

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

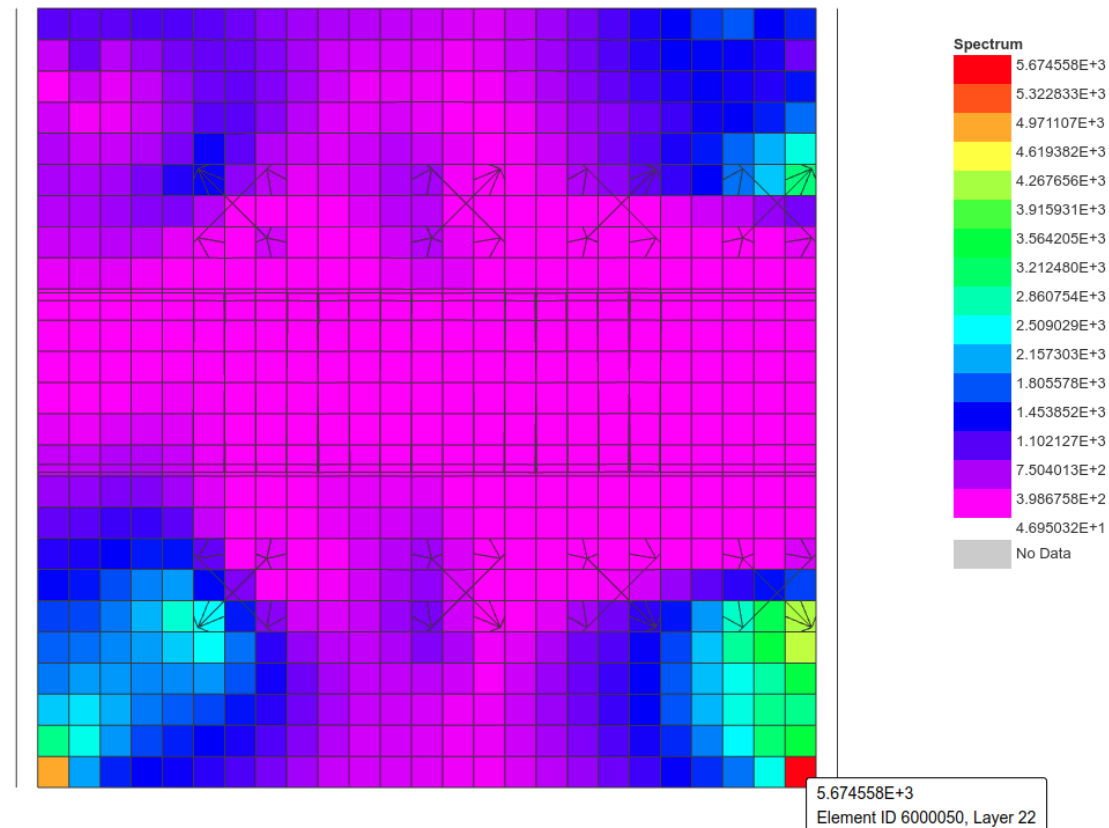
A SOL 200 WEB APP TUTORIAL

Before Starting

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

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

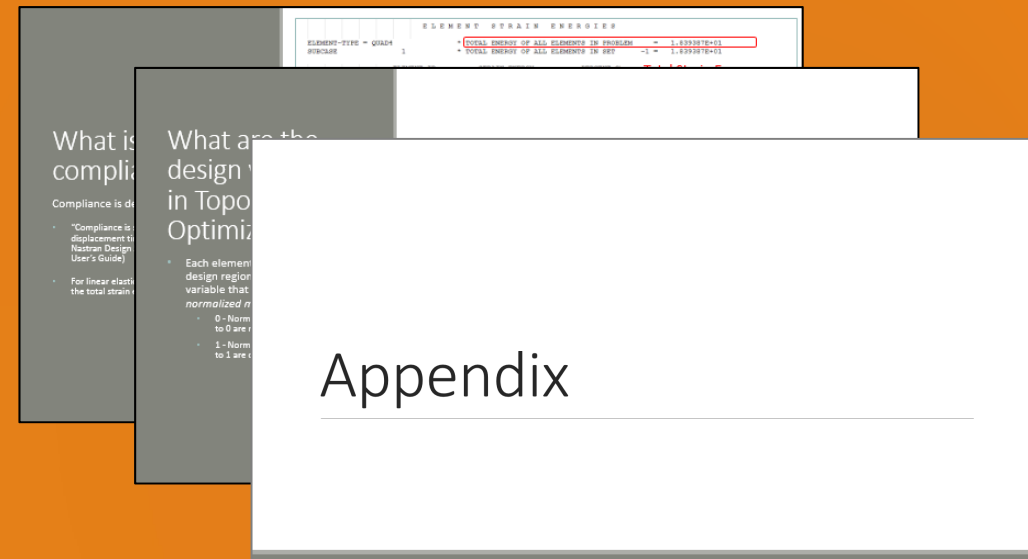
- Open the Post-processor web app and inspect Nastran results
 - Nastran Displacement Results
 - Nastran Stress Results - CQUAD4 and PSHELL
 - Nastran Composite Stress Results - CQUAD8 and PCOMP
 - Nastran Beam Force and Stress Results - CBEAM and PBMSECT
 - Nastran CBUSH Force Results - CBUSH and PBUSH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

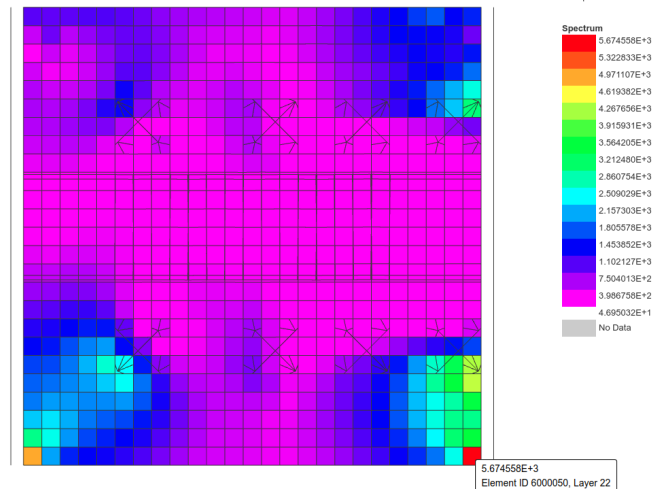
Tutorial

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.

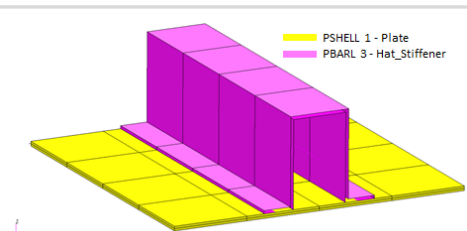
Compatibility

- 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.

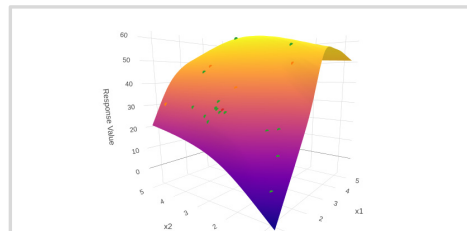
Benefits

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

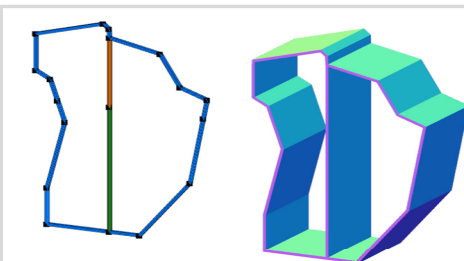
Web Apps



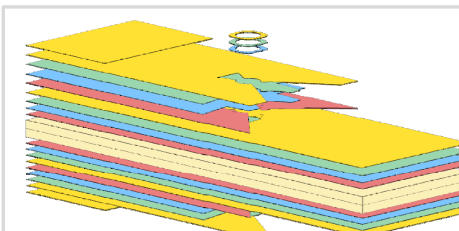
Web Apps for MSC Nastran SOL 200
Pre/post for MSC Nastran SOL 200.
Support for size, topology, topometry, topography, multi-model optimization.



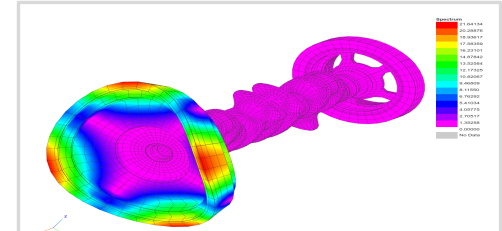
Machine Learning Web App
Bayesian Optimization for nonlinear response optimization (SOL 400)



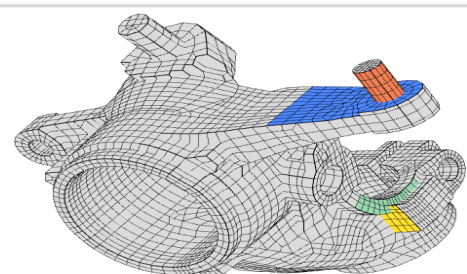
PBMSECT Web App
Generate PBMSECT and PBRSECT entries graphically



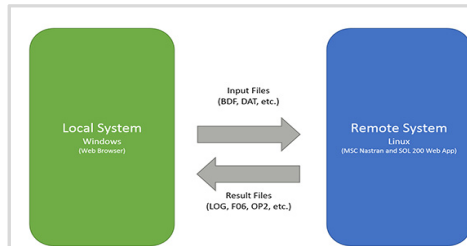
Ply Shape Optimization Web App
Optimize composite ply drop-off locations, and generate new PCOMPG entries



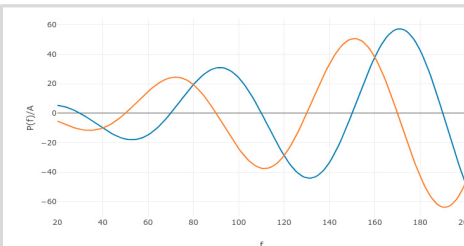
Post-processor Web App
View MSC Nastran results in a web browser on Windows and Linux



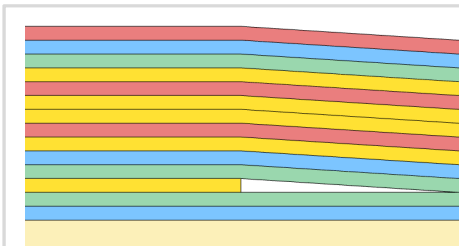
Shape Optimization Web App
Use a web application to configure and perform shape optimization.



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

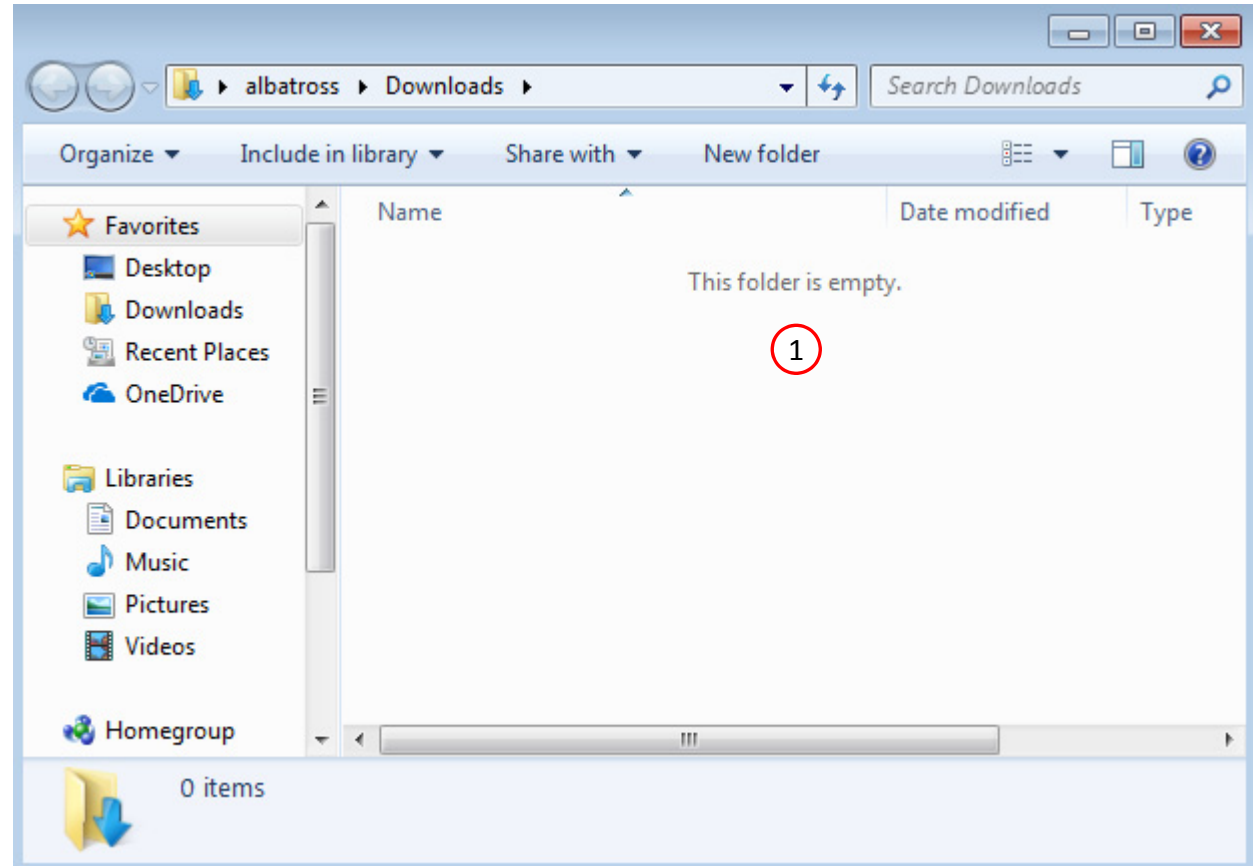


HDF5 Explorer Web App
Create graphs (XY plots) using data from the H5 file

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.

The screenshot displays the SOL 200 Web App interface. At the top, it says "SOL 200 Web App" and "Select a web app to begin". Below this, there are five main categories of tools, each with a representative image:

- Optimization for SOL 200:** Shows a 3D model of a mechanical part in two states: "Before" and "After" optimization.
- Multi Model Optimization:** Shows a 3D model of a part with arrows pointing to a graph showing multiple optimization curves.
- Machine Learning | Parameter Study:** Shows four different mesh deformation or stress distribution visualizations.
- HDF5 Explorer:** Shows a line graph with multiple data series plotted against a parameter.
- Viewer:** Shows a 3D cube with a color gradient from red to blue, representing a field of values.

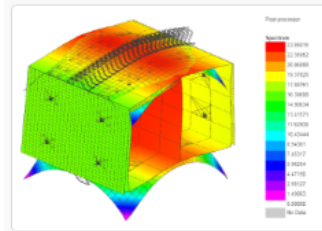
At the bottom of the interface, there is a navigation bar with two items:

- 1 Tutorials and User's Guide** (highlighted with a red box and a circled '1')
- Full list of web apps

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

1

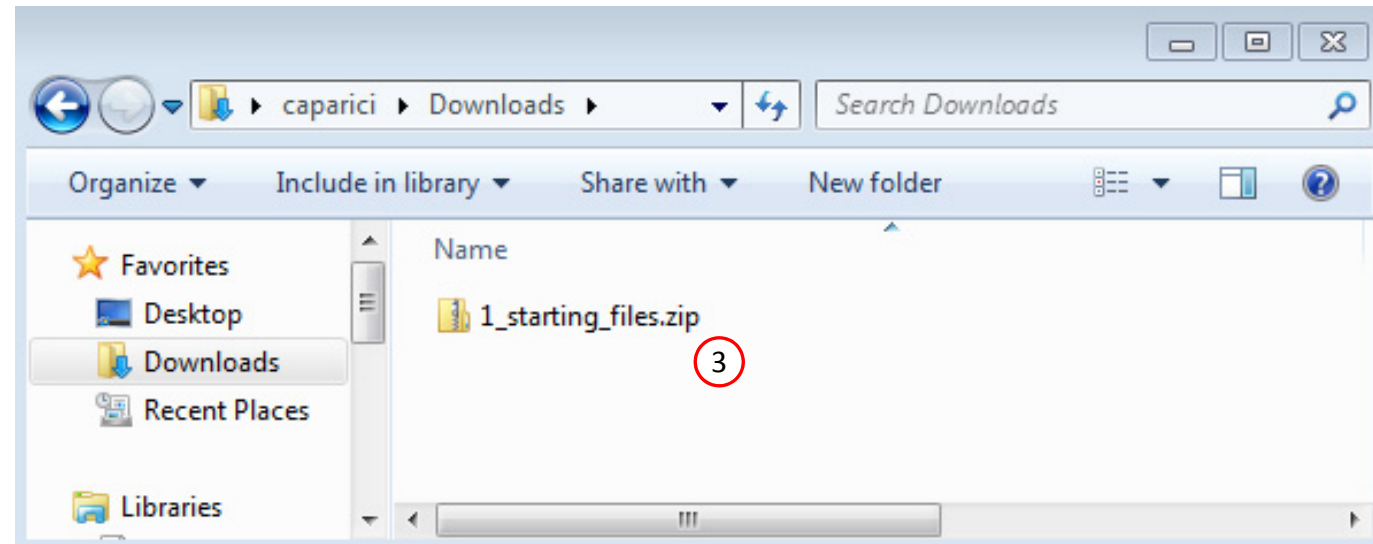
This tutorial is an introduction to Nastran results.

- 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](#)

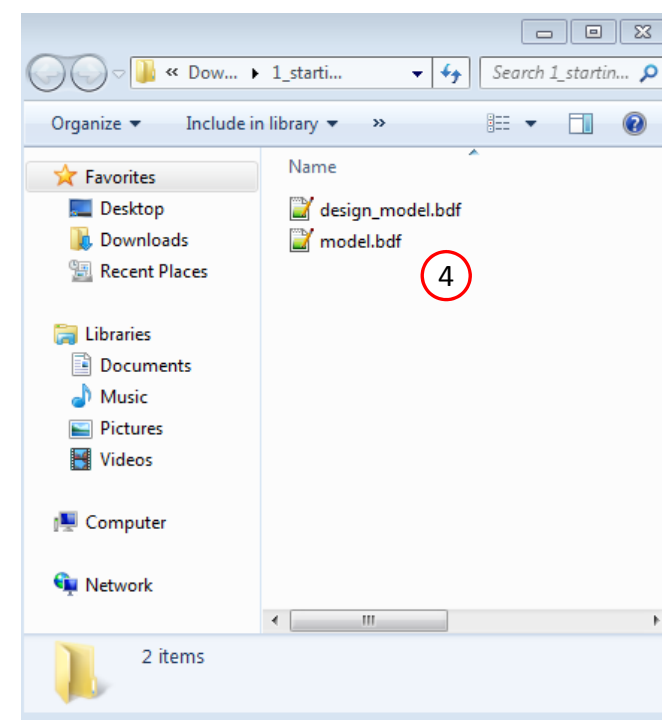
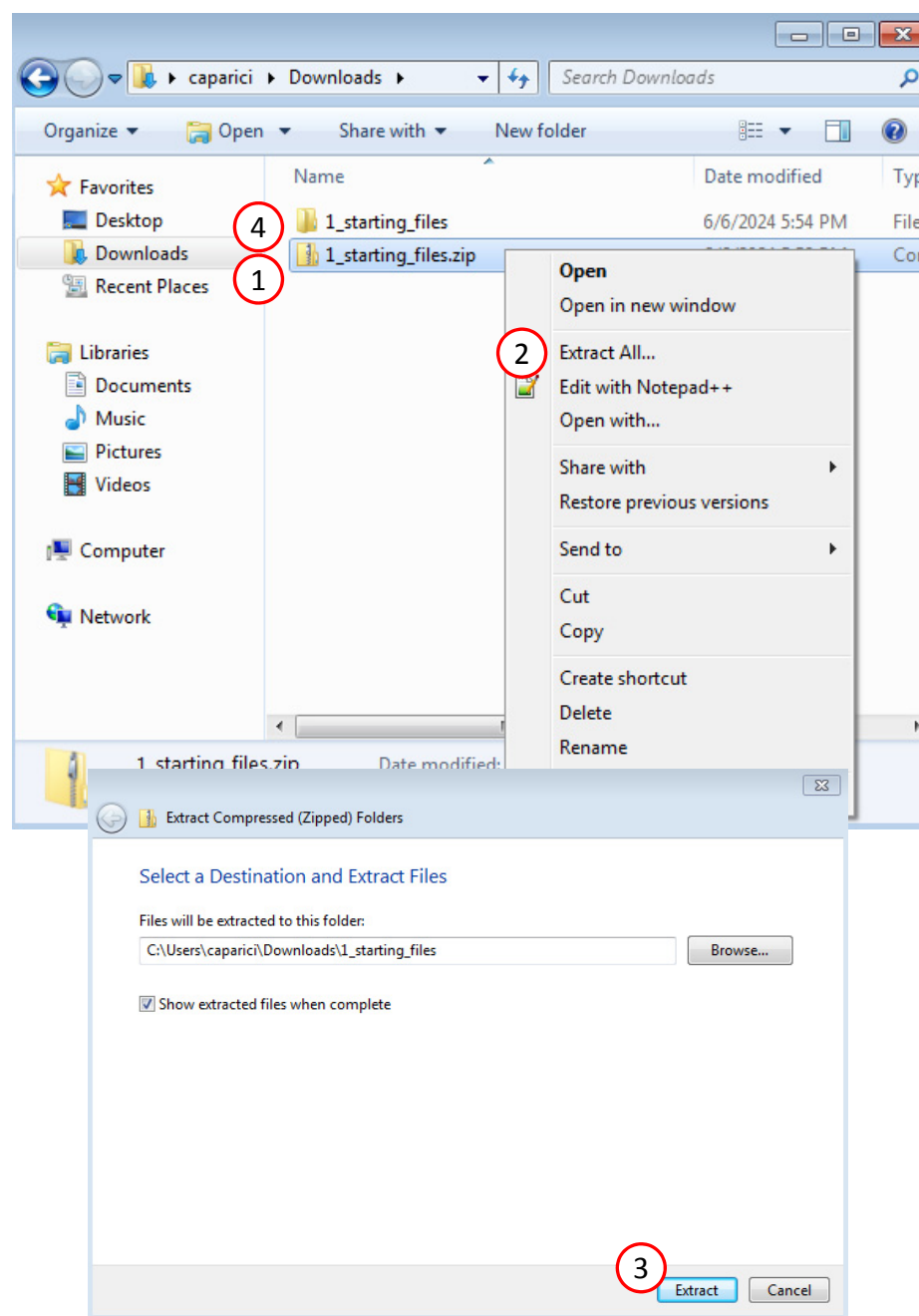
2



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

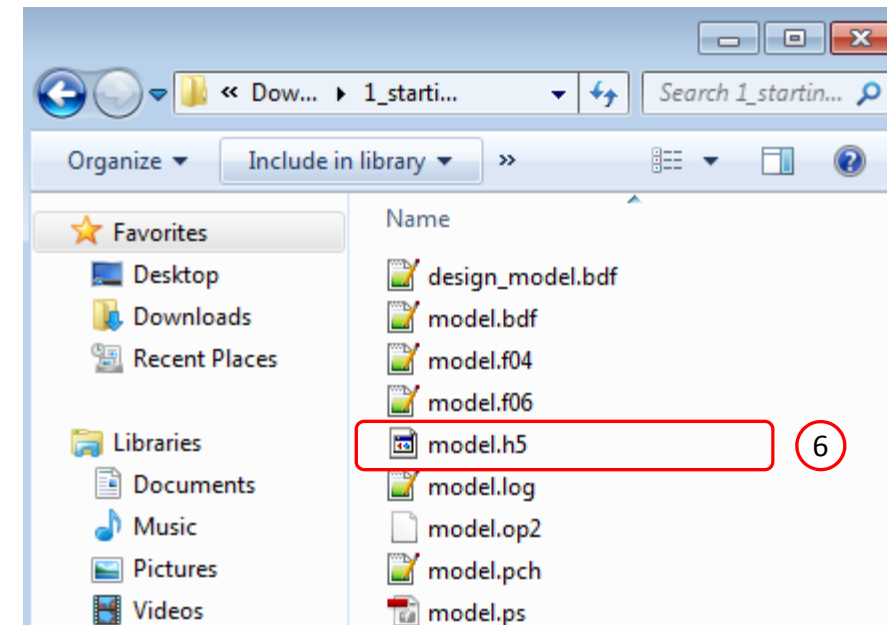
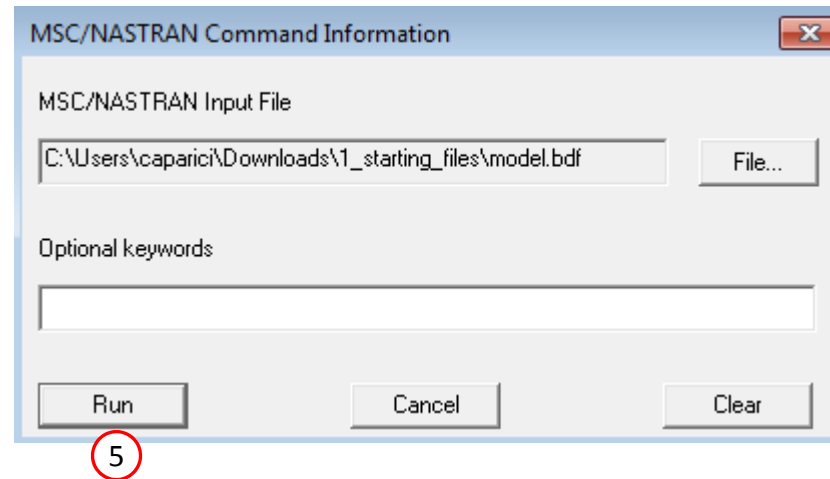
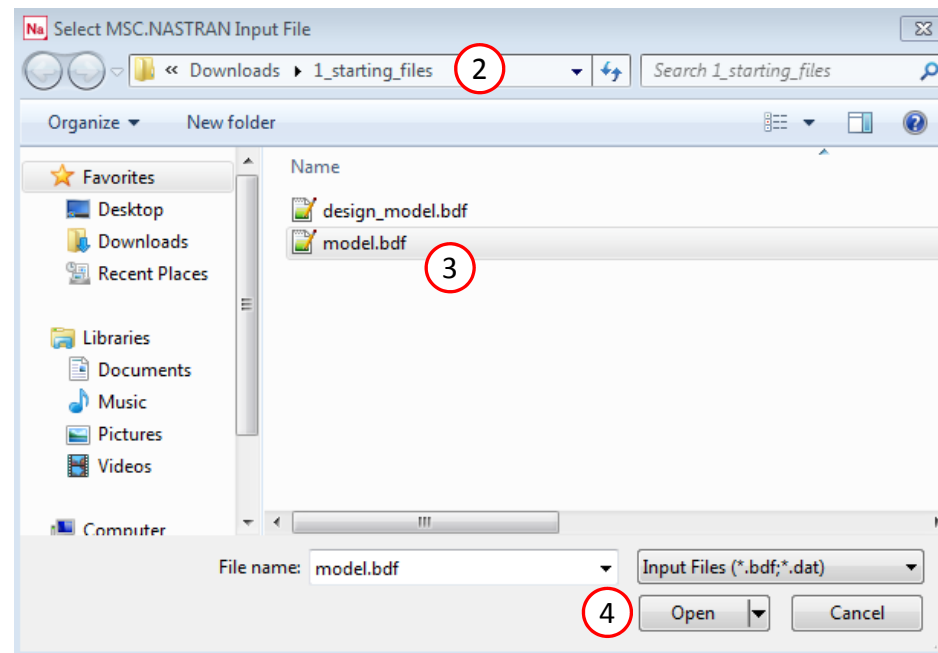
- This example is using a previously created design model. The design model is a model that has been converted to SOL 200 and contains bulk data entries describing the optimization problem statement, e.g. variables, objective and constraints.



