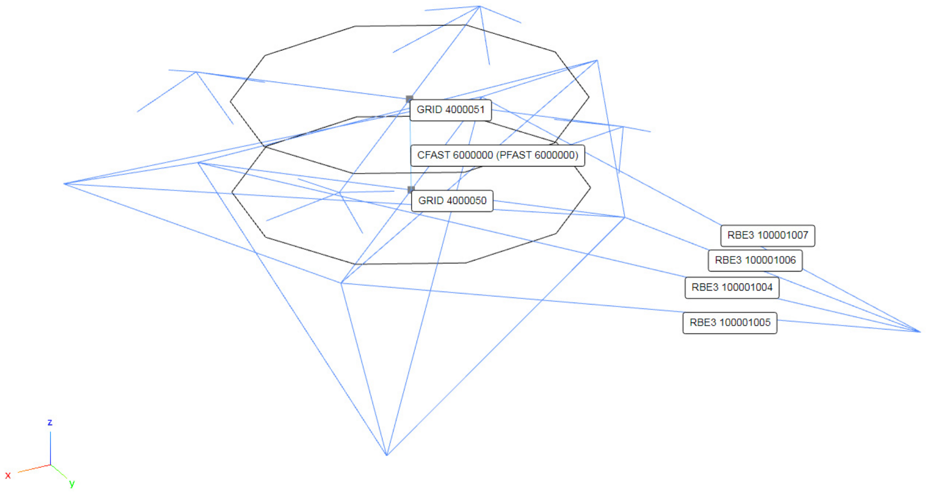
swldprm,prtsw,100

The punch file is uploaded to the Viewer so the equivalent entries are imported and may be displayed.



model.bdf

```
PFAST     6000000      10.        0        0 362000.  25100.  17800.     100.+
+            1.E+9    1.E+9
CFAST     6000000 6000000     ELEM     1032 6000182 4000049 4000050 4000051
```

