Introduction to MSC Nastran Results

Introduction to MSC Nastran Results

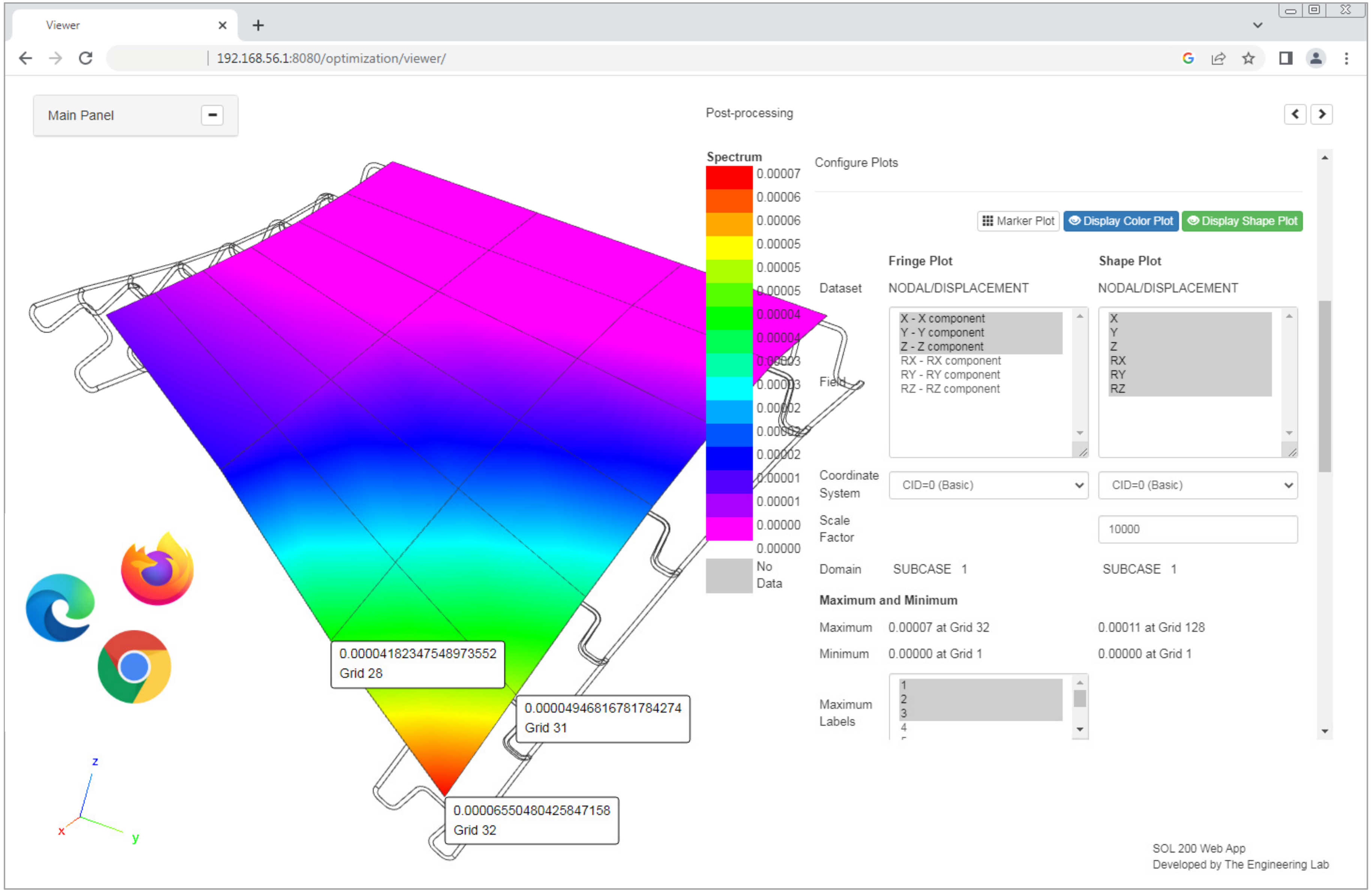
The goal of this document is to help readers interpret the results generated by MSC Nastran. This document briefly covers how to interpret displacements in various coordinate systems and element results for 3D, 2D and 1D elements, e.g. CHEXA, CQUAD4 and CBAR.

Introduction to the Post-processor Web App

During this presentation, the Post-processor Web App is used to display MSC Nastran results.

5 Things to Know about the Post-processor Web App

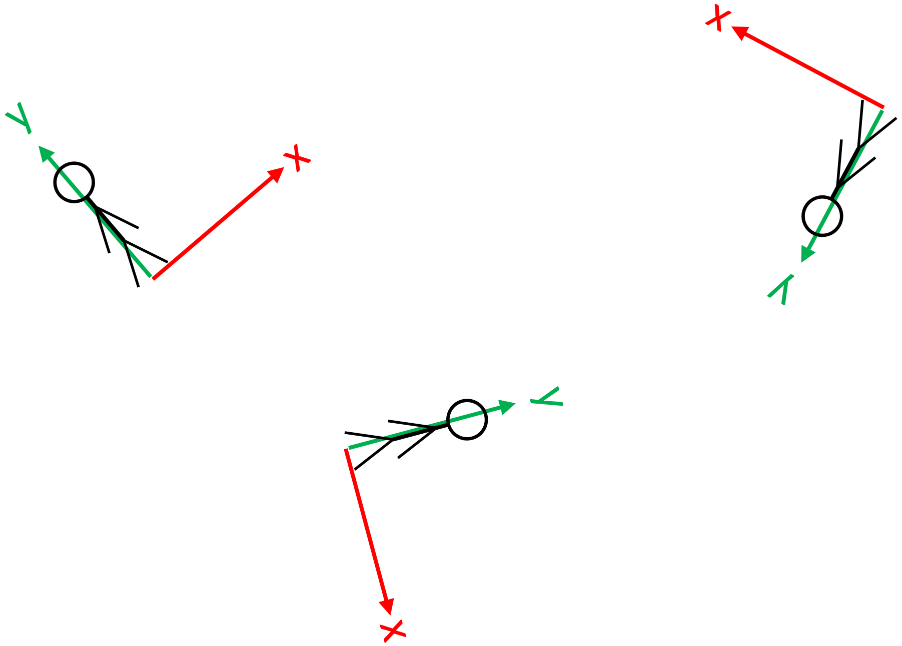
1. The Post-processor Web app is used to create fringe and marker plots, XYPLOTs (graphs) of MSC Nastran results. MSC Nastran support only.
2. Accessed via Google Chrome, Mozilla Firefox or Microsoft Edge
3. Support for Windows, Linux, Mac
4. **Free** to MSC Nastran users
5. Visit this link for access:
   - the-engineering-lab.com



MSC Nastran Results

When inspecting Nastran results, there is one question that should always be asked.

- **In which coordinate system are the Nastran results being observed in?**

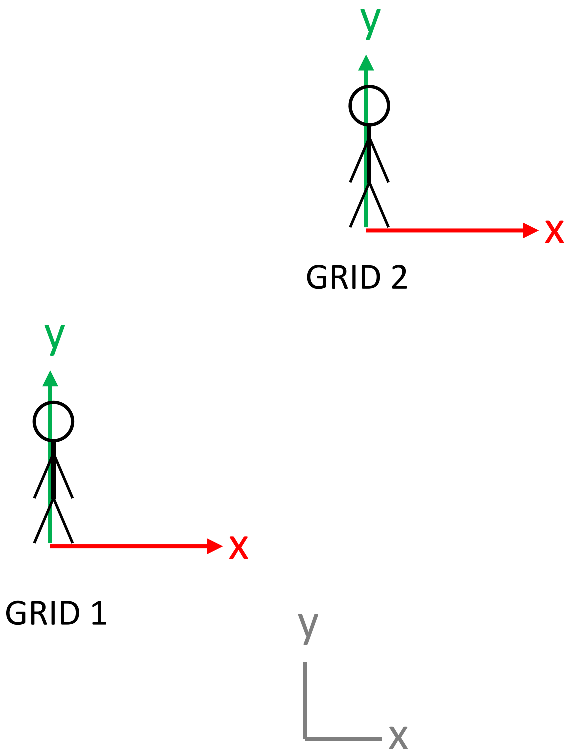


Displacements/Vectors

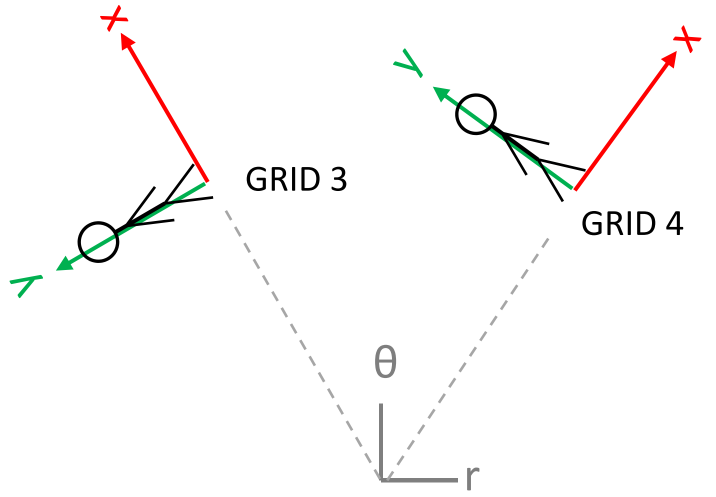
Displacements/Vectors

- Displacements at GRIDs (nodes) are reported as vectors in the GRID's displacement coordinate system
- Every GRID (node) has its own displacement coordinate system
- Consider GRIDs (nodes) with displacement coordinate systems in the rectangular, cylindrical and spherical coordinate systems

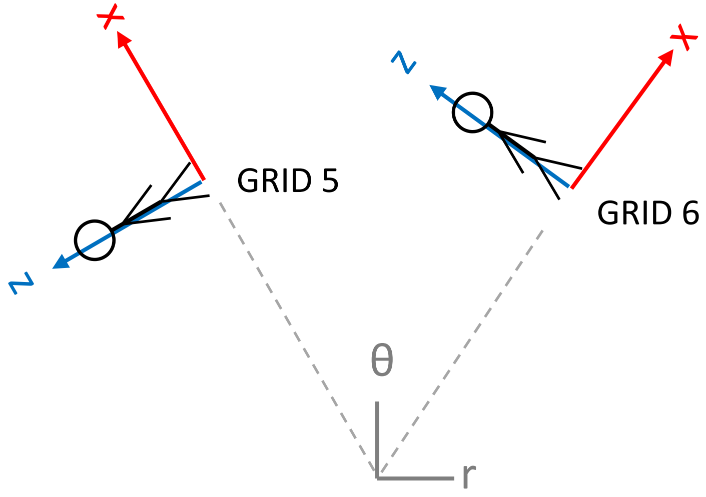
GRID Displacement Coordinate Systems



Rectangular
Position: (x, y, z)
Vector: <x, y, z>



Cylindrical
Position: (r, θ, z)
Vector: <x, y, z>



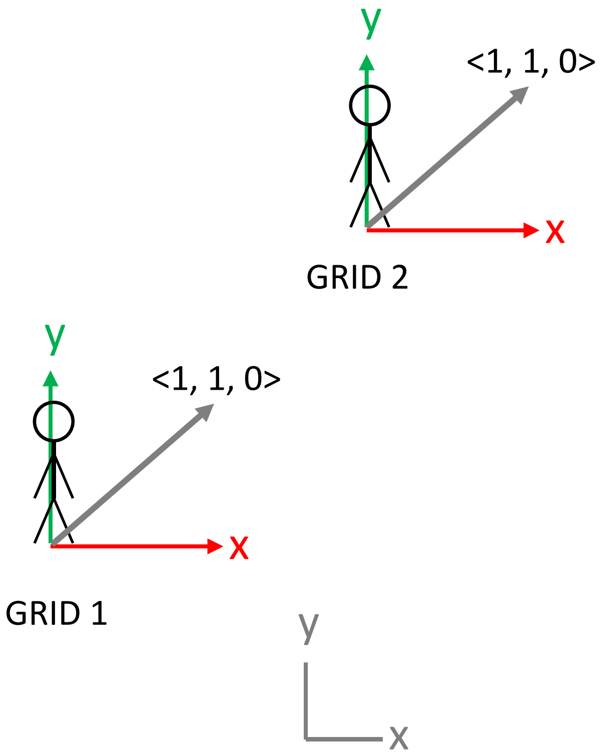
Spherical
Position: (r, θ, φ)
Vector: <x, y, z>

GRID Displacement Coordinate Systems

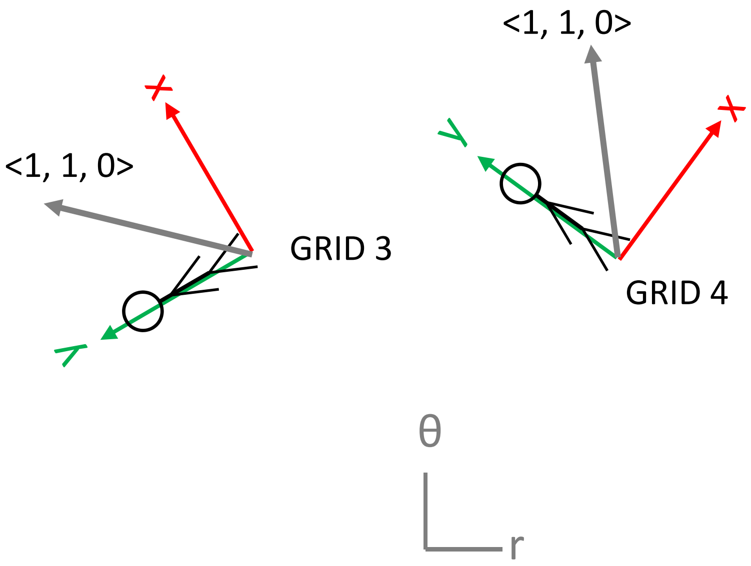
- If a rectangular coordinate system is used, the displacement coordinate systems are parallel. If a cylindrical or spherical coordinate system is used, then the displacement coordinate system orientations depend on the position of the GRID.