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 screenshot displays the SOL 200 Web App interface. At the top, it says "SOL 200 Web App" and "Select a web app to begin". Below this, there are five main categories of web apps, each with a representative image:

- Optimization for SOL 200:** Shows a 3D model of a mechanical part in two states: "Before" and "After" optimization.
- Multi Model Optimization:** Shows a 3D model of a part with arrows pointing to a graph showing multiple optimization curves.
- Machine Learning | Parameter Study:** Shows four small images representing different mesh or parameter study results.
- HDF5 Explorer:** Shows a line graph with multiple data series.
- Viewer:** Shows a 3D cube with a color gradient from red to blue, representing a visualization of results. A red circle with the number "1" is overlaid on this icon.

At the bottom of the interface, there are two additional options: "Tutorials and User's Guide" and "Full list of web apps".

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.

The screenshot illustrates the file upload process in a software application. The interface is divided into a Main Panel on the left and a File Upload window on the right. The Main Panel has sections for File Upload, Camera, Misc., View, and Tools. The File Upload window shows a progress bar for '1. Select files' (2 files selected) and '2. Upload files' (100% complete). A Windows File Explorer window is open, showing the 'Downloads' folder with the '1_starting_files' subfolder selected. The '1_starting_files' folder contains four files: 'design_model.bdf', 'model.bdf', 'model_bm1_01.bdf', and 'model_bm4_01.bdf'. The 'design_model.bdf' file is selected. The File name field in the File Explorer shows 'design_model.bdf' and 'model.bdf'. The 'Open' button is highlighted.

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.

Main Panel

File Upload

File Upload

Camera

First Person

Misc.

Center Model
Fit Model
Background Color

View

Front

Rear

Top

Bottom

Left

Iso 1

Iso 2

Iso 3

Iso 1B

Right

Rotate
Rotate

Tools

Model Display Panel
FEM Label

Model Display Panel

Hide All
Reset Table

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
Search	Search	S	▼	▼	S	S	Se	Search	▼	▼
			<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
PBUSH	101		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
PBUSH	201		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
PCOMP	11		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
					1		45°		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
					2		90°		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
					3		-45°		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
					4		-45°		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
					5		0°		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
RBE3	PFAST=6000022		<input checked="" type="checkbox"/>							
RBE3	PFAST=6000023		<input checked="" type="checkbox"/>							
RBE3	PFAST=6000024		<input checked="" type="checkbox"/>							
RBE3	PFAST=6000025		<input checked="" type="checkbox"/>							

100
200
500
1000

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.

Main Panel

First Person

Misc.

Center Model (2)

Fit Model (1)

Background Color (3)

View

Front, Rear, Top, Bottom, Left, Right, Iso 1, Iso 2, Iso 3, Iso 1B

Rotate, Rotate

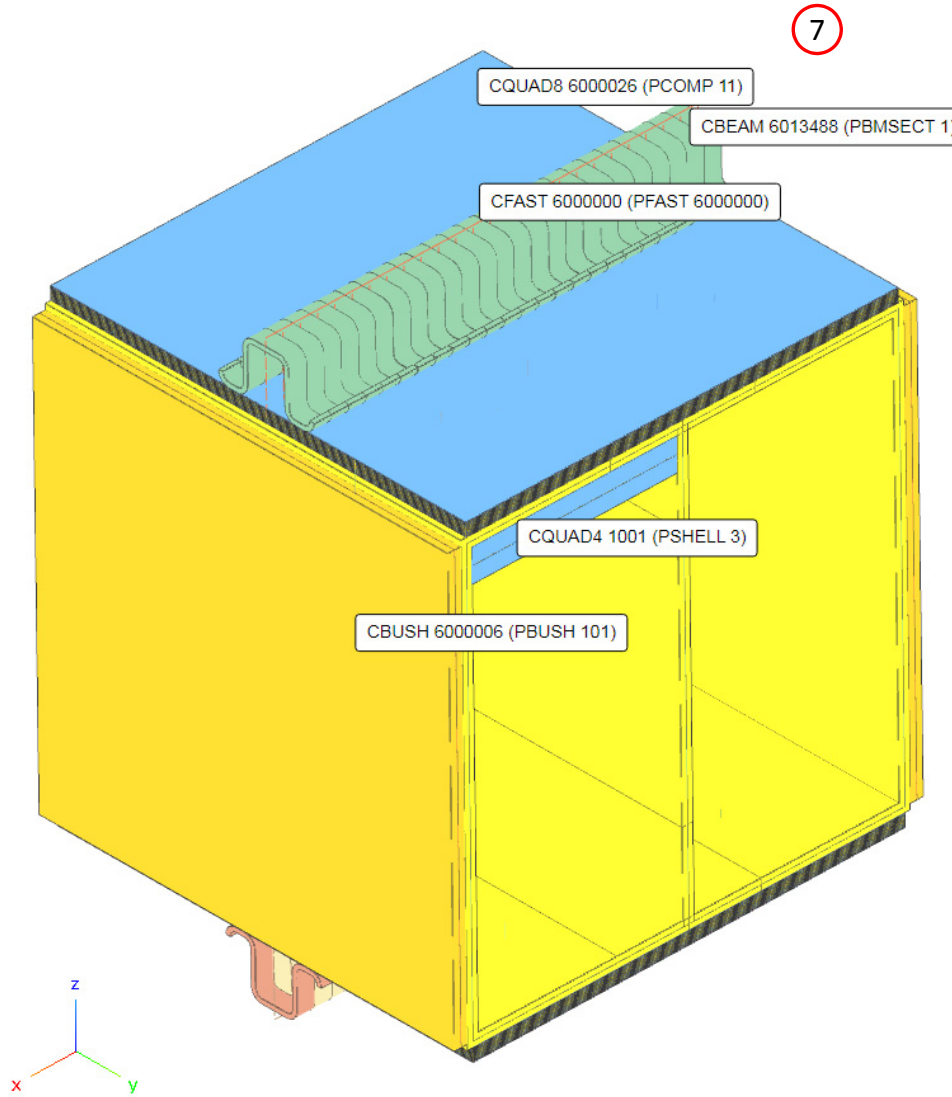
Tools

Model Display Panel (8)

FEM Label (4)

Results

Post-processor



FEM Label

Element IDs (5)

6000000, 6000006, 6013488, 1001, 6000026

Display associated GRID IDs

Display parent property IDs (6)

GRID IDs

Pick Mode

Enter Pick Mode

Options

Reset Labels

Display Associated FEM Entities

Other

Display Displacement CS (DOFs)

Display Coordinate Systems

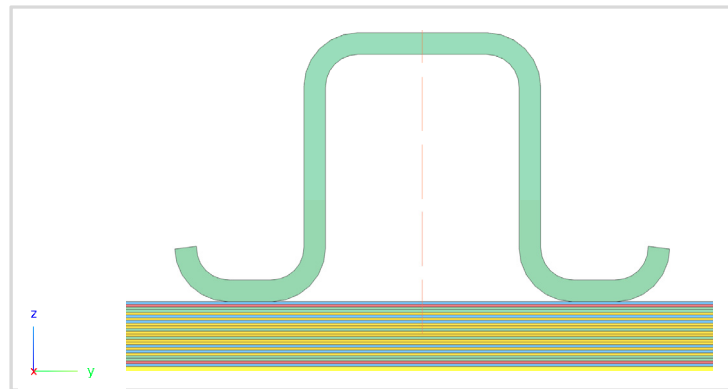
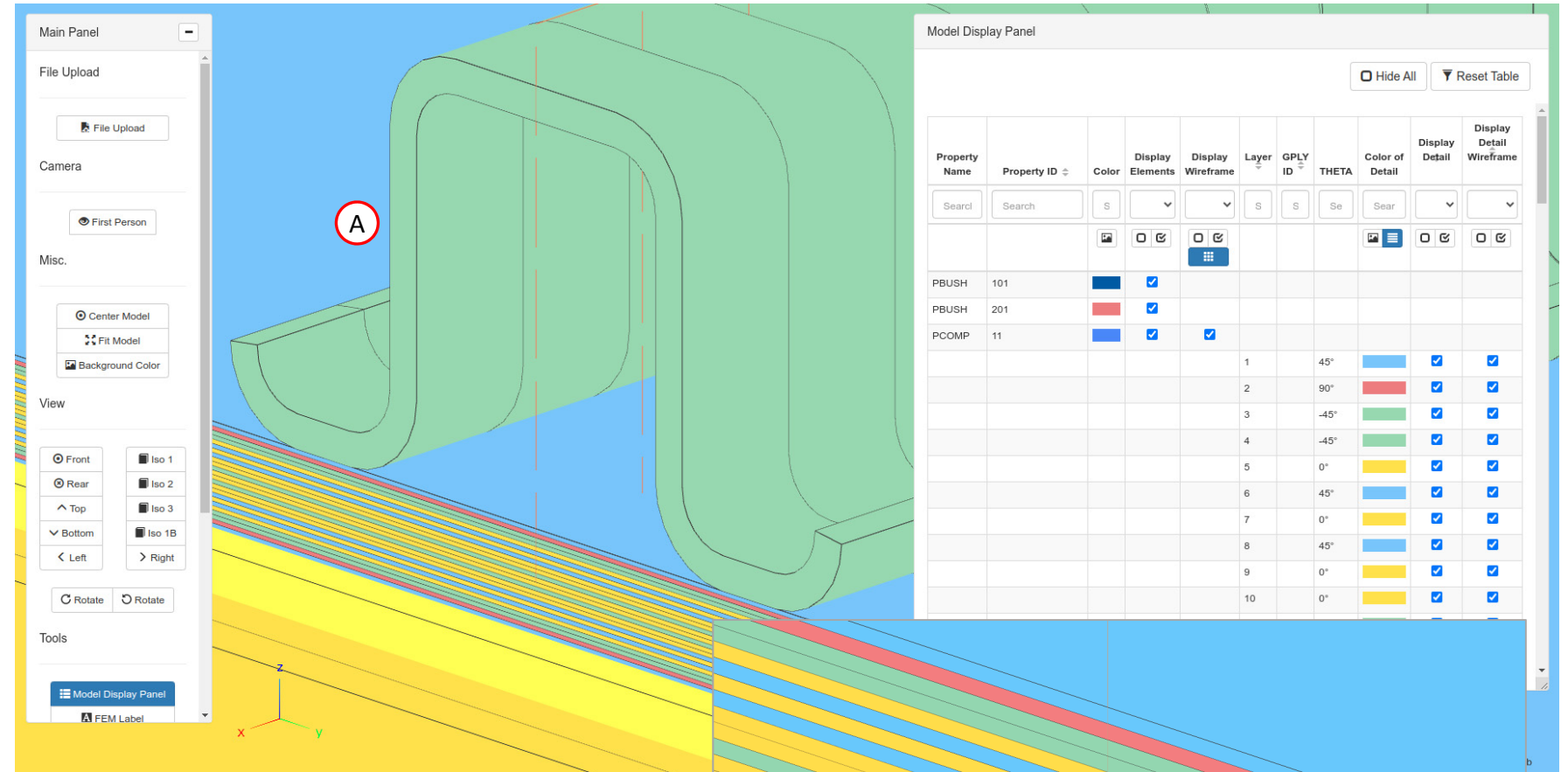
Display Orientation Vectors

A maximum of 500 labels is supported

Beam Cross Section and Composite Laminate Layers

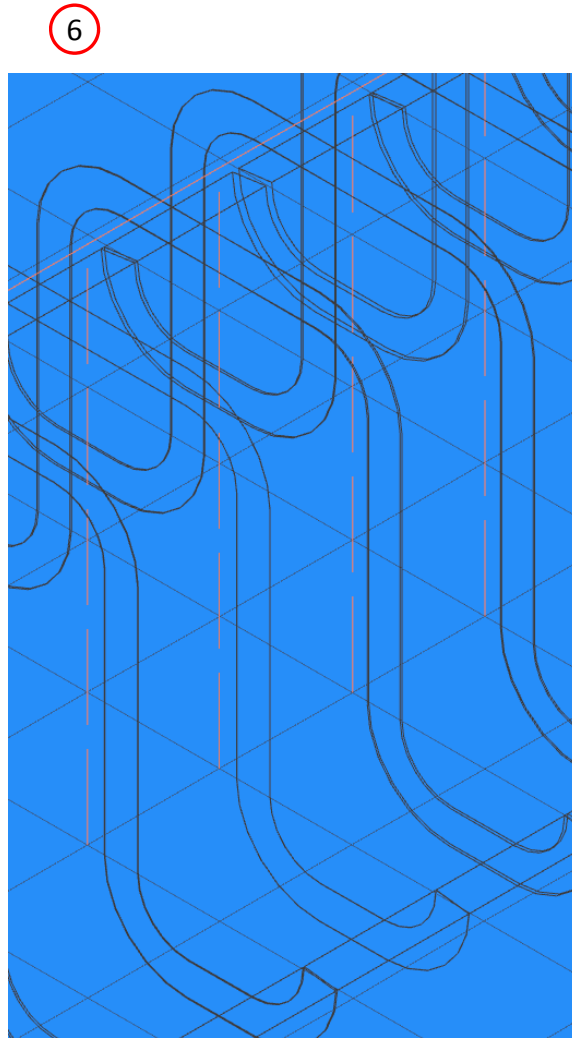
Close inspection of the model reveals some interesting details.

- 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: pbmse
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)



Model Display Panel

Hide All Reset Table

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
Search	Search	Se	▼	▼	S	S	Se	Search	▼	▼
PBUSH	101		<input checked="" type="checkbox"/>						<input type="checkbox"/>	<input type="checkbox"/>
PBUSH	201		<input checked="" type="checkbox"/>						<input type="checkbox"/>	<input type="checkbox"/>
PCOMP	11		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					<input type="checkbox"/>	<input type="checkbox"/>

Model Display Panel

Reset Table

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
pbmse	Search	Se	▼	▼	Se	Se	Sea	Search	▼	▼
PBMSECT	4		<input checked="" type="checkbox"/>						<input type="checkbox"/>	<input checked="" type="checkbox"/>
PBMSECT	5		<input checked="" type="checkbox"/>						<input type="checkbox"/>	<input checked="" type="checkbox"/>
PBMSECT	1		<input checked="" type="checkbox"/>						<input type="checkbox"/>	<input checked="" type="checkbox"/>

Upload Files

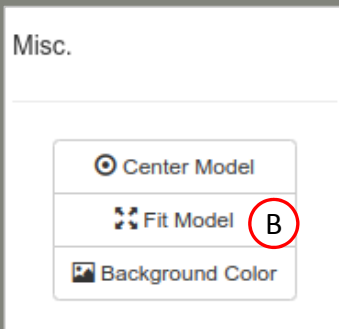
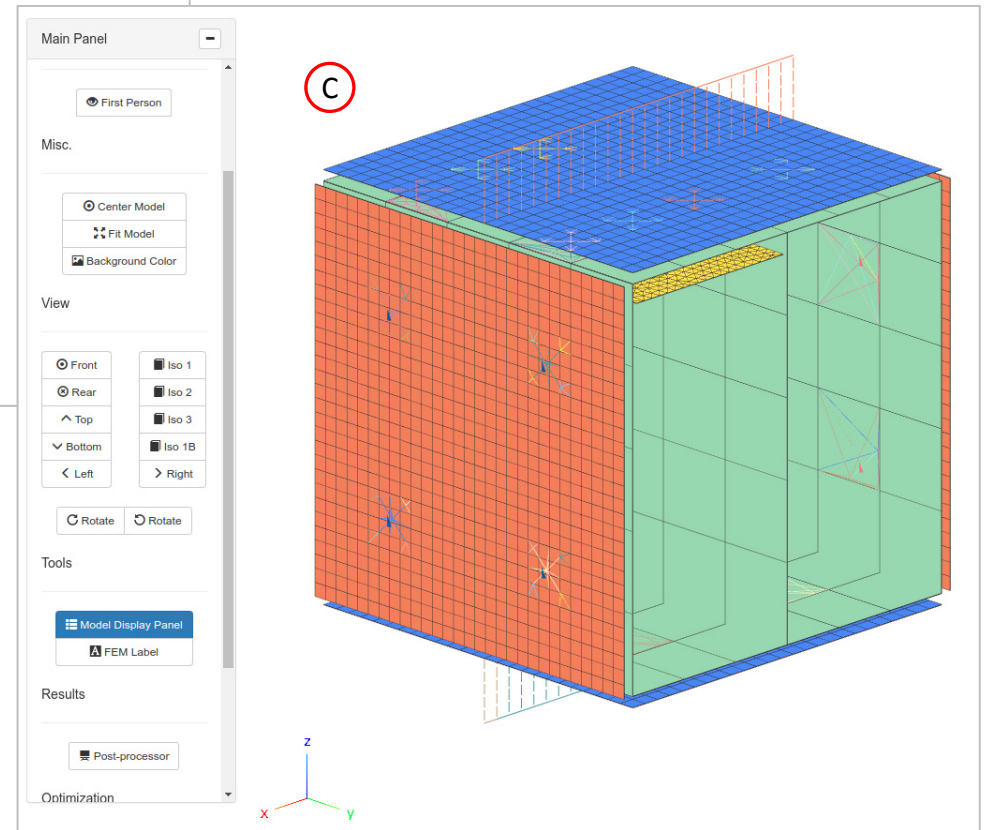
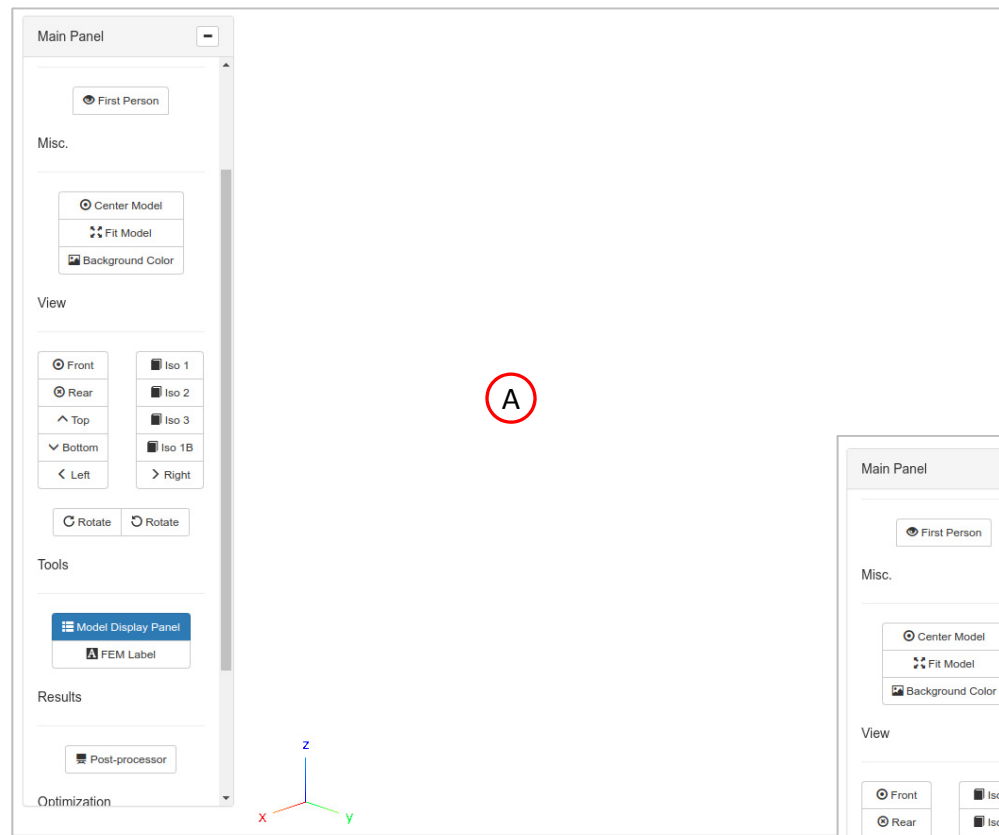
1. Click File Upload
2. Click H5
3. Click Select files
4. Navigate to the directory 1_starting_files
5. Select the following files:
 - model.h5
6. Click Open
7. Click Upload files
8. Wait for both progress bars to reach 100% completion, then continue

- 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.

The screenshot illustrates the file upload process in a web-based post-processor. The interface is divided into several panels: a left sidebar with navigation and camera controls, a central 3D model view, and a right-side control panel. The 'File Upload' section is active, showing the selection of an H5 file. The 'Open' dialog box is used to navigate to the '1_starting_files' directory and select the 'model.h5' file. The progress bars indicate that the upload and loading processes are complete.

Before Continuing

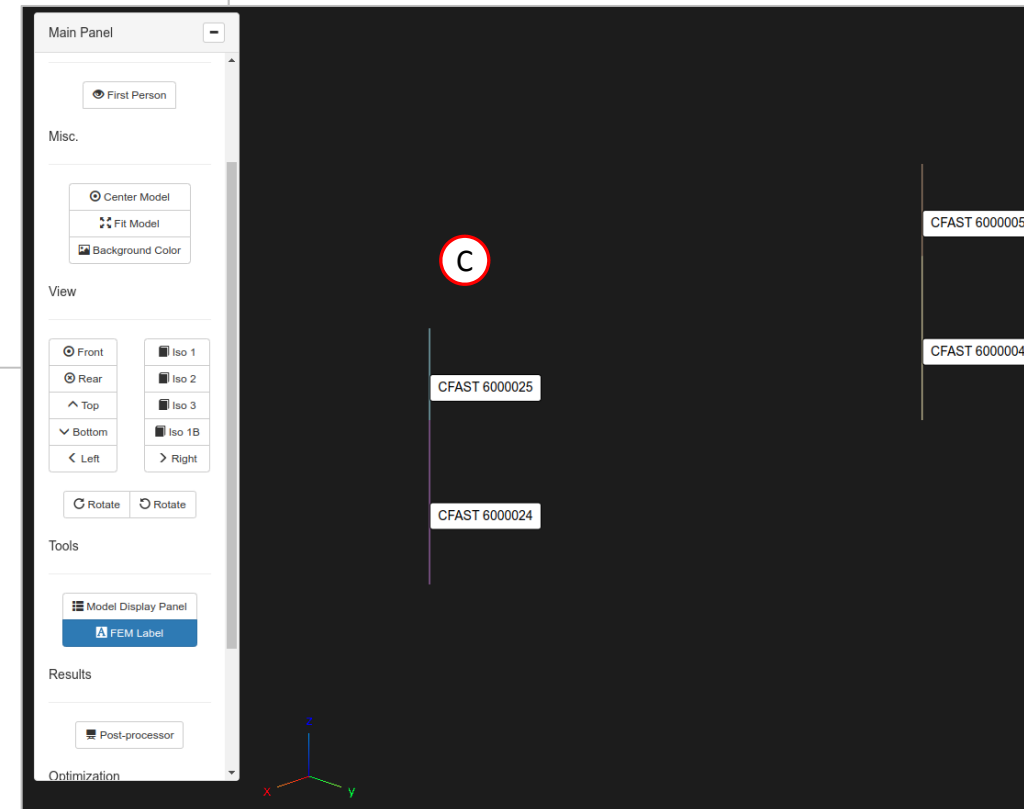
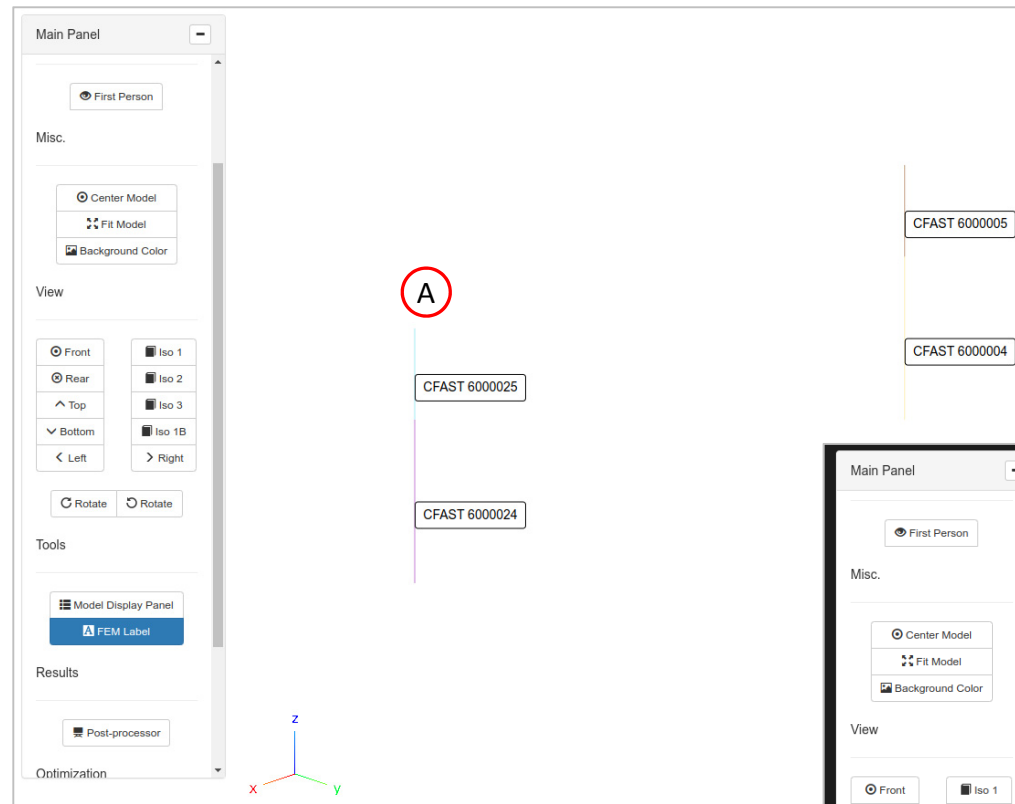
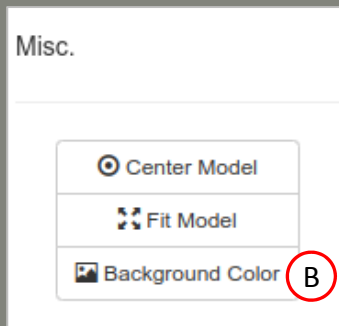
- A. As you hide and show elements and details, and as you navigate the model, you might reach a point where disorientation occurs.
- B. Click Fit Model.
- C. This will restore the model to a visible state.