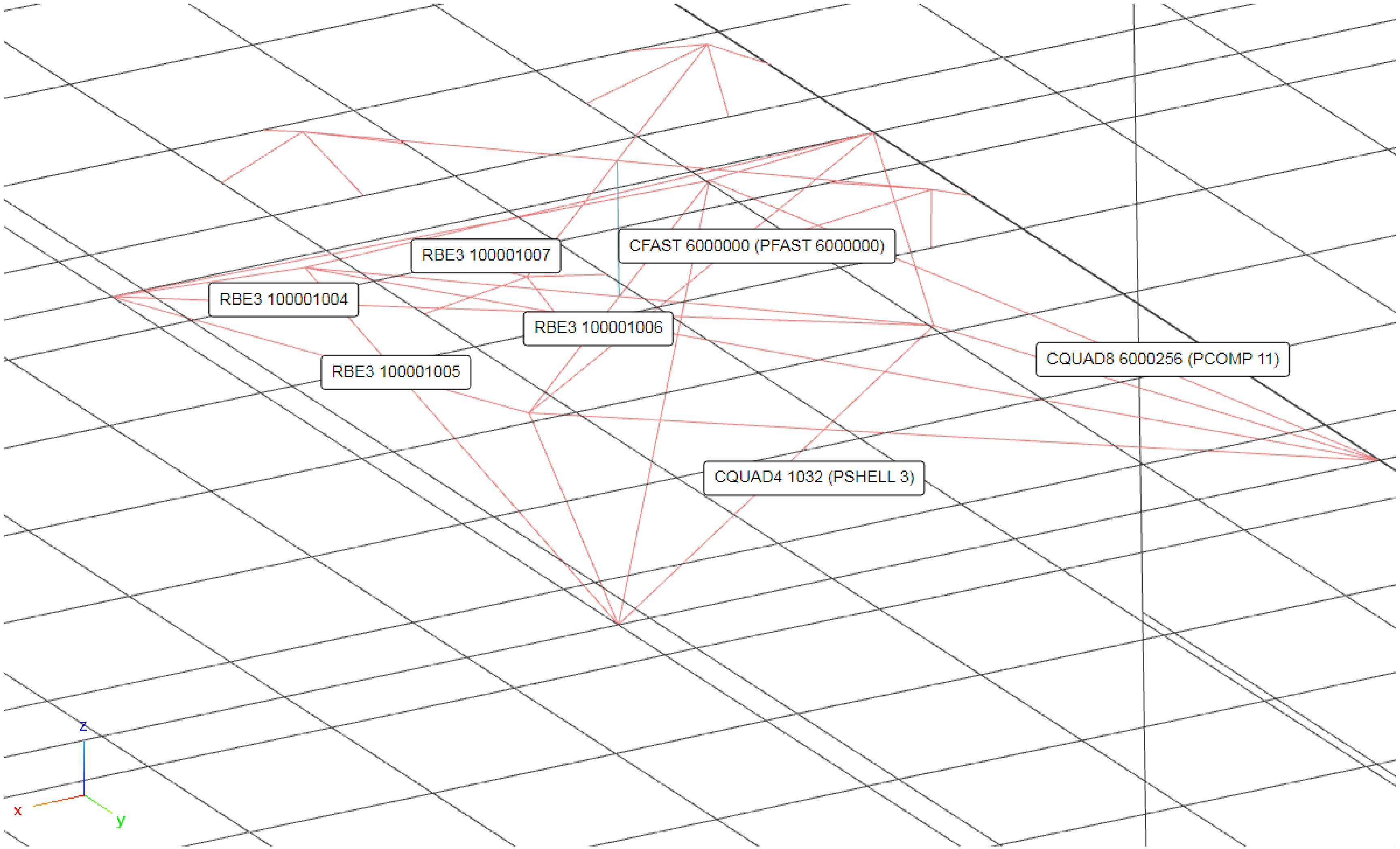


model.pch

```
$ CFAST EID= 6000000
RBE3*          100001002                         4000050          123456
*           1.00000E+00             123         10000003        10000004
*              10000005        10000006
*
RBE3*          100001003                         4000051          123456
*           1.00000E+00             123         10000007        10000008
*              10000009        10000010
*
RBE3*          100001004                        10000003             123
*           7.44085E-01             123             1043
*           3.72043E-02             123             1044
*           3.72043E-02             123             1050
*           2.27471E-01             123             1049
*
[…]

GRID*            4000050                     2.50000E+01     2.50000E+01
*           1.00000E+02
GRID*            4000051                     2.50000E+01     2.50000E+01
*           1.02800E+02
GRID*           10000003                     2.94311E+01     2.05689E+01
*           1.00000E+02
GRID*           10000004                     2.94311E+01     2.94311E+01
*           1.00000E+02
GRID*           10000005                     2.05689E+01     2.94311E+01
*           1.00000E+02
GRID*           10000006                     2.05689E+01     2.05689E+01
            1.00000E+02
[…]
```

Why is the PCH file uploaded to the Viewer?

- To the right, is the display of a CFAST element after the punch file (.pch) is uploaded to the Viewer and the equivalent entries are imported.
- It is shown the CFAST element is connecting elements from PCOMP 11 and PSHELL 3.