- Consider vector <1, 1, 0> in each displacement coordinate system

GRID Displacement Coordinate Systems

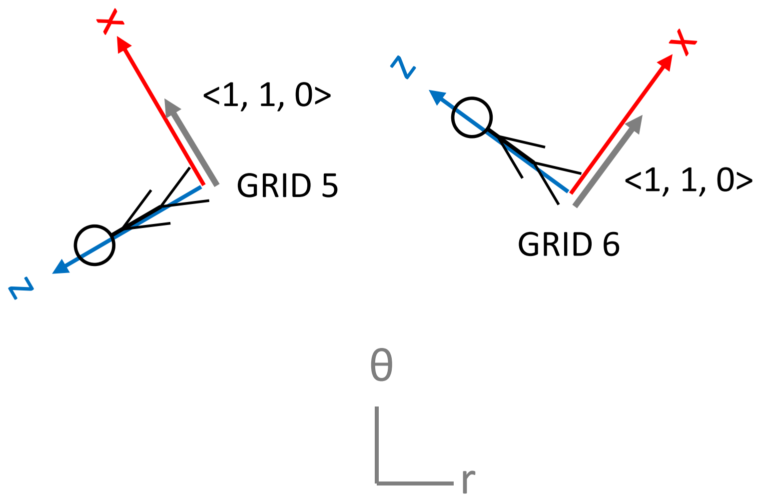


Rectangular
Position: (x, y, z)
Vector: <x, y, z>



Cylindrical
Position: (r, θ, z)
Vector: <x, y, z>

The y-axis points inward, so the vector <1, 1, 0> points inward.



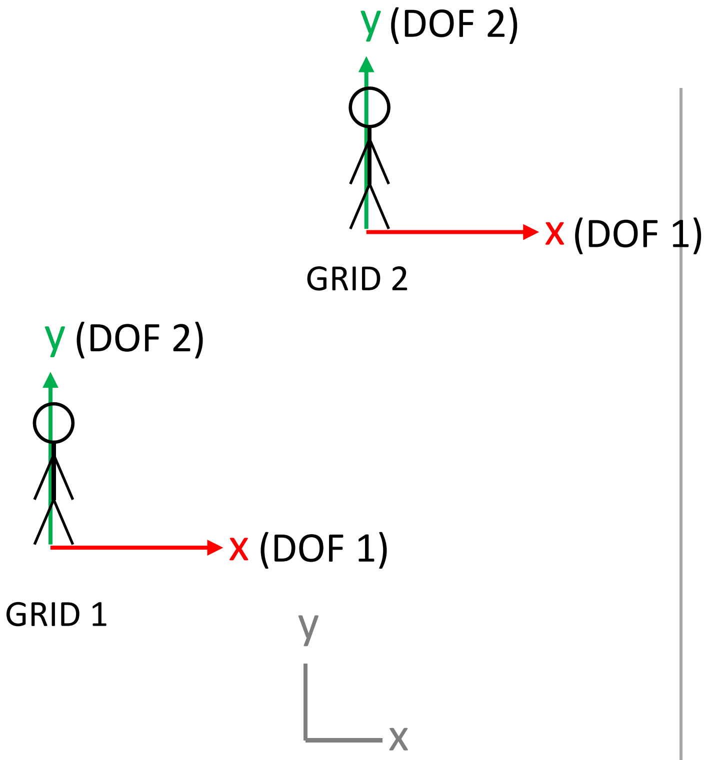
Spherical
Position: (r, θ, φ)
Vector: <x, y, z>

GRID Displacement Coordinate Systems

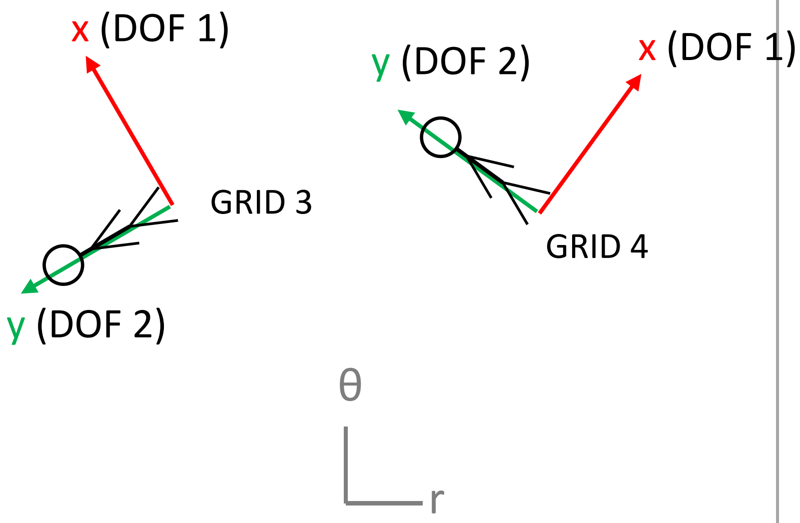
- Vectors, such as beam orientation vectors, beam offsets, displacements, are observed in the GRID displacement coordinate systems.

- The GRID displacement coordinate systems also determine the degrees of freedom (DOFs) used to define the stiffness matrix [K].

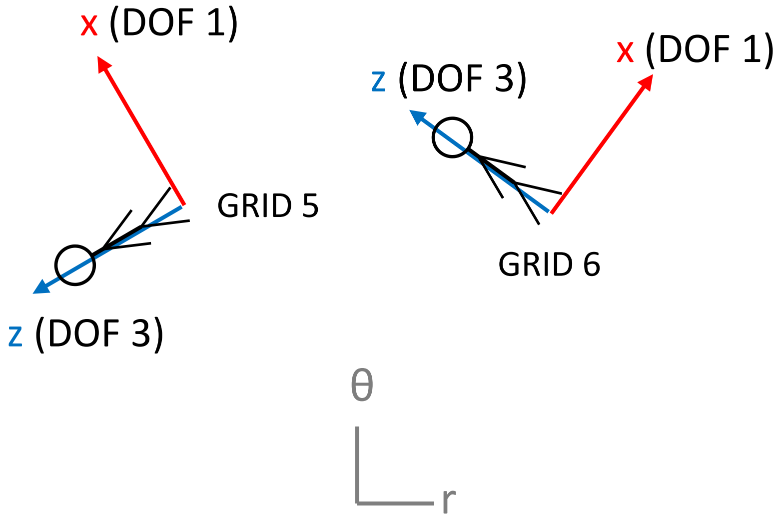
GRID Displacement Coordinate Systems



Rectangular
Position: (x, y, z)
Vector: <x, y, z>



Cylindrical
Position: (r, θ, z)
Vector: <x, y, z>



Spherical
Position: (r, θ, φ)
Vector: <x, y, z>

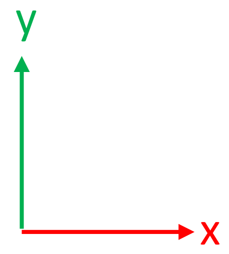
GRID Displacement Coordinate Systems

Field 7 of the GRID entry specifies the coordinate system ID (CID) of the displacement coordinate system.

```
GRID    1       1       3.      0.      0.      1
GRID    2       1       3.24314 0.      0.      1
GRID    3       1       3.50273 0.      0.      1
GRID    4       1       3.77988 0.      0.      1
CORD2C  1               0.      0.      0.      0.      0.      1.      +
+     AT 1.     0.      0.
```

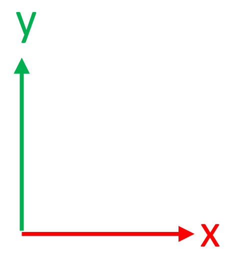
First Coordinate System

When constructing a finite element model, there is always the first coordinate system that is defined. All subsequent coordinate systems are relative to the first coordinate system.



First Coordinate System

The name of the first coordinate system varies



FEA Class

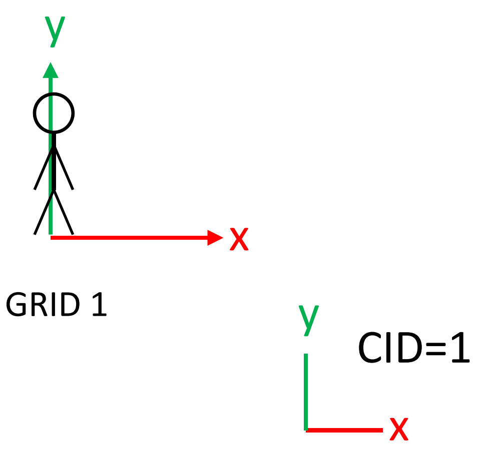
“Global Coordinate System”

MSC Nastran

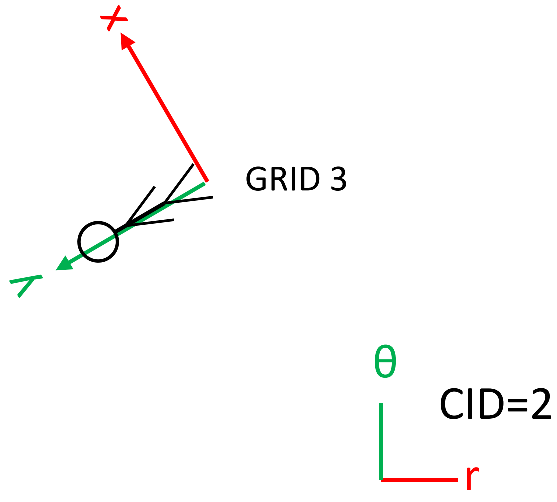
“Basic Coordinate System”

Global Coordinate System

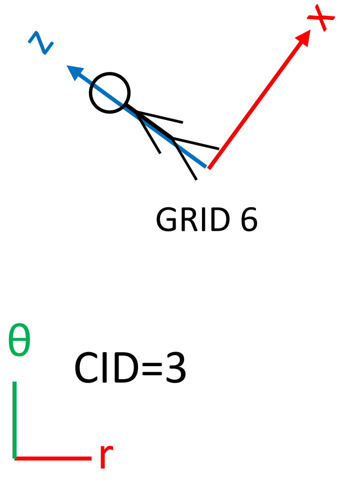
MSC Nastran defines the global coordinate system as all the coordinate systems used to define the DOFs of the stiffness matrix. For the example below, the global coordinate system consists of coordinate systems 1, 2 and 3 to define the displacement coordinate systems of GRIDs 1, 3, and 6.



Rectangular
Position: (x, y, z)
Vector: <x, y, z>



Cylindrical
Position: (r, θ, z)
Vector: <x, y, z>



Spherical
Position: (r, θ, φ)
Vector: <x, y, z>

MSC Nastran Coordinate Systems

Per the MSC Nastran Reference Guide,

"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."

3D Element Stresses and Strains

3D Element Stresses and Strains

- The F06 file reports 3D element stresses in the material coordinate system. This is controlled via the PSOLID entry's CORDM field.
- 3D element stresses and strains are reported at the center and corners of the element.

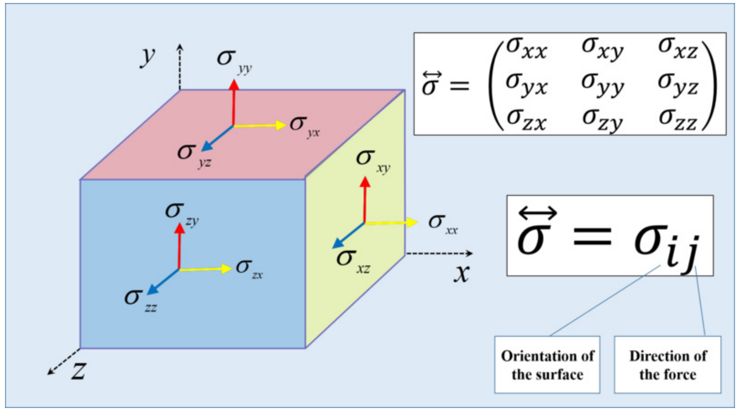
F06 Output

```
                         S T R A I N S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S   ( H E
0                   CORNER         ------CENTER AND CORNER POINT  STRAINS---------        DIR.  COSINES
  ELEMENT-ID       GRID-ID         NORMAL                  SHEAR              PRINCIPAL       -A-  -B-  -C-
0         638          99GRID CS   8 GP
0                   CENTER  X   1.5      E-03   XY   5.2      E-10    A    1.616575E-03  LX 0.99-0.10 0.00   -1.
                            Y  -4.4      E-04   YZ   5.6      E-10    B   -7.094377E-04  LY 0.00-0.00-1.00
                            Z  -6.8      E-04   ZX   4.7      E-04    C   -4.464904E-04  LZ 0.10 0.99-0.00
0                      199  X   2.8      E-03   XY   8.5      E-10    A    2.894503E-03  LX 1.00-0.06 0.00   -3.
                            Y  -7.891483E-04    YZ   2.644298E-10     B   -1.093295E-03  LY 0.00-0.00-1.00
                            Z  -1.079092E-03    ZX   4.751221E-04     C   -7.891483E-04  LZ 0.06 1.00-0.00
[... Data for 6 nodes hidden ...]
0                      293  X   3.632144E-04    XY   1.822795E-10     A    4.421220E-04  LX 0.95-0.32 0.00    4.
                            Y  -2.205963E-04    YZ   8.727141E-10     B   -3.519919E-04  LY 0.00-0.00-1.00
                            Z  -2.730842E-04    ZX   4.751221E-04     C   -2.205963E-04  LZ 0.32 0.95-0.00
```

