Altair HyperView 2019 Tutorials

HV-3055: Comparing Scalar and Tensor Results from Abaqus in HyperView

HV-3055: Comparing Scalar and Tensor Results from Abaqus in HyperView

The following exercise illustrates how scalar and tensor results are contoured in HyperView using both shell and solid elements.

Exercise

This exercise uses the model file `Postprocessing_demo.inp` and the corresponding results file `Postprocessing_demo.odb`.

Step 1: Set up two windows and import the file `Postprocessing_demo.odb` into both windows.

1. Within HyperView, click on the drop-down menu next to the **Page Window Layout** icon ☐ ˅ (on the **PageControls** toolbar), and select the following **Two Window Layout** icon ⊞.

2. Go to the **Load Model** panel by selecting *File > Open > Model* from the menu bar.

3. Load the model file `Postprocessing_demo.inp` and the results file `Postprocessing_demo.odb`, located in the `animation` folder.

4. Click on the *Reader Options...* button.

    The **Reader Options** dialog is displayed.
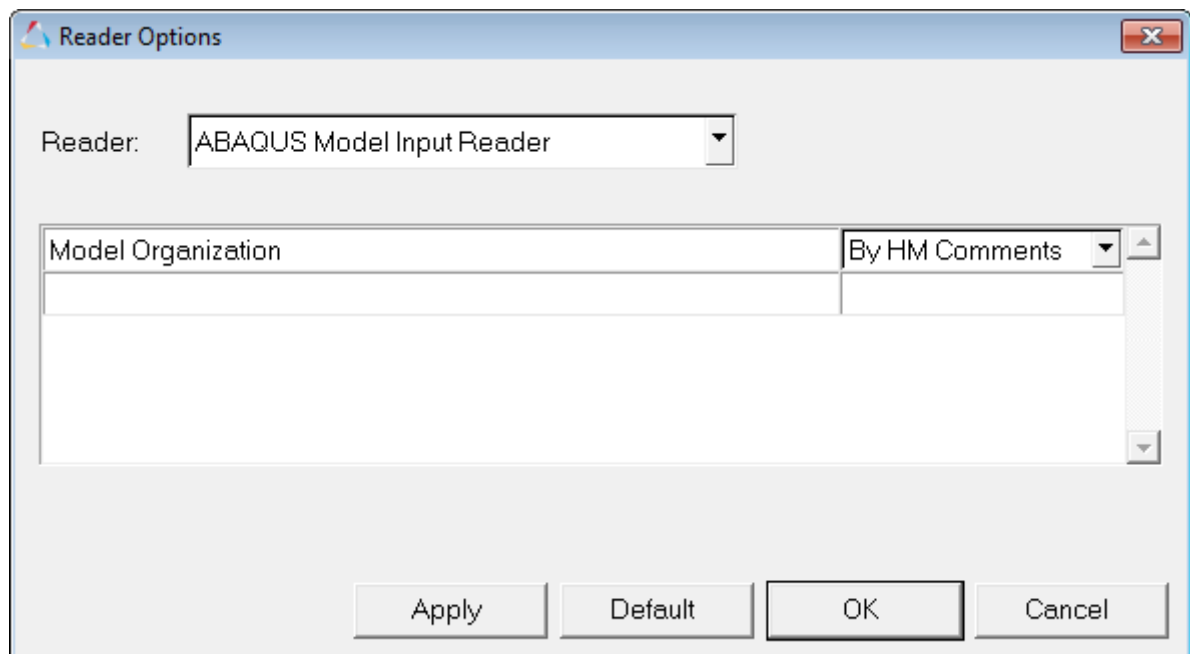

5. Select the *ABAQUS ODB Result Reader* option from the **Reader** drop-down menu, Review the options that can be set, and verify that **Element results position** is set to *All*. Click *OK* to close the window.

△ Altair

6. Click *Apply* to bring the model and results into HyperView.

7. Click in the second window in the graphics area to make it the active window (you can tell it is the active window because there is a cyan box highlighting the window).