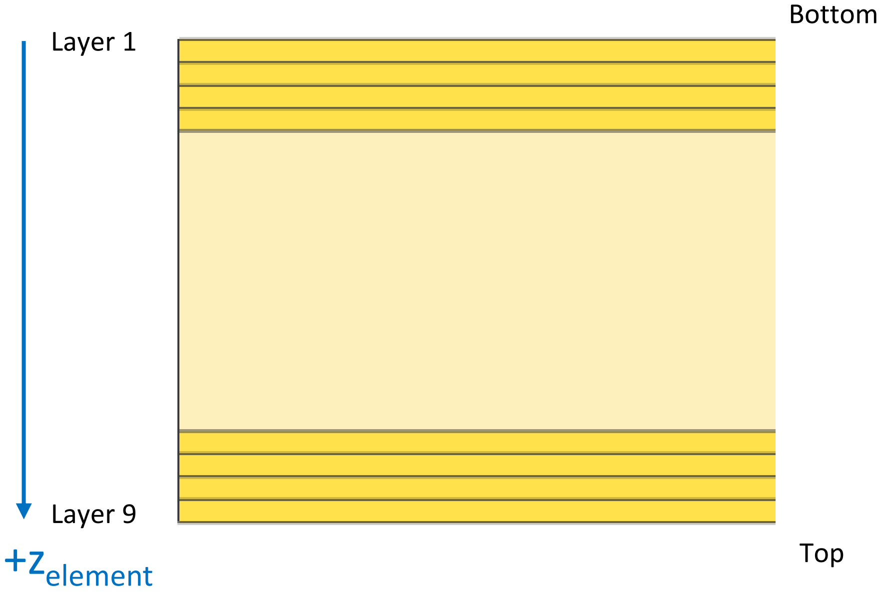
What are layers in a PCOMP and PCOMPG entry?

PCOMP and PCOMPG entries are used to define *layers*.

A layer can have multiple meanings. A layer may be used to define:

- One ply
- Multiple Plies
- Core

For the example on the right, PCOMP 1 defines 4 layers for the 4 top plies. For PCOMP 2, 1 layer is used to represent the 4 top plies. Both configurations yield the same ABD matrices.



```
PCOMP   1
        1       .0458   0.      YES
        1       .0458   0.      YES
        1       .0458   0.      YES
        1       .0458   0.      YES
        1       .6      90.     YES
        1       .0458   0.      YES
        1       .0458   0.      YES
        1       .0458   0.      YES
        1       .0458   0.      YES
```

```
PCOMP   2
        1       .1832   0.      YES
        1       .6      90.     YES
        1       .1832   0.      YES
```

What are layers in a PCOMP and PCOMPG entry?

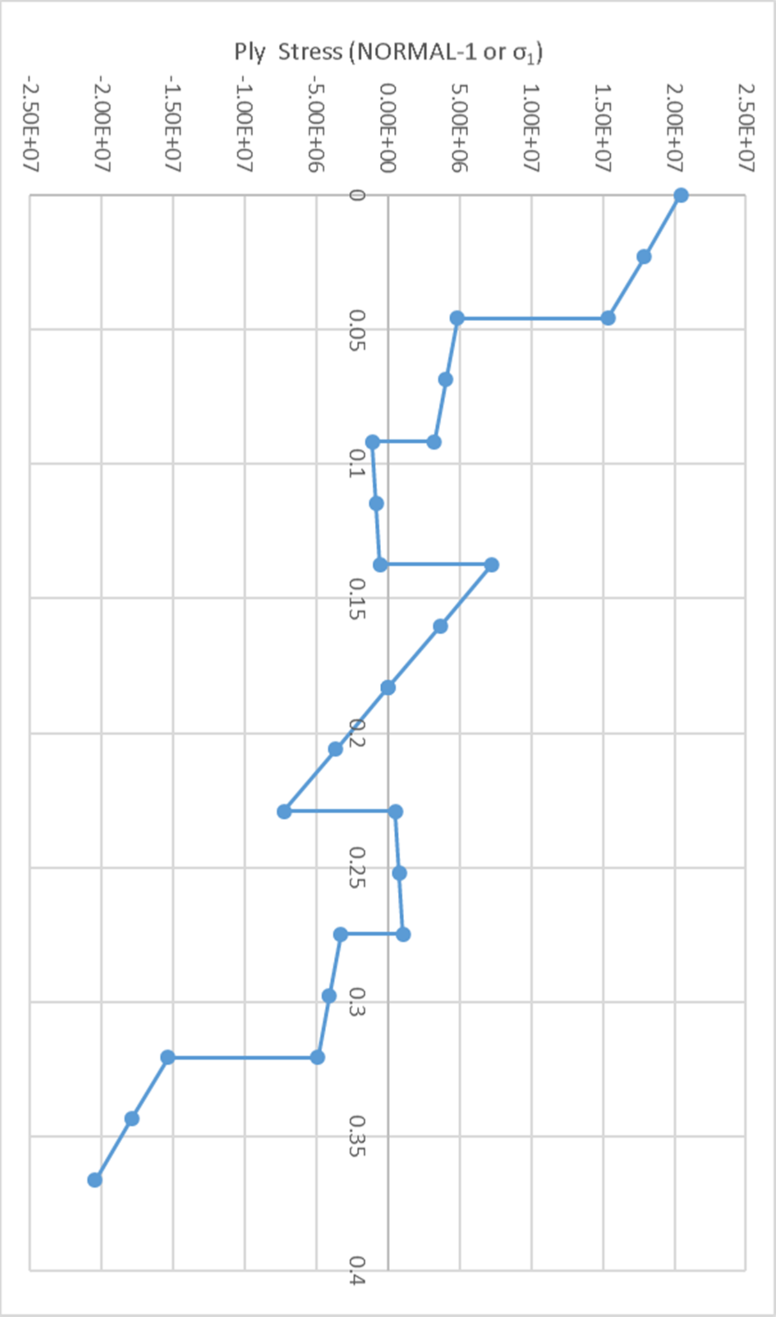
PCOMP and PCOMPG entries are used to define *layers*.

