

# ANSYS - Semi-monocoque shell - Step 8

## Problem Specification

1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
- 8. Postprocess the results**
9. Validate the results

## Step 8: Postprocess the results

### Plot Deformed Shape

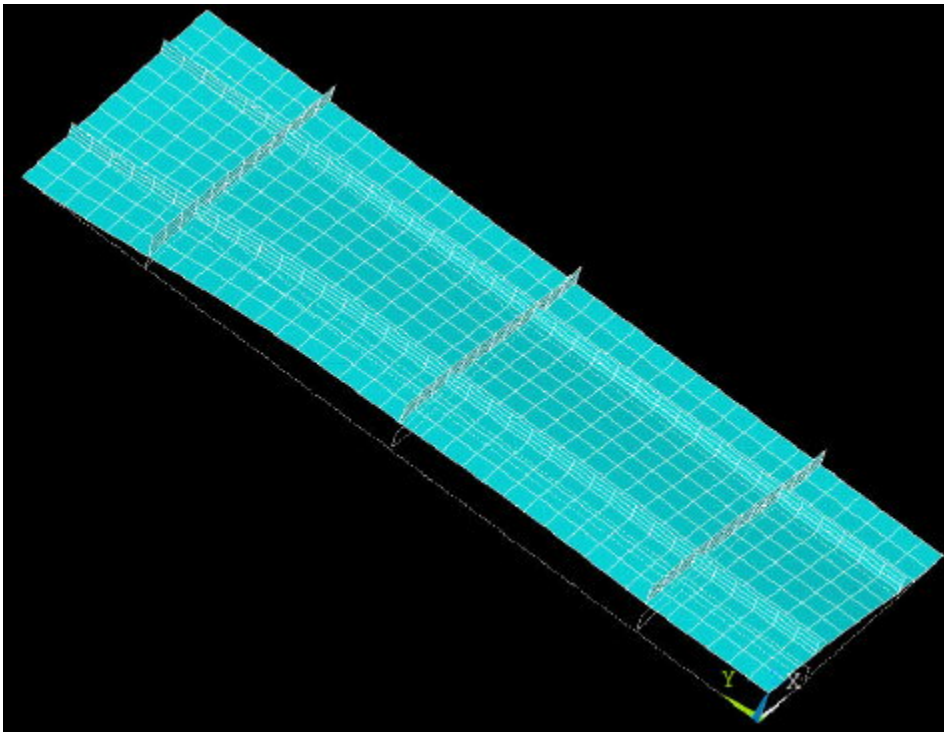
The *Results Viewer* provides a convenient way to review results.

#### Main Menu > General Postproc > Results Viewer

The title bar for the *Results Viewer* indicates that the results are being read from the *shell.rst* file. Rearrange the windows slightly so that you can access the *Results Viewer*, ANSYS GUI and tutorial simultaneously.

Under **Choose a result item**, select **Nodal Solution > DOF Solution > Displaced structure**. Click on the **Plot Results** icon.

Hold down the *Ctrl* key and use the mouse to **manipulate the model view**; check that the BC's and loads have been imposed correctly. The snapshot below shows one view that is useful in checking the BC's. Are the symmetry and clamped conditions satisfied at the corresponding edges for the facesheet as well as the stiffeners? Is the model deformation as you'd expect for a uniform pressure on the facesheet in the +z direction?



The maximum displacement  $DMX=3.88$  mm. Where do you think this occurs? We'll check this in a minute.

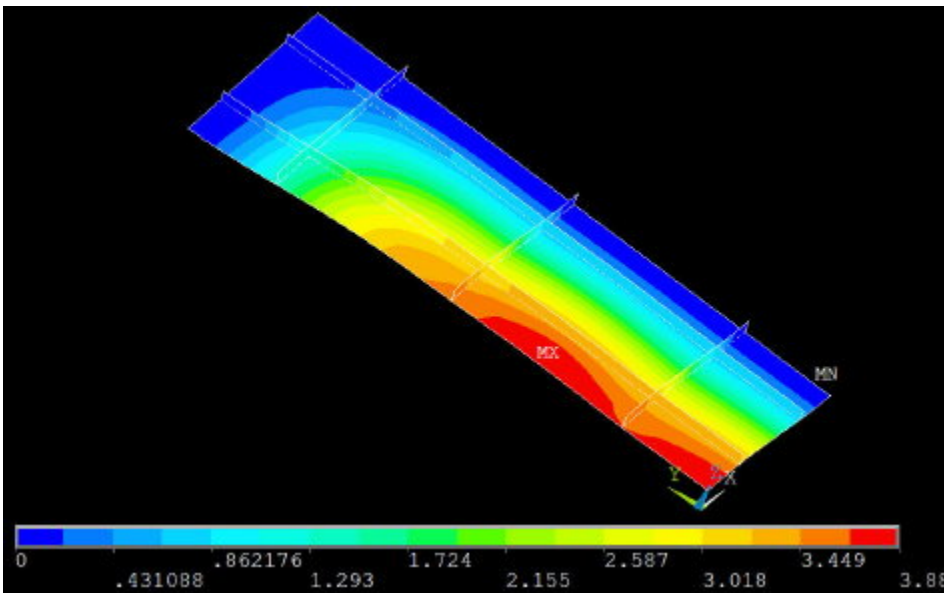
Animate the deformation and obsessively recheck the BC's:

**Results Viewer > Animate Results > Deformed Shape > OK > Def + undef edge > OK**

Select **Forward Only** in the *Animation Controller*.

### Plot w Displacement

In the *Results Viewer*, in place of **Displaced structure**, choose **Nodal Solution > DOF Solution > Z-Component of displacement**. Click on the **Plot Results** icon.



The location of maximum deflection  $w_{\max}$  is labeled MX. This occurs not at the facesheet center but on the edge  $x=0$ , at approximately  $y=L/3$ . As expected, the deflection is zero along the clamped edges.

To save a copy of this plot, select **Results Viewer > Image Capture > Capture to file > OK**.

For filename, enter `w_contours`. This creates the file `_w_contours.png` in your working directory. Check that you can open this file using your pet image viewer.

### Deflection at Facesheet Center

To query the value of the deflection at the center of the facesheet, select the **Query Results** icon in the *Results Viewer*.

Pick the node at  $X=0$ ,  $Y=0$ . The coordinates of the picked node and the corresponding value of *UZ* (i.e.  $w$  displacement) are displayed in the *Query Subgrid Results* menu. The deflection at the center of the facesheet is 3.8 mm.

**Cancel** the *Query Subgrid Results* menu.

### $\sigma_{xx}$ for Facesheet

For clarity, let's first plot  $\sigma_{xx}$  for just the facesheet. Select the elements for the facesheet in two steps:

1. Select areas corresponding to the facesheet.
2. Select elements attached to these previously selected areas. Close the *Results Viewer*.

**Utility Menu > Plot > Elements**

**Utility Menu > Select > Entities**

*Select Entities* menu: Select **Areas**, **By Location** and **Z coordinates**. Under **Min,Max**, enter 0. Make sure **From Full** is selected. Click **Apply**.

Following this, select **Elements**, **Attached to**, **Areas** and **From Full**. Click **Apply** and then **Replot**. You should see that only elements for the facesheet are currently selected.

**Main Menu > General Postproc > Results Viewer > Nodal Solution > Stress > X-Component of stress**

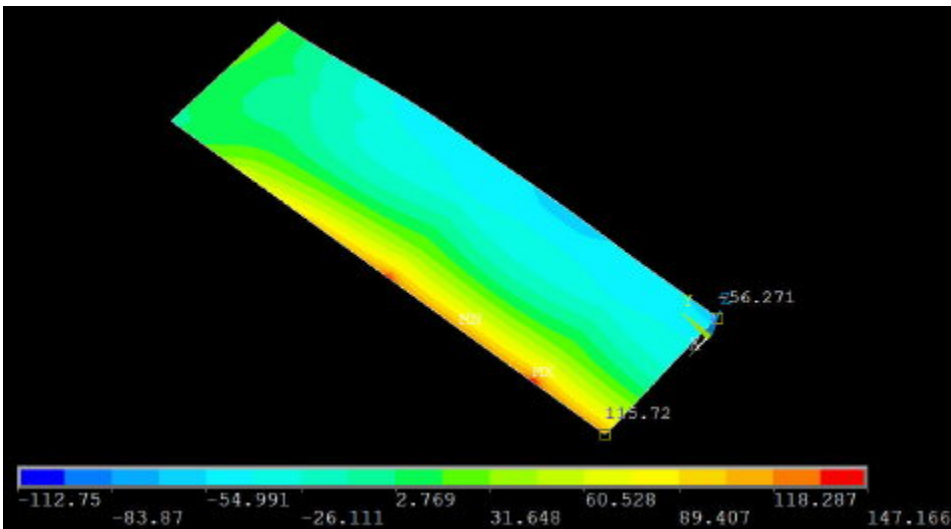
Click **Plot Results** icon.