8. Click **Apply** to bring the model and result file into the second window.
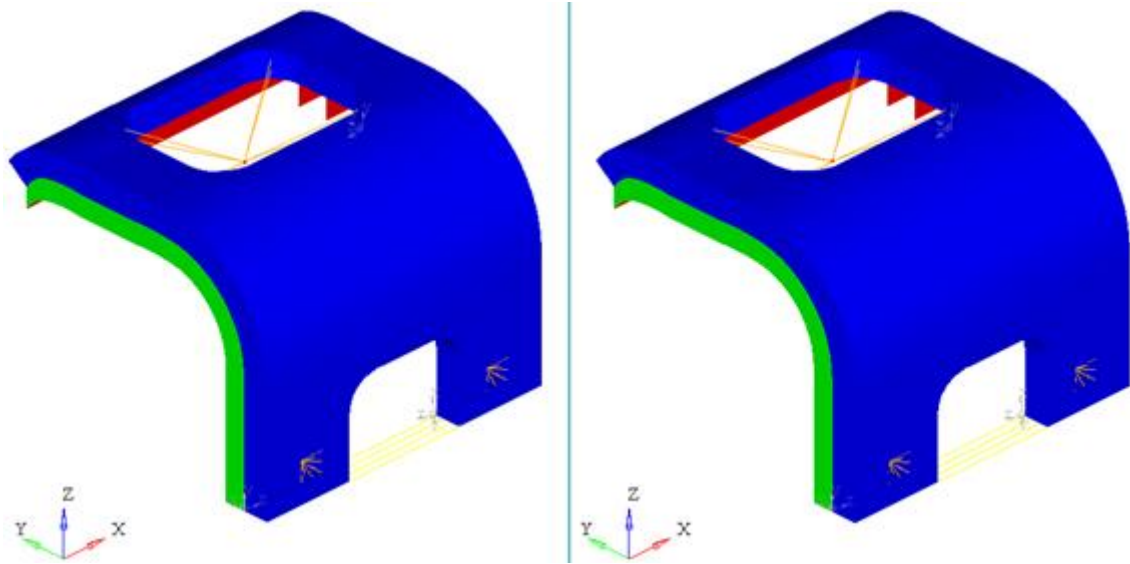
Step 2: Update the display and turn on the display of mesh lines.

1. From the **Visualization** toolbar, click on the **Shaded Elements and Mesh Lines** icon to turn on the display of the mesh lines.

2. Repeat Step 1 in the other window so that both windows display the model with mesh lines.
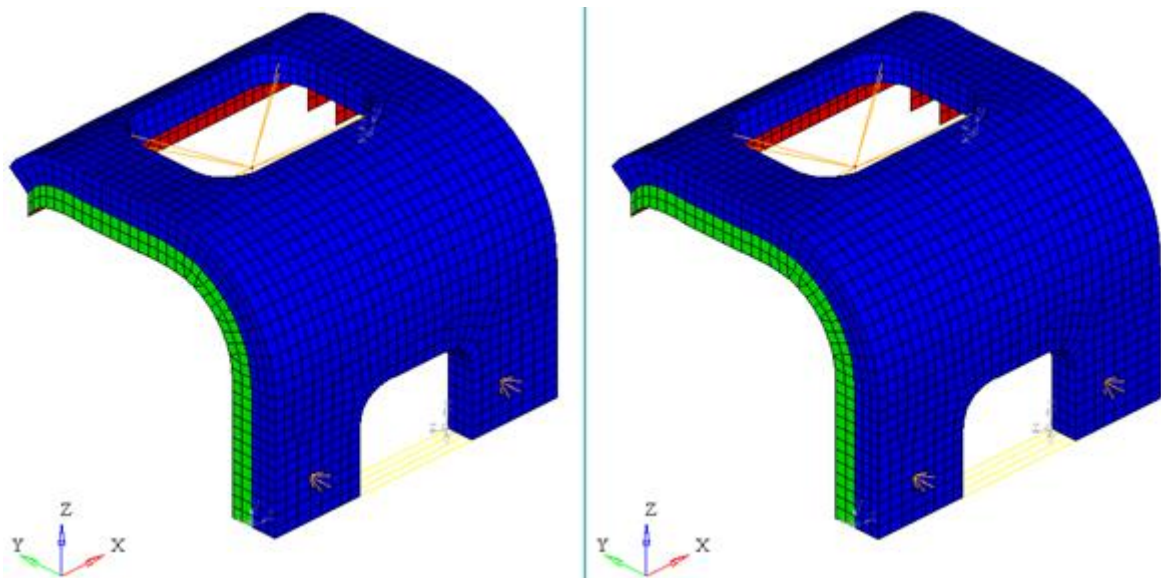
3. From the **Results Browser**, use the **Change load case** drop-down menu to set the load case to **Step-1**.
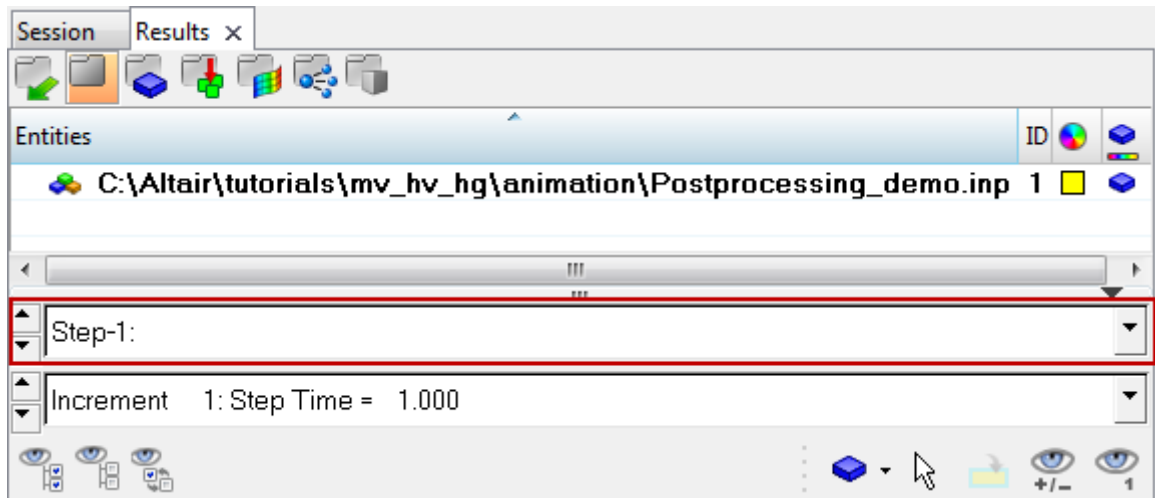