A layer can have multiple meanings. A layer may be used to define:

- One ply
- Multiple Plies
- Core
- Dummy Layers
    - Responses are output for the midplane of each layer. For the example on the right, dummy layers with very small thicknesses are used to acquire the ply stress at the top and bottom of each ply.

```
PCOMP     3
          1         1.0E-10 45.     YES
          1         .0458   45.     YES
          1         1.0E-10 45.     YES
          1         1.0E-10 -45.    YES
          1         .0458   -45.    YES
          1         1.0E-10 -45.    YES
          1         1.0E-10 90.     YES
          1         .0458   90.     YES
          1         1.0E-10 90.     YES
          1         1.0E-10 0.      YES
          1         .0458   0.      YES
          1         1.0E-10 0.      YES
          1         1.0E-10 0.      YES
          1         .0458   0.      YES
          1         1.0E-10 0.      YES
          1         1.0E-10 90.     YES
          1         .0458   90.     YES
          1         1.0E-10 90.     YES
          1         1.0E-10 -45.    YES
          1         .0458   -45.    YES
          1         1.0E-10 -45.    YES
          1         1.0E-10 45.     YES
          1         .0458   45.     YES
          1         1.0E-10 45.     YES
```

Layer	Ply (Position)
1	Ply 1 (Bottom)
2	Ply 1 (Middle)
3	Ply 1 (Top)
4	Ply 2 (Bottom)
5	Ply 2 (Middle)
6	Ply 2 (Top)
7	Ply 3 (Bottom)
8	Ply 3 (Middle)
9	Ply 3 (Top)
10	Ply 4 (Bottom)
11	Ply 4 (Middle)
12	Ply 4 (Top)
13	Ply 5 (Bottom)
14	Ply 5 (Middle)
15	Ply 5 (Top)
16	Ply 6 (Bottom)
17	Ply 6 (Middle)
18	Ply 6 (Top)
19	Ply 7 (Bottom)
20	Ply 7 (Middle)
21	Ply 7 (Top)
22	Ply 8 (Bottom)
23	Ply 8 (Middle)
24	Ply 8 (Top)



What are displacement coordinate systems?

Displacement Coordinate Systems

- The displacement coordinate system of nodes (GRIDs) determines the direction of the 6 degrees of freedom.
- Most software applications do not display the DOFs, so the following slides display the DOFs when a rectangular or cylindrical coordinate system is used to define the displacement coordinate systems of nodes.