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

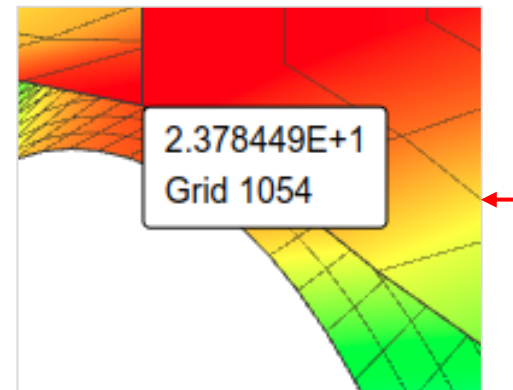
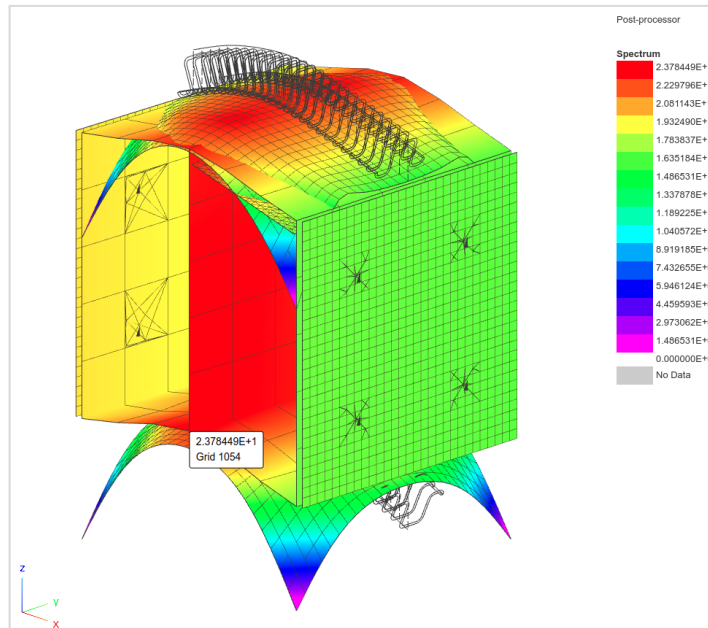
F06 File

0

SUBCASE 1

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

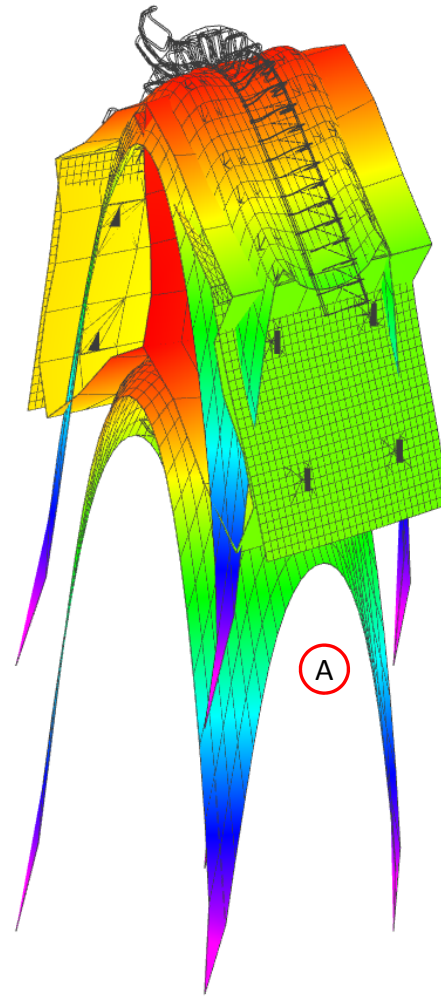
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1054	G	4.533402E-02	1.694495E+00	2.378449E+01	-2.665313E-02	1.776151E-02	2.036122E-03



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

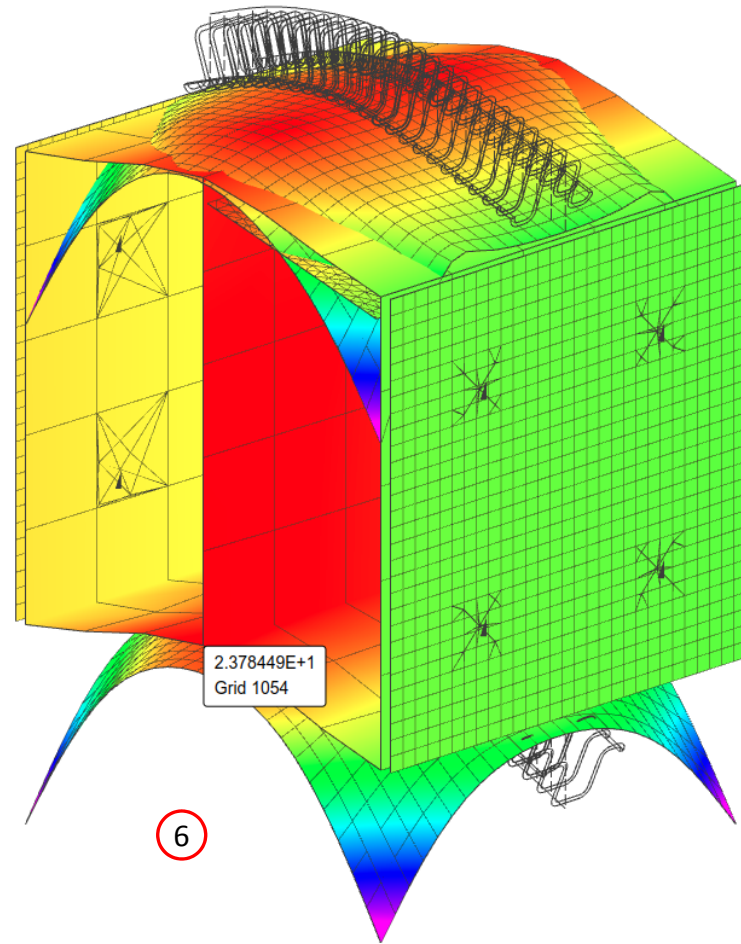
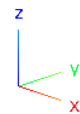
A. The scale factor is used to make the displacements more pronounced and noticeable. The default scale factor is 10 and in this case leads to a significantly deformed shape. On the next page, the scale factor is reduced.



Selected	DOMAIN_ID	DESIGN_CYCLE	SUBCASE
<input checked="" type="checkbox"/>	1	0	1
<input type="checkbox"/>	2	0	2
<input type="checkbox"/>	3	0	3
<input type="checkbox"/>	9	2	1
<input type="checkbox"/>	10	2	2
<input type="checkbox"/>	11	2	3

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



Post-processor

Spectrum

2.378449E+1
2.229796E+1
2.081143E+1
1.932490E+1
1.783837E+1
1.635184E+1
1.486531E+1
1.337878E+1
1.189225E+1
1.040572E+1
8.919185E+0
7.432655E+0
5.946124E+0
4.459593E+0
2.973062E+0
1.486531E+0
0.000000E+0
No Data

Configure Plots **1**

Marker Plot
 Display Color Plot
 Display Shape Plot

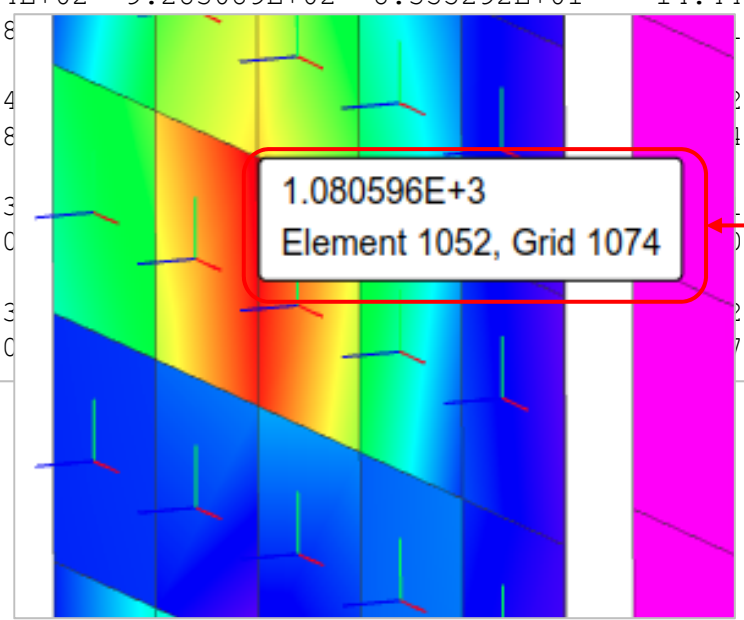
	Fringe Plot	Shape Plot
Dataset	NODAL/DISPLACEMENT	NODAL/DISPLACEMENT
Field	X - X component Y - Y component Z - Z component 2 RX - RX component RY - RY component RZ - RZ component	X - X component Y - Y component Z - Z component 3 RX - RX component RY - RY component RZ - RZ component
Coordinate System	CID=0 (Basic)	CID=0 (Basic)
Scale Factor		4 2
Domain	DESIGN_CYCLE 0 SUBCASE 1	DESIGN_CYCLE 0 SUBCASE 1
Maximum and Minimum		
Maximum	2.378449E+1 at Grid 1054	2.378449E+1 at Grid 1054
Minimum	0.000000E+0 at Grid 4000125	0.000000E+0 at Grid 4000049
Maximum Labels	1 5 2 3 4 5 6	

Nastran Stress Results - CQUAD4 and PSHELL

Goal: Display PSELL stresses

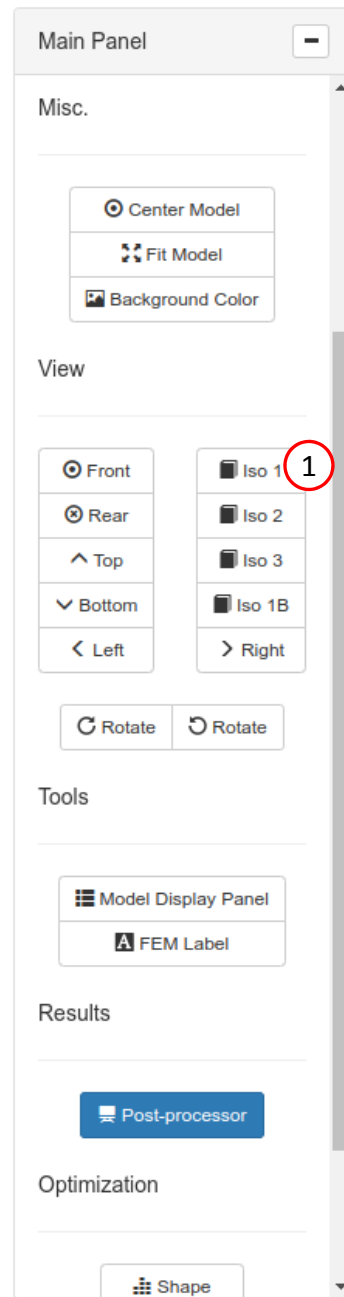
F06 File
 S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT		FIBER		STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			
ID	GRID-ID	DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	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			11.94	9.477564E+02	6.042814E+02	8.310756E+02
	1068		-1.000000E+00	-6.255414			12.77	-6.136088E+02	-1.210584E+03	1.048430E+03
			1.000000E+00	6.263418			11.41	1.206677E+03	6.141238E+02	1.045069E+03
	1074		-1.000000E+00	-8.623293			11.05	-8.427159E+02	-1.218265E+03	1.080596E+03
			1.000000E+00	8.638710			10.22	1.214672E+03	8.436585E+02	1.078155E+03
	1073		-1.000000E+00	-8.623293			12.70	-8.054891E+02	-9.851490E+02	9.087379E+02
			1.000000E+00	8.638710			11.93	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



Acquire Dataset **2**

Select Dataset

ELEMENTAL/STRESS/QUAD_CN **3**

Select Domain

Reset Table Uncheck visible boxes Check visible boxes

Selected	DOMAIN_ID	DESIGN_CYCLE	SUBCASE
		0 2	1 2 3
<input type="checkbox"/>	1	0	1
<input type="checkbox"/>	2	0	2
<input type="checkbox"/>	3	0	3
<input type="checkbox"/>	9	2	1
<input type="checkbox"/>	10	2	2
4 <input checked="" type="checkbox"/>	11	2	3

5

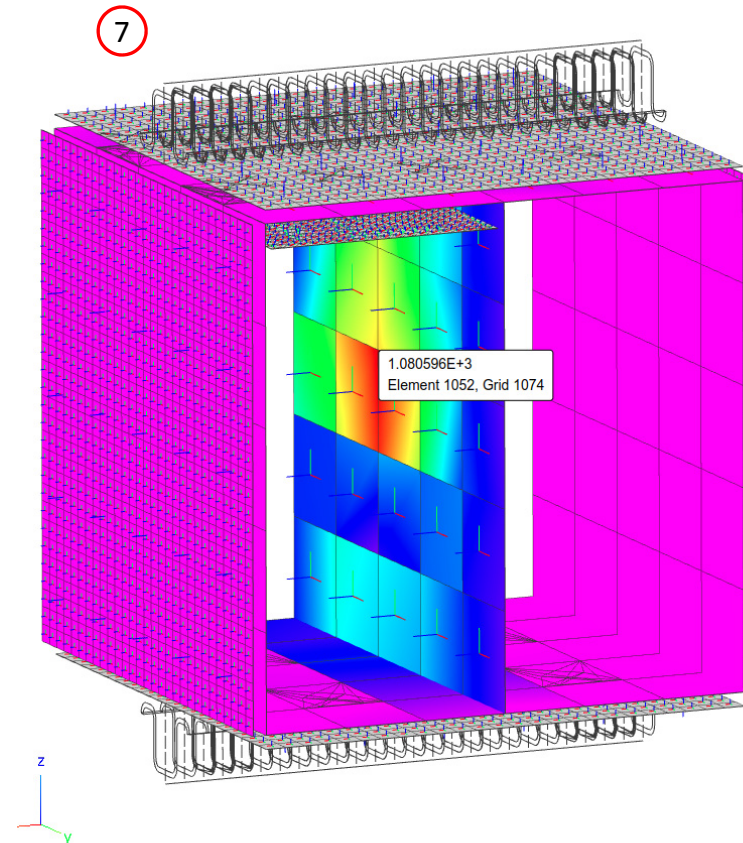
Acquisition complete and successful

10 20 50

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.



Post-processor

Spectrum

1.080596E+3
1.013064E+3
9.455308E+2
8.779981E+2
8.104654E+2
7.429327E+2
6.754000E+2
6.078674E+2
5.403347E+2
4.728020E+2
4.052693E+2
3.377366E+2
2.702039E+2
2.026712E+2
1.351385E+2
6.760580E+1
7.310496E-2
No Data

Configure Plots **1**

Display Color Plot Display Shape Plot **2**

Fringe Plot

Dataset: ELEMENTAL/STRESS/QUAD_CN

Field: MAX(VM1,VM2) - Max. of VM1 and VM2** **3**

Coordinate System: [Default]

Location: CORNER **4**

Domain: DESIGN_CYCLE 2
SUBCASE 3

Display Directions: Yes **5**

Maximum and Minimum

Maximum: 1.080596E+3 at Element 1052, Grid 1074

Minimum: 7.310496E-2 at Element 6011993, Grid 4012338

Maximum Labels: 1 **6**, 2, 3, 4, 5

Minimum Labels: 1, 2, 3, 4, 5

Nastran Stress Results - CQUAD4 Corner and Center Output

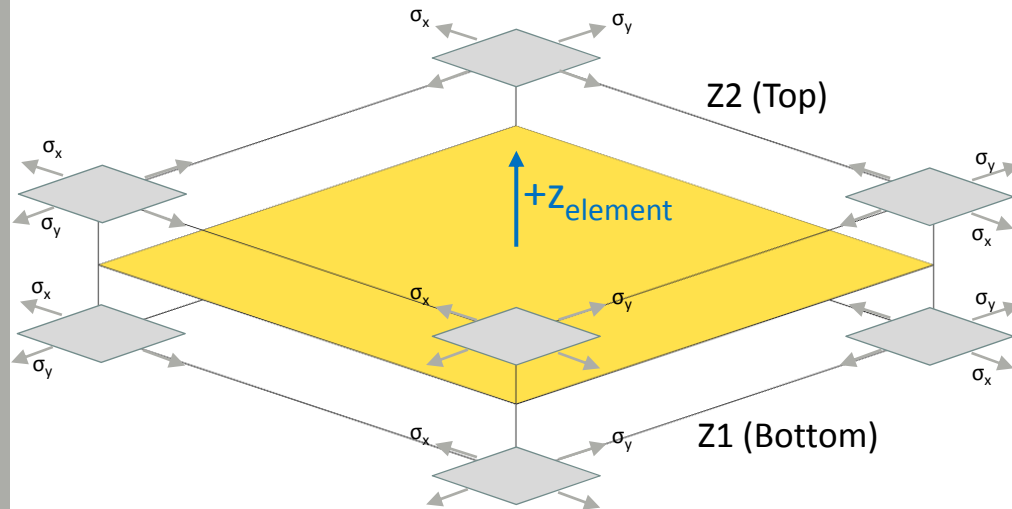
1. Refer to the red box.

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.

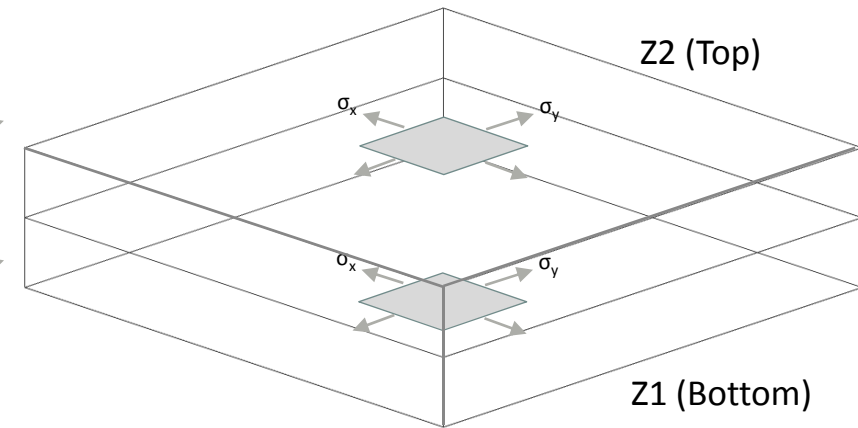
- NORMAL-X: σ_x
- NORMAL-Y: σ_y
- SHEAR-XY: τ_{xy}

F06 File
 STRESSES IN QUADRILATERAL ELEMENTS (QUAD4) OPTION = BIL

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (Z ANGLE MAJOR)		
			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.

Configure Plots

Display Color Plot Display Shape Plot

Fringe Plot

Dataset: ELEMENTAL/STRESS/QUAD_CN

Field: X1 - Normal in x at Z1 (Bottom)

Coordinate System: CORNER

Domain: DESIGN_CYCLE 2
SUBCASE 3

Display Directions: Yes

Maximum and Minimum

Maximum: 2.604272E+2 at Element 1062, Grid 1038

Minimum: -8.623293E+2 at Element 1052, Grid 1074

Minimum Labels: 1

Minimum Labels: 1

-8.623293E+2
Element 1052, Grid 1074

CQUAD4 Corner Output

Configure Plots

Display Color Plot Display Shape Plot

Fringe Plot

Dataset: ELEMENTAL/STRESS/QUAD_CN

Field: X1 - Normal in x at Z1 (Bottom)

Coordinate System: CENTER

Domain: DESIGN_CYCLE 2
SUBCASE 3

Display Directions: Yes

Maximum and Minimum

Maximum: 1.353330E+2 at Element 1062

Minimum: -7.439354E+2 at Element 1052

Minimum Labels: 1

Minimum Labels: 1

-7.439354E+2
Element 1052

CQUAD4 Center Output

Nastran Stress Results - CQUAD4 and PSHELL

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

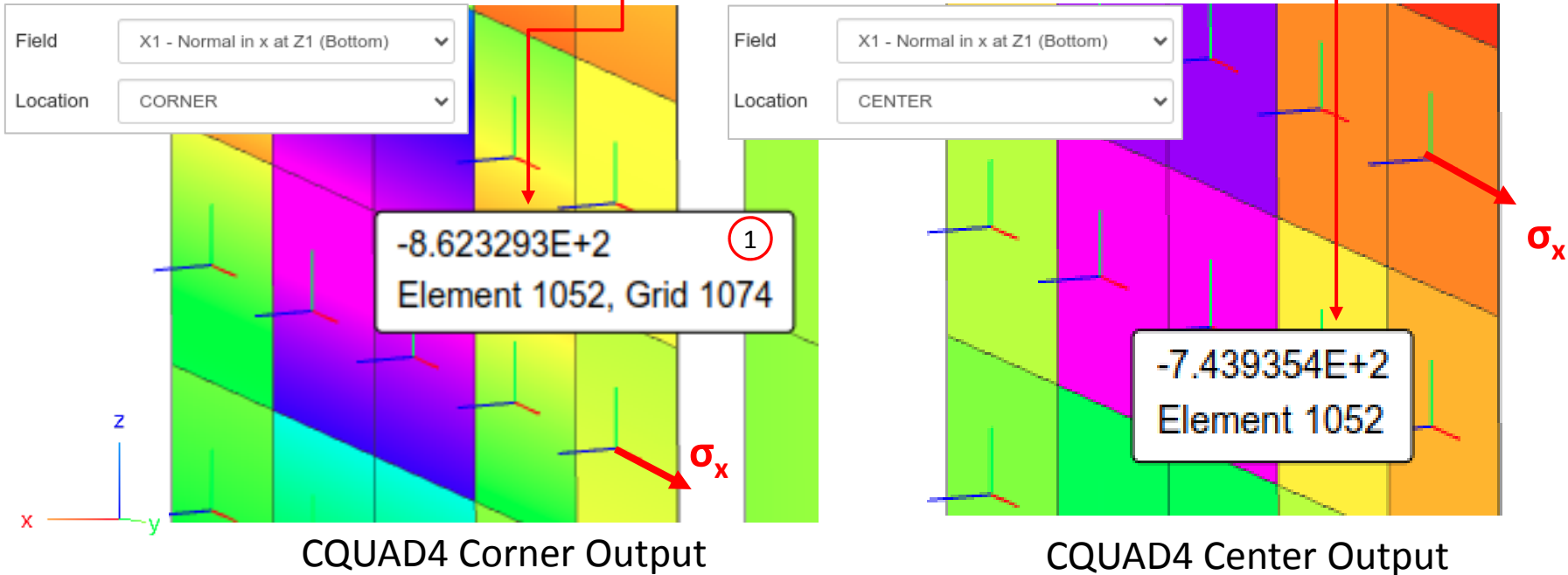
As shown to the right, the X1 field (NORMAL-X/ σ_x) is displayed for the bottom (Z1) of the thickness for both the center and corner locations.

- 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.