σ_{xx}, σ_{yy}, σ_{zz}, σ_{xy}, σ_{yz}, σ_{zx}

BDF File

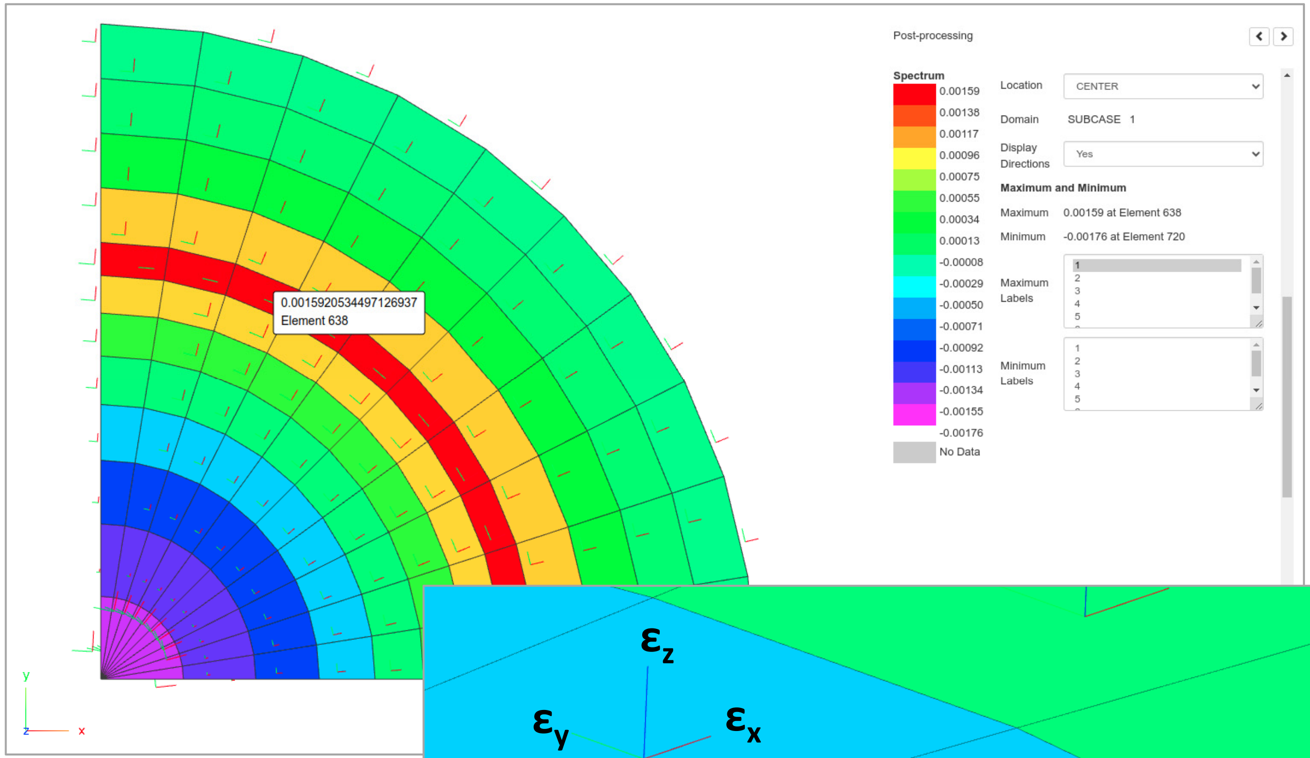
```
PSOLID   1       1       99
CORD2C   99      0.      0.      0.      0.      0.      1.      +       AAQ
+     AAQ 1.     0.      0.
```

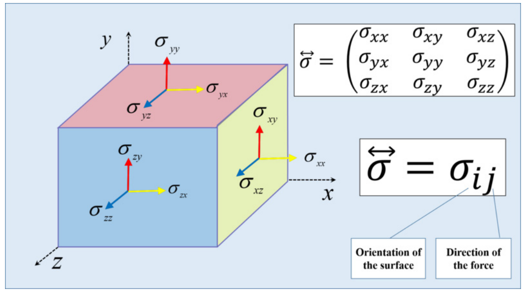


Source: Wiki

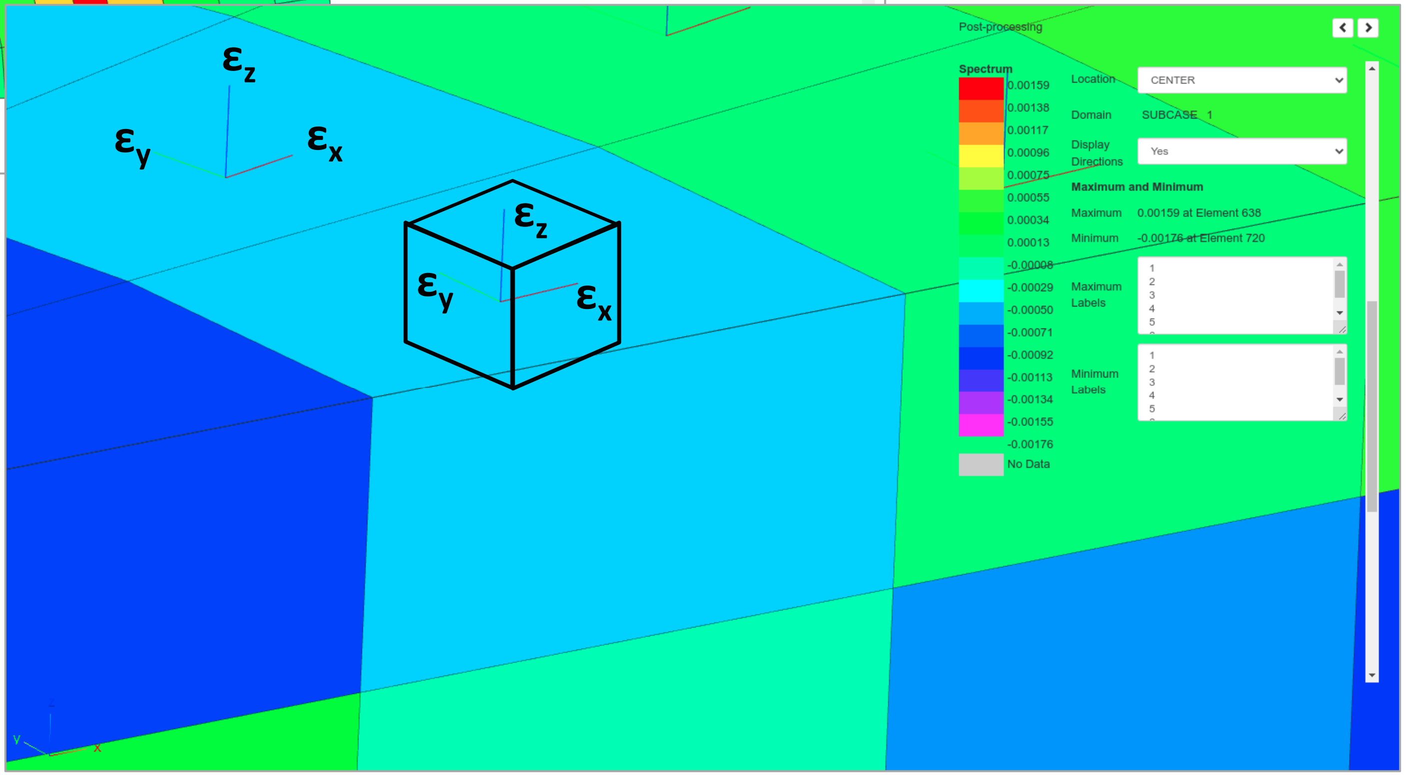
3D Element Stresses and Strains

- In this example, the Post-processor Web App is used to display the x component of strain ($\boldsymbol{\varepsilon}_x$) at the center of each element.
- A coordinate system is displayed for each element and indicates the general direction of the components of the strain or stress tensor.



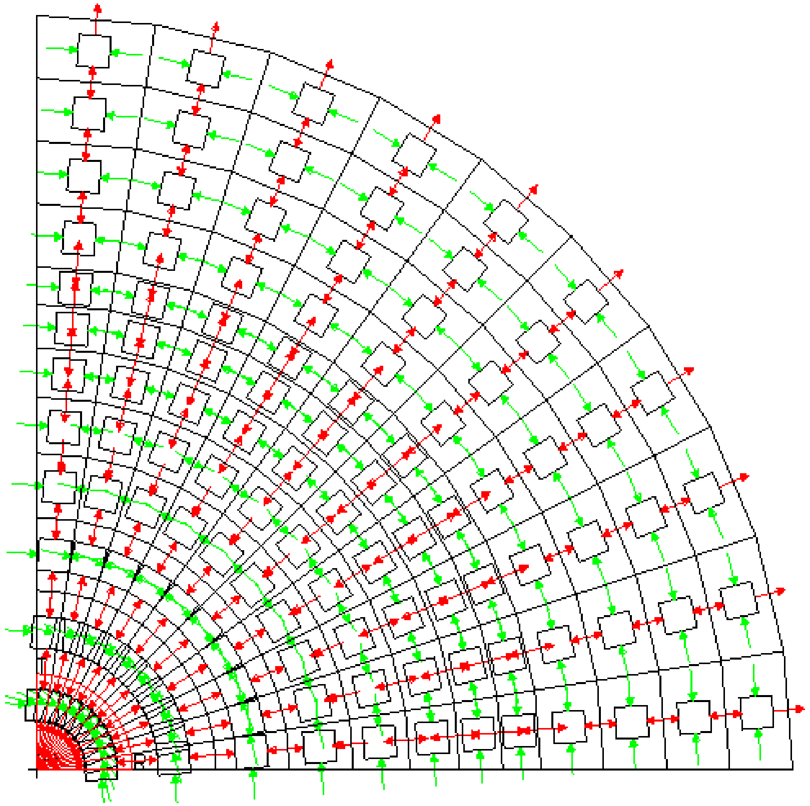


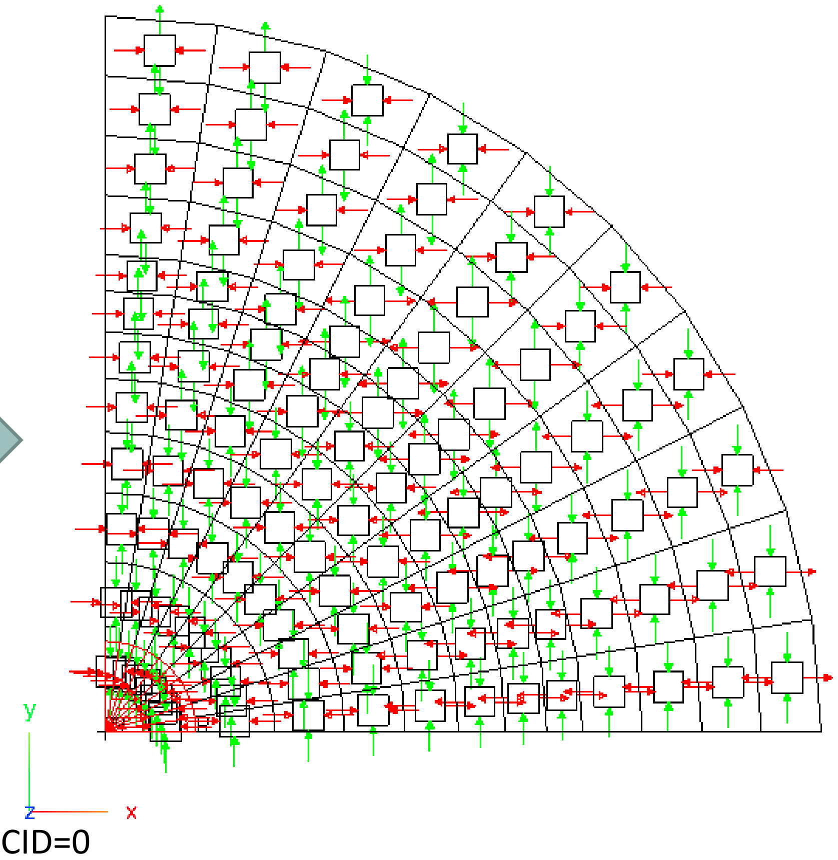
Source: Wiki

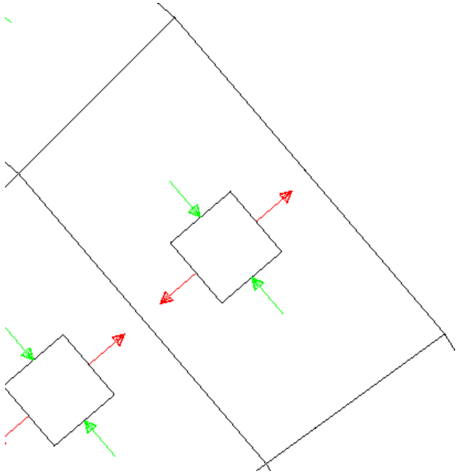


3D Element Stresses and Strains

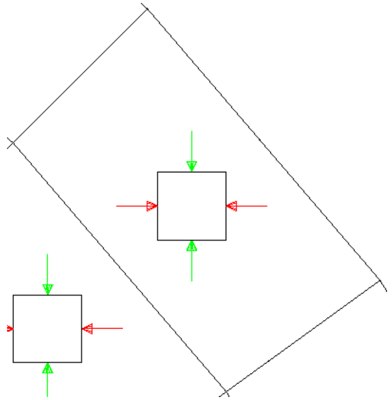
- Alternatively, a post-processor may be used to transform the stresses or strains to a user defined coordinate system.
- In this example, Patran is used to transform and display the strain tensors.







Element results observed in CID=1 (Cylindrical)



Element results observed in CID=0 (Basic CS)(Rectangular)

2D Element Stresses and Strains

2D Element Stresses and Strains

- The F06 file reports 2D element stresses in one of 2 coordinate systems.
  - Element coordinate system (Default)
  - Material coordinate system
    - Requires defining the material coordinate system for each element.
    - Use PARAM,OMID,YES or BOTH to output tensors in the material coordinate system.
- Alternatively, a post-processor may be used to transform the results.