"Six rectangular displacement components (three translations and three rotations) are defined at each grid point. The displacement coordinate system, which is used to define the directions of motion, may be different from the 'location coordinate system,' which is used to locate the grid point. Both the location coordinate system and the displacement coordinate system are specified on the GRID entry for each geometric grid point. The orientation of displacement components depends on the type of local coordinate system used to define the displacement components. If the defining local system is rectangular, the displacement components are parallel to the local system and are independent of the grid point location as indicated in Figure 2-2 (a). If the local system is cylindrical, the displacement components are in the radial, tangential, and axial directions as indicated in Figure 2-2 (b). If the local system is spherical, the displacement components are in the radial, meridional, and azimuthal directions as indicated in Figure 2-2 (c).

[…]

Each geometric grid point may have a unique displacement coordinate system associated with it. The collection of all displacement coordinate systems is known as the global coordinate system. All matrices are formed and all displacements are output in the global coordinate system. The symbols T1, T2 and T3 on the printed output indicate translations in the 1, 2, and 3-directions, respectively, for each grid point. The symbols R1, R2, and R3 indicate rotations about the three axes."

MSC Nastran Reference Guide

Displacement Coordinate Systems

Consider these 2 examples.

1. For example model_a.bdf, the displacement coordinate system for each node are based on a cylindrical coordinate system. Field 7 is set to CD=1, which points to CORD2C ID=1 and is a cylindrical coordinate system. The position of the node determines the orientation of the displacement coordinate systems. For example, the T1 DOF is radial and varies depending on the position of the node.

2. For model_b.bdf, field 7 is blank, so the default displacement coordinate system is CD=0, which is the basic coordinate system and is a rectangular coordinate system. When a rectangular coordinate system is used to define the displacement coordinate system of each node, the displacement coordinate systems are parallel.

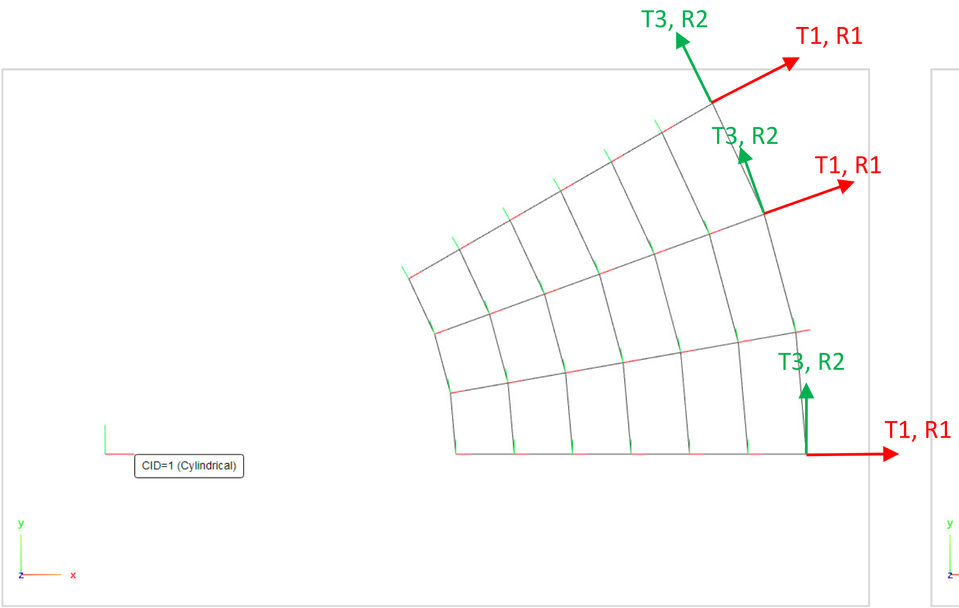
T1, T2, T3 represent the translation DOFs. R1, R2, R3 represent the rotation DOFs.