Display Directions

F06 File
 STRESSES IN QUADRILATERAL ELEMENTS (QUAD4) OPTION = BIL

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM			PRINCIPAL STRESSES (Z ANGLE MAJOR)	
			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



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

Field

The screenshot shows a dropdown menu for field selection. The menu is currently set to 'FD1 - Z1 = Fiber distance'. The following fields are listed:

- FD1 - Z1 = Fiber distance
- X1 - Normal in x at Z1 (Bottom)
- Y1 - Normal in y at Z1 (Bottom)
- TXY1 - Shear in xy at Z1 (Bottom)
- FD2 - Z2 = Fiber distance
- X2 - Normal in x at Z2 (Top)
- Y2 - Normal in y at Z2 (Top)
- TXY2 - Shear in xy at Z2 (Top)
- VM1 - von Mises Stress at Z1 (Bottom)**
- VM2 - von Mises Stress at Z2 (Top)**
- MAX(X1,X2) - Max. of X1 and X2**
- MAX(Y1,Y2) - Max. of Y1 and Y2**
- MAX(TXY1,TXY2) - Max. of TXY1 and TXY2**
- MAX(VM1,VM2) - Max. of VM1 and VM2**
- MIN(X1,X2) - Min. of X1 and X2**
- MIN(Y1,Y2) - Min. of Y1 and Y2**
- MIN(TXY1,TXY2) - Min. of TXY1 and TXY2**
- MIN(VM1,VM2) - Min. of VM1 and VM2**

Annotations:

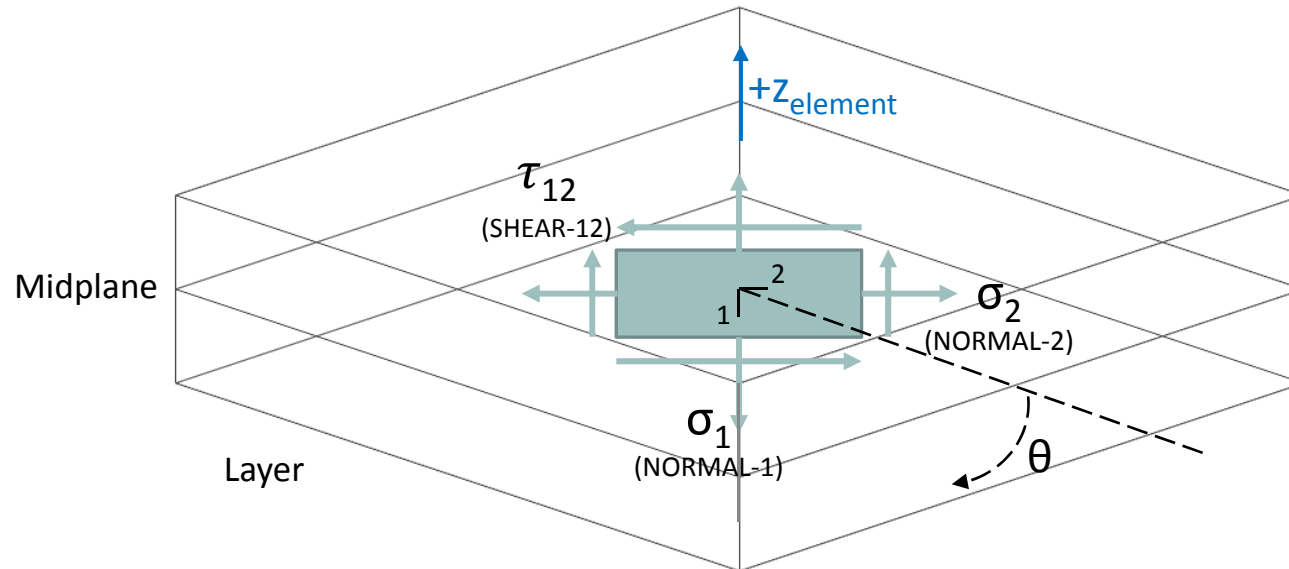
- A: A red box highlights the fields X1, Y1, and TXY1, with a circled 'A' to its right.
- B: A red box highlights the fields X2, Y2, and TXY2, with a circled 'B' to its right.
- C: A red box highlights the fields MAX(X1,X2), MAX(Y1,Y2), MAX(TXY1,TXY2), MAX(VM1,VM2), MIN(X1,X2), MIN(Y1,Y2), MIN(TXY1,TXY2), and MIN(VM1,VM2), with a circled 'C' to its right.

Nastran Composite Stress Results - CQUAD8 and PCOMP

Goal: Display PCOMP layer stresses

F06 File

SUBCASE 1										
STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD8)										
ELEMENT ID	PLY ID	STRESSES IN FIBER AND MATRIX DIRECTIONS			INTER-LAMINAR STRESSES		PRINCIPAL STRESSES (ZERO SHEAR)			MAX SHEAR
		NORMAL-1	NORMAL-2	SHEAR-12	SHEAR XZ-MAT	SHEAR YZ-MAT	ANGLE	MAJOR	MINOR	
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

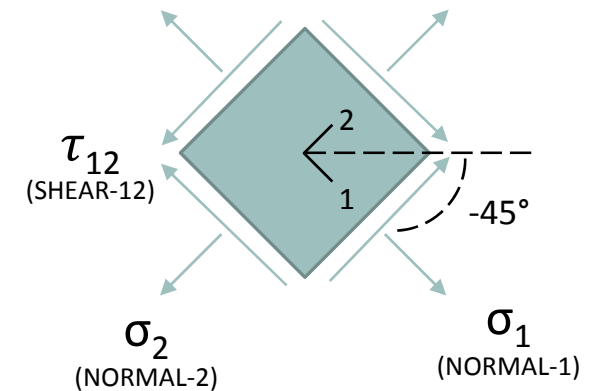
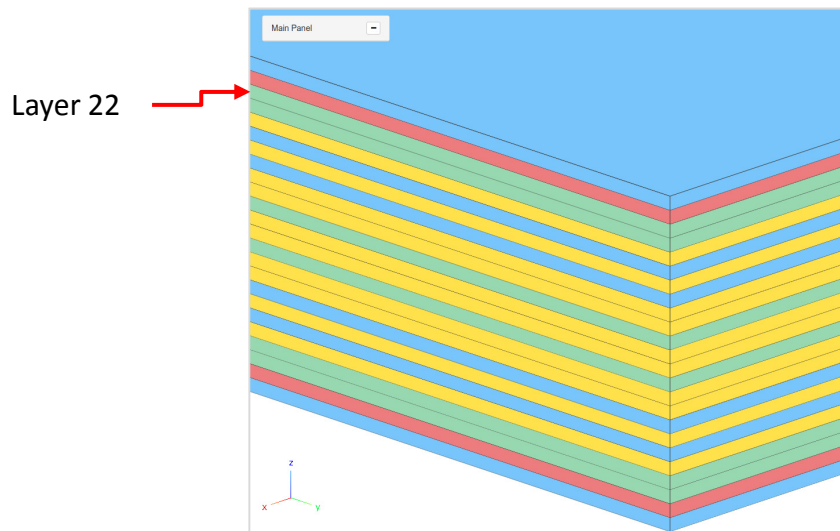


- Layer responses are reported in the 1-2 coordinate system
- Responses are reported at the midplane of each layer

Goal: Display PCOMP layer stresses

F06 File

SUBCASE 1										
STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD8)										
ELEMENT ID	PLY ID	STRESSES IN FIBER AND MATRIX DIRECTIONS			INTER-LAMINAR STRESSES	PRINCIPAL STRESSES (ZERO SHEAR)		MAX SHEAR		
		NORMAL-1	NORMAL-2	SHEAR-12	SHEAR XZ-MAT	SHEAR YZ-MAT	ANGLE	MAJOR	MINOR	
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



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

Acquire Dataset **1**

Select Dataset

ELEMENTAL/STRESS/QUAD8_COMP **2**

Select Domain

Reset Table Uncheck visible boxes Check visible boxes

Selected	DOMAIN_ID	DESIGN_CYCLE	SUBCASE
3 <input checked="" type="checkbox"/>	1	0	1
<input type="checkbox"/>	2	0	2
<input type="checkbox"/>	3	0	3
<input type="checkbox"/>	9	2	1
<input type="checkbox"/>	10	2	2
<input type="checkbox"/>	11	2	3

4

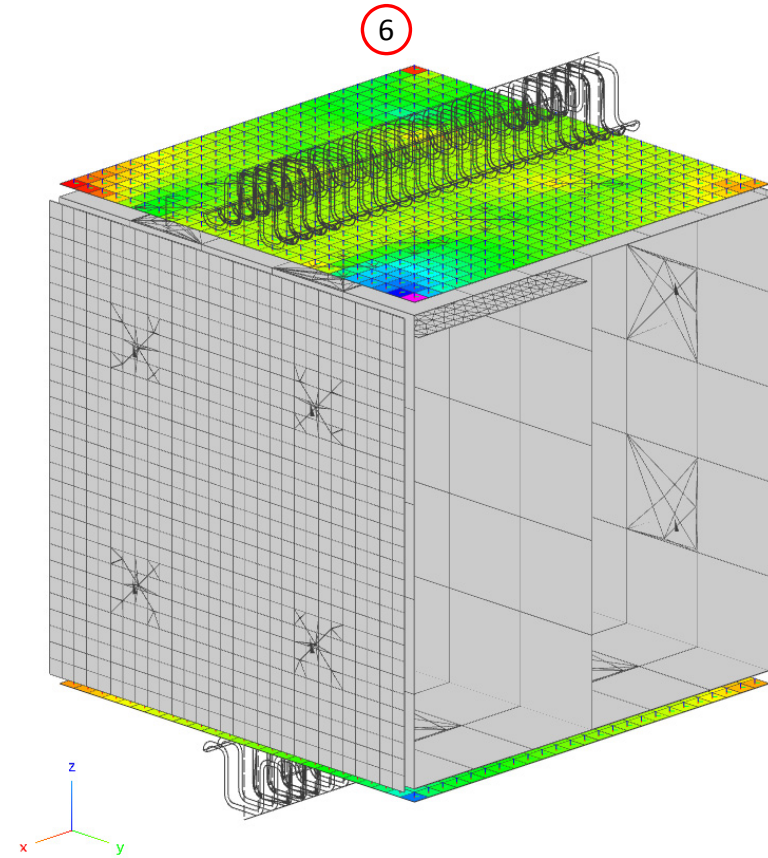
10 20 50

Acquisition complete and successful

Nastran Composite Stress Results - CQUAD8 and PCOMP

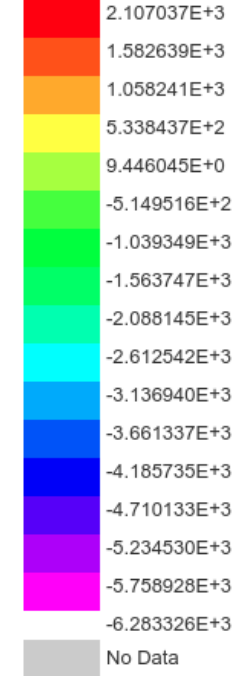
1. Use the vertical scroll bar to navigate to section Configure Plots
2. Click Display Shape Plot to hide the deformation
3. Set Field to X1 - Normal 1
 - This option displays the layer stress in the 1 direction
4. Set Layer to 1
 - This select the layer stress of layer 1 of the composite laminate
5.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



Post-processor

Spectrum



Configure Plots **1**

Display Color Plot Display Shape Plot **2**

Fringe Plot

Dataset: ELEMENTAL/STRESS/QUAD8_COMP

Field: X1 - Normal-1 **3**

Coordinate System: [Dropdown]

Layer: 1 **4**

Max/Min Option: MAX

Domain: DESIGN_CYCLE 0, SUBCASE 1

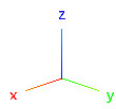
Display Directions: Yes **5**

Maximum and Minimum

Maximum: 2.107037E+3 at Element ID 6000050, Layer 1

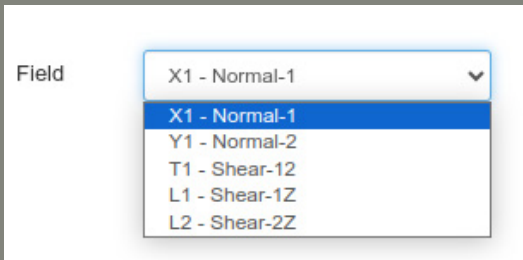
Minimum: -6.283326E+3 at Element ID 6000650, Layer 1

1

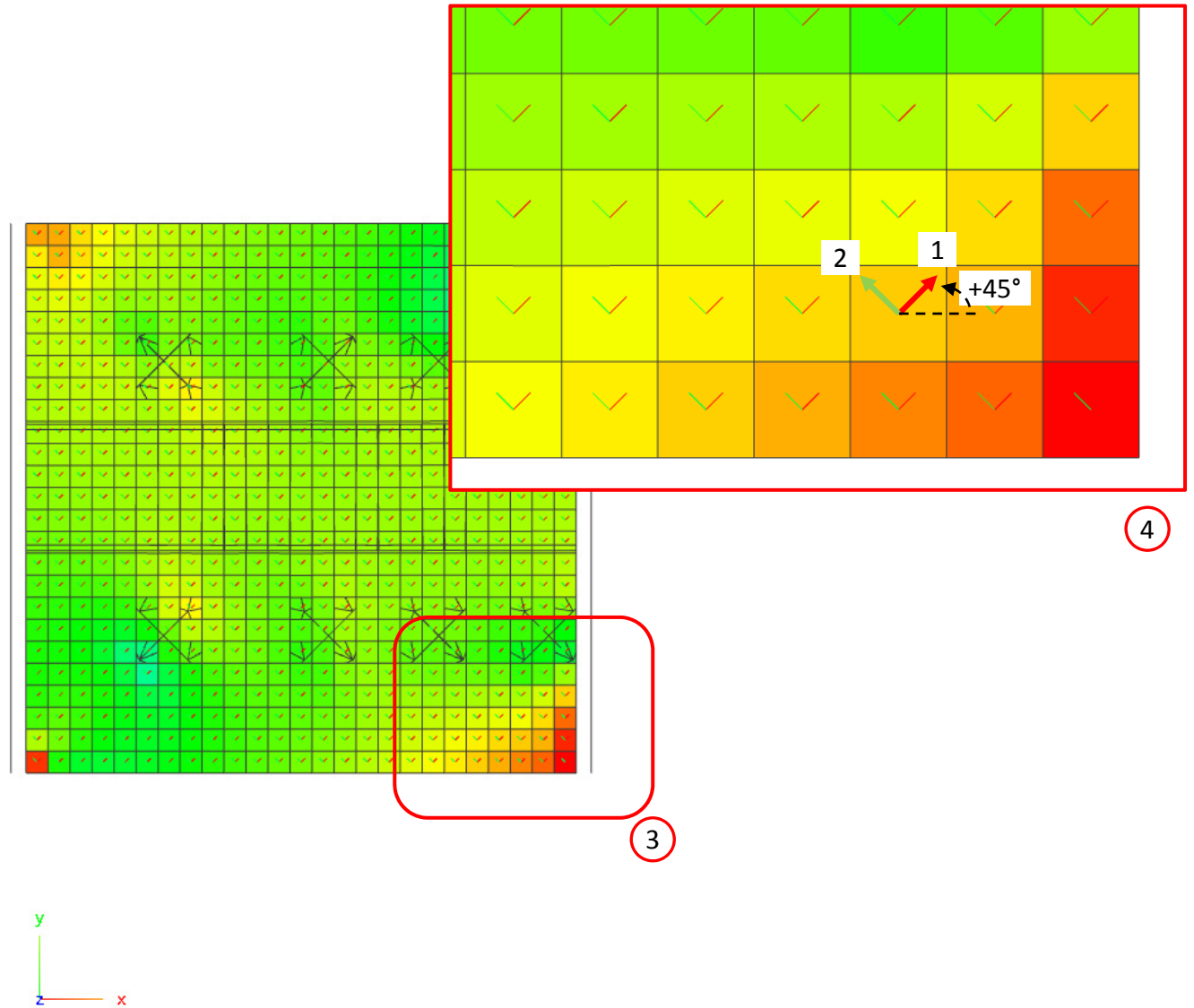
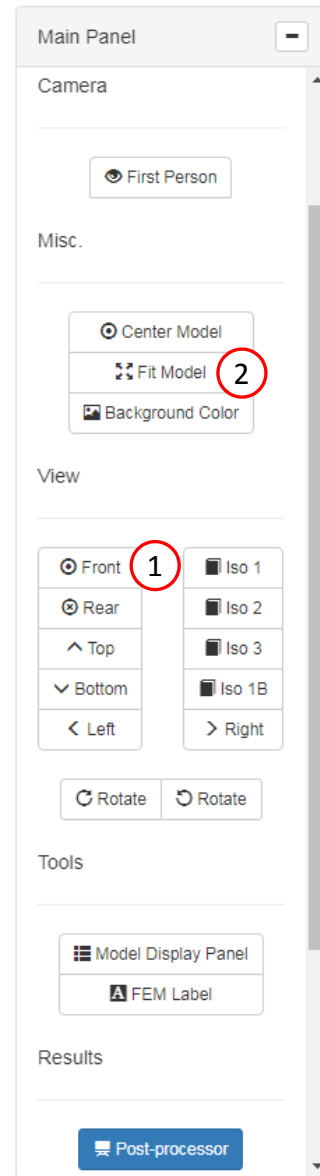


Nastran Composite Stress Results - CQUAD8 and PCOMP

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}

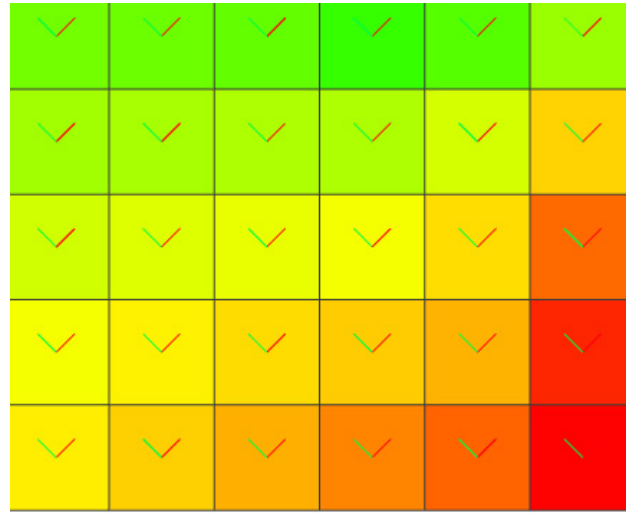


Nastran Composite Stress Results - CQUAD8 and PCOMP

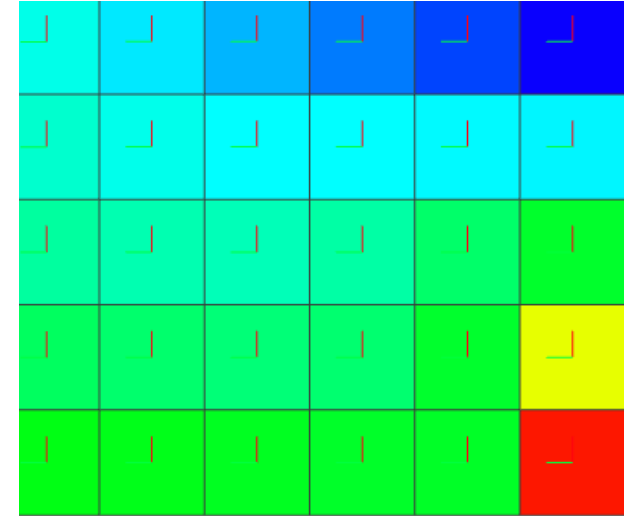
- The X1 (σ_1) layer stresses for layers 1, 2, 3 and 5 are displayed

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

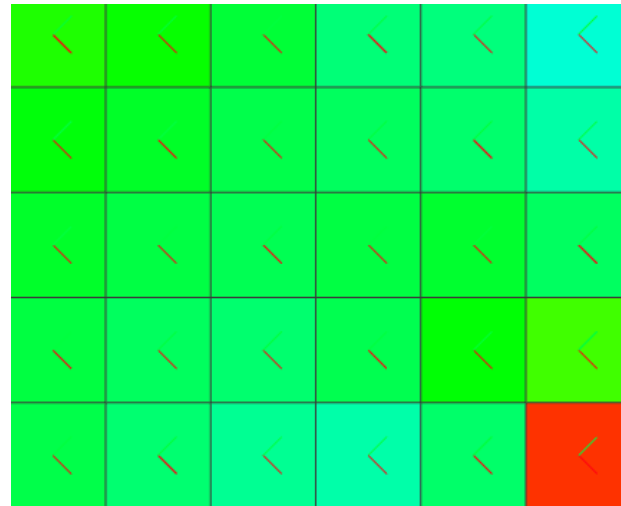
Layer 1 (45°)



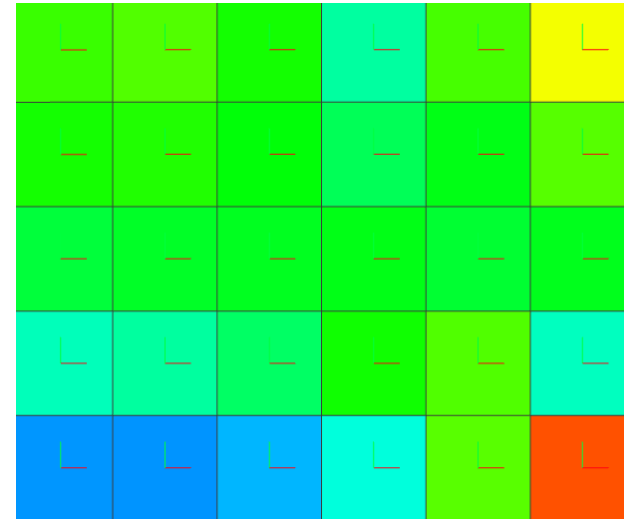
Layer 2 (90°)



Layer 3 (-45°)



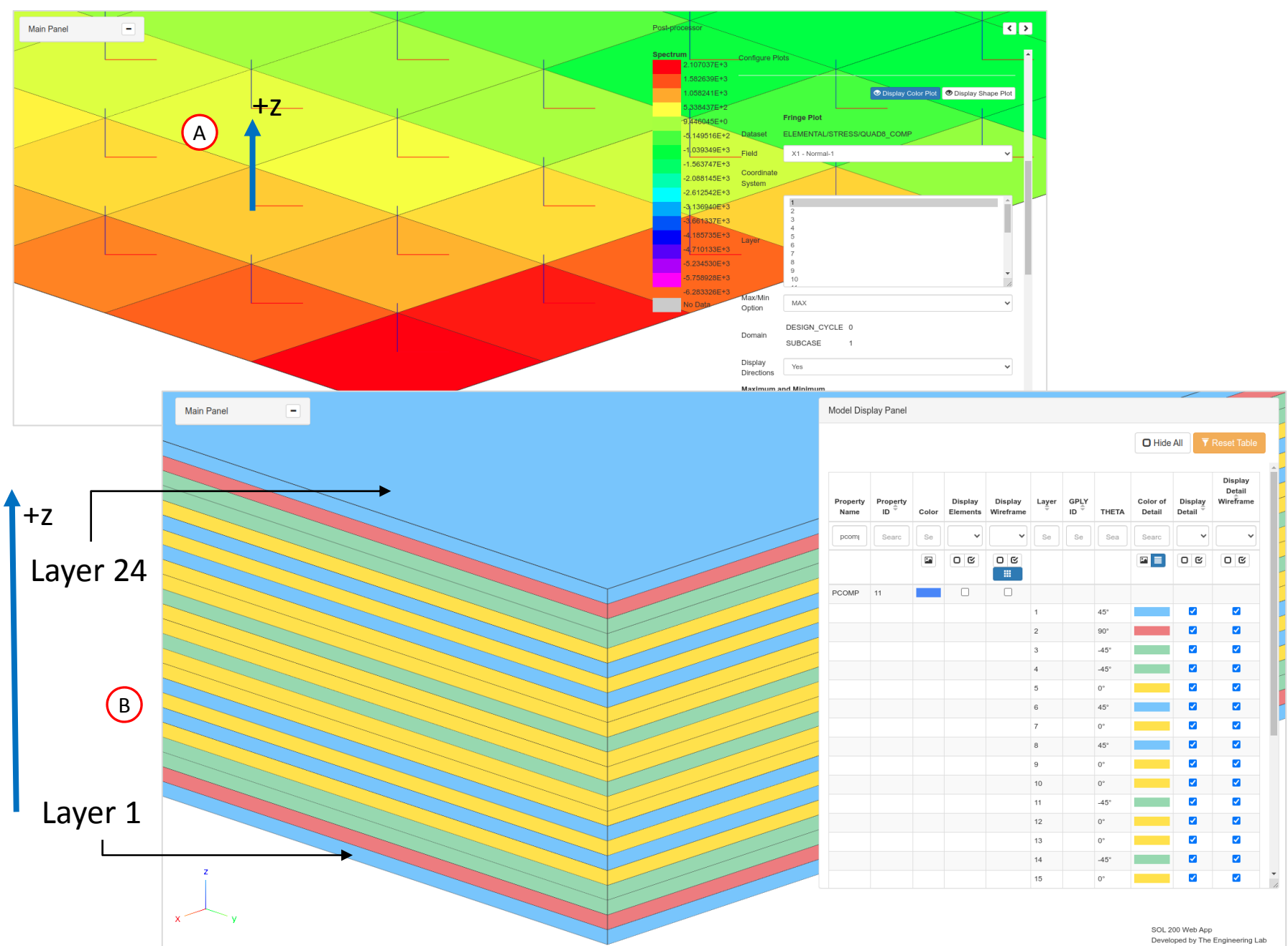
Layer 5 (0°)



Nastran Composite Stress Results - CQUAD8 and PCOMP

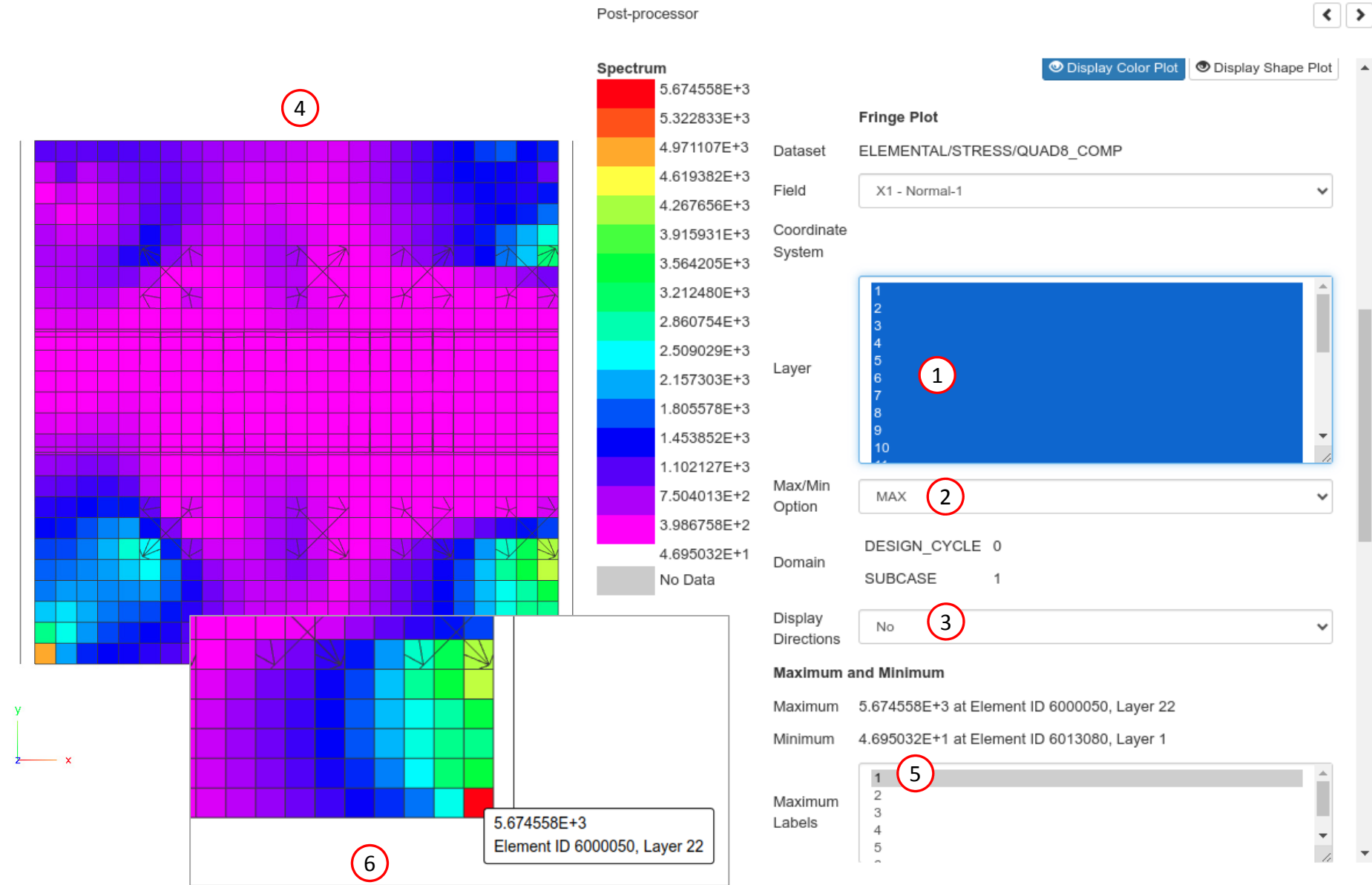
- A. The Display Directions option is useful for determining the order of the layers. Layers are numbered in ascending order along the +z direction.
- B. For comparison, the actual layer thicknesses are displayed via the Model Display Panel.

