

Plate With a Hole - Numerical Solution

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

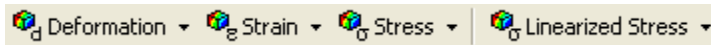
[Comments](#)

Numerical Solution

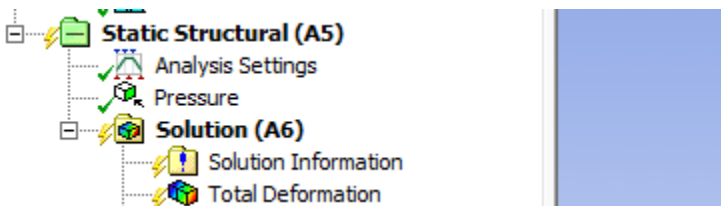
Now we are ready to choose what kind of results we would like to see.

Deformation

To add deformation to the solution, first click **Solution** to add the solution sub menu to menu bar



Now in the solution sub menu click **Deformation > Total** to add the total deformation to the solution. It should appear in the outline tree.



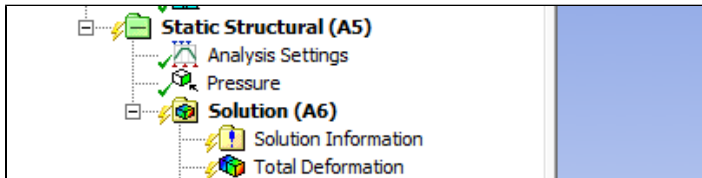
Normal Stresses

Sigma_xx

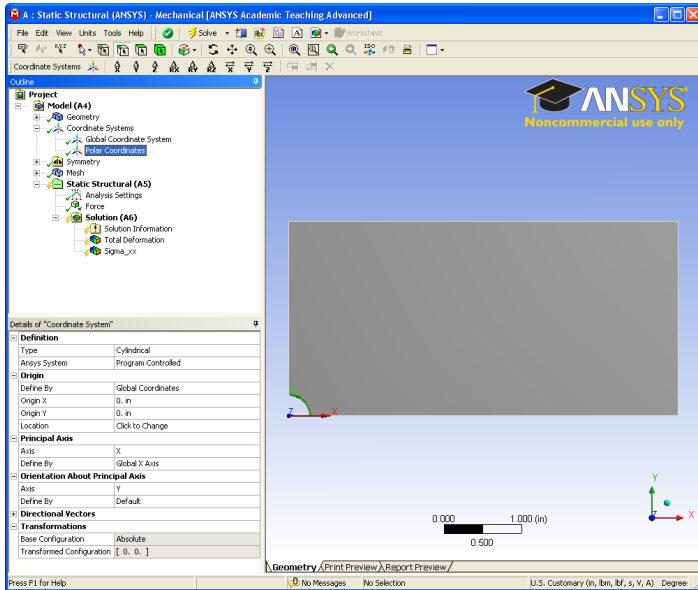
To add the normal stress in the x-direction, in the solution sub menu go to **Stress > Normal**. In the details view window ensure that the **Orientation** is set to **X Axis**. Let's rename the stress to Stress_xx by right clicking the stress, and going to rename.

Sigma_r

To add the polar stresses, we need to first define a polar coordinate system. In the outline tree, right click **Coordinate System > Insert > Coordinate System**.



This will create a new Cartesian Coordinate System. To make the new coordinate system a polar one, look to the details view and change the **Type** Parameter from Cartesian to Cylindrical. To define the origin, change the **Define By** parameter from Geometry to Global Coordinate System. Put the origin coincident with the global coordinate systems origin ($x = 0$, $y = 0$). Now that the polar coordinates have been created, let's rename the coordinate system to make it more distinguishable. Right click on the coordinate system you just created, and go to **Rename**. For simplicity sake, let's just name it Polar Coordinates.



[Click here to enlarge image](#)

Now, we can define the radial stress using the new coordinate system. Click **Solution > Stress > Normal**. This will create "Normal Stress 2", and list its parameters in the details view. We want to change the coordinate system to the polar one we just created; so in the details view window, change the **Coordinate System** parameter from "Global Coordinate System" to "Polar Coordinates". Ensure that the orientation is set to the x-axis, as defined by our polar coordinate system. Now the stress is ready. Let's rename it to Sigma_r and keep going.

Sigma_theta

Now let's add the theta stress. This is too a normal stress, so create a new normal stress as you did for Sigma_xx and Sigma_r. Now, change the coordinate system to Polar Coordinates, as you did for Sigma-r. Next, change the Orientation to the Y axis. The Y axis should be in the theta direction by default. Rename the stress to Sigma_theta.

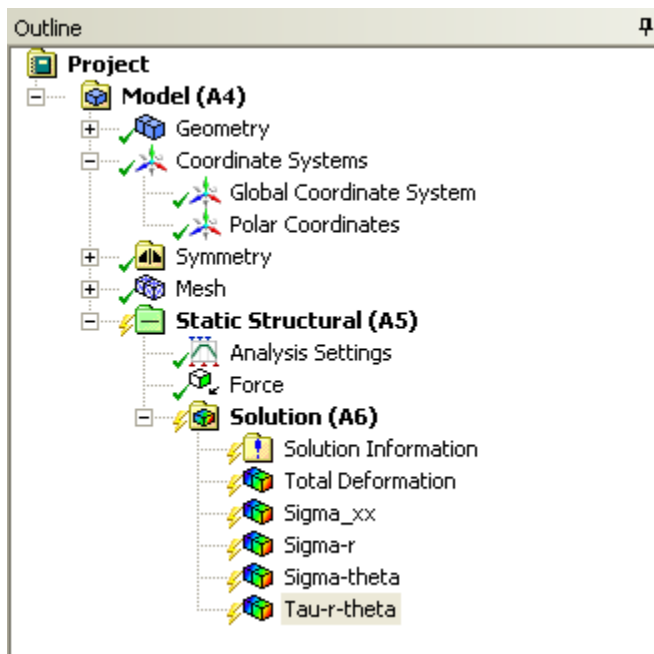
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Normal Stress
Orientation	Y Axis
By	Time
Display Time	Last
Coordinate System	Polar Coordinates
Calculate Time History	Yes
Identifier	
Integration Point Results	
Display Option	Averaged
Results	
☐ Minimum	
☐ Maximum	
Information	

Tau_r-theta

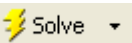
Finally, let's add the shear stress in the r-theta direction. To do this, we go to **Solution > Stress > Shear**. You'll notice that now, in the details view window, the stress needs two directions to define it. In order to solve for the r-theta shear, we need to change the **Coordinate System** parameter from the Global Coordinate System to Polar Coordinates. Also, ensure that the Orientation is in the XY direction (in polar, this will be r_theta by the coordinate system we created). Rename the stress to Tau_r - theta.

Details of "Shear Stress"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Shear Stress
Orientation	XY Plane
By	Time
Display Time	Last
Coordinate System	Polar Coordinates
Calculate Time History	Yes
Identifier	
Integration Point Results	
Display Option	Averaged
Results	
☐ Minimum	
☐ Maximum	
Information	

This is what your outline tree should look like at this point:



Solve!

To solve for the stresses and deformation, we now hit the solve button. 

Keep going! Almost done!

[Go to Step 6: Numerical Results](#)

[Go to all ANSYS Learning Modules](#)