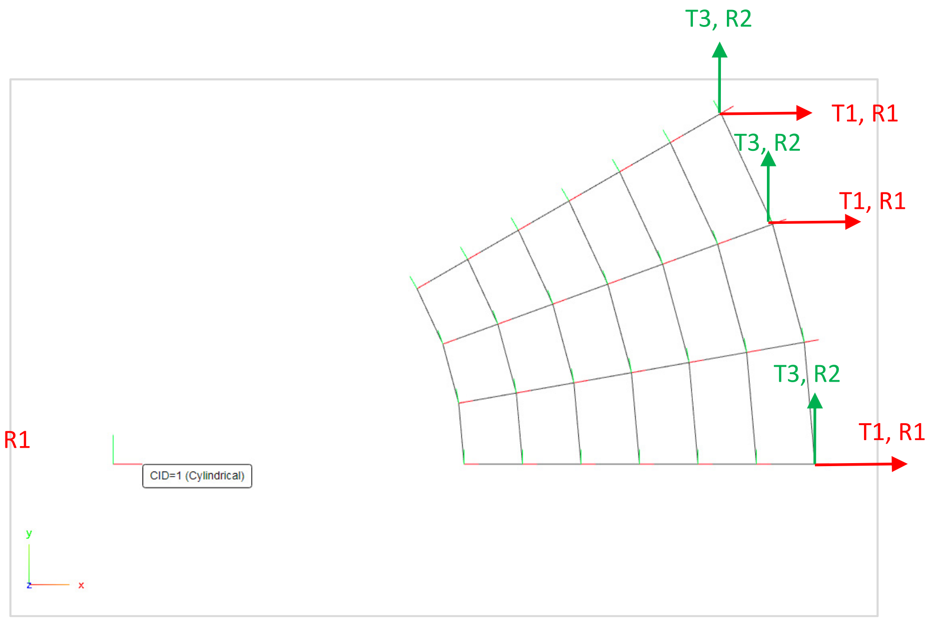
1

```
model_a.bdf
GRID     1       1       6.      0.      0.      1
GRID     2       1       7.      0.      0.      1
GRID     3       1       8.      0.      0.      1
GRID     4       1       9.      0.      0.      1
GRID     5       1       10.     0.      0.      1
GRID     6       1       11.     0.      0.      1
GRID     7       1       12.     0.      0.      1
GRID     8       1       6.      9.99998 0.      1
GRID     9       1       7.      9.99998 0.      1
GRID     10      1       8.      9.99999 0.      1
CORD2C   1               0.      0.      0.      0.
         1.      0.      0.
```

2

```
model_b.bdf
GRID     1       1       6.      0.      0.
GRID     2       1       7.      0.      0.
GRID     3       1       8.      0.      0.
GRID     4       1       9.      0.      0.
GRID     5       1       10.     0.      0.
GRID     6       1       11.     0.      0.
GRID     7       1       12.     0.      0.
GRID     8       1       6.      9.99998 0.
GRID     9       1       7.      9.99998 0.
GRID     10      1       8.      9.99999 0.
```