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



Nastran Composite Stress Results - CQUAD8 and PCOMP

- 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.
- Set Max/Min Option to MAX
- Set Display Directions to No
- The fringe plot is updated to show the maximum X1 layer stress for all layers
- Select the first maximum label
- A label is displayed indicating the maximum response and corresponding element ID and layer number



Nastran Beam Force and Stress Results - CBEAM and PBMSECT

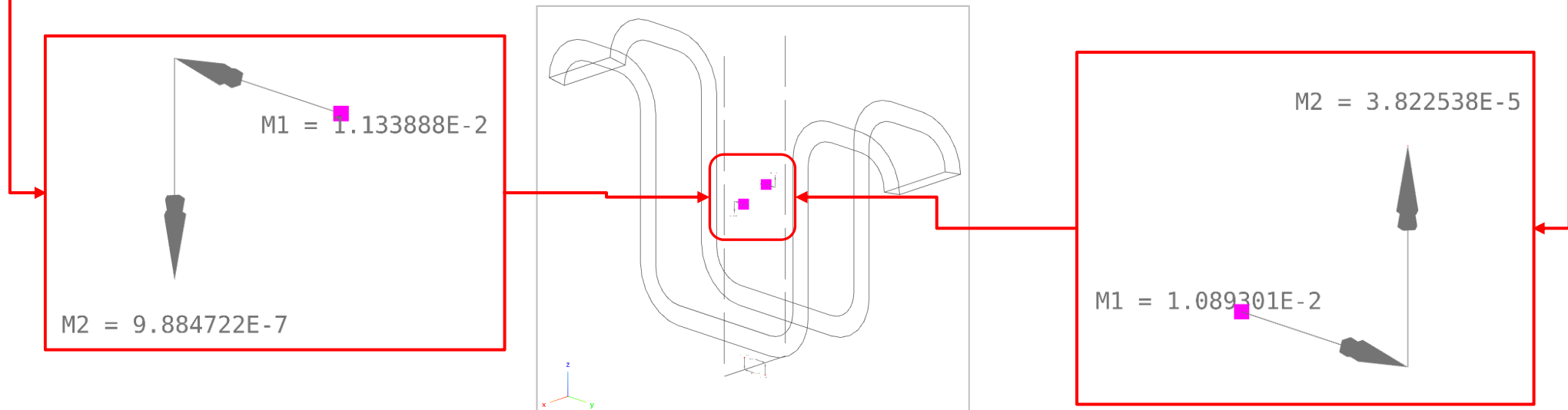
Goal: Display CBEAM element forces

F06 File

0

SUBCASE 1

FORCES IN BEAM ELEMENTS (C B E A M)									
ELEMENT-ID	GRID	STAT DIST/ LENGTH	- BENDING MOMENTS -		- WEB SHEARS -		AXIAL FORCE	TOTAL TORQUE	WARPING TORQUE
			PLANE 1	PLANE 2	PLANE 1	PLANE 2			
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

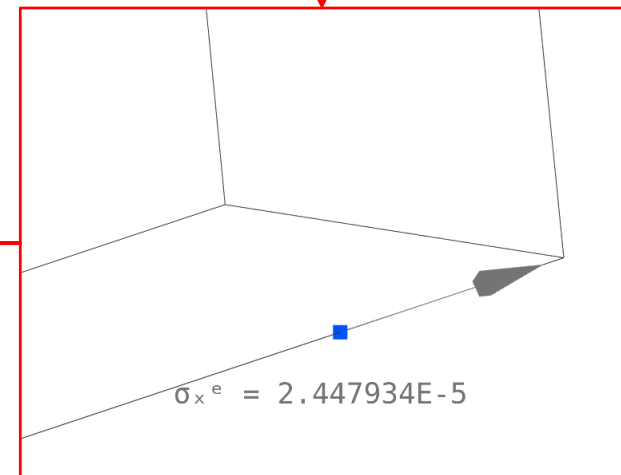
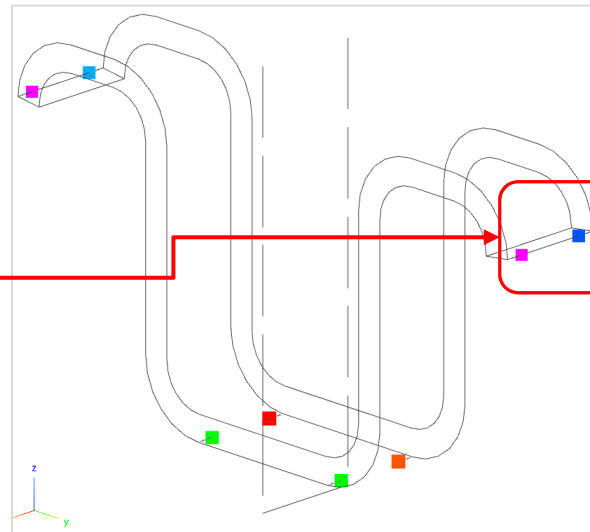
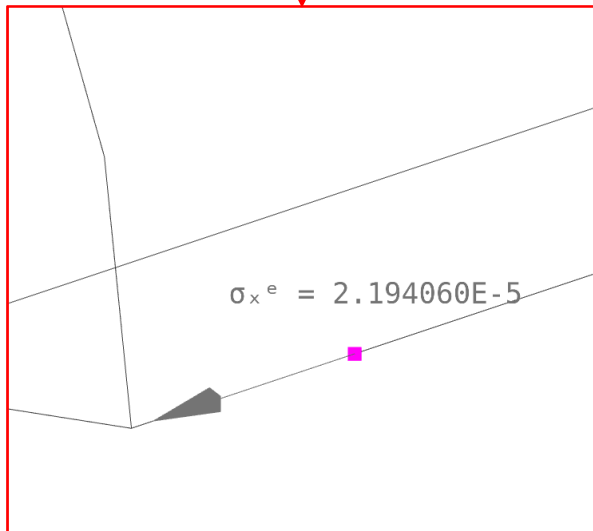
F06 File

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)

ELEMENT-ID	GRID	STAT DIST/ 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

Acquire Dataset 1

Select Dataset

ELEMENTAL/ELEMENT_FORCE/BEAM 2

Select Domain

Uncheck visible boxes Check visible boxes

Selected	DOMAIN_ID	DESIGN_CYCLE	SUBCASE
3 <input checked="" type="checkbox"/>	1	0	1
<input type="checkbox"/>	2	0	2
<input type="checkbox"/>	3	0	3
<input type="checkbox"/>	9	2	1
<input type="checkbox"/>	10	2	2
<input type="checkbox"/>	11	2	3

4

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

Configure Plots 1

4 Arrow Plot Marker Plot Display Color Plot Display Shape Plot 2

Fringe Plot

Dataset ELEMENTAL/ELEMENT_FORCE/BEAM

Field

- SD - Station distance divided by length
- BM1 - Bending moment plane 1**
- BM2 - Bending moment plane 2** 3
- TS1 - Shear plane 1
- TS2 - Shear plane 2
- AF - Axial Force
- TTRQ - Total Torque
- WTRQ - Warping Torque

Coordinate System

Domain DESIGN_CYCLE 0
SUBCASE 1

Nastran Beam Force and Stress Results - CBEAM and PBMSECT

1. Click Model Display Panel
2. Click Hide All

Main Panel

File Upload

File Upload

Camera

First Person

Misc.

Center Model

Fit Model

Background Color

View

Front, Rear, Top, Bottom, Left, Right

Iso 1, Iso 2, Iso 3, Iso 1B

Rotate, Rotate

Tools

Model Display Panel **1**

FEM Label

Model Display Panel

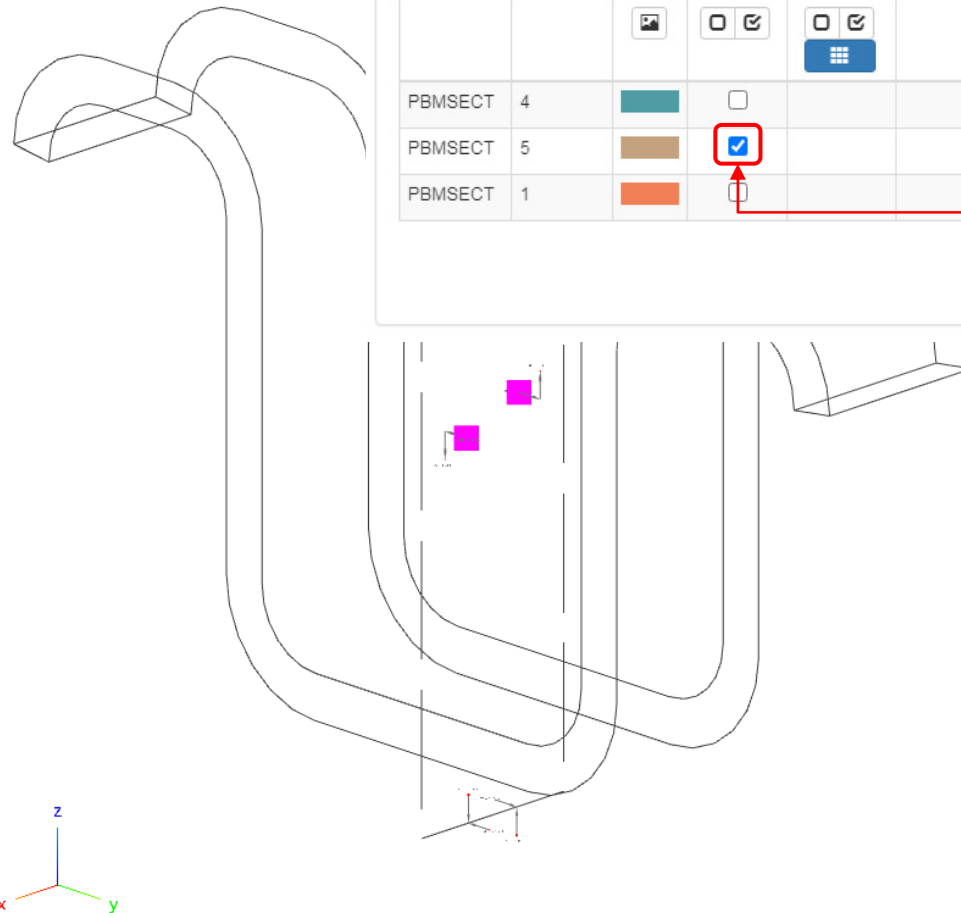
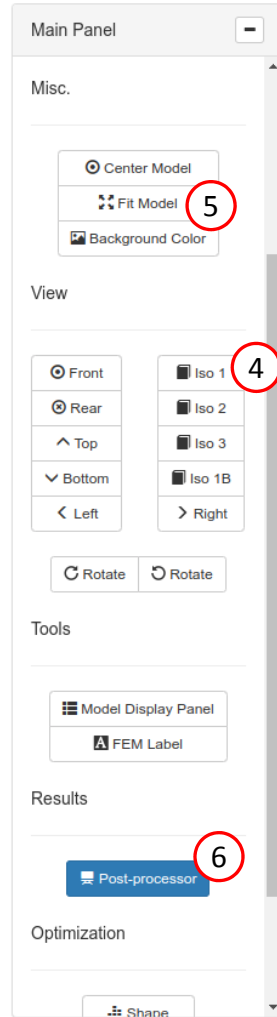
2 Hide All Reset Table

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
Search	Search	S			S	S	Se	Search		
PBUSH	101									
PBUSH	201									
PCOMP	11									
					1		45°			
					2		90°			
					3		-45°			
					4		-45°			
					5		0°			
					6		45°			
					7		0°			
					8		45°			
					9		0°			
					10		0°			
					11		-45°			

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.



Model Display Panel

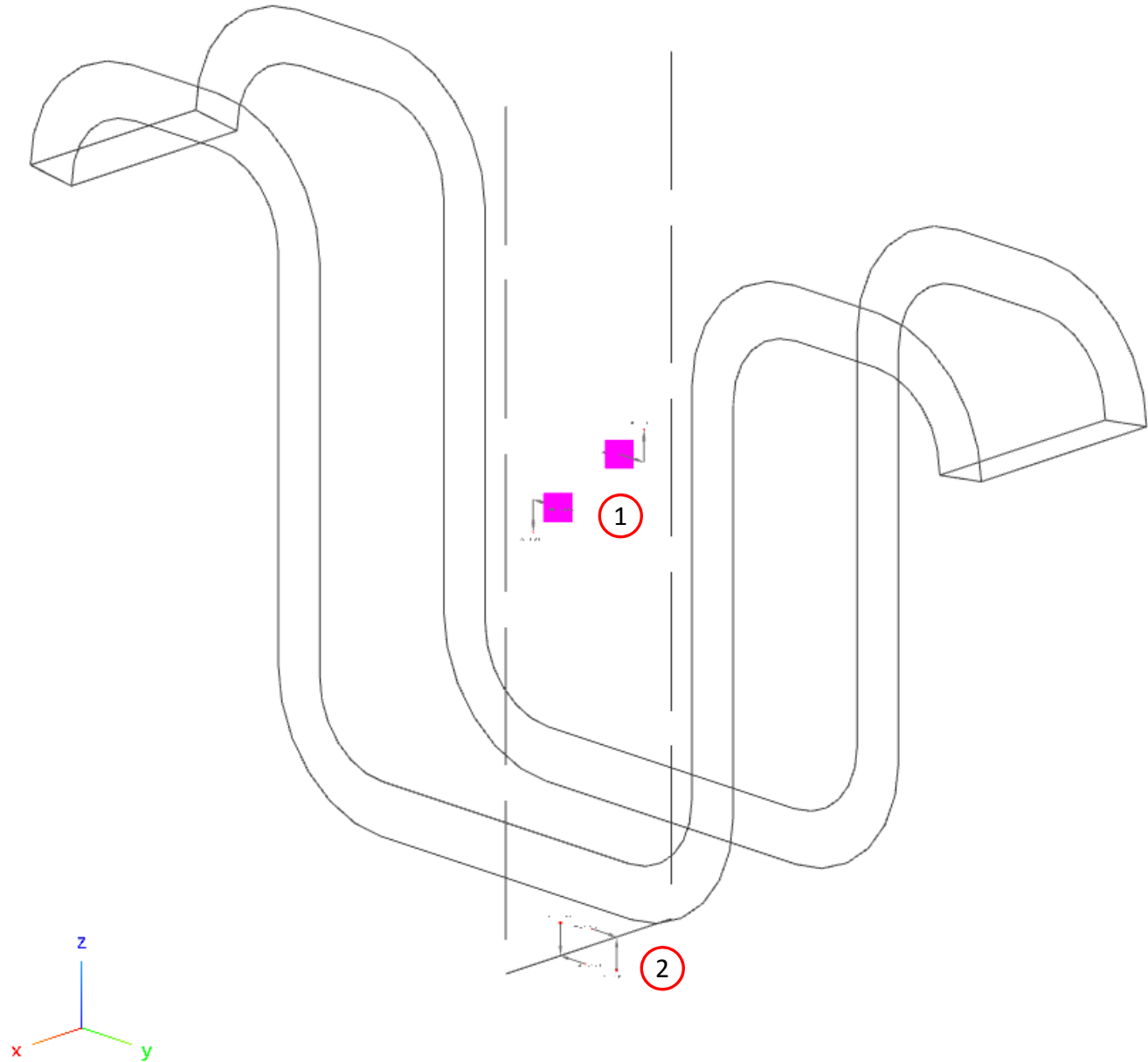
Hide All 3

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
1 pbmsect	Search	Se	<input type="checkbox"/>	<input type="checkbox"/>	Se	Se	Sea	Search	<input type="checkbox"/>	<input type="checkbox"/>
PBMSECT	4		<input type="checkbox"/>	<input type="checkbox"/>					<input type="checkbox"/>	<input type="checkbox"/>
PBMSECT	5		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					<input type="checkbox"/>	<input checked="" type="checkbox"/>
PBMSECT	1		<input type="checkbox"/>	<input type="checkbox"/>					<input type="checkbox"/>	<input type="checkbox"/>

100 200 500 1000

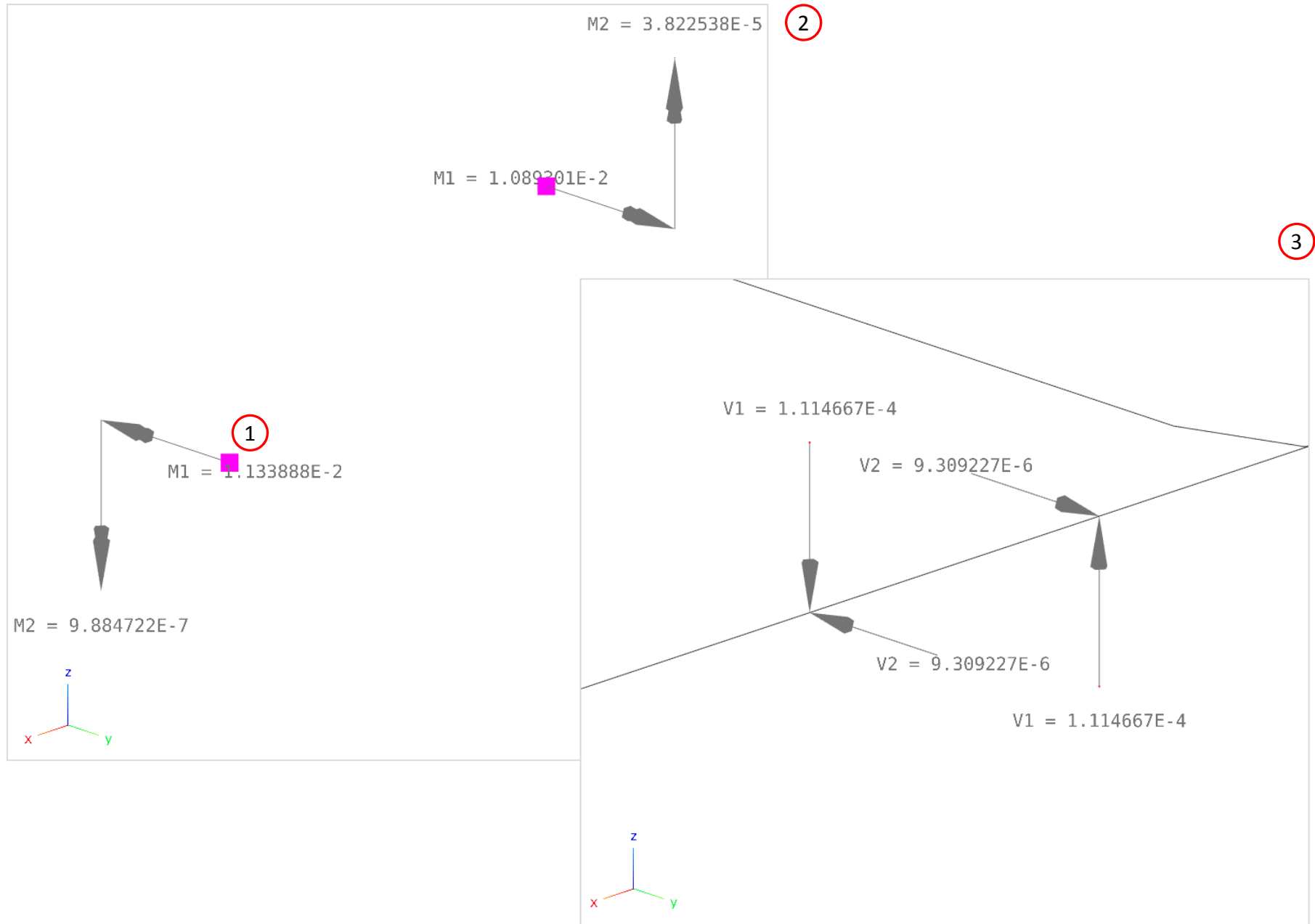
Nastran Beam Force and Stress Results - CBEAM and PBMSECT

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

Acquire Dataset **1**

Select Dataset

ELEMENTAL/STRESS/BEAM **2**

Select Domain

Uncheck visible boxes Check visible boxes

Selected	DOMAIN_ID	DESIGN_CYCLE	SUBCASE
		0 2	1 2 3
3 <input checked="" type="checkbox"/>	1	0	1
<input type="checkbox"/>	2	0	2
<input type="checkbox"/>	3	0	3
<input type="checkbox"/>	9	2	1
<input type="checkbox"/>	10	2	2
<input type="checkbox"/>	11	2	3

4

Acquisition complete and successful

10 20 50

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

Configure Plots **1**

4 Arrow Plot Marker Plot Display Color Plot Display Shape Plot **2**

Fringe Plot

Dataset ELEMENTAL/STRESS/BEAM

Field

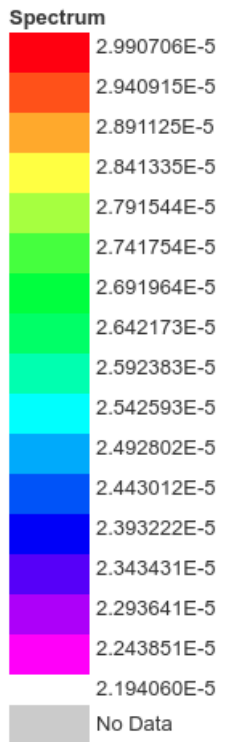
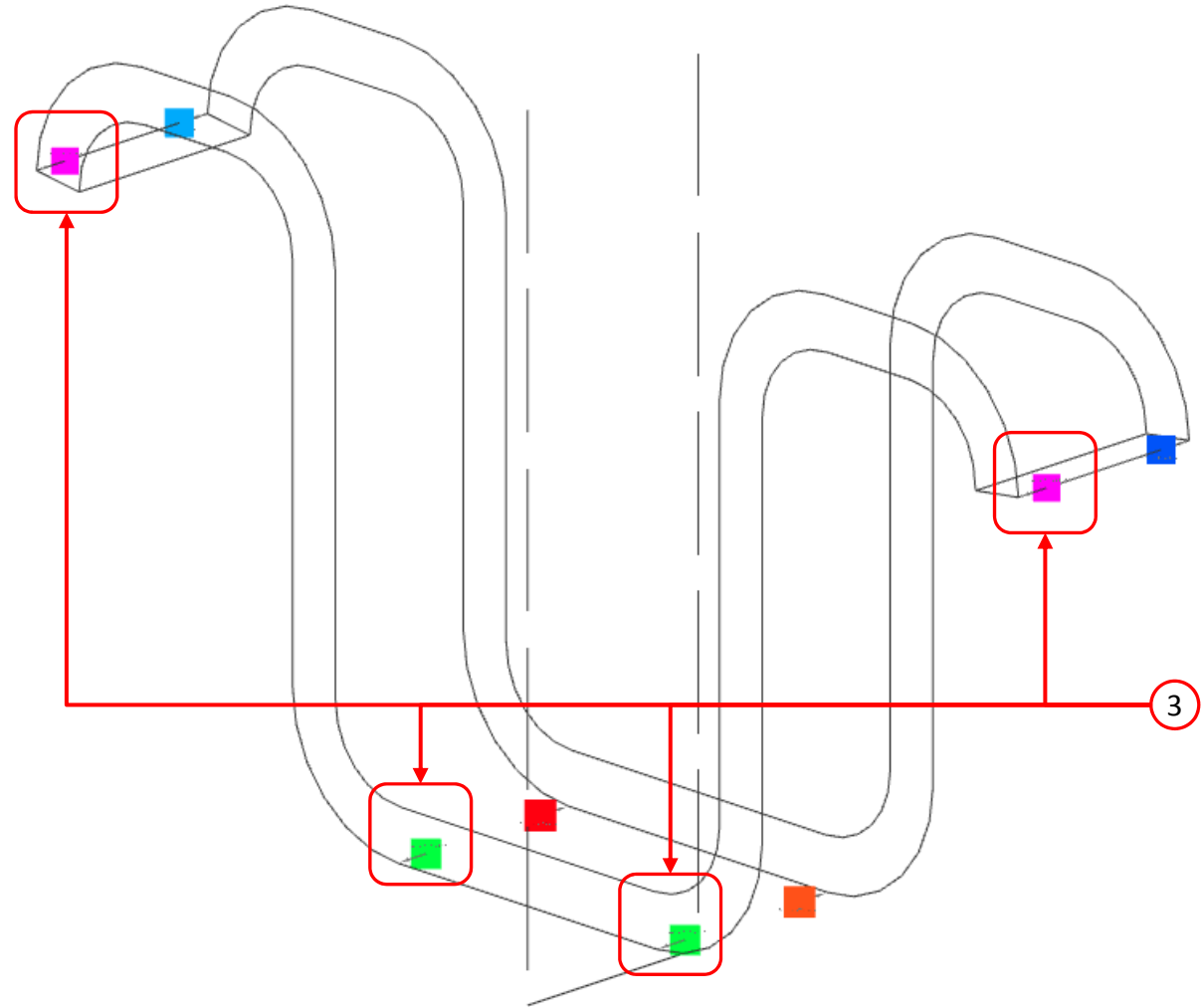
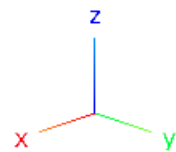
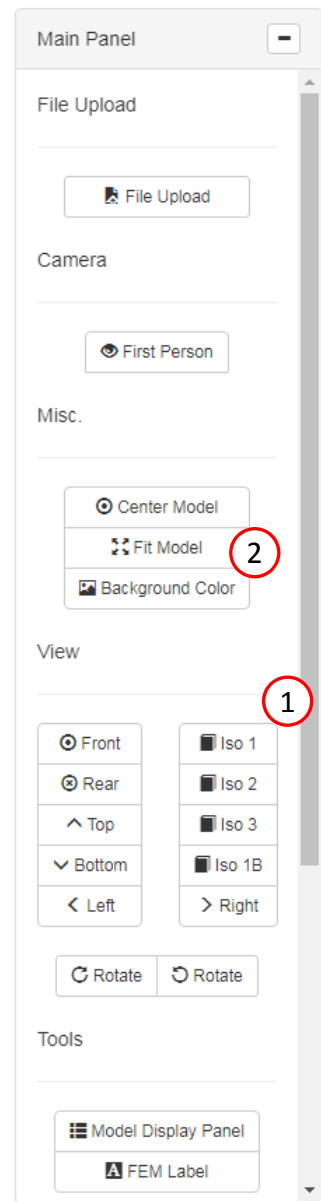
- SD - Station distance divided by length
- XC - Longitudonal Stress or Strain at Point C** **3**
- XD - Longitudonal Stress or Strain at Point D
- XE - Longitudonal Stress or Strain at Point E
- XF - Longitudonal Stress or Strain at Point F
- MAX - Maximal Stress or Strain
- MIN - Minimal Stress or Strain
- MST - Margin of Safety in Tension
- MSC - Margin of Safety in Compression