F06 Output (Default)

```
                        S T R E S S E S   I N   T R I A N G U L A R   E L E M E N T S   ( T R I A 6 )

    ELEMENT                 FIBER            STRESSES IN ELEMENT COORD SYSTEM          PRINCIPAL STRESSES (ZERO SHEAR)
      ID       GRID-ID    DISTANCE         NORMAL-X      NORMAL-Y      SHEAR-XY       ANGLE         MAJOR           MINOR
0       211     CEN/6   -5.000000E-01  -9.419119E+01  1.194960E+02 -6.050703E+00   -88.3794   1.196672E+02 -9.436238
                          5.000000E-01  -9.419119E+01  1.194960E+02 -6.050703E+00   -88.3794   1.196672E+02 -9.436238

                  363   -5.000000E-01  -1.000644E+02  1.239453E+02  4.536549E-02    89.9884   1.239453E+02 -1.000645
                          5.000000E-01  -1.000644E+02  1.239453E+02  4.536549E-02    89.9884   1.239453E+02 -1.000645

                  365   -5.000000E-01  -8.373954E+01  1.094619E+02  4.331688E-01    89.8715   1.094629E+02 -8.374051
                          5.000000E-01  -8.373954E+01  1.094619E+02  4.331688E-01    89.8715   1.094629E+02 -8.374051

                  425   -5.000000E-01  -9.876957E+01  1.250808E+02 -1.863064E+01   -85.2747   1.266208E+02 -1.003096
                          5.000000E-01  -9.876957E+01  1.250808E+02 -1.863064E+01   -85.2747   1.266208E+02 -1.003096
```

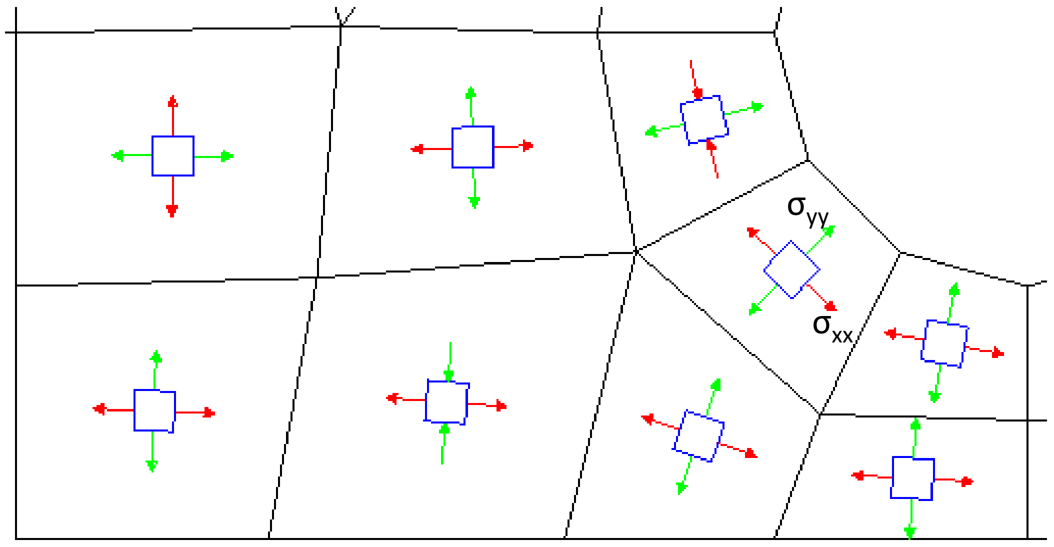
F06 Output (PARAM,OMID,BOTH)

```
                          S T R E S S E S   I N   T R I A N G U L A R   E L E M E N T S   ( T R I A 6 )

    ELEMENT                 FIBER            STRESSES IN MATERIAL COORD SYSTEM         PRINCIPAL STRESSES (ZERO SHEAR)
      ID       GRID-ID    DISTANCE         NORMAL-X      NORMAL-Y      SHEAR-XY       ANGLE         MAJOR           MINOR
0       211     CEN/6   -5.000000E-01  -9.436238E+01  1.196672E+02  1.069910E-02    89.9971   1.196672E+02 -9.436238
                          5.000000E-01  -9.436238E+01  1.196672E+02  1.069910E-02    89.9971   1.196672E+02 -9.436238

                  363   -5.000000E-01  -9.988267E+01  1.237635E+02  6.378907E+00    88.3676   1.239453E+02 -1.000645
                          5.000000E-01  -9.988267E+01  1.237635E+02  6.378907E+00    88.3676   1.239453E+02 -1.000645

                  365   -5.000000E-01  -8.355938E+01  1.092817E+02  5.912978E+00    88.2454   1.094629E+02 -8.374051
                          5.000000E-01  -8.355938E+01  1.092817E+02  5.912978E+00    88.2454   1.094629E+02 -8.374051

                  425   -5.000000E-01  -9.969705E+01  1.260083E+02 -1.177387E+01   -87.0220   1.266208E+02 -1.003096
                          5.000000E-01  -9.969705E+01  1.260083E+02 -1.177387E+01   -87.0220   1.266208E+02 -1.003096
```

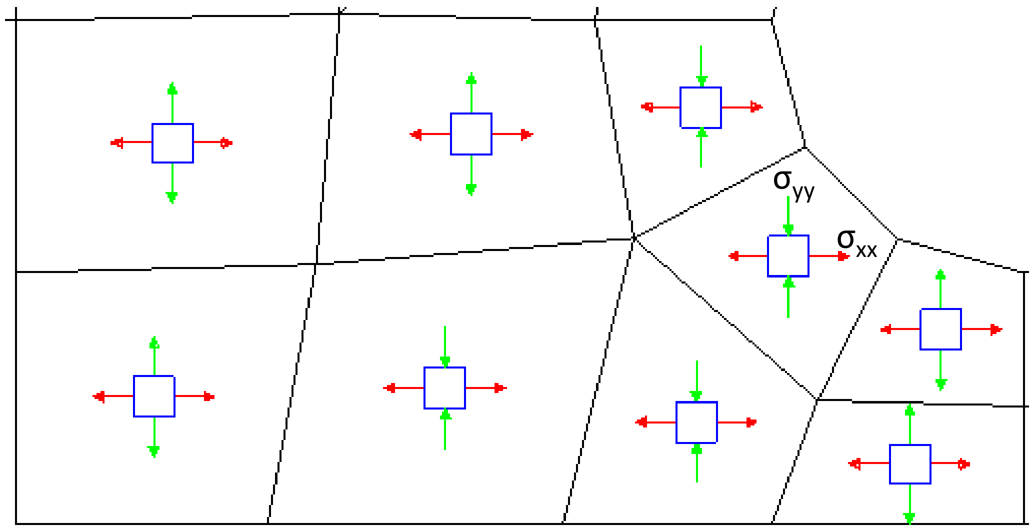
2D Element Stresses and Strains

- Caution! The default output of 2D element stresses is in the element coordinate system. When inspecting the stresses or strains for 2D elements in the element coordinate system, the components of the stress tensor have different directions as shown in the image to the right.
- It is recommended that the results be viewed in the material coordinate system OR a post-processor is used to transform the results to a consistent coordinate system.

Output observed in element coordinate systems

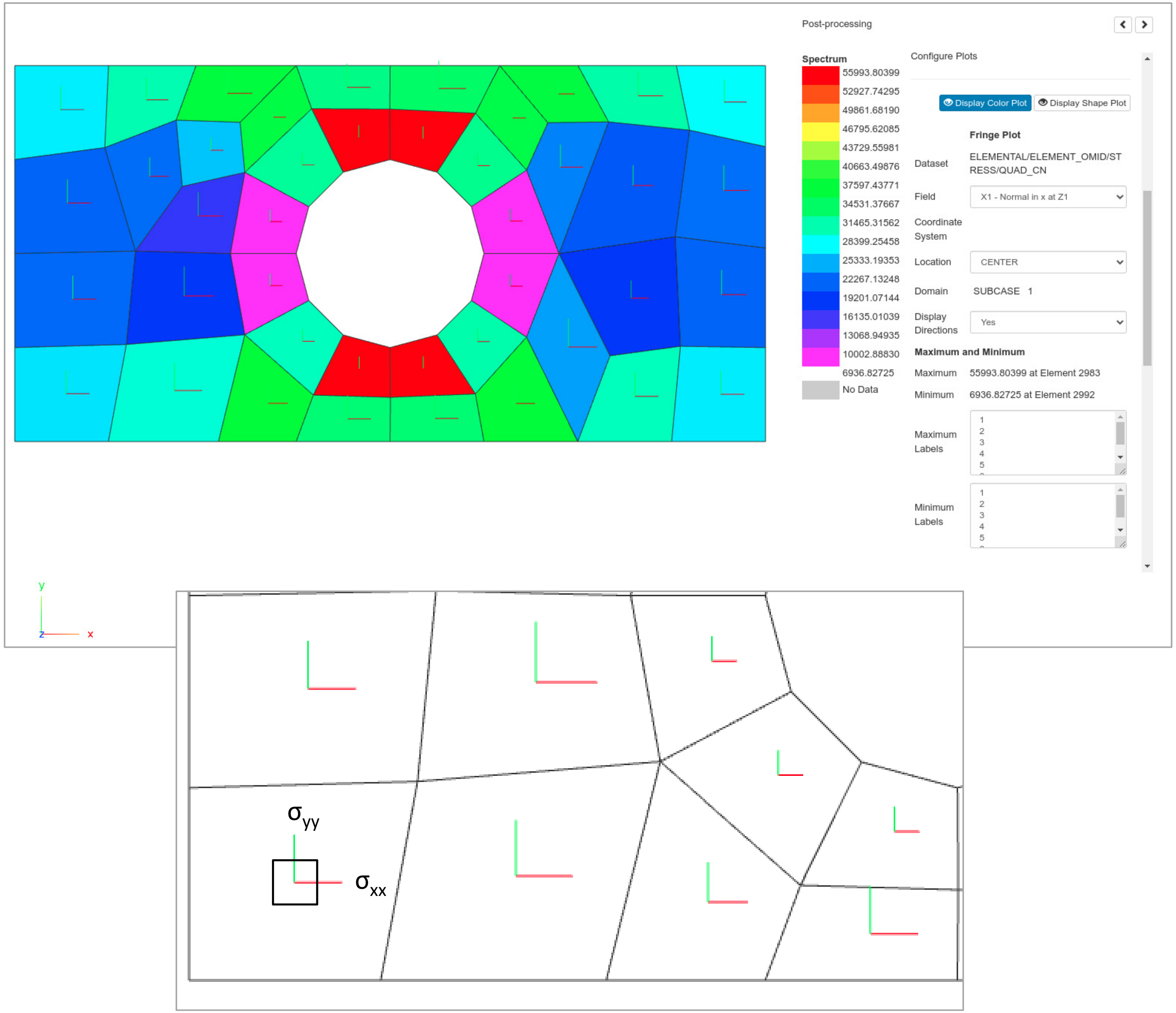


Output observed in material coordinate systems



2D Element Stresses and Strains

- The Post-processor Web App is used to display the x component of stress (σ_x) as observed in the material coordinate system CID=0.

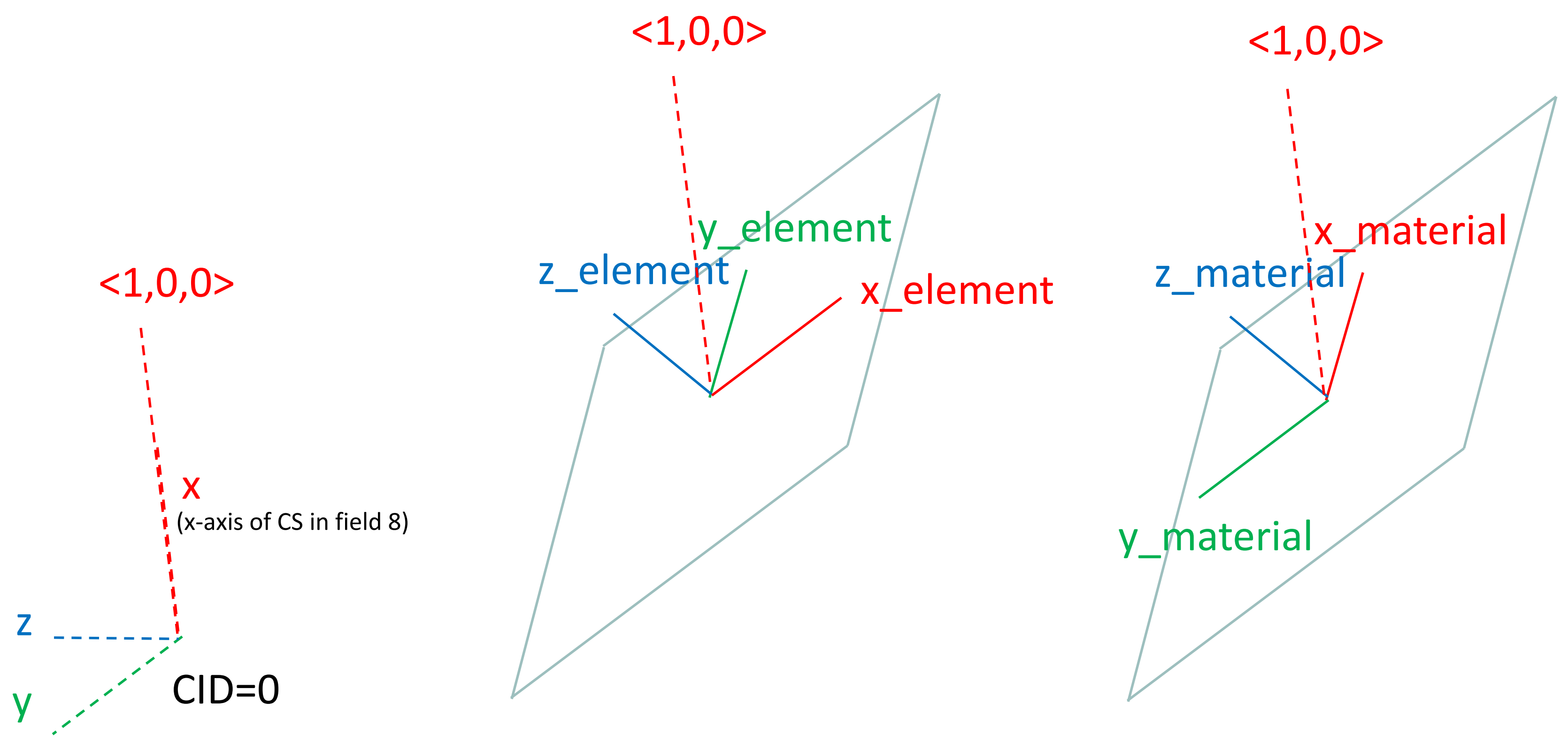


2D Element Stresses and Strains

- The material coordinate system is specified in field 8 of the CQUAD4 element. For other 2D elements, the field number varies.
- In this example, CID=0 (basic coordinate system) is used to align the material stiffness (material coordinate system) for the element. Stress outputs will be output in the material coordinate system as long as one of these bulk data entries is used.
  - PARAM,OMID,YES
  - PARAM,OMID,BOTH
- The material coordinate system is determine as follows and involves cross products.
  1. z_material = z_element
  2. y_material = z_element X <1, 0, 0>
  3. x_material = y_material X z_material

  It should be observed that these cross products project the x-axis of the specified coordinate system, in this case CID=0, onto the element face.
- Important! The x-axis of the coordinate system defined in field 8 must not be parallel to the element's z-axis, else the cross product for y_material cannot be defined.