> **Note** The **Change load case** toolbar visibility can be toggled on/off using the **Configure Browser** option (located in the **Results Browser** context menu).

4. Use the second drop-down menu to set the simulation to **Increment 1: Step Time = 1.000**.

Step 3: Apply a contour of the scalar stress to the shell elements in the left window.

1. Click on the left window in the graphics area to make this window the active window.

2. Click the **Contour** panel button on the **Result** toolbar to enter the **Contour** panel.

3. For **Result type:** select **S – Stress components IP (s)**.

   This is the scalar stress and it is in a local coordinate system.

4. Under **S – Stress components IP (s)** click on the drop-down menu and select **S22**.
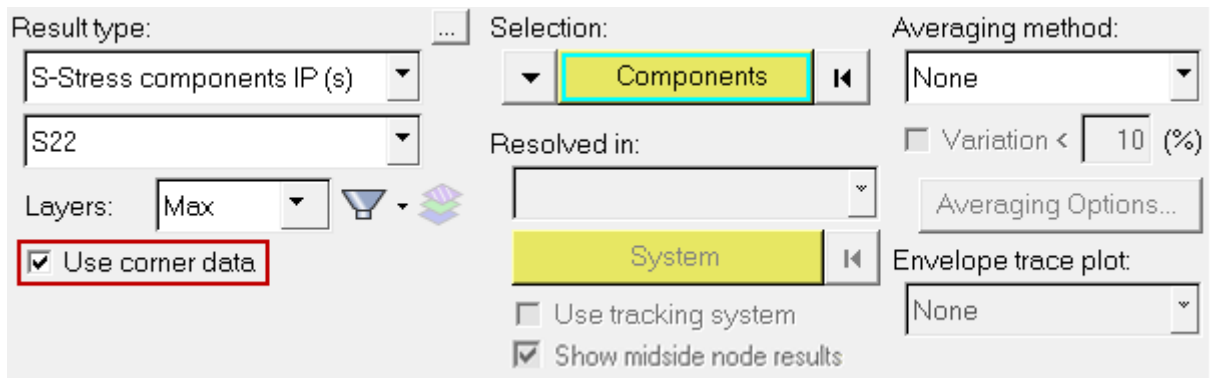
5. Activate the **Use corner data** option check box.