Coordinate System

Domain DESIGN_CYCLE 0
SUBCASE 1

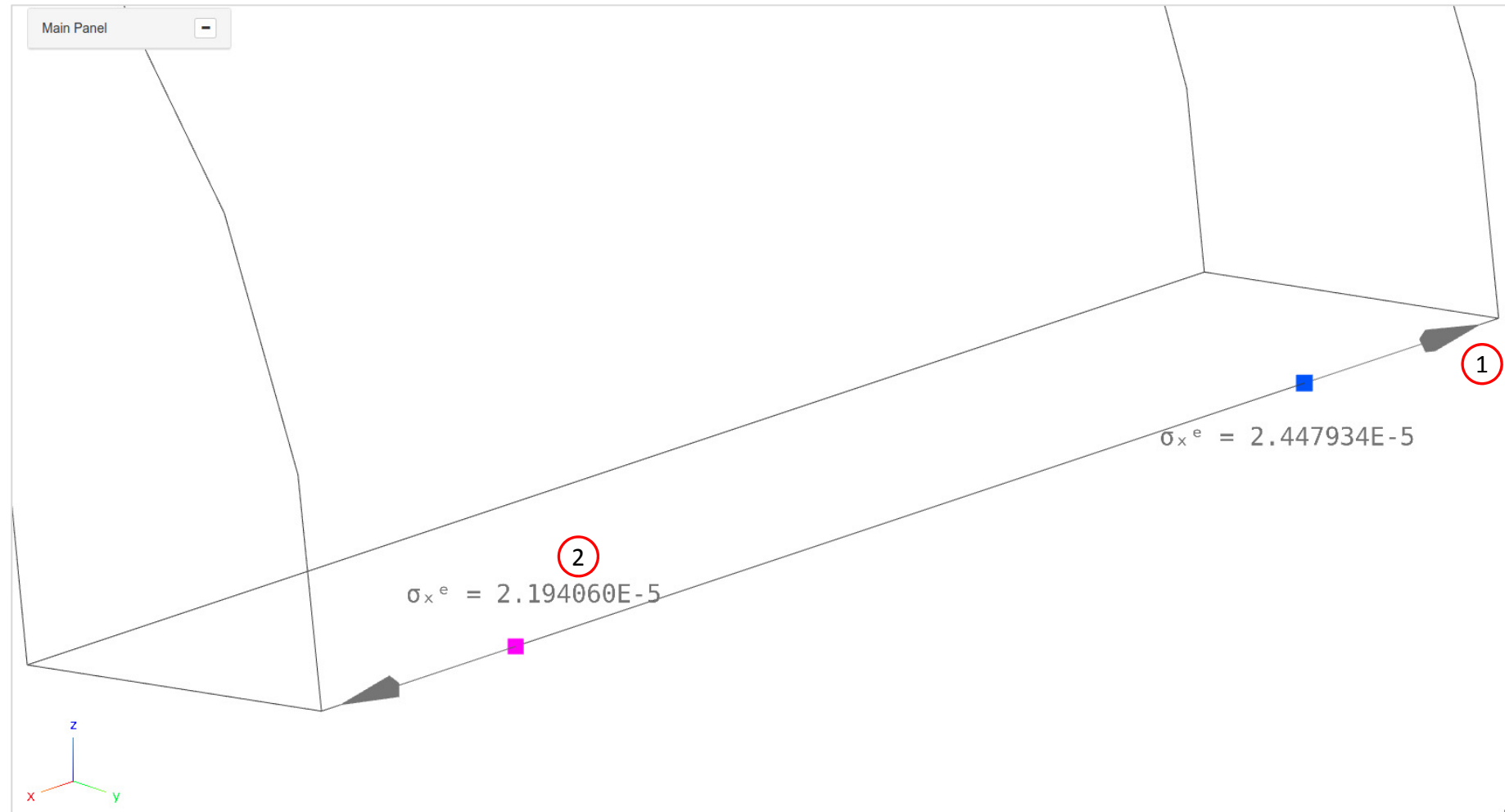
Nastran Beam Force and Stress Results - CBEAM and PBMSECT

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

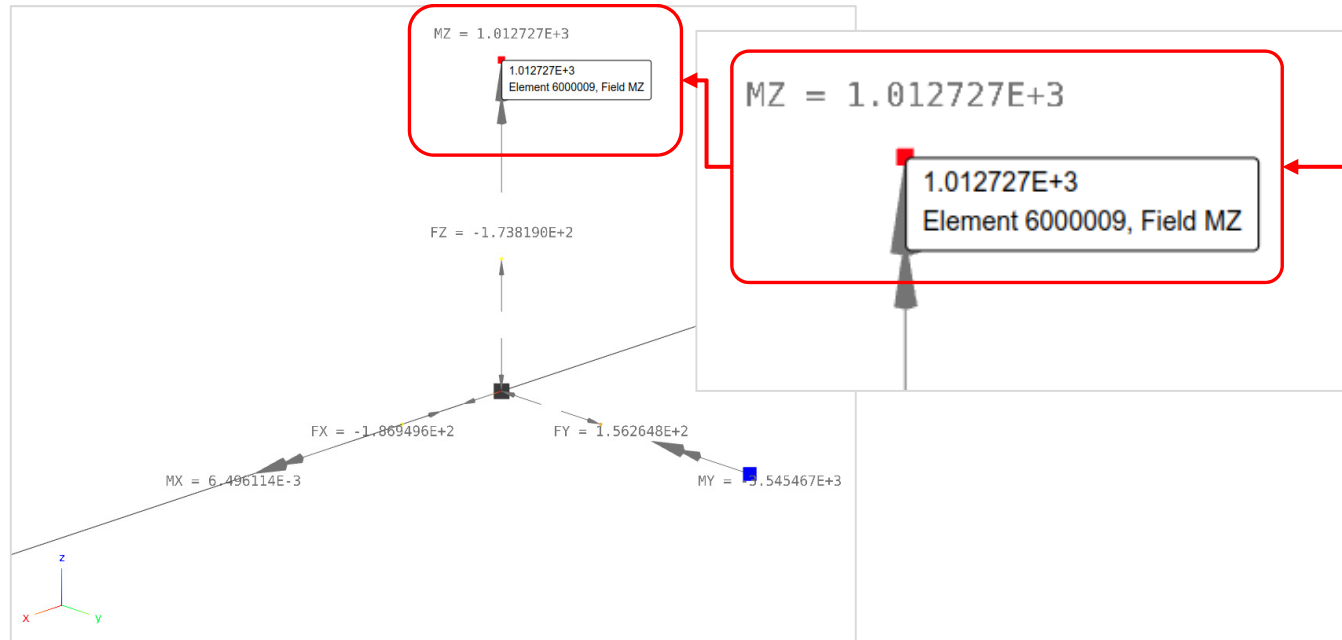
F06 File

0

SUBCASE 1

FORCES IN BUSH ELEMENTS (CBUSH)

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

Main Panel

Misc.

- Center Model
- Fit Model
- Background Color

View

- Front
- Rear
- Top
- Bottom
- Left
- Right
- Iso 1
- Iso 2
- Iso 3
- Iso 1B

Rotate Rotate

Tools

- Model Display Panel** 1
- FEM Label

Results

- Post-processor

Optimization

- Shape

Model Display Panel

2 Hide All 3 Reset Table

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
Search	Search	S			S	S	Se	Search		
PBUSH	101									
PBUSH	201									
PCOMP	11									
					1		45°			
					2		90°			
					3		-45°			
					4		-45°			
					5		0°			
					6		45°			
					7		0°			
					8		45°			
					9		0°			
					10		0°			
					11		-45°			

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

Main Panel

Misc.

- Center Model
- Fit Model **3**
- Background Color

View

- Front
- Rear
- Top
- Bottom
- Left
- Right
- Iso 1 **2**
- Iso 2
- Iso 3
- Iso 1B

Tools

- Model Display Panel
- FEM Label

Results

- Post-processor

Optimization

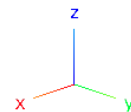
- Shape

Model Display Panel

Hide All Reset Table

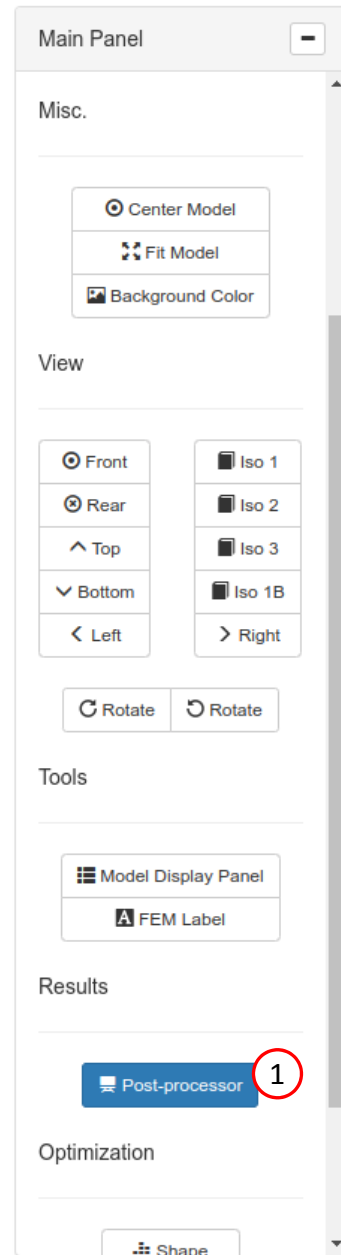
Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
Search	Search	S			S	S	Se	Search		
			<input checked="" type="checkbox"/>	<input type="checkbox"/>					<input type="checkbox"/>	<input type="checkbox"/>
PBUSH	101	Blue	<input checked="" type="checkbox"/>	<input type="checkbox"/>					<input type="checkbox"/>	<input type="checkbox"/>
PBUSH	201	Red	<input type="checkbox"/>	<input type="checkbox"/>					<input type="checkbox"/>	<input type="checkbox"/>
PCOMP	11	Blue	<input type="checkbox"/>	<input type="checkbox"/>					<input type="checkbox"/>	<input type="checkbox"/>

1



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



Acquire Dataset **2**

Select Dataset

ELEMENTAL/ELEMENT_FORCE/BUSH **3**

Select Domain

Reset Table Uncheck visible boxes Check visible boxes

Selected	DOMAIN_ID	DESIGN_CYCLE	SUBCASE
		0 2	1 2 3
4 <input checked="" type="checkbox"/>	1	0	1
<input type="checkbox"/>	2	0	2
<input type="checkbox"/>	3	0	3
<input type="checkbox"/>	9	2	1
<input type="checkbox"/>	10	2	2
<input type="checkbox"/>	11	2	3

5

Acquisition complete and successful

10 20 50

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

Configure Plots **1**

4 Arrow Plot Marker Plot Display Color Plot Display Shape Plot **2**

Fringe Plot

Dataset ELEMENTAL/ELEMENT_FORCE/BUSH

Field FX - Force x
FY - Force y
FZ - Force z
MX - Membrane force in x
MY - Membrane force in y
MZ - Bending moment in z-direction **3**

Coordinate System DESIGN_CYCLE 0

Domain SUBCASE 1

Maximum and Minimum

Maximum 1.012724E+3 at Element 6000009, Field MZ

Minimum -5.025716E+3 at Element 6000006, Field MY

Maximum Labels 1 **5**
2
3
4
5
6

Minimum Labels 1
2
3
4

Nastran CBUSH Force Results - CBUSH and PBUSH

1. Zoom in to the indicated region

Main Panel

File Upload

File Upload

Camera

First Person

Misc.

Center Model

Fit Model

Background Color

View

Front

Rear

Top

Bottom

Left

Right

Iso 1

Iso 2

Iso 3

Iso 1B

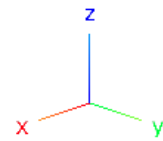
Rotate

Rotate

Tools

Model Display Panel

FEM Label

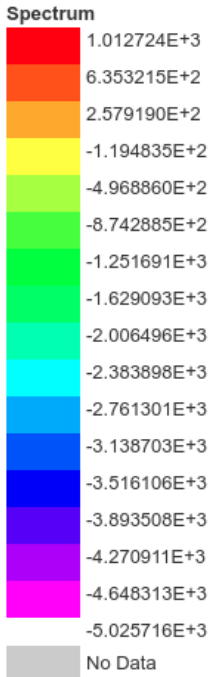


1.880123E+2
Element 6000008, Field FX

7.691939E+2
Element 6000006, Field MZ

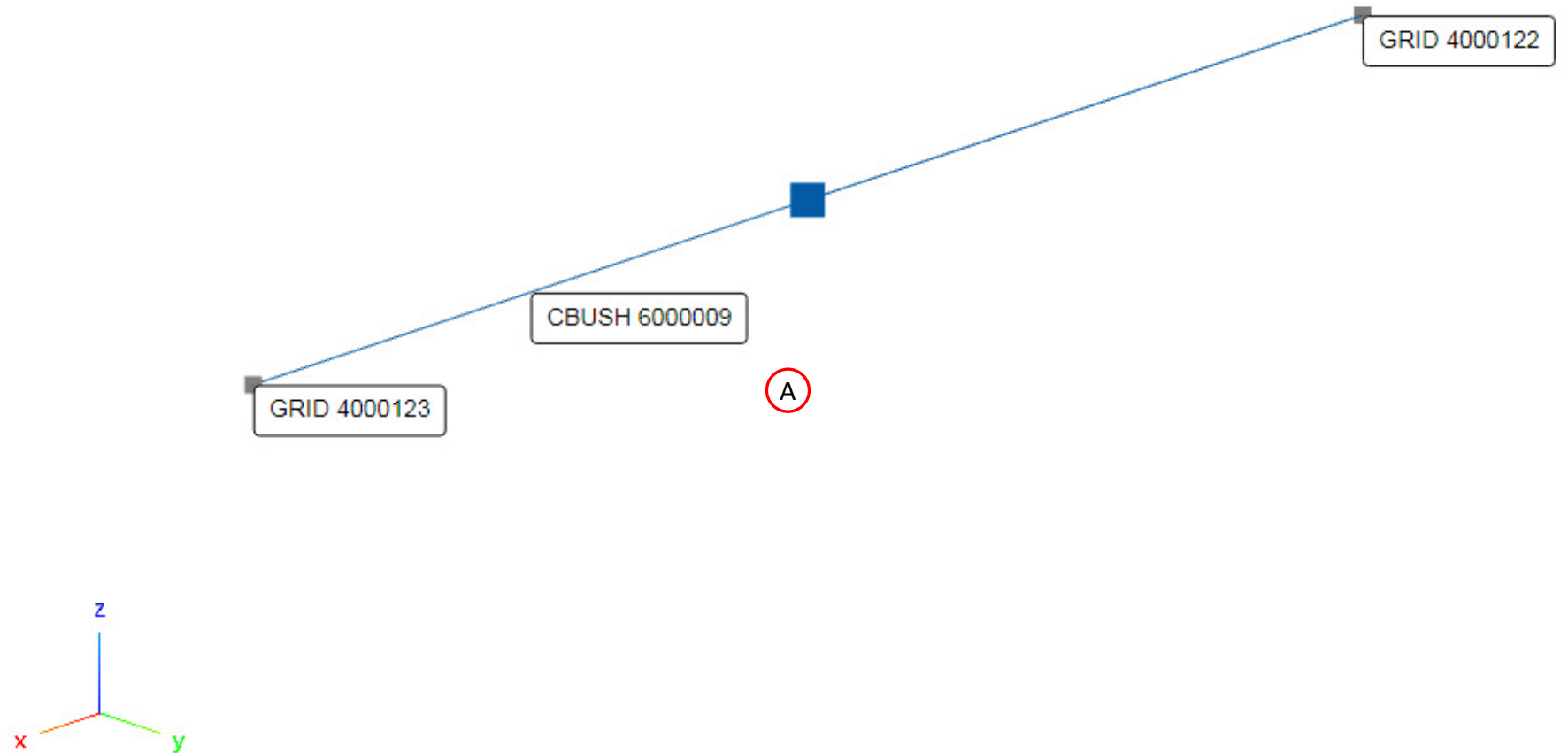
1.012724E+3
Element 6000009, Field MZ

1



Nastran CBUSH Force Results - CBUSH and PBUSH

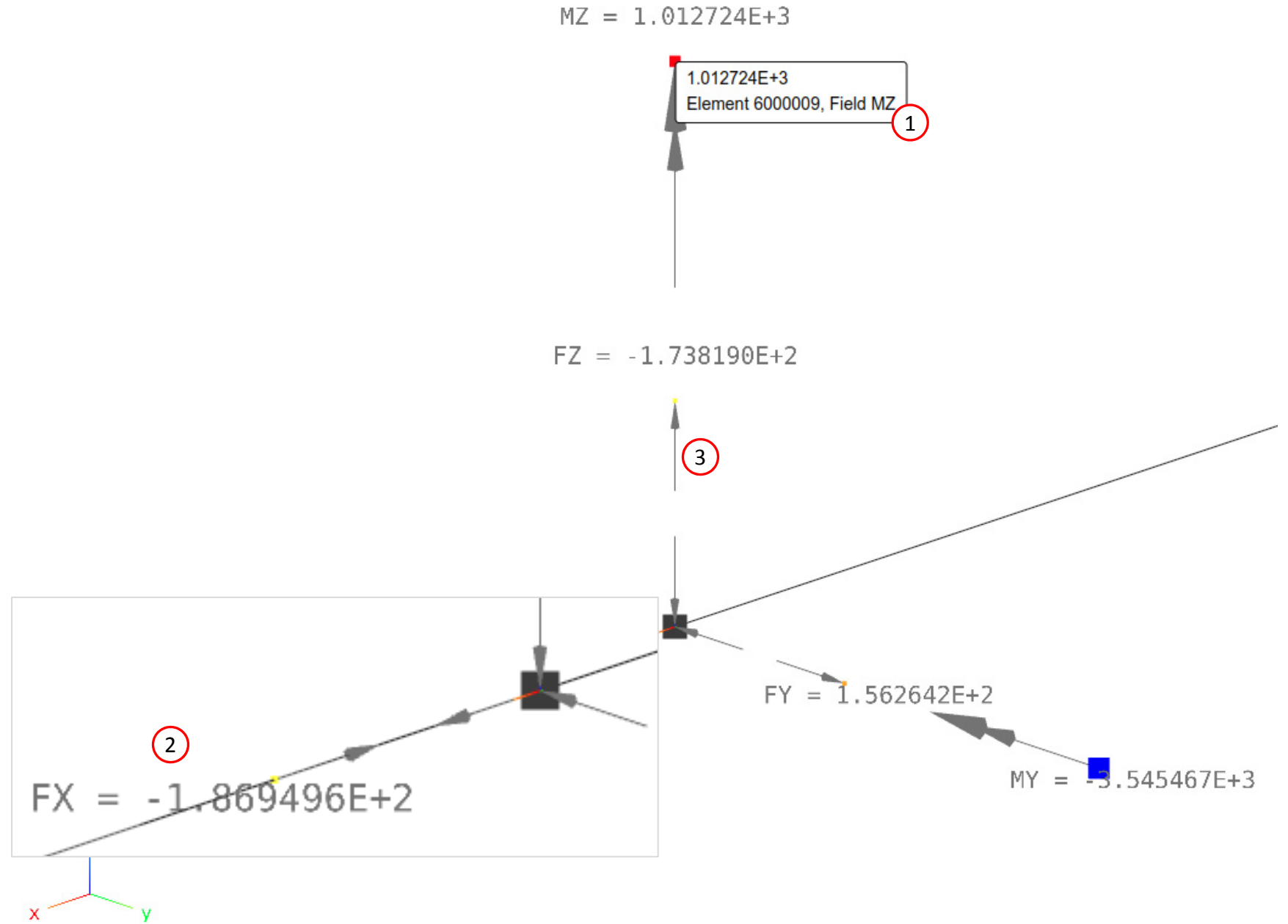
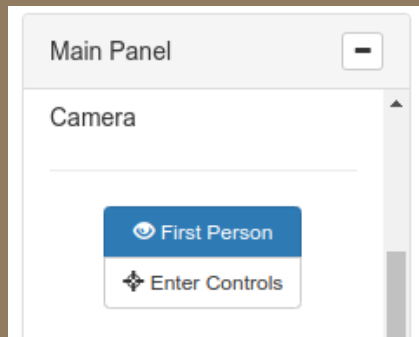
- A. As you zoom in, note that this CBUSH element is used to connect two grids (nodes). The FEM Label Tool was used to display the element and grid IDs.



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.

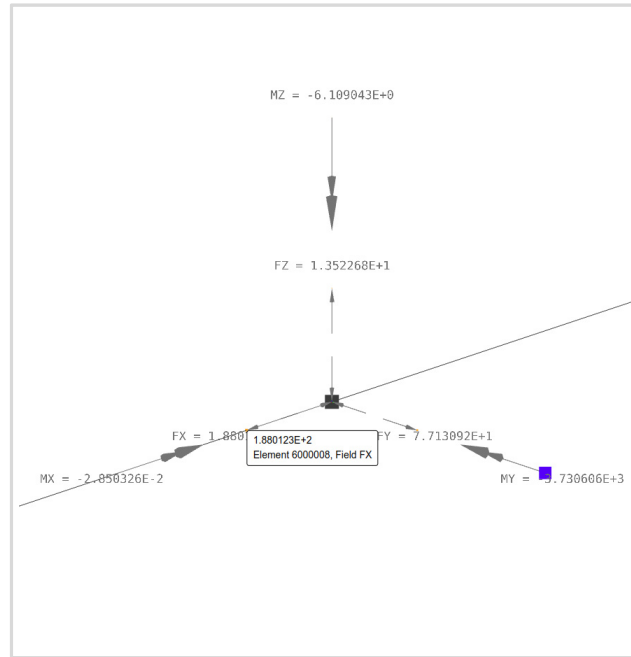
- If the default method of inspecting the model is not suitable for inspecting the CBUSH element forces, consider using the First Person option.



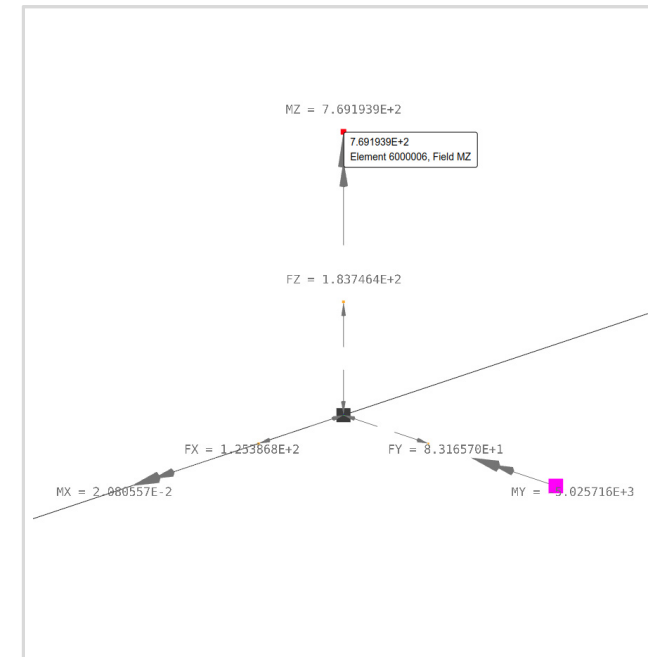
Nastran CBUSH Force Results - CBUSH and PBUSH

- The forces for the other CBUSH elements are displayed.