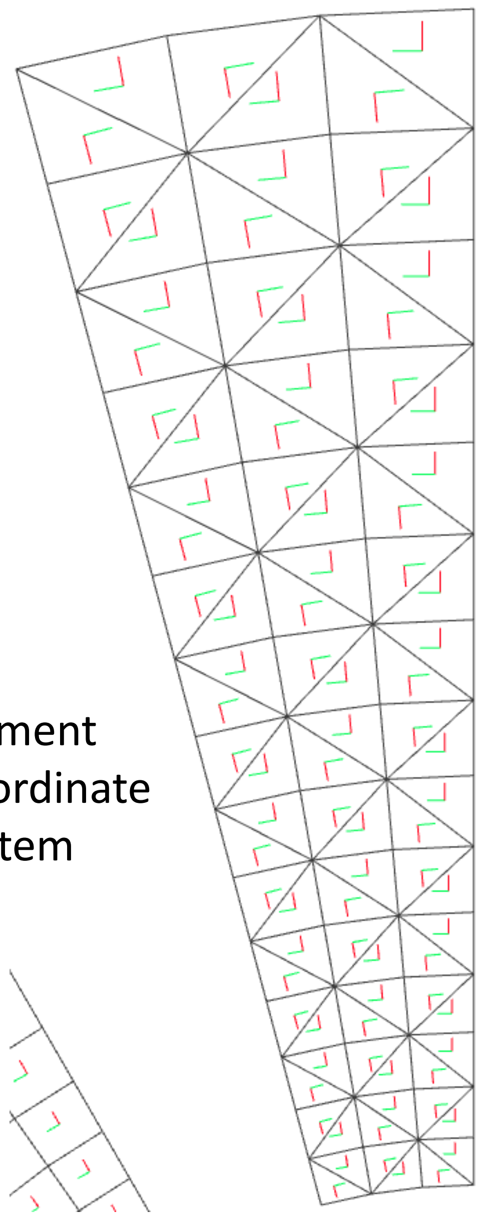
```
PARAM    OMID       YES

CQUAD4   2983     1          3162     3163     3193     3194     0
CQUAD4   2984     1          3163     3164     3192     3193     0
CQUAD4   2985     1          3164     3165     3191     3192     0
CQUAD4   2986     1          3165     3166     3190     3191     0
```



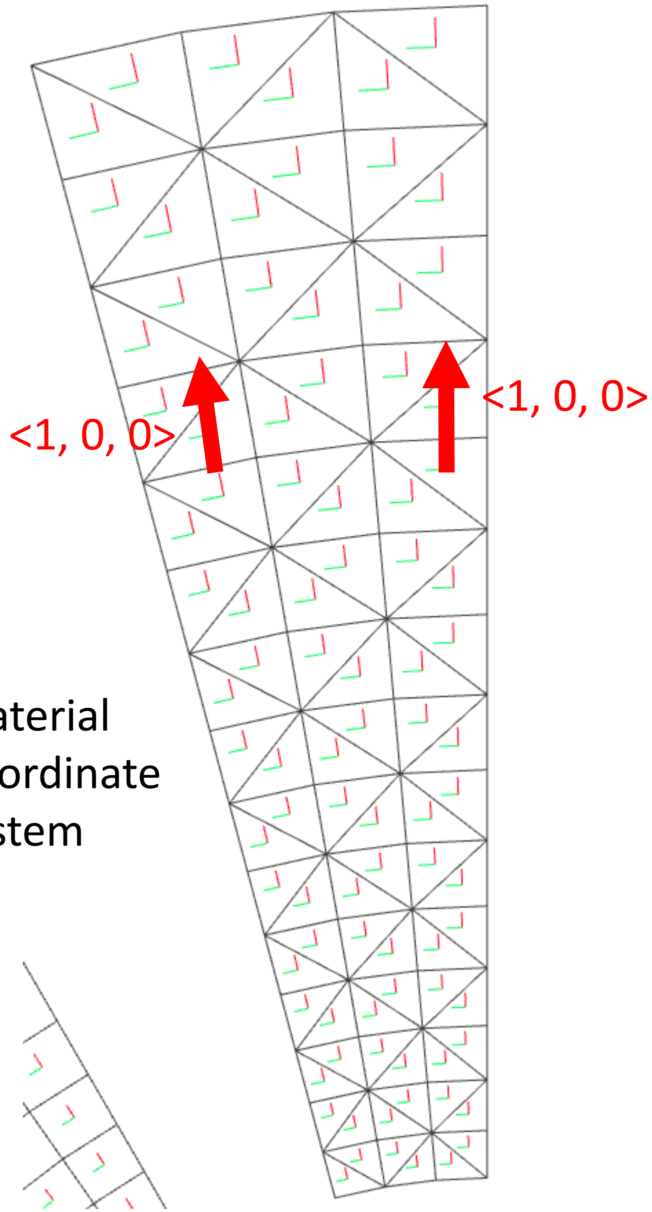
2D Element Stresses and Strains

- Note that if the material coordinate system is based on a cylindrical or spherical coordinate system, the vector <1, 0, 0> varies depending on position.
- The example uses a cylindrical coordinate system to align the material coordinate system for the elements. When this is done, the x_material axis is in the radial direction of the cylindrical coordinate system.

Element Coordinate System



Material Coordinate System



Shaders

Shaders

- MSC Nastran outputs data at select points.
  - 3D elements: responses are output at the corners or centers of the elements.
  - 2D elements: responses are output at the corners or centers of the elements. When PSHELL is used, responses are available for the top and bottom of the thickness. When PCOMP or PCOMPG is used, responses are output at the midplane of each layer.
  - 1D elements: Responses, such as shear forces, moments, beam stresses, etc., are output at the ends of the 1D elements. If stations are used, responses are output at the ends of the stations.

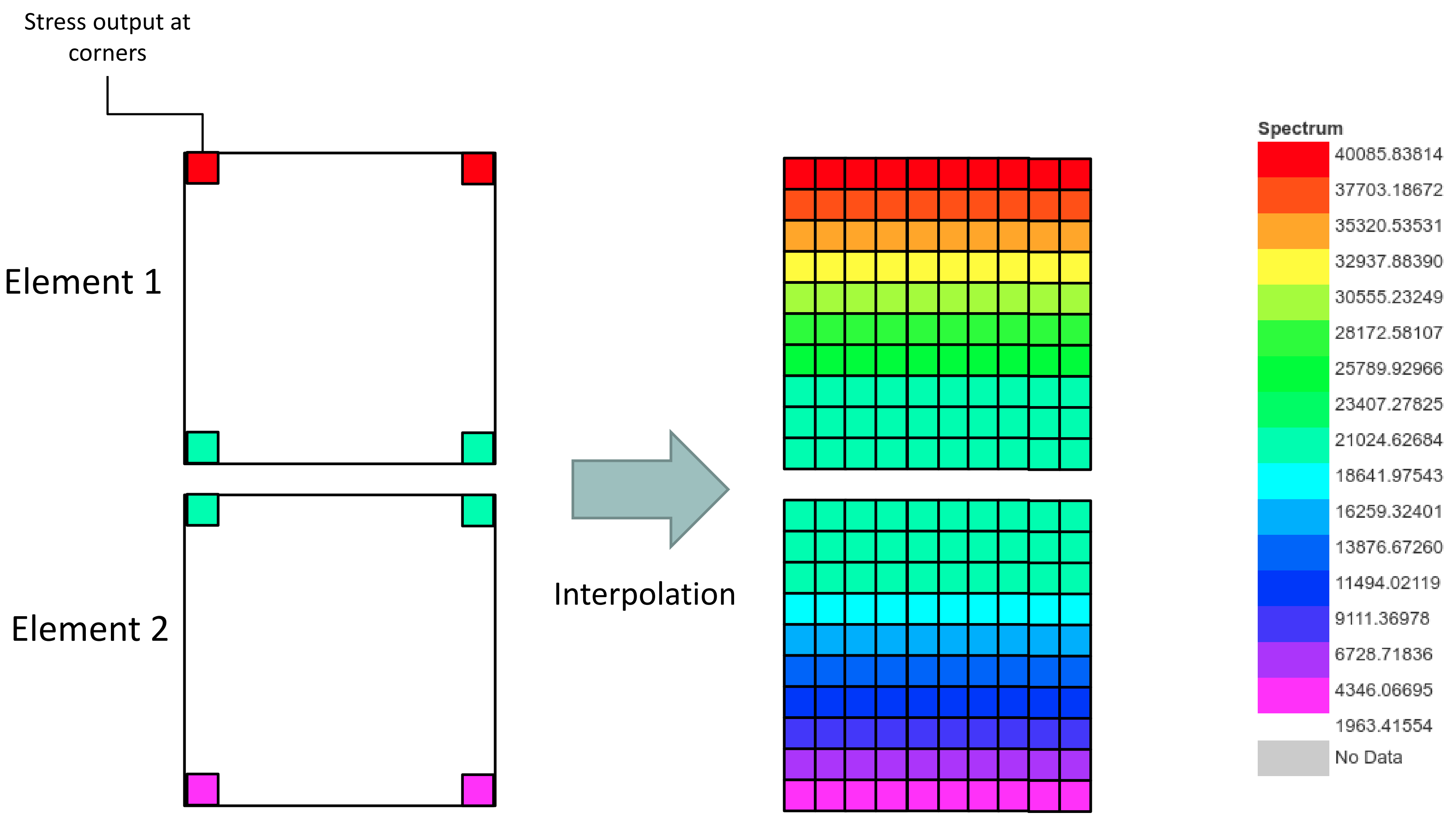
- Consider σ_x at the corners for two elements. Only the responses at the top of the thickness are considered.

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

	ELEMENT ID	GRID-ID	FIBER DISTANCE		STRESSES IN MATERIAL COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR
0	2983	CEN/4	-5.000000E-03	Bottom	5.599380E+04	4.091693E+03	-2.046786E+03	-2.2548	5.607439E+04	4.011102E+
			5.000000E-03	Top	5.599380E+04	4.091693E+03	-2.046786E+03	-2.2548	5.607439E+04	4.011102E+
	Corner 1	3162	-5.000000E-03		7.889764E+04	3.416357E+03	-5.055151E+03	-3.8145	7.923469E+04	3.079307E+
			5.000000E-03		7.889764E+04	3.416357E+03	-5.055151E+03	-3.8145	7.923469E+04	3.079307E+
	Corner 2	3163	-5.000000E-03		7.429754E+04	5.287339E+03	-3.969823E+03	-3.2815	7.452516E+04	5.059725E+
			5.000000E-03		7.429754E+04	5.287339E+03	-3.969823E+03	-3.2815	7.452516E+04	5.059725E+
	Corner 3	3193	-5.000000E-03		4.054612E+04	4.465995E+03	-4.027739E+01	-0.0640	4.054617E+04	4.465951E+
			5.000000E-03		4.054612E+04	4.465995E+03	-4.027739E+01	-0.0640	4.054617E+04	4.465951E+
	Corner 4	3194	-5.000000E-03		3.571606E+04	3.144009E+03	1.881411E+02	0.3309	3.571714E+04	3.142922E+
			5.000000E-03		3.571606E+04	3.144009E+03	1.881411E+02	0.3309	3.571714E+04	3.142922E+
0	2984	CEN/4	-5.000000E-03		3.089604E+04	-4.791212E+03	-5.696117E+03	-8.8522	3.178315E+04	-5.678329E+
			5.000000E-03		3.089604E+04	-4.791212E+03	-5.696117E+03	-8.8522	3.178315E+04	-5.678329E+
		3163	-5.000000E-03		4.008584E+04	3.912176E+03	-3.052811E+03	-4.7902	4.034167E+04	3.656349E+
			5.000000E-03		4.008584E+04	3.912176E+03	-3.052811E+03	-4.7902	4.034167E+04	3.656349E+
		3164	-5.000000E-03		2.845331E+04	-7.774744E+03	-1.437158E+04	-19.2142	3.346200E+04	-1.278344E+
			5.000000E-03		2.845331E+04	-7.774744E+03	-1.437158E+04	-19.2142	3.346200E+04	-1.278344E+
		3192	-5.000000E-03		2.526295E+04	-1.004106E+04	-6.304080E+03	-9.8266	2.635487E+04	-1.113298E+
			5.000000E-03		2.526295E+04	-1.004106E+04	-6.304080E+03	-9.8266	2.635487E+04	-1.113298E+
		3193	-5.000000E-03		3.118665E+04	-4.044532E+03	-3.123219E+00	-0.0051	3.118665E+04	-4.044532E+
			5.000000E-03		3.118665E+04	-4.044532E+03	-3.123219E+00	-0.0051	3.118665E+04	-4.044532E+

Shaders

- Shaders (fragment shaders and vertex shaders) are responsible for coloring the faces of elements and determine the color of each pixel on the screen. For 3D and 2D elements, responses are output at the corners and center of the element. For intermediate points/pixels between the corners and center, an interpolation is performed to determine the color of the intermediate points/pixels.
- In the image to the right, pixels are represented with small squares.
- Some post-processors assume linear relationships between points.
- The example on the right shows the stresses output at the corners of the element. The shaders use linear interpolation to determine the colors of pixels for intermediate points.