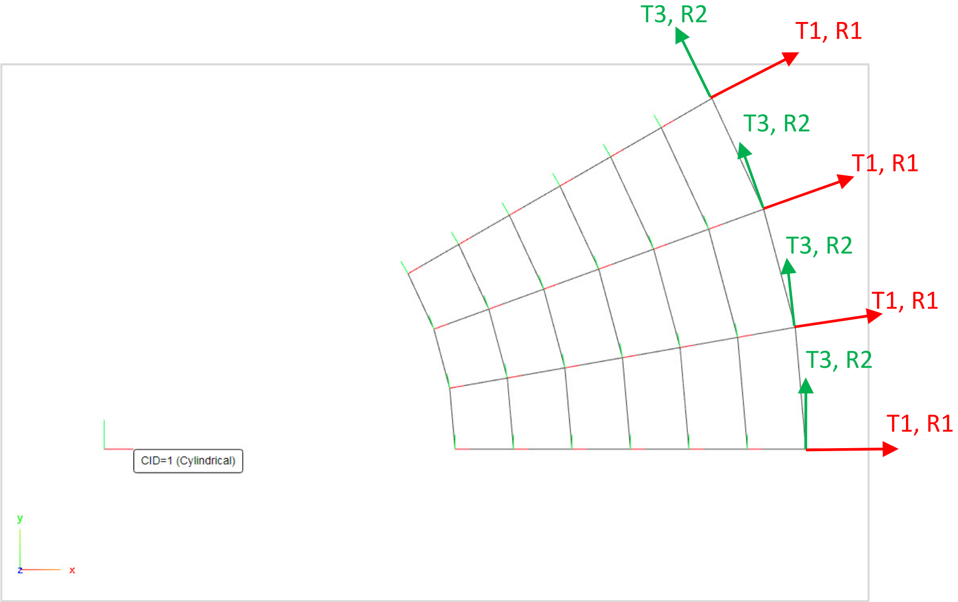
0 SUBCASE 1

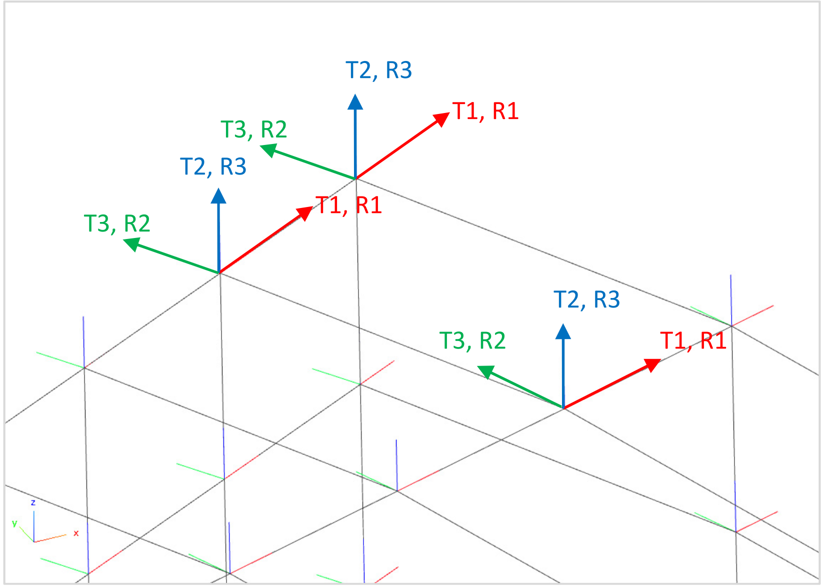
D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	3.325737E-06	0.0	0.0	0.0	0.0	0.0
2	G	3.057431E-06	0.0	0.0	0.0	0.0	0.0
[...]							
21	G	2.662868E-06	4.414948E-13	0.0	0.0	0.0	0.0
22	G	3.325737E-06	0.0	0.0	0.0	0.0	0.0
23	G	3.057431E-06	0.0	0.0	0.0	0.0	0.0
24	G	2.883740E-06	0.0	0.0	0.0	0.0	0.0
25	G	2.773202E-06	0.0	0.0	0.0	0.0	0.0

Displacement Coordinate Systems

- When inspecting the F06 results output file, the displacements are reported in the displacement coordinate system.
- T1, T2, T3 represent the translation DOFs. R1, R2, R3 represent the rotation DOFs.
    - T1, T2, T3 are in units of length, e.g. mm, m, in., etc.
    - R1, R2, R3 are typically in radians.