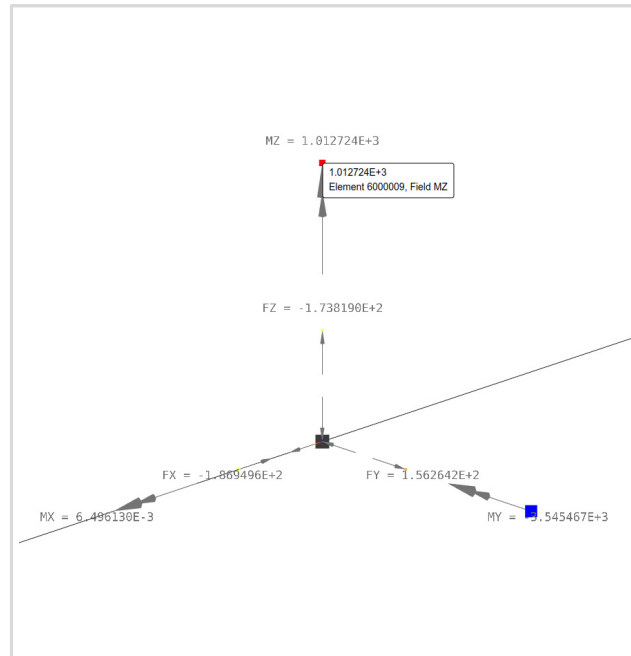
CBUSH
6000008



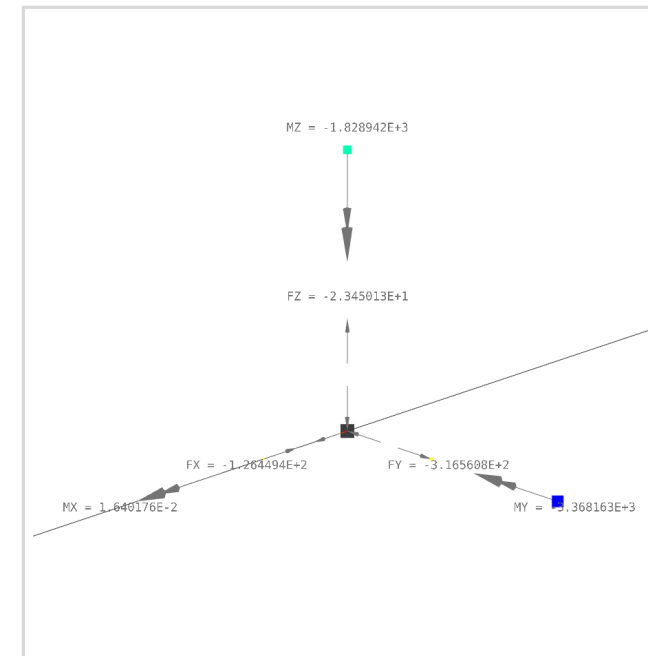
CBUSH
6000006



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

SOL 200 Web App - HDF5 Explorer Acquire Dataset Plots Browser Combine Plots Last Plot Added Home

Acquire Dataset Session ID: 7016 HDF5

Select Dataset

- ELEMENTAL/ELEMENT_FORCE/BEAM
- ELEMENTAL/ELEMENT_FORCE/BUSH 2
- ELEMENTAL/ELEMENT_FORCE/FAST
- ELEMENTAL/ELEMENT_FORCE/QUAD4_C
- ELEMENTAL/ELEMENT_FORCE/QUAD8
- ELEMENTAL/ELEMENT_FORCE/...

Specify Entities

ALL 4

Element identification number (EID)
Examples: 6000006, 6000007, 6000008, etc.

Auto Execute 3

Select Domains

Acquire Dataset 5

Acquisition complete and successful

Acquired Dataset Create Plot Reset Filters Download CSV 7

ELEMENTAL/ELEMENT_FORCE/BUSH - ALL

EID	FX	FY	FZ	MX	MY	MZ	SAMPLE	DOMAIN_ID
Element identification number	Force x	Force y	Force z	Membrane force in x	Membrane force in y	Bending moment in z-direction	Name of H5 File**	Domain identi
6000006	125.38677215576...	83.16570281982422	183.74644470214...	0.0208055693656...	-5025.7158203125	769.1939086914062	model	1
6000007	-126.44943237304...	-316.560791015625	-23.450128555297...	0.0164017565548...	-3368.1628417968...	-1828.9415283203...	model	1
6000008	188.0122528076172							
6000009	-186.94960021972...							
6000014	132.1404266357422							
6000015	-137.3411407470703							
6000016	86.4691162109375							
6000017	-81.268402099609...							
6000006	-125.38677215576...							
6000007	126.44943237304...							
6000008	-188.01225280761...							
6000009	186.94960021972...							
6000014	-132.14042663574...							
6000015	137.3411407470703							
6000016	-86.4691162109375							

acquired_dataset.csv - Excel

File Home Insert Page Layout Formulas Data Review View Tell me what you want to do... Sign in Share

Clipboard Font Alignment Number Styles

A1 Dataset: ELEMENTAL/ELEMENT_FORCE/BUSH

1	Dataset: ELEMENTAL/ELEMENT_FORCE/BUSH									
2	EID	FX	FY	FZ	MX	MY	MZ	SAMPLE	DOMAIN	SUBCASE
3	Element i	Force x	Force y	Force z	Membran	Membran	Bending n	Name of f	Domain id	Subcase n
4	6000006	125.3868	83.1657	183.7464	0.020806	-5025.72	769.1939	model	1	1
5	6000007	-126.449	-316.561	-23.4501	0.016402	-3368.16	-1828.94	model	1	1
6	6000008	188.0123	77.13092	13.52268	-0.0285	-3730.61	-6.10904	model	1	1
7	6000009	-186.95	156.2642	-173.819	0.006496	-3545.47	1012.724	model	1	1
8	6000014	132.1404	-199.981	-44.6648	0.00074	-3685.56	1901.261	model	1	1
9	6000015	-137.341	170.0618	225.8021	-0.00795	-1203.08	-1808.24	model	1	1
10	6000016	86.46912	381.1184	-146.611	0.014054	-3046.13	-2645.56	model	1	1
11	6000017	-81.2684	-351.199	-34.5261	0.004712	-2996.82	2297.517	model	1	1
12	6000006	-125.387	-83.1657	-183.746	-0.02081	5025.716	-769.194	model	2	2
13	6000007	126.4494	316.5608	23.45013	-0.0164	3368.163	1828.942	model	2	2
14	6000008	-188.012	-77.1309	-13.5227	0.028503	3730.606	6.109043	model	2	2
15	6000009	186.9496	-156.264	173.819	-0.0065	3545.467	-1012.72	model	2	2
16	6000014	-132.14	199.9807	44.66481	-0.00074	3685.564	-1901.26	model	2	2
17	6000015	137.3411	-170.062	-225.802	0.007947	1203.082	1808.242	model	2	2
18	6000016	-86.4691	-381.118	146.6112	-0.01405	3046.131	2645.565	model	2	2

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/MS_C_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

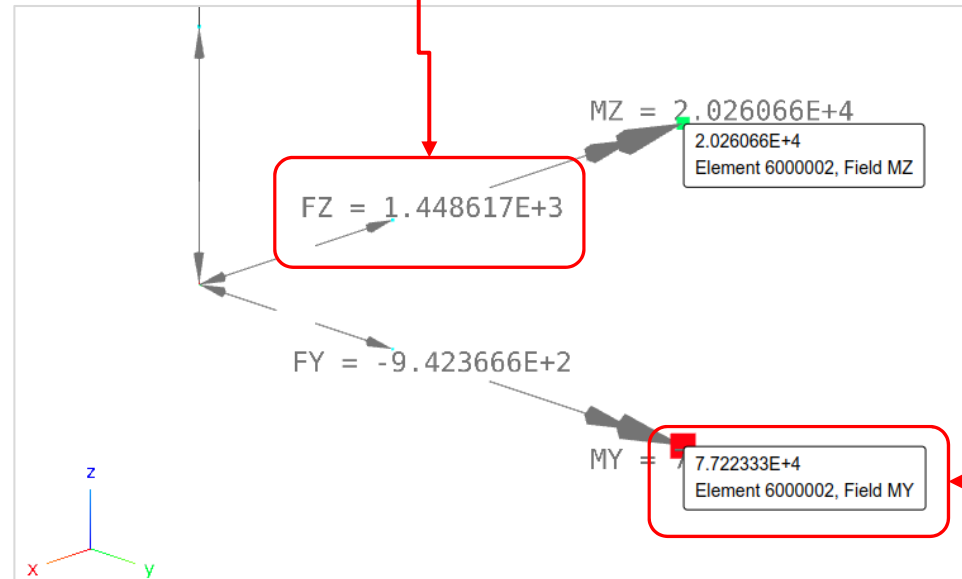
F06 File

0

SUBCASE 1

FORCES IN FASTENER ELEMENTS (CFAST)

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

Main Panel

File Upload

File Upload

Camera

First Person

Misc.

Center Model

Fit Model

Background Color

View

Front, Rear, Top, Bottom, Left, Right

Iso 1, Iso 2, Iso 3, Iso 1B

Rotate, Rotate

Tools

Model Display Panel **1**

FEM Label

Model Display Panel

2 Hide All Reset Table

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
Search	Search	S			S	S	Se	Search		
PBUSH	101									
PBUSH	201									
PCOMP	11									
					1		45°			
					2		90°			
					3		-45°			
					4		-45°			
					5		0°			
					6		45°			
					7		0°			
					8		45°			
					9		0°			
					10		0°			
					11		-45°			

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

The Main Panel interface includes sections for File Upload, Camera (First Person), Misc. (Center Model, Fit Model, Background Color), View (Front, Rear, Top, Bottom, Left, Right, Iso 1, Iso 2, Iso 3, Iso 1B), Rotate, and Tools (Model Display Panel, FEM Label). Red circles with numbers 1 through 5 highlight the search bar, checkboxes in the Display Elements column, Iso 1, Fit Model, and the Model Display Panel button respectively.

The Model Display Panel interface features a table with columns: Property Name, Property ID, Color, Display Elements, Display Wireframe, Layer, GPLY ID, THETA, Color of Detail, Display Detail, and Display Detail Wireframe. Red circles with numbers 1 through 5 highlight the search bar, checkboxes in the Display Elements column, Iso 1, Fit Model, and the Model Display Panel button respectively.

Property Name	Property ID	Color	Display Elements	Display Wireframe	Layer	GPLY ID	THETA	Color of Detail	Display Detail	Display Detail Wireframe
pfast	Search	Se			Se	Se	Sea	Search		
			<input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/>				<input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/>
PFAST	6000000	Blue	<input checked="" type="checkbox"/>					Green	<input type="checkbox"/>	<input type="checkbox"/>
PFAST	6000001	Yellow	<input checked="" type="checkbox"/>					Blue	<input type="checkbox"/>	<input type="checkbox"/>
PFAST	6000002	Green	<input checked="" type="checkbox"/>					Orange	<input type="checkbox"/>	<input type="checkbox"/>
PFAST	6000003	Orange	<input checked="" type="checkbox"/>					Brown	<input type="checkbox"/>	<input type="checkbox"/>



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

Main Panel

Misc.

- Center Model
- Fit Model
- Background Color

View

- Front
- Rear
- Top
- Bottom
- Left
- Right
- Iso 1
- Iso 2
- Iso 3
- Iso 1B

Tools

- Model Display Panel
- FEM Label

Results

Post-processor **1**

Optimization

Share

Acquire Dataset **2**

Select Dataset

ELEMENTAL/ELEMENT_FORCE/FAST **3**

Select Domain

Reset Table Uncheck visible boxes Check visible boxes

Selected	DOMAIN_ID	DESIGN_CYCLE	SUBCASE
		0 2	1 2 3
4 <input checked="" type="checkbox"/>	1	0	1
<input type="checkbox"/>	2	0	2
<input type="checkbox"/>	3	0	3
<input type="checkbox"/>	9	2	1
<input type="checkbox"/>	10	2	2
<input type="checkbox"/>	11	2	3

5

Acquisition complete and successful

10 20 50

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

Main Panel

File Upload

File Upload

Camera

First Person

Misc.

Center Model

Fit Model

Background Color

View

Front

Rear

Top

Bottom

Left

Right

Iso 1

Iso 2

Iso 3

Iso 1B

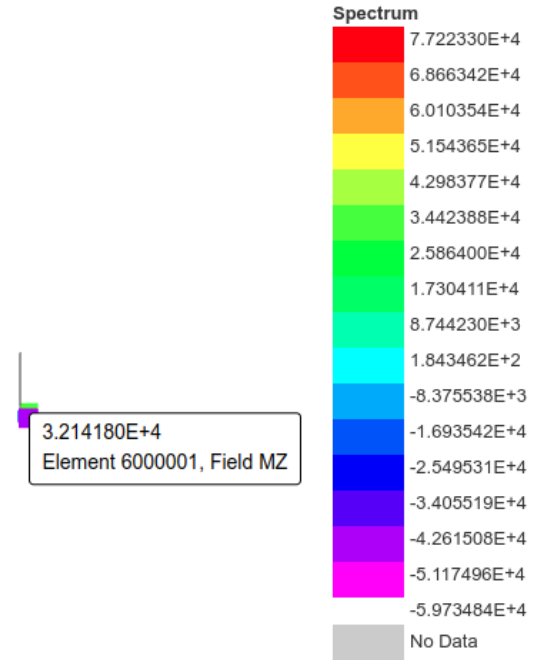
Rotate

Rotate

Tools

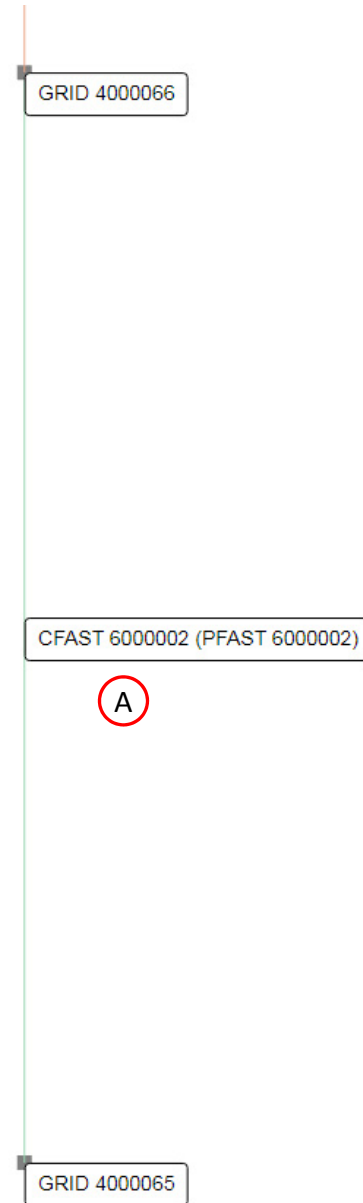
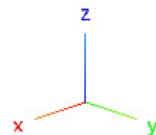
Model Display Panel

FEM Label



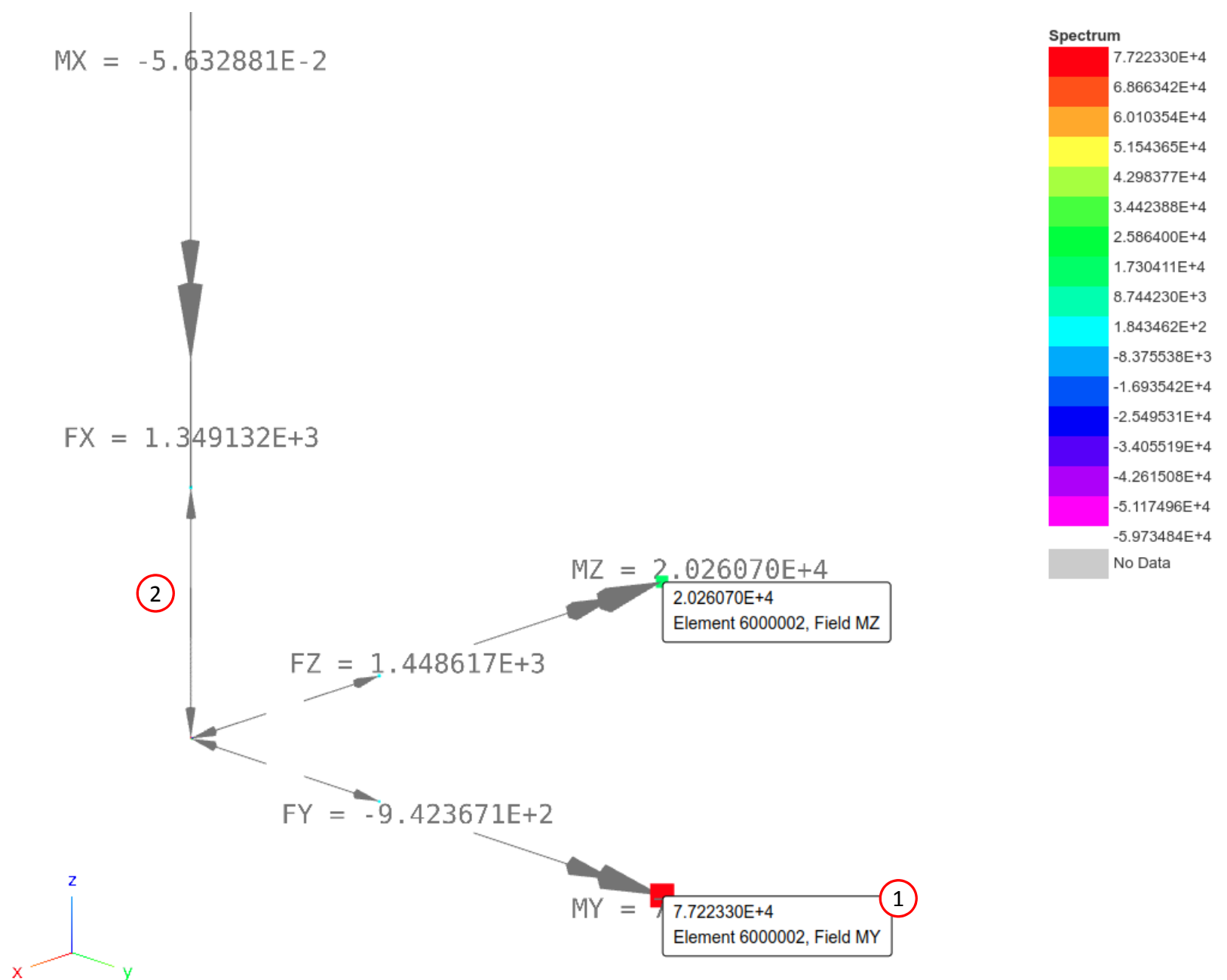
Nastran CFAST Force Results - CFAST and PFAST

- 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

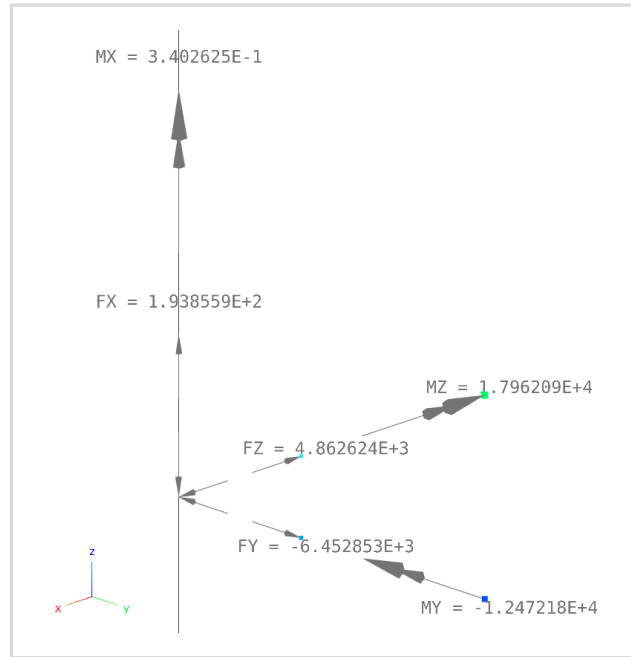
1. The label indicates the maximum response value and corresponding element ID and field name
2. For force along the x-axis (FX) of the CFAST element coordinate system (ECS), the sign is positive. The sign of the force does NOT always indicate tension or compression. Instead, use the displayed arrows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



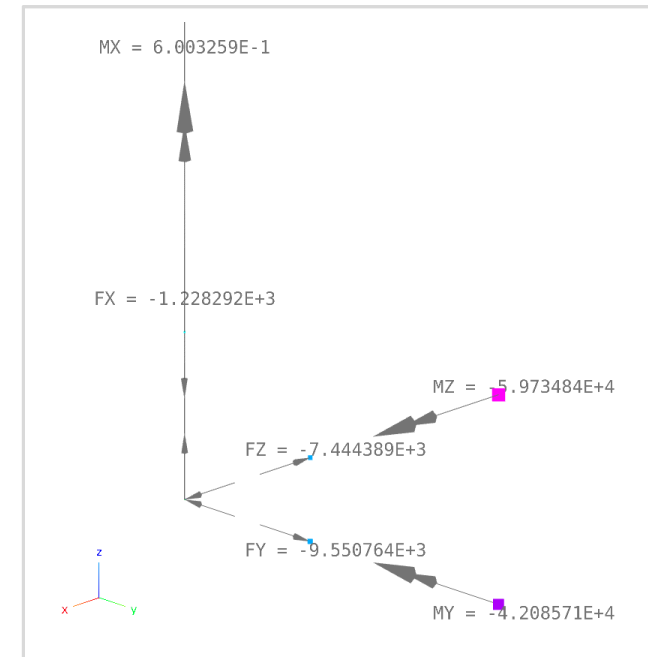
Nastran CFAST Force Results - CFAST and PFAST

- The forces for the other CFAST elements are displayed.

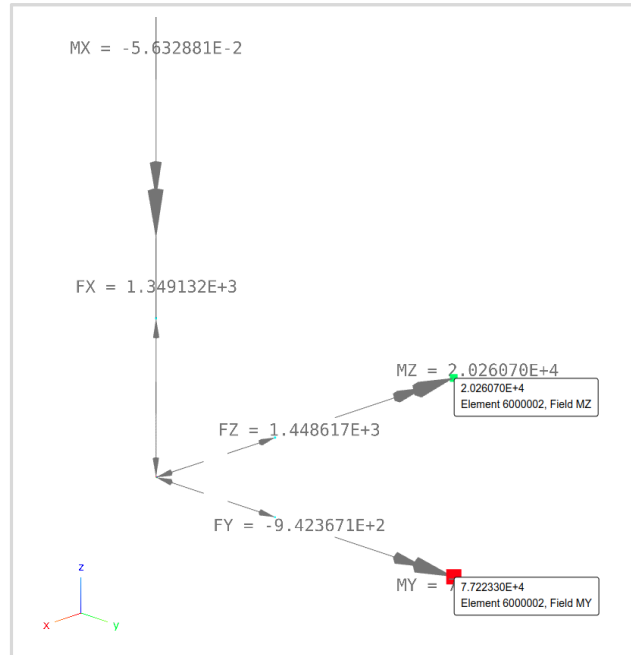
CFAST
6000003



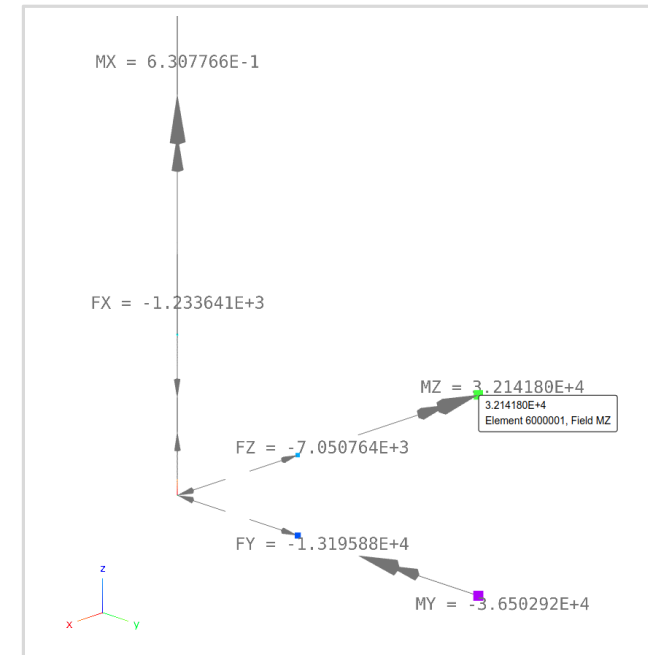
CFAST
6000000



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

SOL 200 Web App - HDF5 Explorer Acquire Dataset Plots Browser Combine Plots Last Plot Added Home

Acquire Dataset Session ID: 7016 HDF5

Select Dataset

- ELEMENTAL/ELEMENT_FORCE/BEAM
- ELEMENTAL/ELEMENT_FORCE/BUSH
- ELEMENTAL/ELEMENT_FORCE/FAST **2**
- ELEMENTAL/ELEMENT_FORCE/QUAD4
- ELEMENTAL/ELEMENT_FORCE/QUAD8
- ELEMENTAL/ELEMENT_FORCE/TRIA9

Specify Entities

ALL **4**

Element identification number (EID)
Examples: 6000000, 6000001, 6000002, etc.