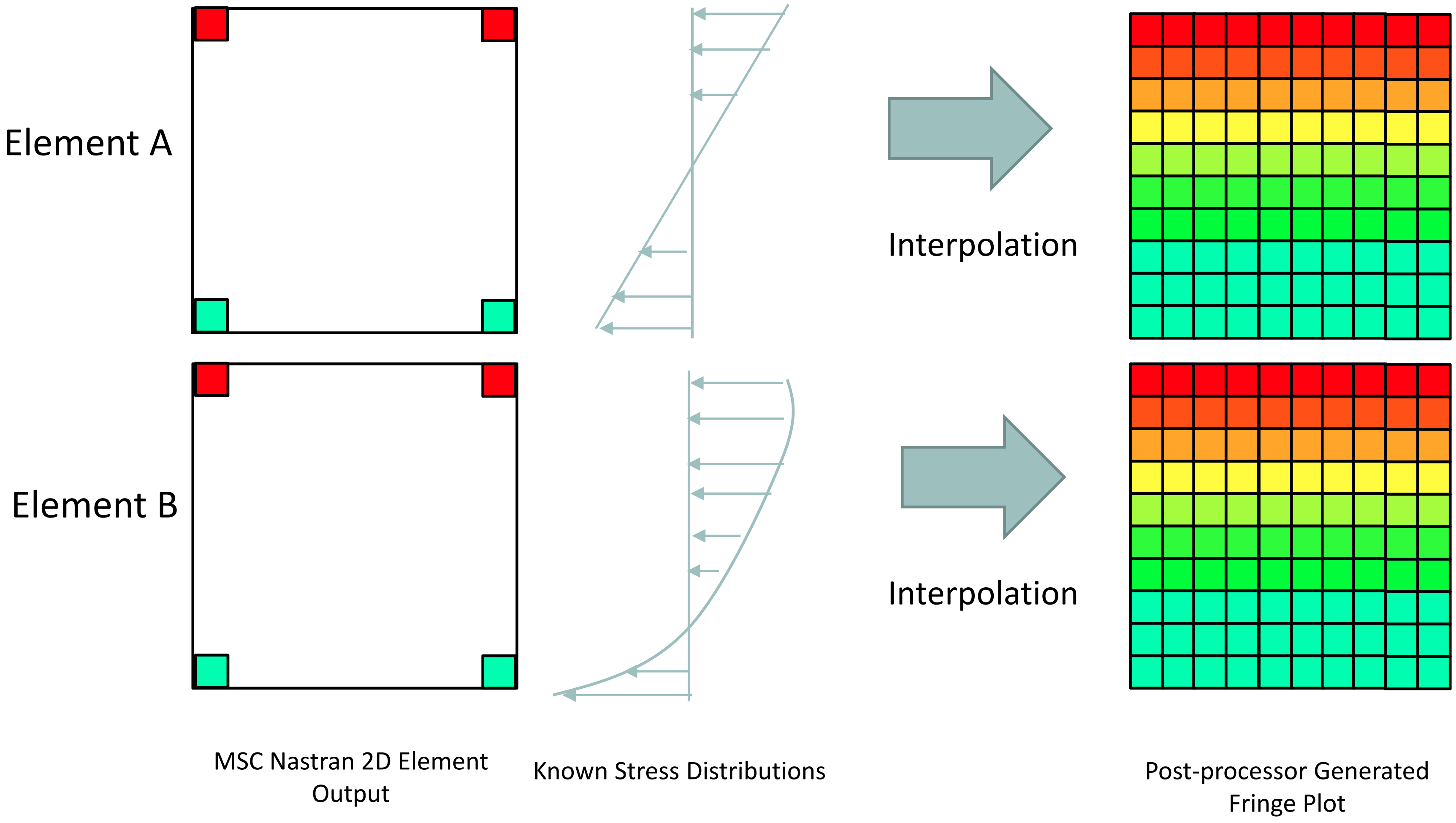


MSC Nastran 2D Element Output

Post-processor Generated Fringe Plot

Shaders

- Consider elements A and B, which have the same corner stresses. Element A is used to model a straight beam, which has a linear stress distribution, whereas element B is used to model a curved beam, which has a nonlinear stress distribution. Both fringe plots look identical, but should not be since the stress distributions are expected to be different. This is because the shaders use linear interpolation.
- As finite element meshes use smaller element sizes, this effect is less pronounced and does not impact everyday results interpretation.
- Again, MSC Nastran outputs responses at points, and post-processors perform interpolation for intermediate points/pixels. Most post-processors perform linear interpolation.

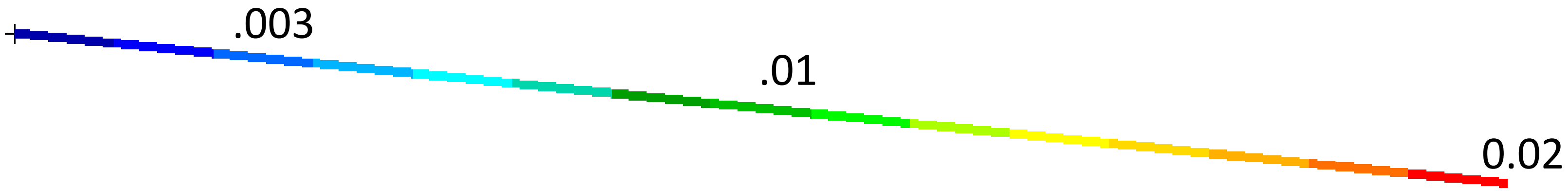


Shaders

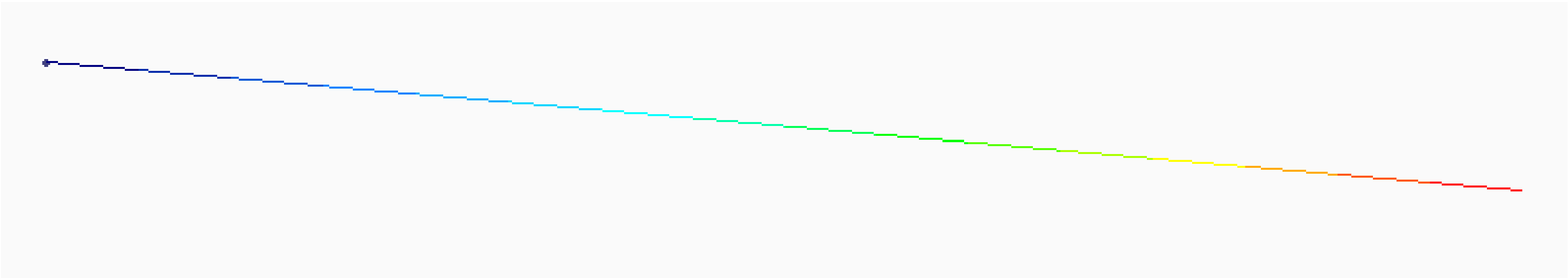
Consider another example.

- Suppose a cantilever beam with a fixed end and a point load at the free end is modeled with one beam element. The expected deflection is parabolic.
- Consider the coloring behavior of 2 different commercial post-processors: Post-processor A and B.
- The deformation is expected to be parabolic, but Post-processor A and B display the deformation as a straight line. Also, notice the color spectrum implies a linear deflection along the length of the beam, which does not align with the expectation that the deflection is parabolic.
- Shaders use linear interpolation to color the points between the nodes.
- When more elements are used, the linear interpolation is less noticeable. Also shown is a scenario when 10 beam elements are used. In this case, the deflection is more parabolic, as is the color plot.

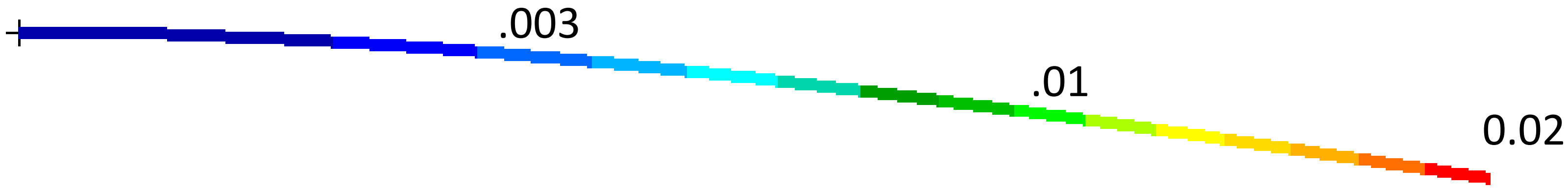
Post-processor A (1 Beam Element)



Post-processor B (1 Beam Element)

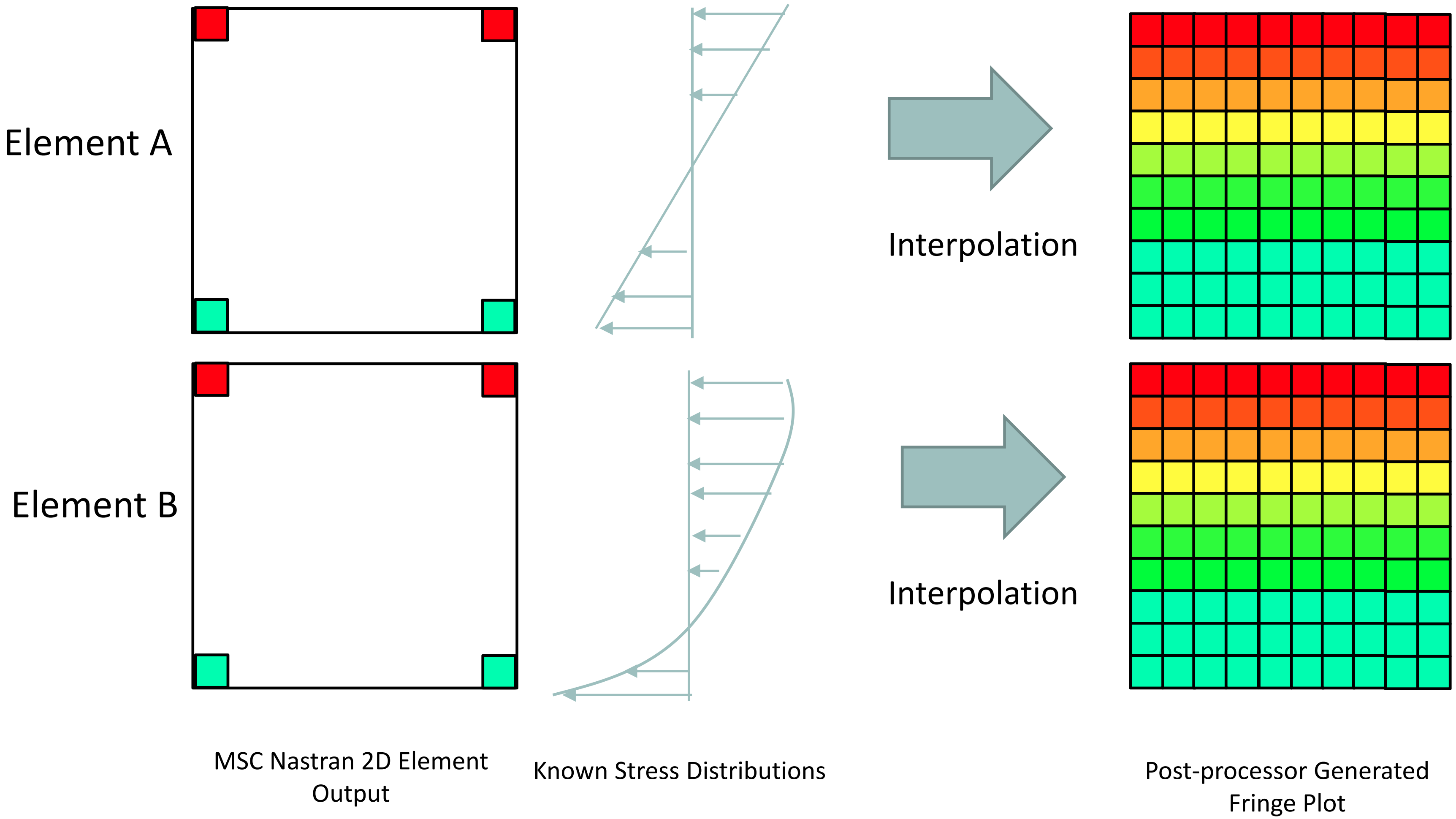


Post-processor A (10 Beam Element)



Shaders

- MSC Nastran outputs results at a finite number points, e.g. corners of 2D elements or ends A and B of 1D elements.
- Post-processors use linear interpolation to color the intermediate points.



1D Element Forces and Stresses

1D Element Forces and Stresses

- Beam forces, stresses and strains are observed in the beam element's element coordinate system.

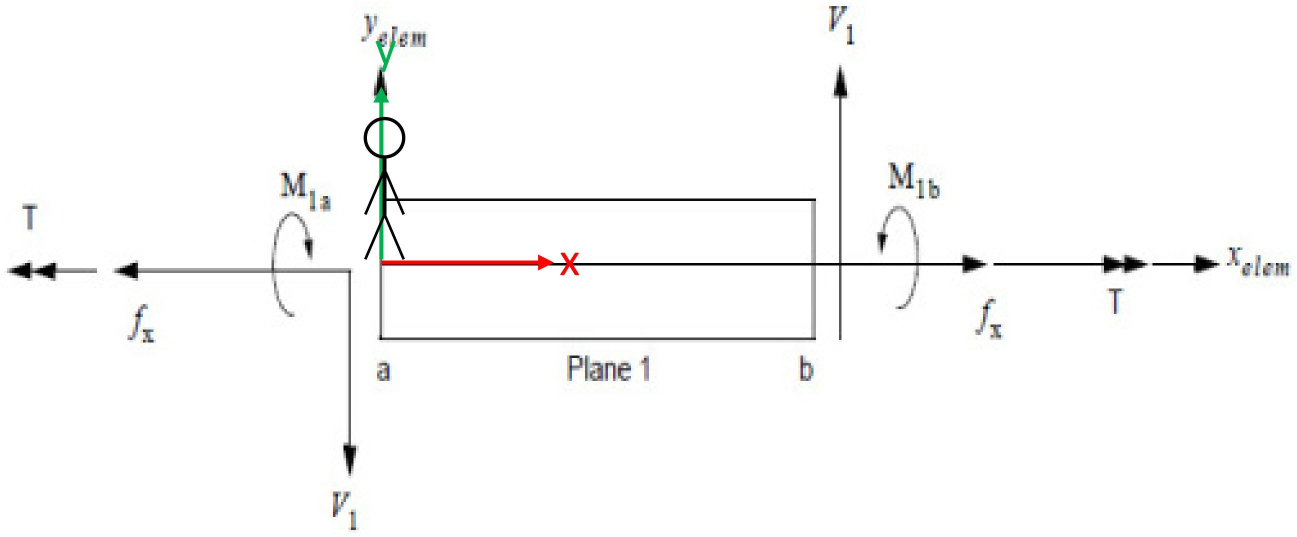


Figure 6-6 CBAR Element Internal Forces and Moments (x-y Plane)

1D Element Forces and Stresses

- Beam forces, stresses and strains are output at the ends of the beam elements.
  - If stations are used, then responses are available at the ends of the stations.
- Figures 6-6 and 6-7 are from the MSC Nastran documentation. The Post-processor Web App uses arrows to denote the direction of the responses, and markers are used to denote the magnitude of the responses.
- Notice the location of the arrows are displayed in the interior of the elements.