6. Notice that the **Components** button under **Selection** is highlighted with a cyan box meaning that it is the current selection method. In the graphics area, click on the green component so that only this component is contoured. This component only contains shell elements.

7. Click **Apply** to contour the model.
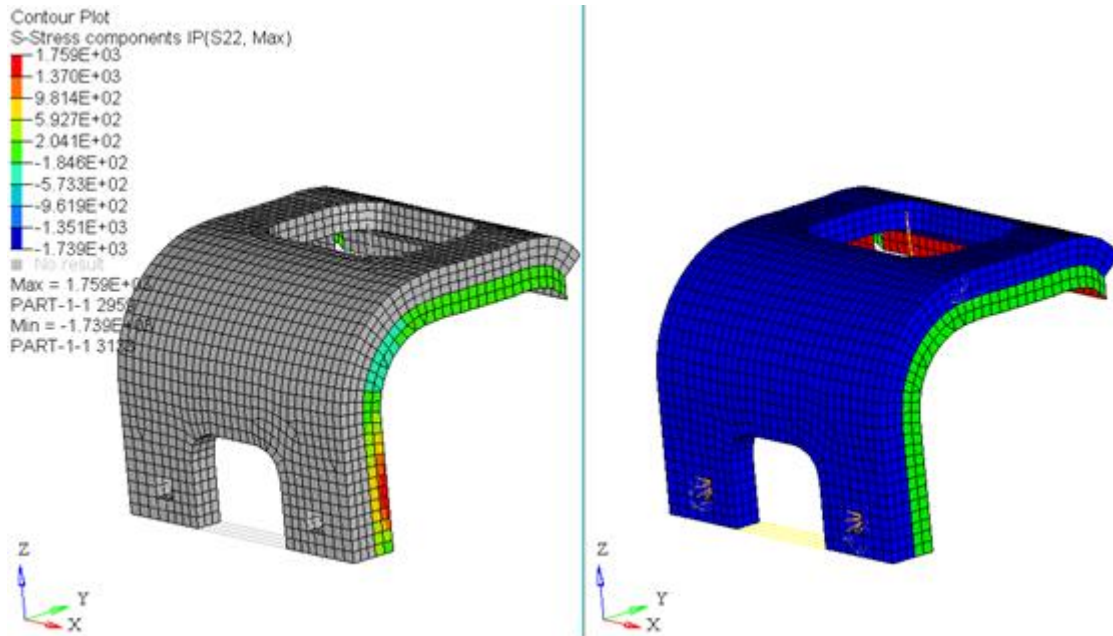
Step 4: Apply a contour of the tensor stress to the shell elements in the right window.

1. Click on the right window in the graphics area to make this window the active window.

2. For **Result type:** select **S – Global Stress components IP (t)**.

   This is the tensor stress and it is in the global coordinate system.

3. Under **S – Global Stress components IP (t)** click on the drop-down menu and select **YY**.

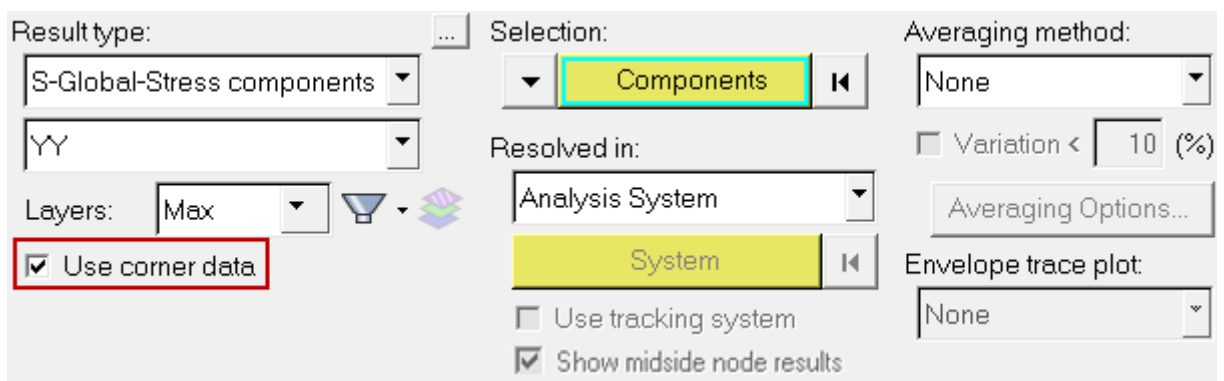4. Activate the **Use corner data** option check box.