Auto Execute **3**

Select Domains

Acquire Dataset **5**

Acquisition complete and successful

Acquired Dataset Create Plot Reset Filters Download CSV **7**

ELEMENTAL/ELEMENT_FORCE/FAST - ALL

EID	FX	FY	FZ	MX	MY	MZ	SAMPLE	DOMAIN_ID
Element identification number	Force x	Force y	Force z	Membrane force in x	Membrane force in y	Bending moment in z-direction	Name of H5 File**	Domain identi
6000000	-1228.2921142578...	-9550.763671875	-7444.388671875	0.600325882434845	-42085.7109375	-59734.84375	model	1
6000001	-1233.6412353515...	-13195.8837890625	-7050.76416015625	0.6307766437530...	-36502.91796875	32141.80078125	model	1
6000002	1349.1318359375							
6000003	193.8559112548828							
6000004	1551.648193359375							
6000005	178.5144500732422							
6000010	2368.254638671875							
6000011	2093.14990234375							
6000012	1973.3665771484...							
6000013	1891.2244873046...							
6000018	-2130.1022949218...							
6000019	-345.10778808593...							
6000020	-384.58908081054...							
6000021	-590.53137207031...							
6000022	-37.870433807373...							

acquired_dataset (1).csv - Excel

File Home Insert Page Layout Formulas Data Review View Tell me what you want to do... Sign in Share

Calibri 11 A A

Clipboard Font Alignment Number Styles Cells Editing

T33 : X ✓ ✗ 0

Dataset: ELEMENTAL/ELEMENT_FORCE/FAST	EID	FX	FY	FZ	MX	MY	MZ	SAMPLE	DOMAIN	SUBCASE	STEP	ANALYSIS	TIME	FREI	EIGI	MODE
1	6000000	-1228.29	-9550.76	-7444.39	0.600326	-42085.7	-59734.8	model	1	1	0	1	0	0	0	0
2	6000001	-1233.64	-13195.9	-7050.76	0.630777	-36502.9	32141.8	model	1	1	0	1	0	0	0	0
3	6000002	1349.132	-942.367	1448.617	-0.05633	77223.3	20260.7	model	1	1	0	1	0	0	0	0
4	6000003	193.8559	-6452.85	4862.624	0.340263	-12472.2	17962.09	model	1	1	0	1	0	0	0	0
5	6000004	1551.648	629.1017	442.8423	-0.06012	58209.53	-49585.4	model	1	1	0	1	0	0	0	0
6	6000005	178.5145	-5362.64	524.3932	0.328909	-8123.26	-28937.5	model	1	1	0	1	0	0	0	0
7	6000010	2368.255	11267.35	-6297.77	-0.05197	34373.32	35731.27	model	1	1	0	1	0	0	0	0
8	6000011	2093.15	7988.549	-4771.32	-0.01568	29944.22	-53492.5	model	1	1	0	1	0	0	0	0
9	6000012	1973.367	8322.856	5183.815	0.033897	-23786.1	-49369.3	model	1	1	0	1	0	0	0	0
10	6000013	1891.224	11202.71	6820.777	0.060717	-22339.5	27039.29	model	1	1	0	1	0	0	0	0
11	6000018	-2130.1	1096.276	-435.714	-0.10776	-33451.9	-2383.29	model	1	1	0	1	0	0	0	0
12	6000019	-345.108	-8292.22	4356.857	0.631806	-40149.5	-9028.26	model	1	1	0	1	0	0	0	0
13	6000020	-384.589	1706.069	-1982.02	-0.10941	-44585	-9808.15	model	1	1	0	1	0	0	0	0
14	6000021	-590.531	-7148.62	1368.651	0.661331	-39492.9	-14847	model	1	1	0	1	0	0	0	0
15	6000022	-37.8704	-1576.56	1103.675	0.006344	1919.087	-1505.78	model	1	1	0	1	0	0	0	0

acquired_dataset (1)

Ready

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/MS_C_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?

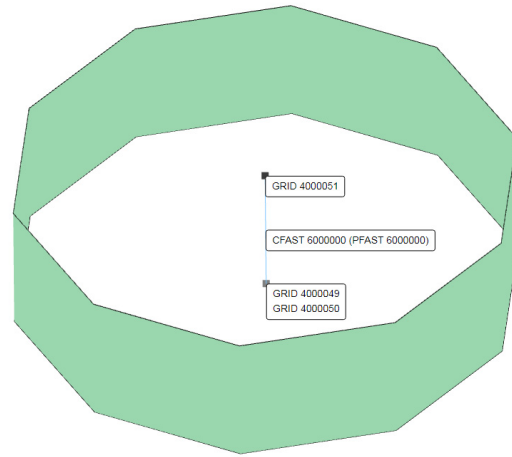
When a connector element, such as a CFAST element, is defined, MSC Nastran internally generates equivalent entries.

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.

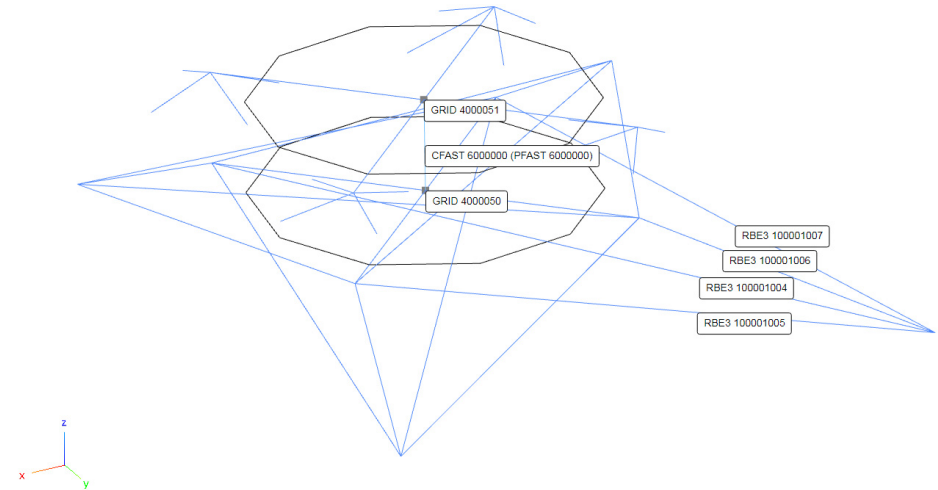
```
swldprm,prtsw,100
```

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
```



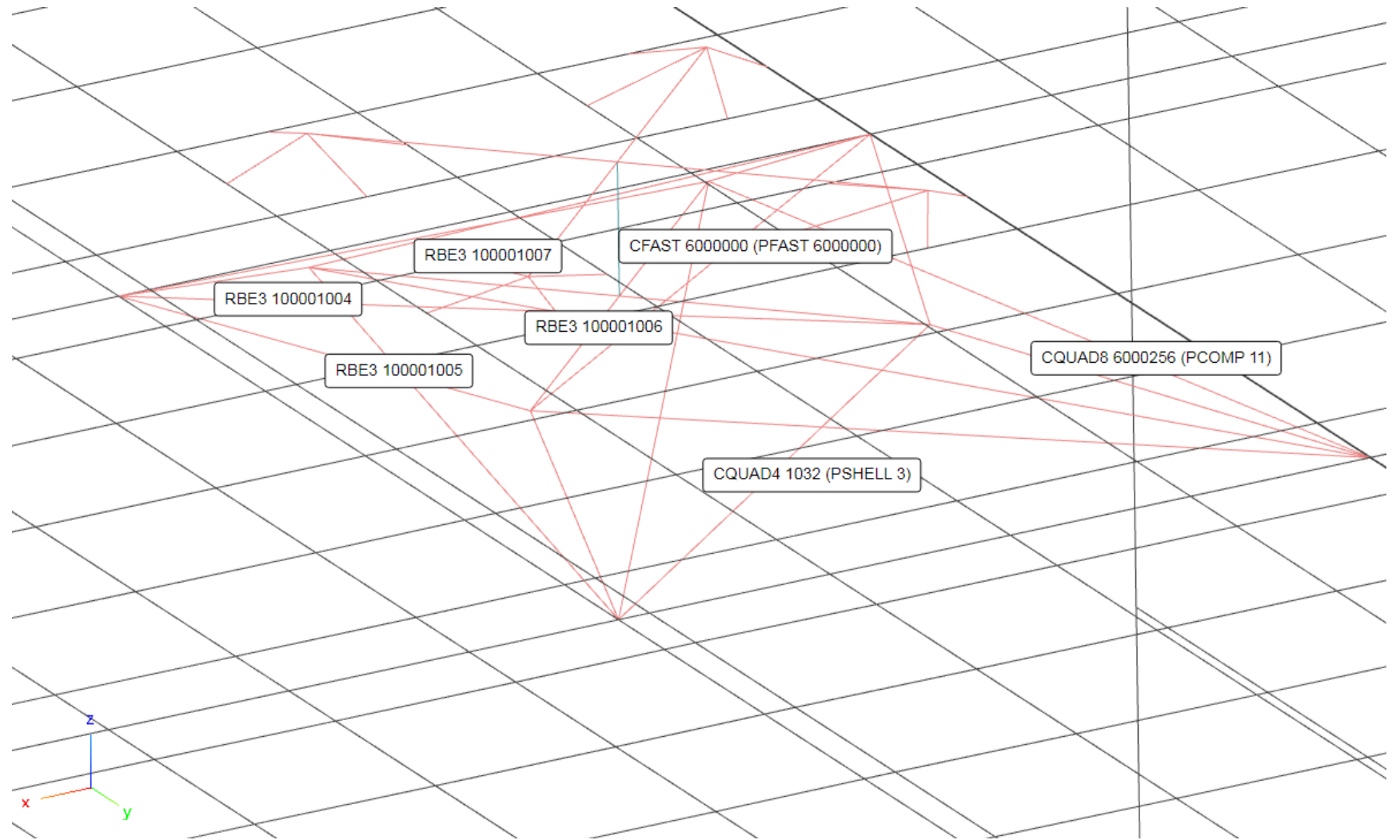
model.pch

```
$ 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 100000003 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.



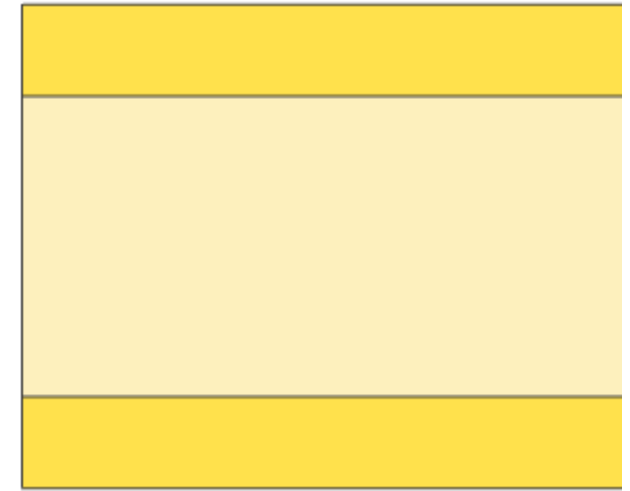
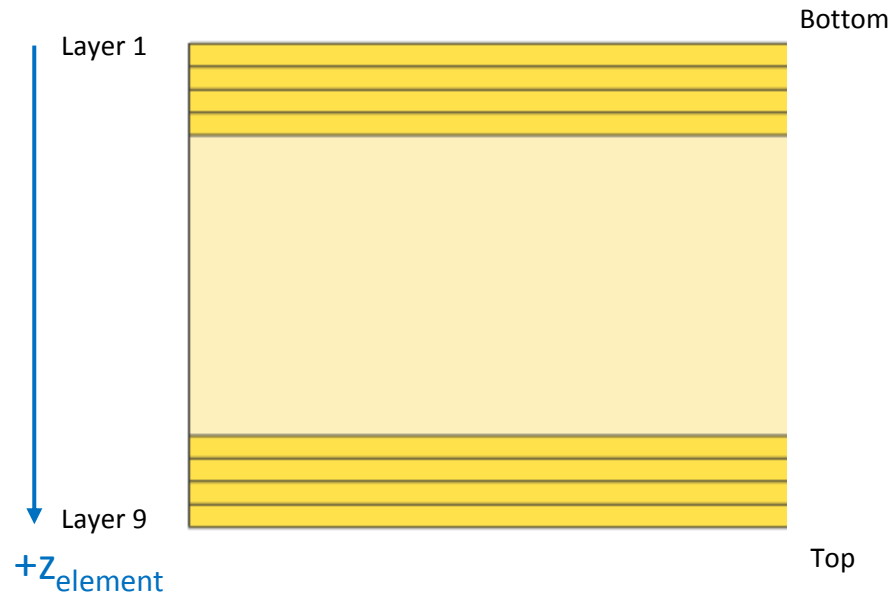
What are layers in a PCOMP and PCOMPG entry?

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?

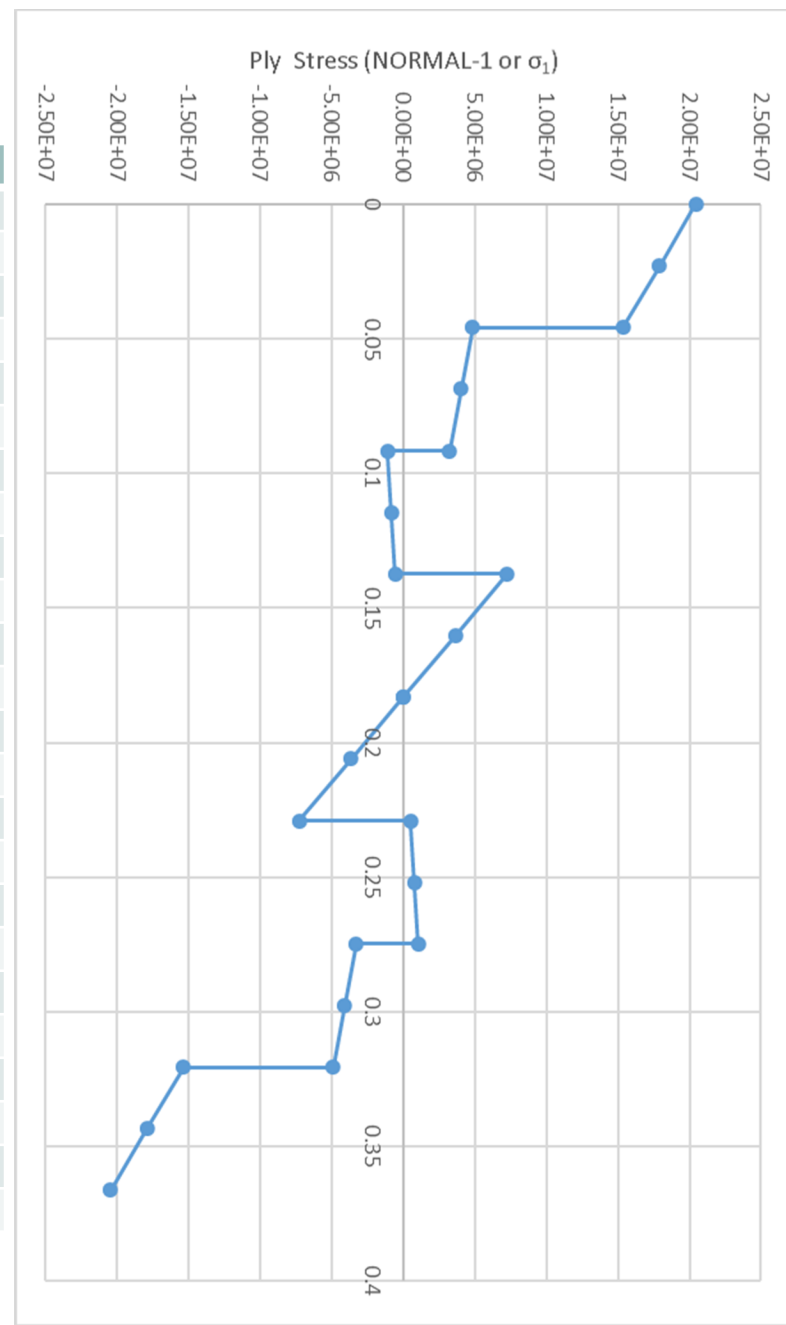
PCOMP and PCOMPG entries are used to define layers.

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	.0458	0.	YES		
1	1.0E-10	0.	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	.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

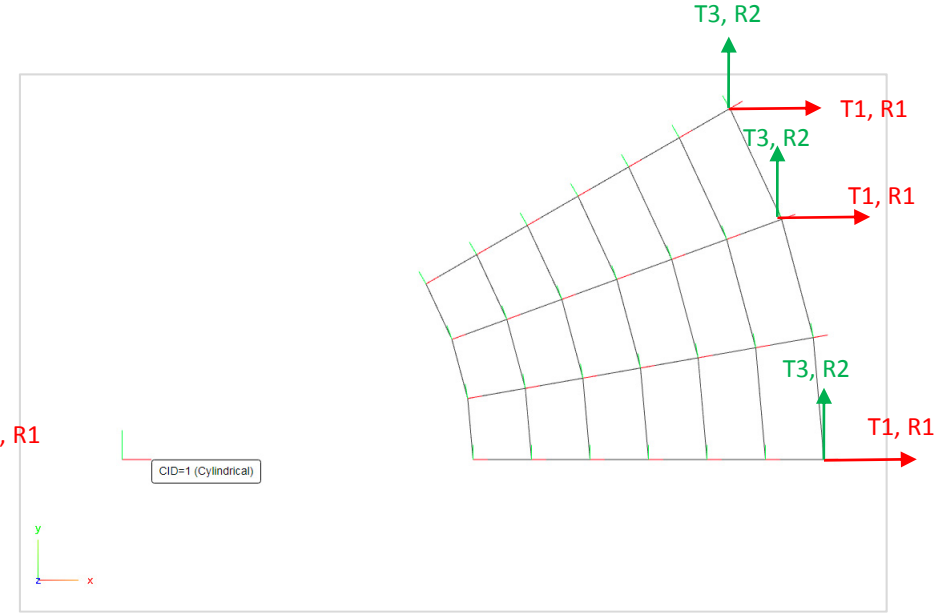
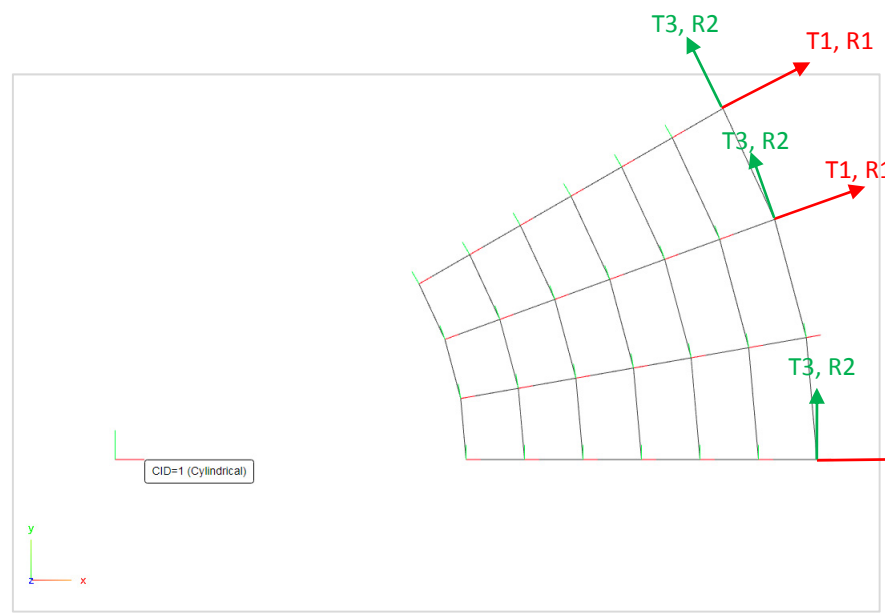
Consider these 2 examples.

- 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.
- 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.

model_a.bdf						①
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.			

model_b.bdf						②
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.	



Displacement Coordinate Systems

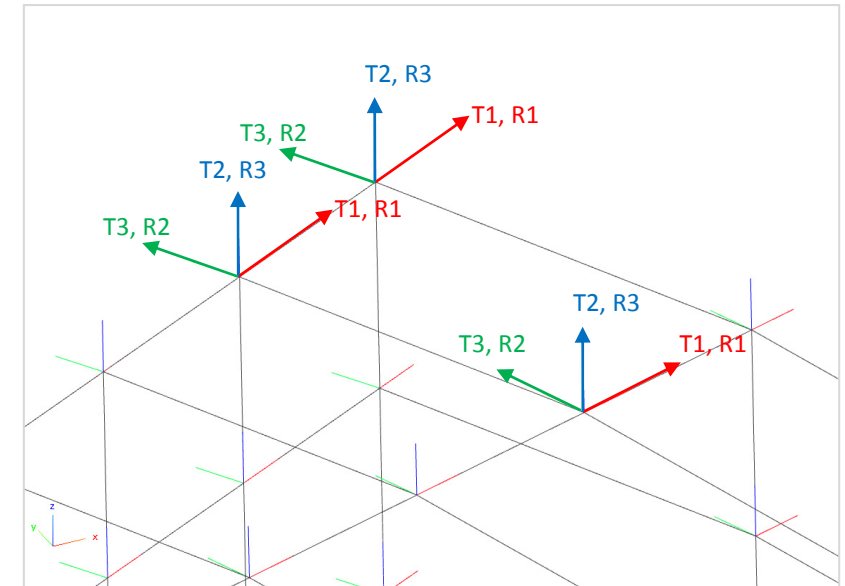
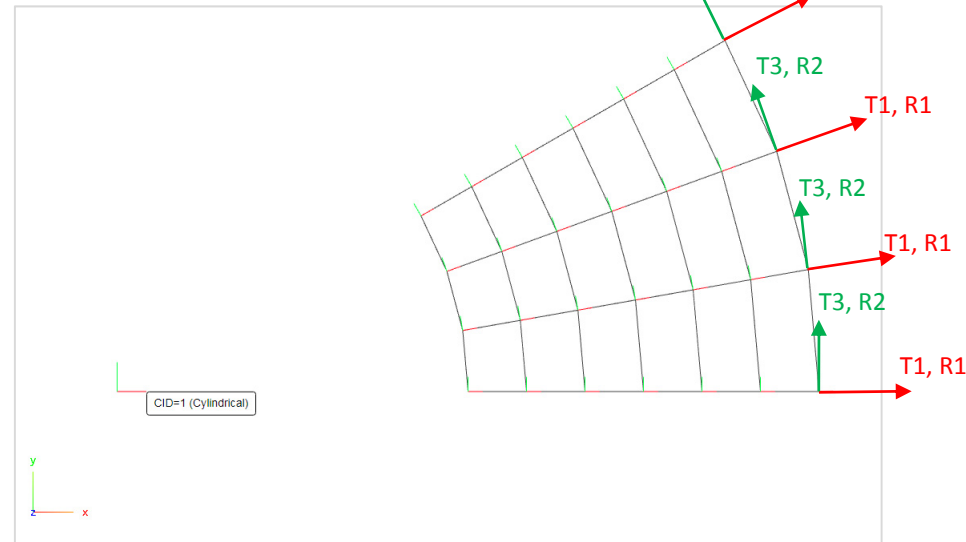
- When inspecting the F06 results output file, the displacements are reported in the displacement coordinate system.
- T1, T2, T3 represent the translation DOFs. R1, R2, R3 represent the rotation DOFs.
 - T1, T2, T3 are in units of length, e.g. mm, m, in., etc.
 - R1, R2, R3 are typically in radians.

0

SUBCASE 1

DISPLACEMENT VECTOR

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

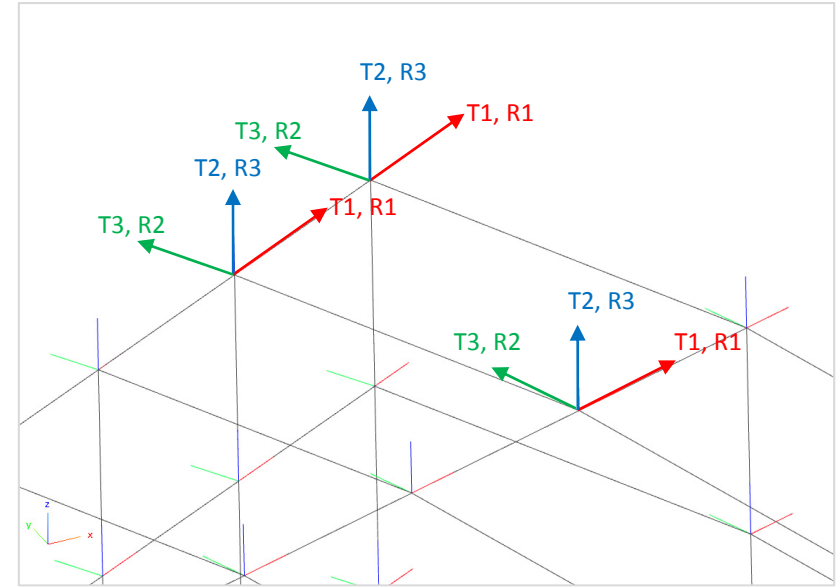
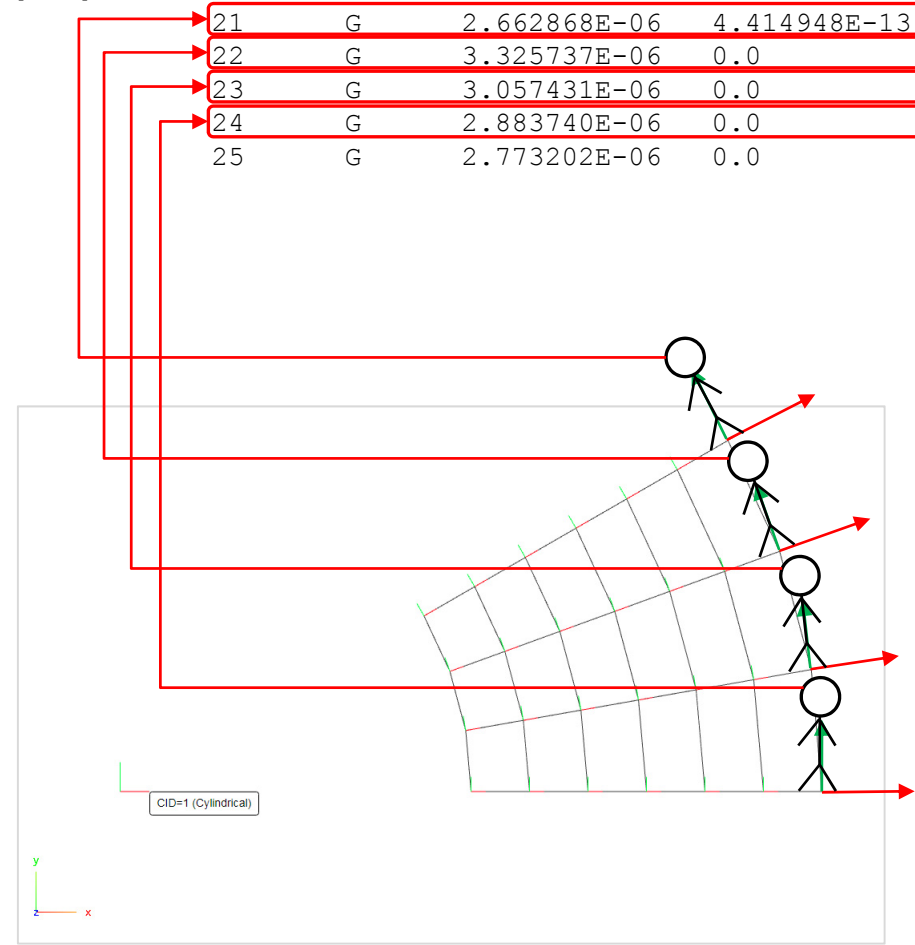
- One helpful strategy when interpreting results is to imagine an observer in each coordinate system. It is from each observer's (displacement coordinate system) perspective that results are reported.

0

SUBCASE 1

DISPLACEMENT VECTOR

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 <td 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.