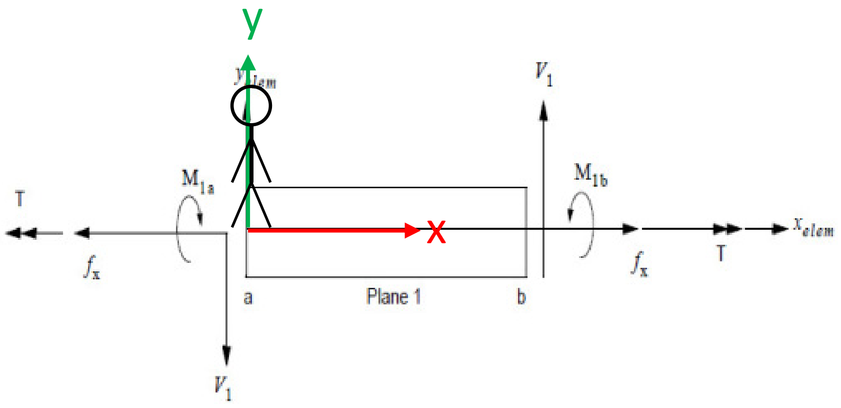


Figure 6-6 CBAR Element Internal Forces and Moments (x-y Plane)

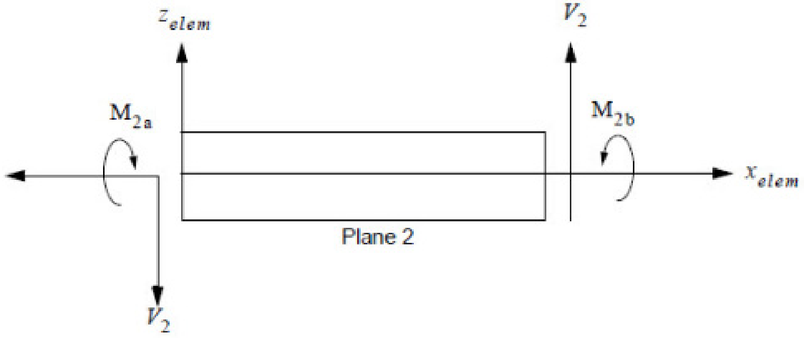
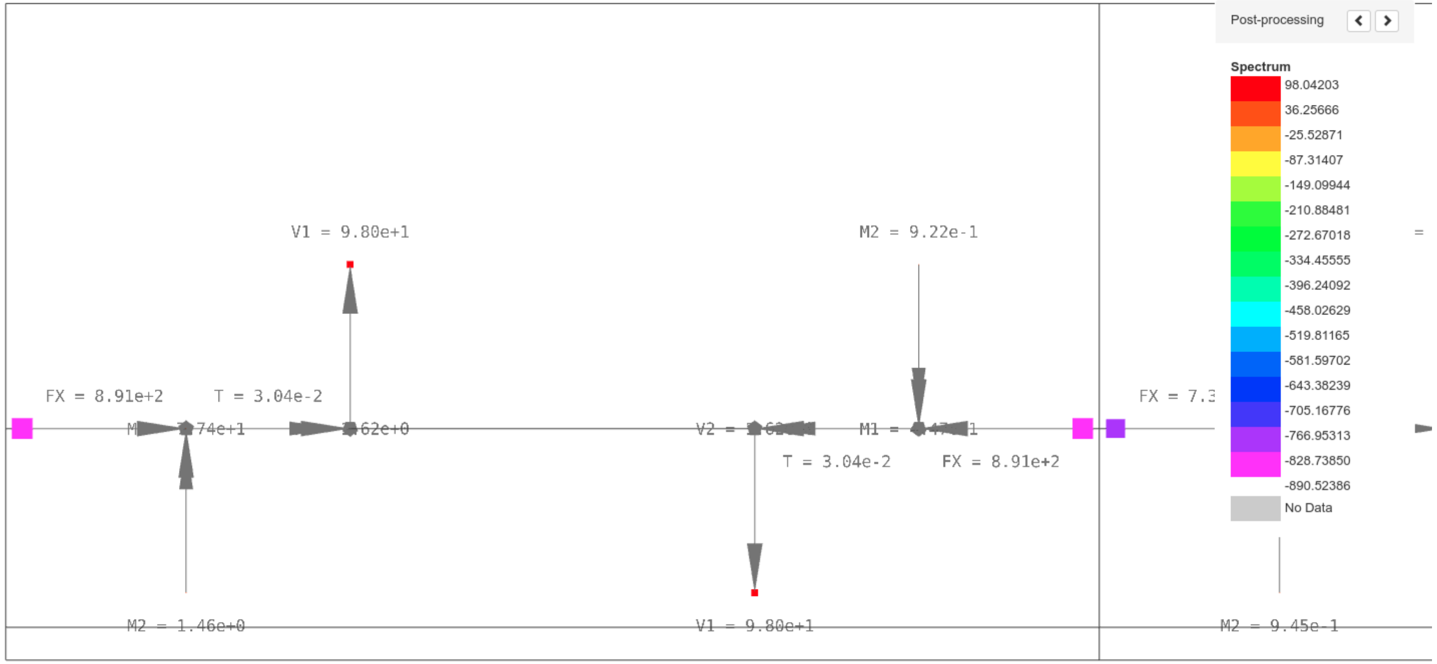


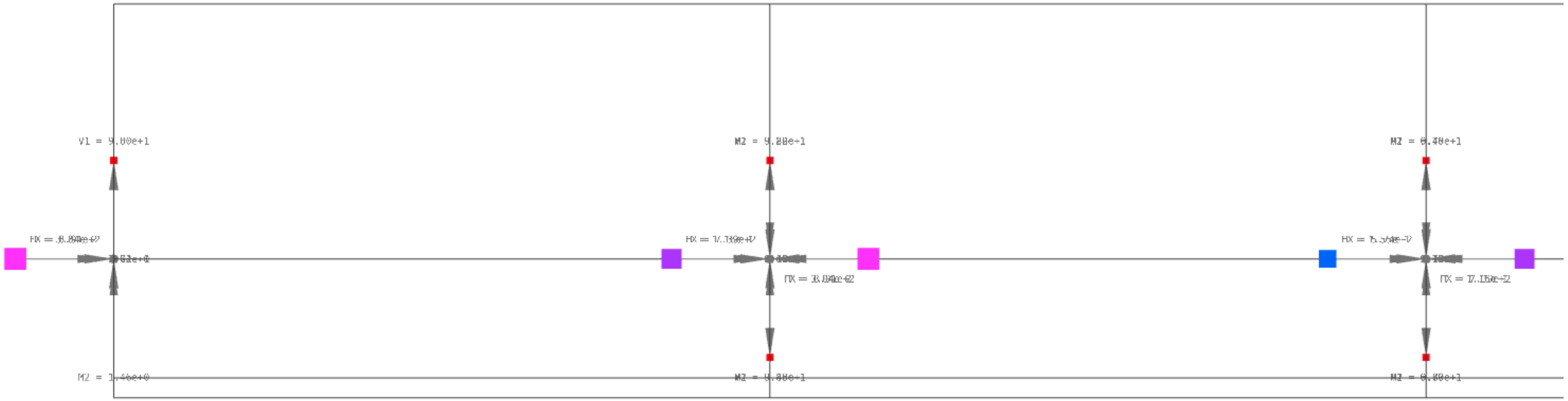
Figure 6-7 CBAR Element Internal Forces and Moments (x-z Plane)



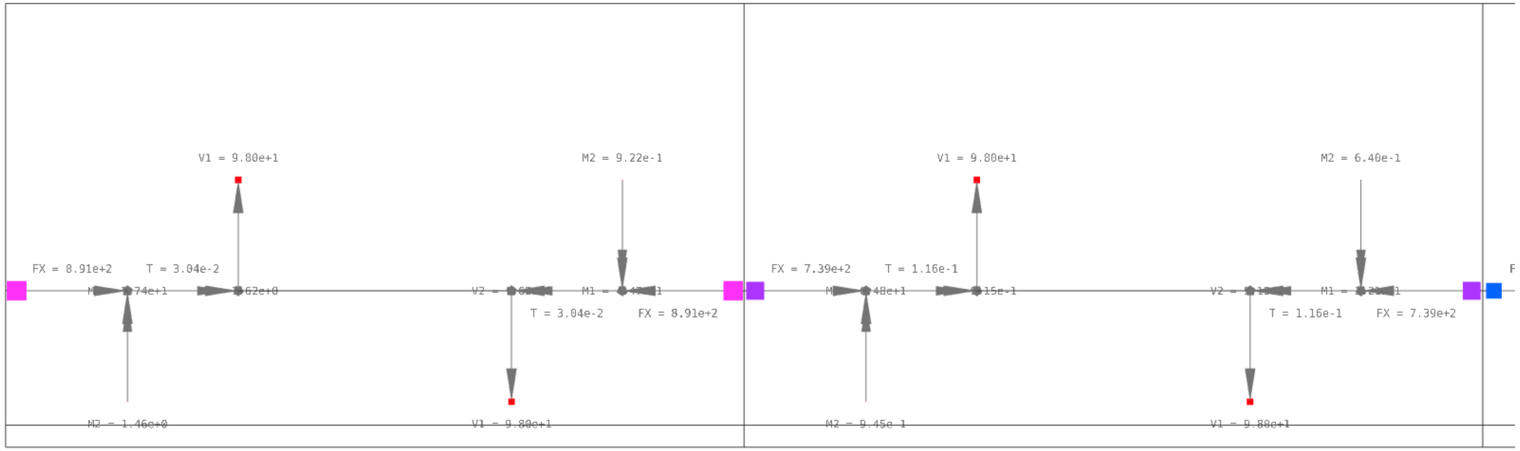
1D Element Forces and Stresses

- Suppose the element forces were displayed at exactly the ends, as is done in traditional post-processors. The data will overlap with numerous other data, which makes results interpretation highly difficult.
- The Post-processor Web App purposely displays the data slightly offset from the ends to avoid overlapping, which makes result interpretation simpler.