△ Altair

5. Notice that the **Components** button under **Selection:** is highlighted with a cyan box meaning that it is the current selection method. In the graphics area, click on the green component so that only this component is contoured. This component only contains shell elements.
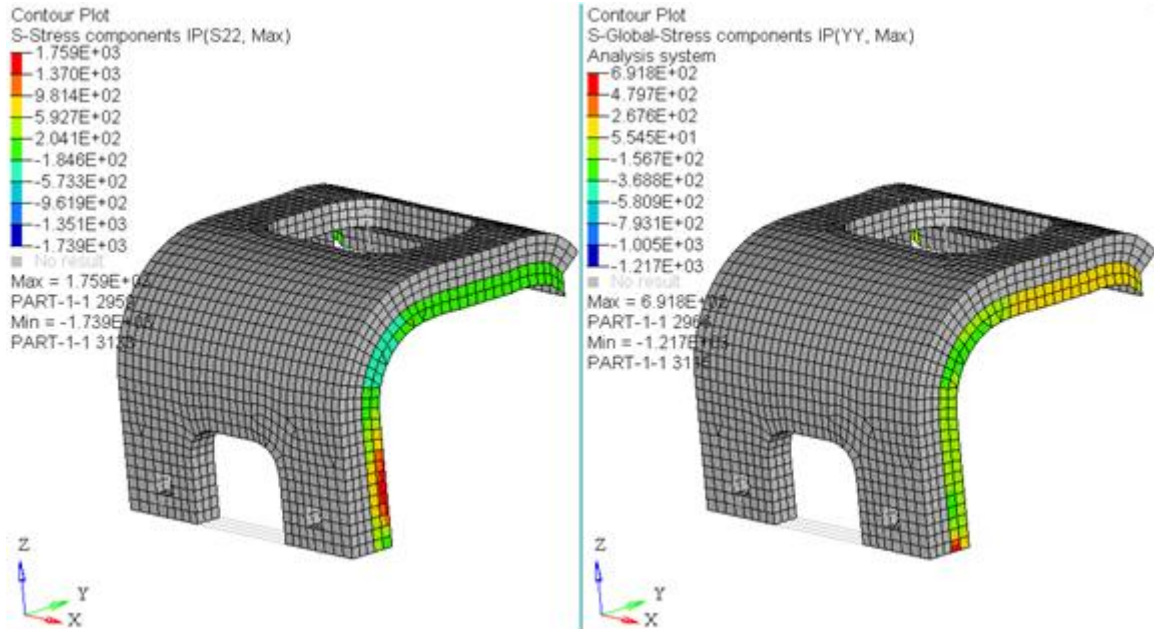
6. Click *Apply* to contour the model.

Notice that the results are different between the scalar S22 and the tensor YY. One would expect that these two options would contour the same results (assuming that S22 corresponds to the Y direction). Remember that the scalar results are in a local coordinate system. For this analysis the local coordinate system has been defined as the following: the first axis is the global Y direction and the second axis is the global Z direction. Therefore, S22 actually corresponds to ZZ.

Step 5: Update the contour of the tensor stress in the right window.