Shell theory provides the stresses through the thickness of the shell. This is illustrated in the help page for SHELL93 element. If, like me, you are looking at the top of the facesheet, then the contours values correspond to  $\sigma_{xx}$  values for the top face. Check the  $\sigma_{xx}$  value for the top face at the facesheet center:

**Results Viewer > Query Results**

Pick the node at  $X=0$ ,  $Y=0$ . I get  $\sigma_{xx}=51$  MPa at the center-top of the facesheet.

Hold down the **Ctrl** key and flip the model over so that you are looking at the *bottom* of the facesheet as in the snapshot below. Now the contours values correspond to  $\sigma_{xx}$  values for the facesheet *bottom*. Use **Query Results** to determine  $\sigma_{xx}$  at the center-*bottom* of the facesheet. I get -56 MPa. What do you get? The negative sign indicates compression.



(For an enlarged view, click and maximize the browser window)

Thus, at the facesheet center,  $\sigma_{xx}$  varies from a tensile stress of 51 MPa at the top to a compressive stress of 56 MPa at the bottom.

Using **Query Results**, I get  $\sigma_{xx}=116$  MPa at the edge point  $x=W1$ ,  $y=0$  for the bottom of the facesheet (see snapshot above). What's the value you finagled out of ANSYS?

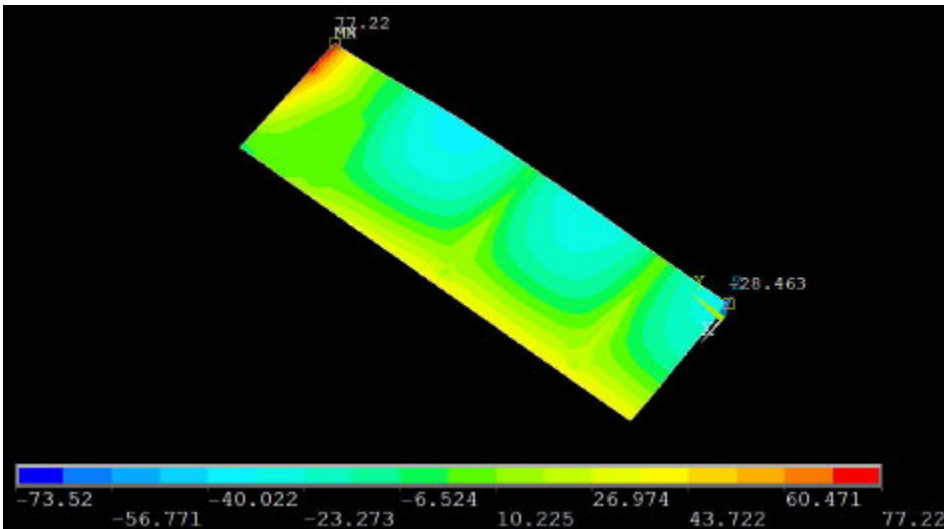
Do these values look plausible? We'll investigate this in [Step 9](#) when we compare the center and edge stress values to plate theory.

**Cancel** the *Query Subgrid Results* menu.

## $\sigma_{yy}$ for Facesheet

**Results Viewer > Nodal Solution > Stress > Y-Component of stress**

Click **Plot Results** icon. The snapshot below shows the  $\sigma_{yy}$  contours.



Using **Query Results**, I get the following values for the bottom of the facesheet (see snapshot above):

1. At the center,  $\sigma_{yy} = -28$  MPa.
2. At the clamped edge location ( $x=0$ ,  $y=L1$ ),  $\sigma_{yy} = 77$  MPa. What are your corresponding values?

## $\sigma_{xx}$ for Stiffeners

Currently, the facesheet is selected and the stiffeners are unselected. To plot stress contours for the stiffeners, we'll unselect the facesheet and select the stiffeners i.e. reverse the selection. This can be done using the *Invert* operation:

**Select Entities menu> Invert**

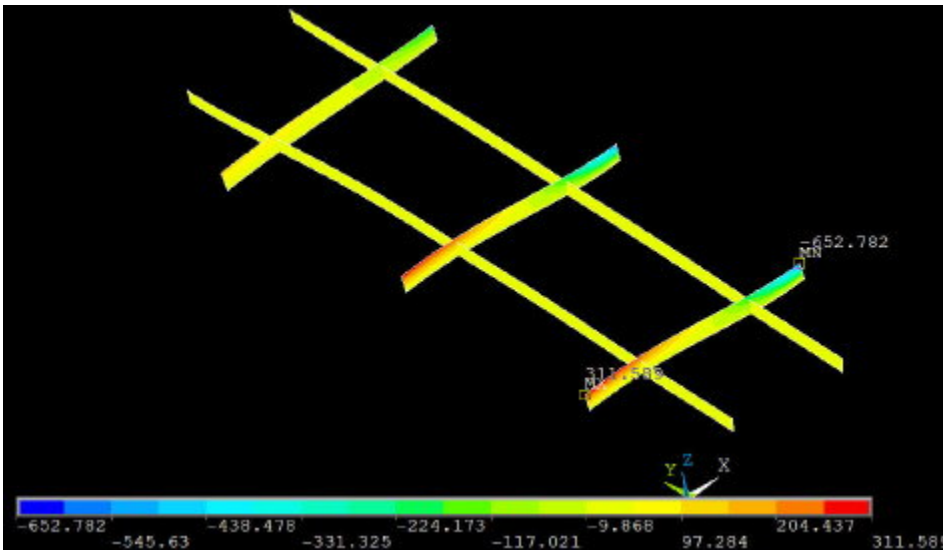
## Select Entities menu> Replot

(In MS-Windows, you can cycle through windows on the desktop by holding down the *Alt* key and repeatedly pressing the *Tab* key. In case the *Select Entities* menu has disappeared, exit the *Results Viewer*, bring up the *Select Entities* menu and re-enter the *Results Viewer*.)

## Results Viewer > Nodal Solution > Stress > X-Component of stress

Click **Plot Results** icon.

Manipulate the view so that you are looking down on the top of the stiffeners as in the snapshot below. You might have to flip the model over.



The  $\sigma_{xx}$  values for the stiffeners in the y-direction is virtually zero; confirm this using **Query Results**. This means that, as expected, the load in the x-direction is carried by the stiffeners in that direction.

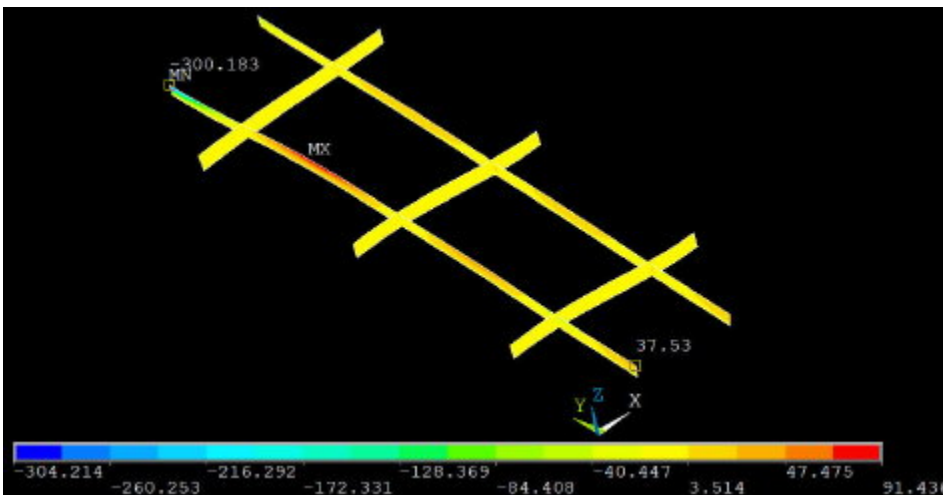
Let's determine  $\sigma_{xx}$  values at the top of the x-stiffener closest to the center; we'll compare these values to plate theory in [Step 9](#). Using **Query Results**, I get the following values at the top of this stiffener (see snapshot above):

1. At  $(x=0, y=L_1/6)$ ,  $\sigma_{xx} \approx 312$  MPa.
2. At the clamped edge location  $(x=W_1, y=L_1/6)$ ,  $\sigma_{xx} \approx -653$  MPa. What are your corresponding values?

## $\sigma_{yy}$ for Stiffeners

## Results Viewer > Nodal Solution > Stress > Y-Component of stress

Click Plot Results icon.



The  $\sigma_{yy}$  values for the stiffeners in the x-direction is virtually zero; confirm this using **Query Results**. This means that, as expected, the load in the y-direction is carried by the stiffeners in that direction.

Let's determine  $\sigma_{yy}$  values at the *top* of the  $y$ -stiffener closest to the center which we'll compare to plate theory in [Step 9](#). Using *Query Results*, I get the following values at the top of the stiffener (see snapshot above):

1. At  $(x=W_1/4, y=0)$ ,  $\sigma_{yy} = 38$  MPa.
2. At the clamped edge location  $(x=W_1/4, y=L_1)$ ,  $\sigma_{yy} = -300$  MPa. What are your corresponding values?

Save: **Toolbar > SAVE\_DB**

Go to [Step 9: Validate the Results](#)

[See and rate the complete Learning Module](#)

Go to [all ANSYS Learning Modules](#)