Traditional Post-processors



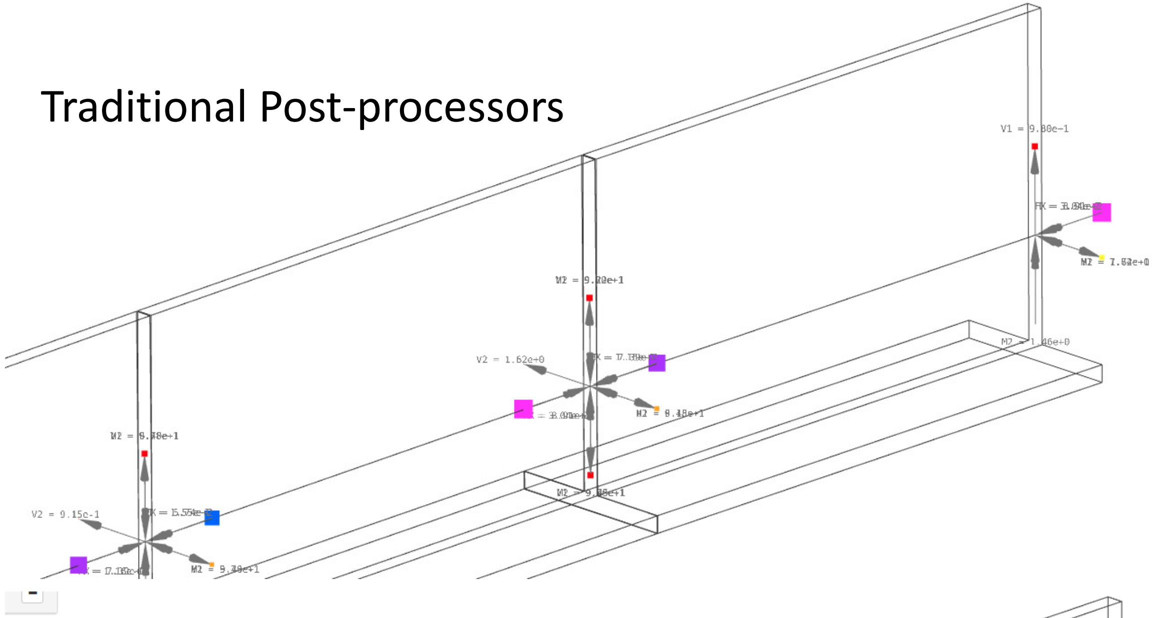
Post-processor Web App

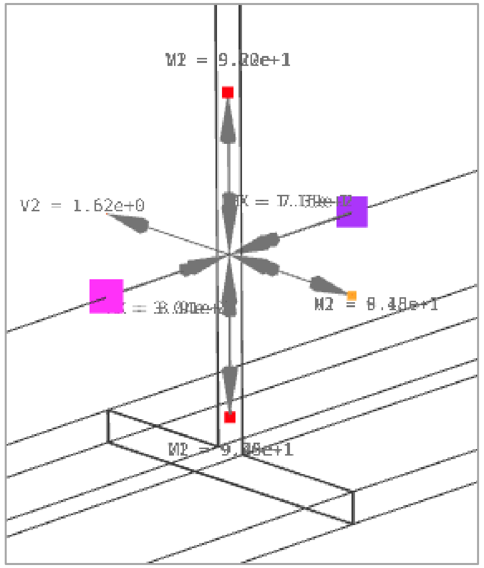


1D Element Forces and Stresses

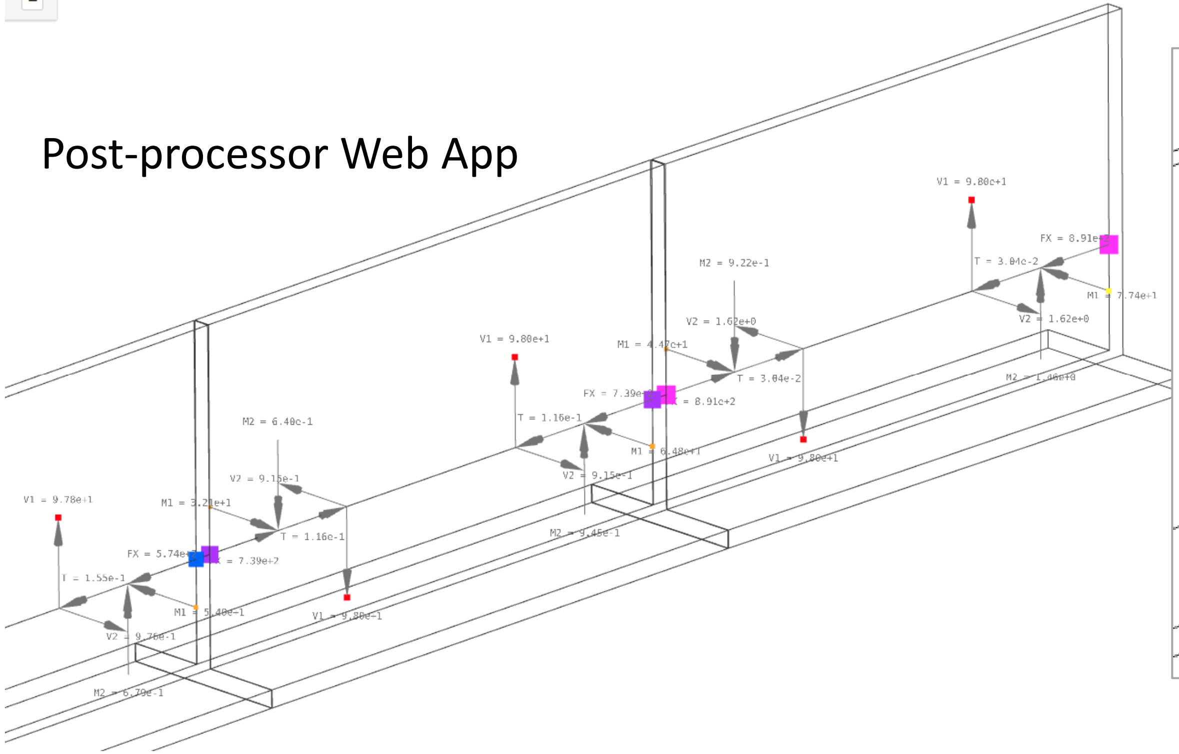
- This is another view of the beam element forces for traditional post-processors and the Post-processor Web App.

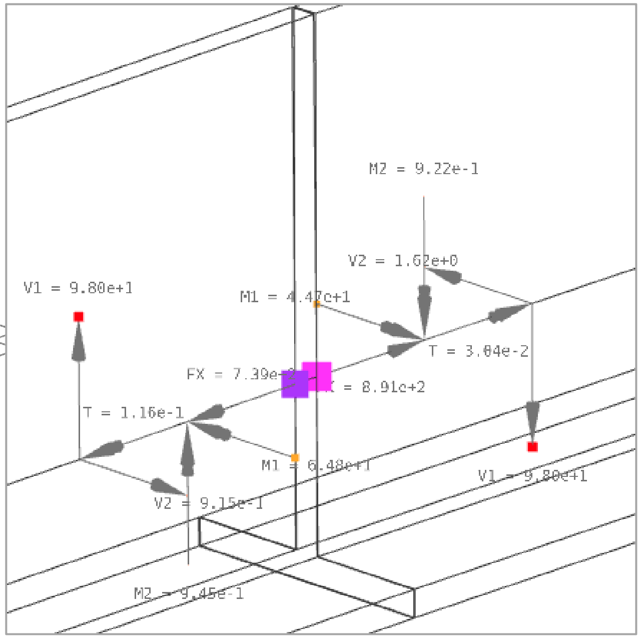
Traditional Post-processors





Post-processor Web App





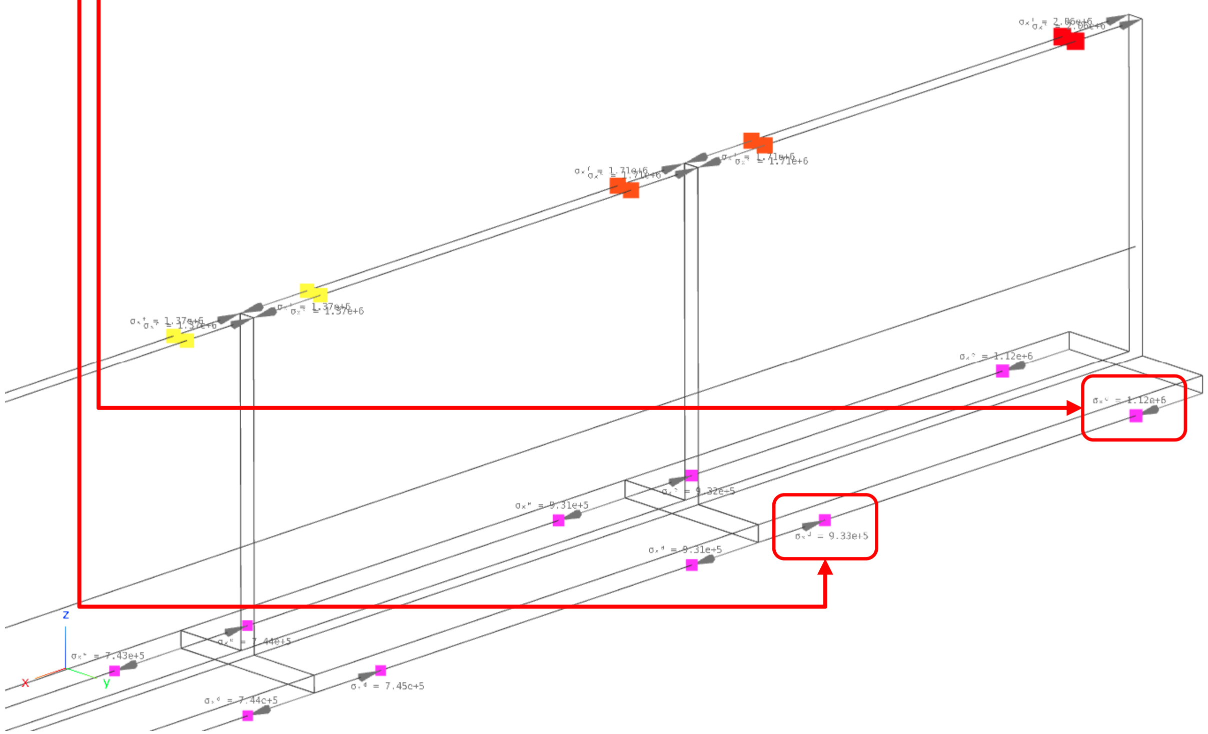
1D Element Forces and Stresses

- Recall MSC Nastran typically outputs responses at points. For 3D and 2D elements, responses are output at the corners and centers of the elements. For 1D elements, responses are output at the ends of the beam elements. If stations are used, responses are output at the ends of each station.
- For stress distributions, we are used to viewing the stress distribution for the entire cross section. Beam stress responses available in the F06 file and are typically output at 4 points on the cross section, as shown in the data and image to the right.

F06 Output

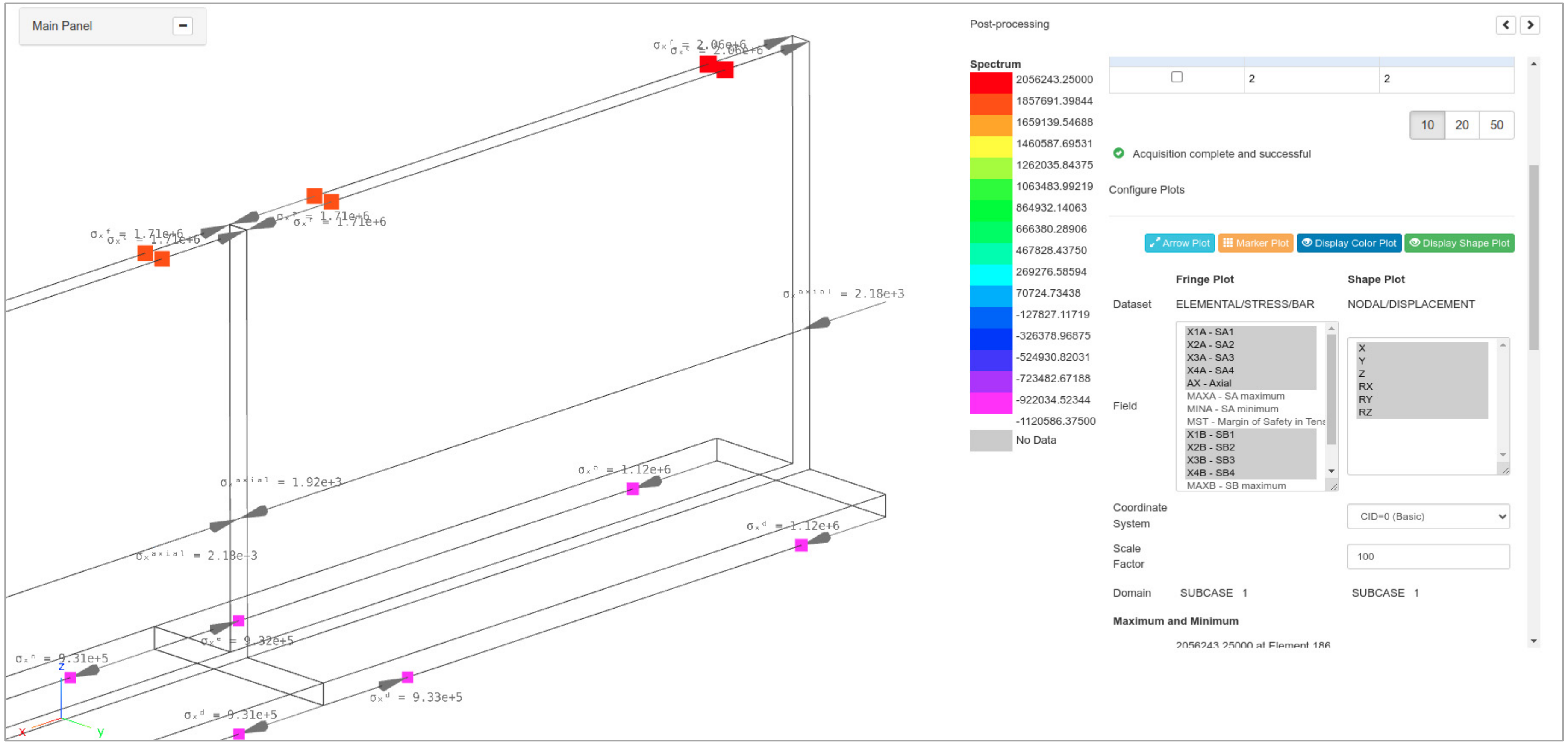
S T R E S S E S   I N   B A R   E L E M E N T S   (C B A R)

	ELEMENT ID.	SA1 / SB1	SA2 / SB2	SA3 / SB3	SA4 / SB4	AXIAL STRESS	SA-MAX / SB-MAX	SA-MIN / SB-MIN
0	186	1.712209E+06	-9.328956E+05	-9.321099E+05	1.712288E+06	-2.182193E+03	1.710105E+06	-9.350777E+05
		2.056086E+06	-1.120586E+06	-1.119018E+06	2.056243E+06		2.054061E+06	-1.122769E+06
0	187	1.366966E+06	-7.449318E+05	-7.440360E+05	1.367056E+06	-1.919324E+03	1.365136E+06	-7.468512E+05
		1.709330E+06	-9.313308E+05	-9.305395E+05	1.709409E+06		1.707490E+06	-9.332501E+05
0	190	1.024060E+06	-5.580100E+05	-5.574415E+05	1.024117E+06	-1.496167E+03	1.022620E+06	-5.595061E+05
		1.365037E+06	-7.438844E+05	-7.429825E+05	1.365127E+06		1.363631E+06	-7.453806E+05
0	194	6.826201E+05	-3.719388E+05	-3.715993E+05	6.826540E+05	-9.980246E+02	6.816560E+05	-3.729368E+05
		1.022712E+06	-5.572800E+05	-5.567039E+05	1.022770E+06		1.021771E+06	-5.582780E+05
0	196	3.420758E+05	-1.862195E+05	-1.863664E+05	3.420611E+05	-5.003812E+02	3.415754E+05	-1.868667E+05
		6.818681E+05	-3.715334E+05	-3.711860E+05	6.819028E+05		6.814024E+05	-3.720338E+05
0	200	5.722028E+02	-3.112152E+02	-3.119939E+02	5.721249E+02	-1.188287E+02	4.533741E+02	-4.308226E+02
		3.418317E+05	-1.860892E+05	-1.862311E+05	3.418175E+05		3.417129E+05	-1.863499E+05



1D Element Forces and Stresses

- An enlarged view of the beam elements does show arrows indicating tension or compression of stresses, and the markers and labels indicate the magnitude of the response.



Conclusion

When reviewing MSC Nastran results, always answer this question:

- In which coordinate system are the Nastran results being observed in?
  - Displacements are always observed in the displacement coordinate system
  - 3D element results are observed in the material coordinate system
  - 2D element results are observed in the element coordinate system or material coordinate system
    - For 3D and 2D elements, stress and strain transformation is possible if the post-processor supports this capability
  - 1D element results are observed in the element coordinate system