1. For **Result type:** leave **S – Global Stress components IP (t)**.

2. Under **S – Global Stress components IP (t)**, update the value from **YY** to *ZZ*.
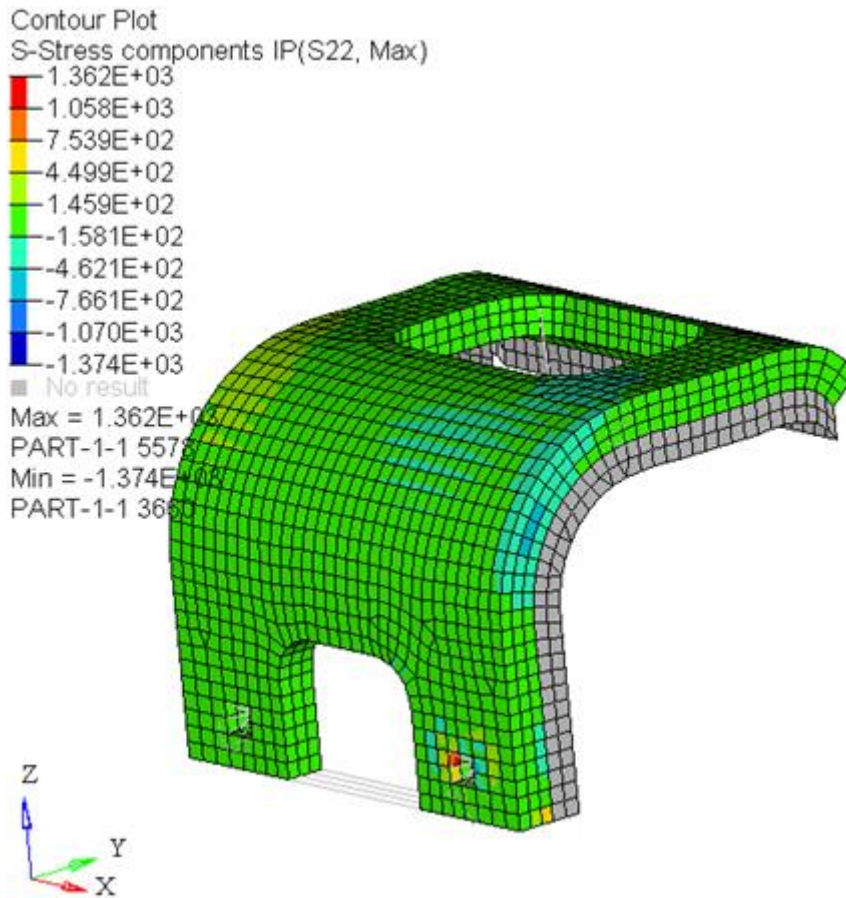
3. Click **Apply** to contour the model.
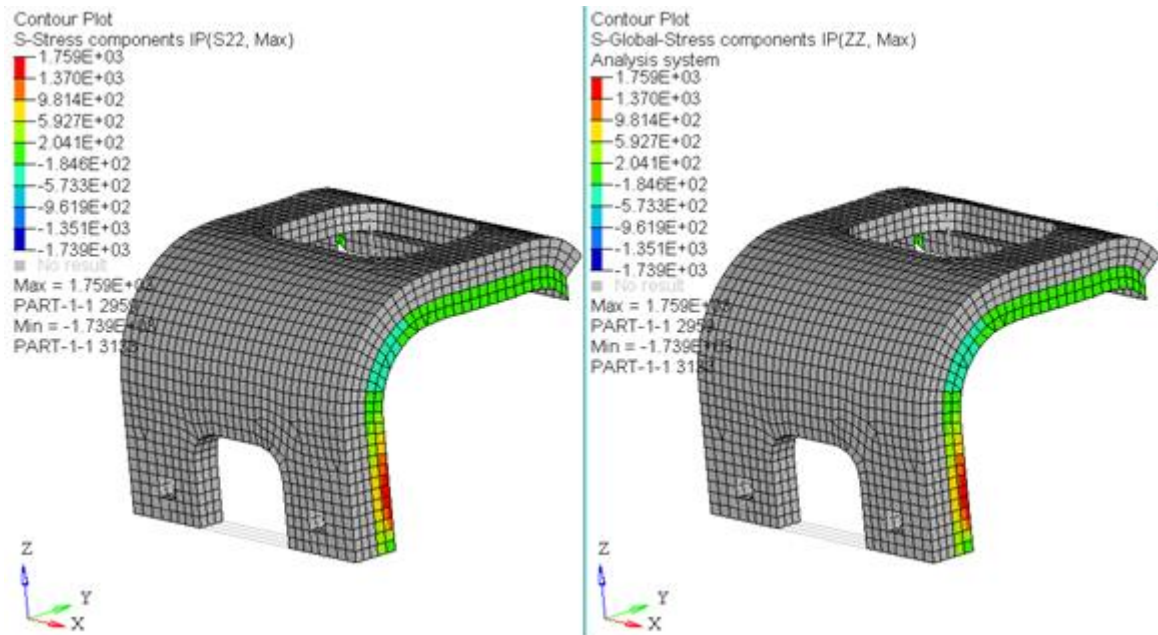
Notice how now the results match. Remember, shell elemental results are the in the local coordinate system for scalar results and in the global coordinate system for tensor results.