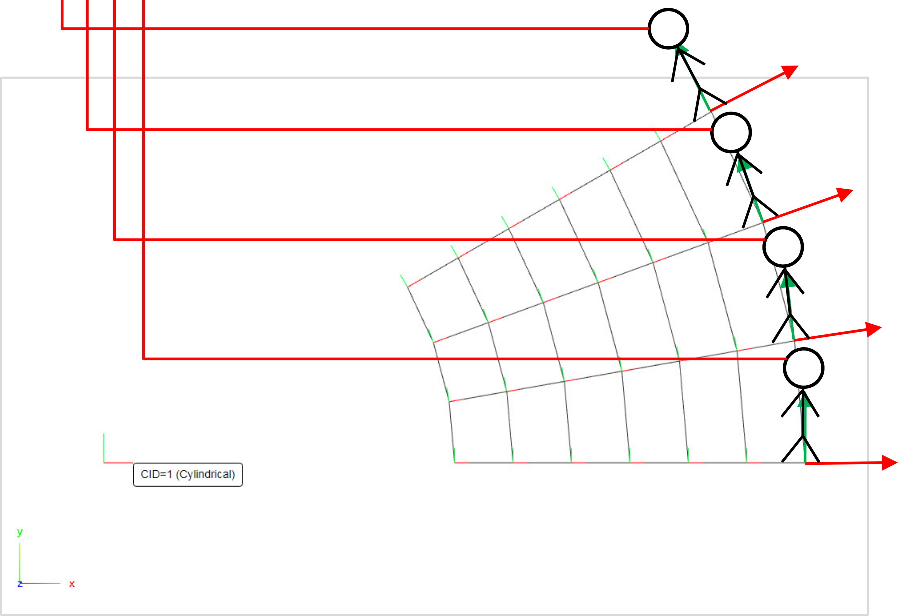
Displacement Coordinate Systems

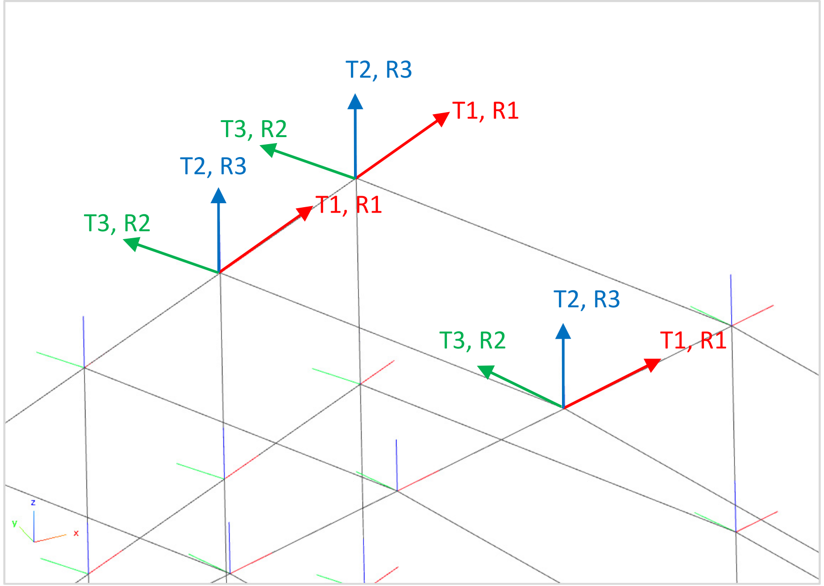
- One helpful strategy when interpreting results is to imagine an observer in each coordinate system. It is from each observer's (displacement coordinate system) perspective that results are reported.

0 SUBCASE 1

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	3.325737E-06	0.0	0.0	0.0	0.0	0.0
2	G	3.057431E-06	0.0	0.0	0.0	0.0	0.0
[...]							
21	G	2.662868E-06	4.414948E-13	0.0	0.0	0.0	0.0
22	G	3.325737E-06	0.0	0.0	0.0	0.0	0.0
23	G	3.057431E-06	0.0	0.0	0.0	0.0	0.0
24	G	2.883740E-06	0.0	0.0	0.0	0.0	0.0
25	G	2.773202E-06	0.0	0.0	0.0	0.0	0.0





Displacement Coordinate Systems

1. The Post-processor web app has an option to display the displacement coordinate system for each node. This is done by opening the FEM Label tool and clicking Display Displacement CS and Display Coordinate System.