Step 6: Apply a contour of the scalar stress to the solid elements in the left window.

1. Click on the left window in the graphics area to make this window the active window.

2. For **Result type:** leave **S – Stress components IP (s)**.

3. Under **S – Stress components IP (s)** verify that **S22** is selected.

4. Verify that the **Use corner data** option is activated.

5. Under **Selection**, reset the components selected by clicking on ⏮ next to **Components**. In the graphics area, select the component displayed in grey (this component contains only solid elements).

△ Altair

6. Click **Apply** to contour the solid elements.

Contour Plot
S-Stress components IP(S22, Max)
```
   1.362E+03
   1.058E+03
   7.539E+02
   4.499E+02
   1.459E+02
  -1.581E+02
  -4.621E+02
  -7.661E+02
  -1.070E+03
  -1.374E+03
   No result
```
Max = 1.362E+0
PART-1-1 557
Min = -1.374E
PART-1-1 36

Step 7: Apply a contour of the tensor stress to the solid elements in the right window.

1. Click on the right window in the graphics area to make this window the active window.

2. For **Result type:** leave **S – Global Stress components IP (t)**.

3. Under **S – Global Stress components IP (t)** update the selection to **YY**.

4. Verify that the **Use corner data option** is activated.

5. Under **Selection**, reset the components selected by clicking on ⏮ next to **Components**. In the graphics area, select the same grey component as was selected in Step 6/sub-Step 5 (above).

△ Altair

6. Click **Apply** to contour the solid elements.
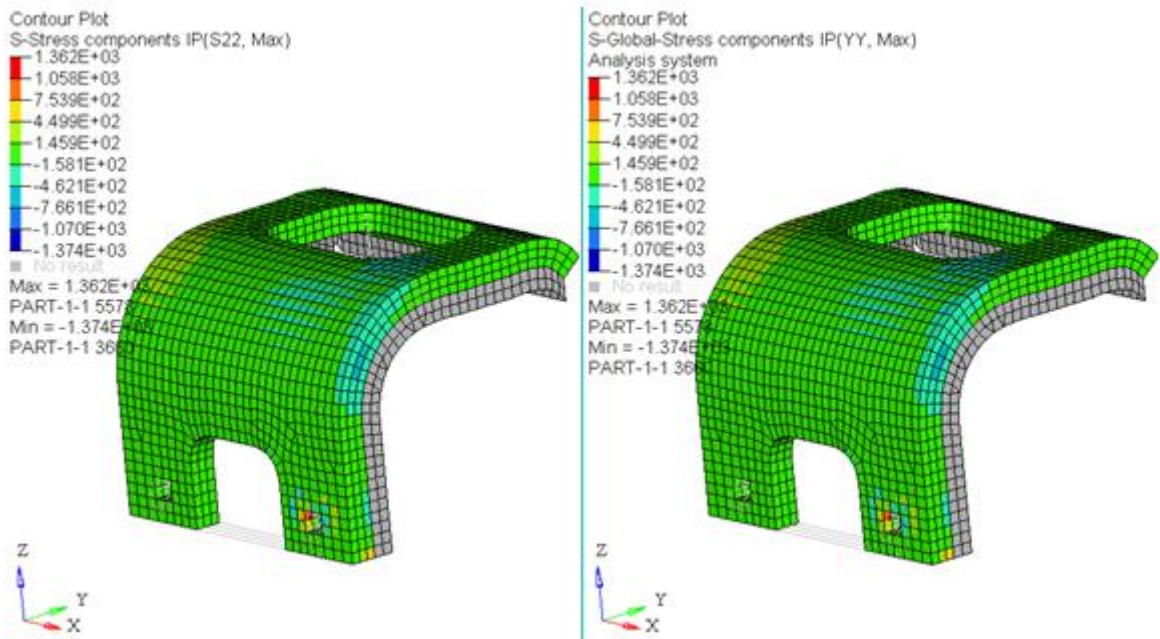
Notice that in this example, the scalar results for S22 and the tensor results for YY match. This is because the elements that are contoured are solid elements. With solid elements, the scalar results are also in the global system.