



Faculty of Engineering and Applied Science

MECE 2310U - Concurrent Engineering and Design

Fall 2013

Laboratory Manual 4

Finite Element Analysis (FEA)

Course Instructor:

Remon Pop-Iliev, Ph.D, P.Eng

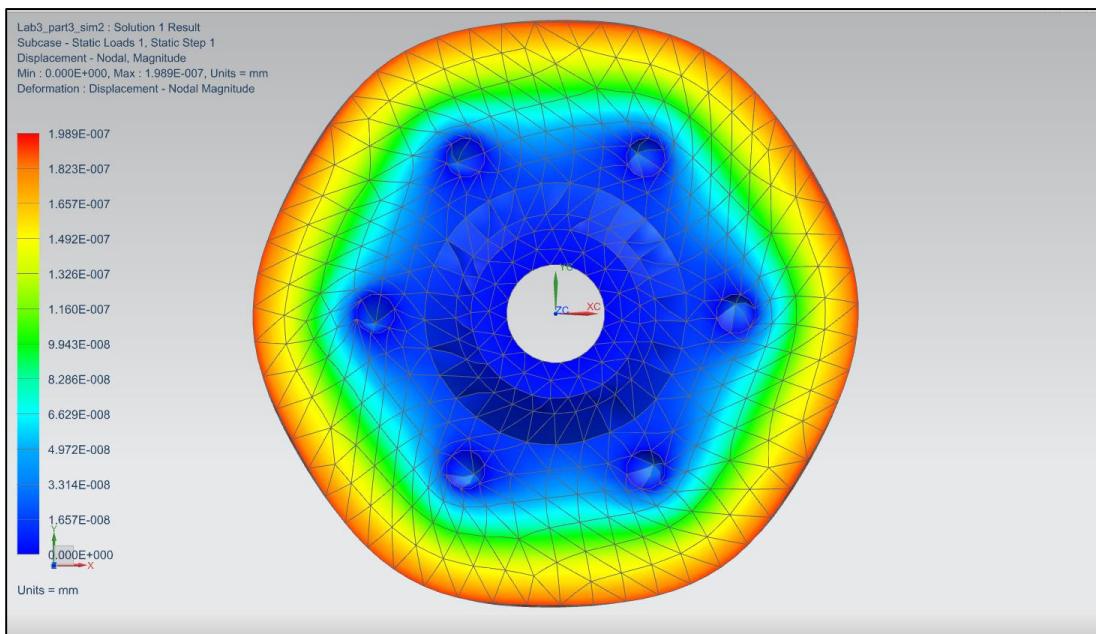
Lab Instructor:

Mike MacLeod (mike.macleod@uoit.ca)

Lab Teaching Assistants:

Adam Reid (adam.reid@uoit.ca)

Ian Wood (ian.wood@uoit.ca)



Laboratory Objectives

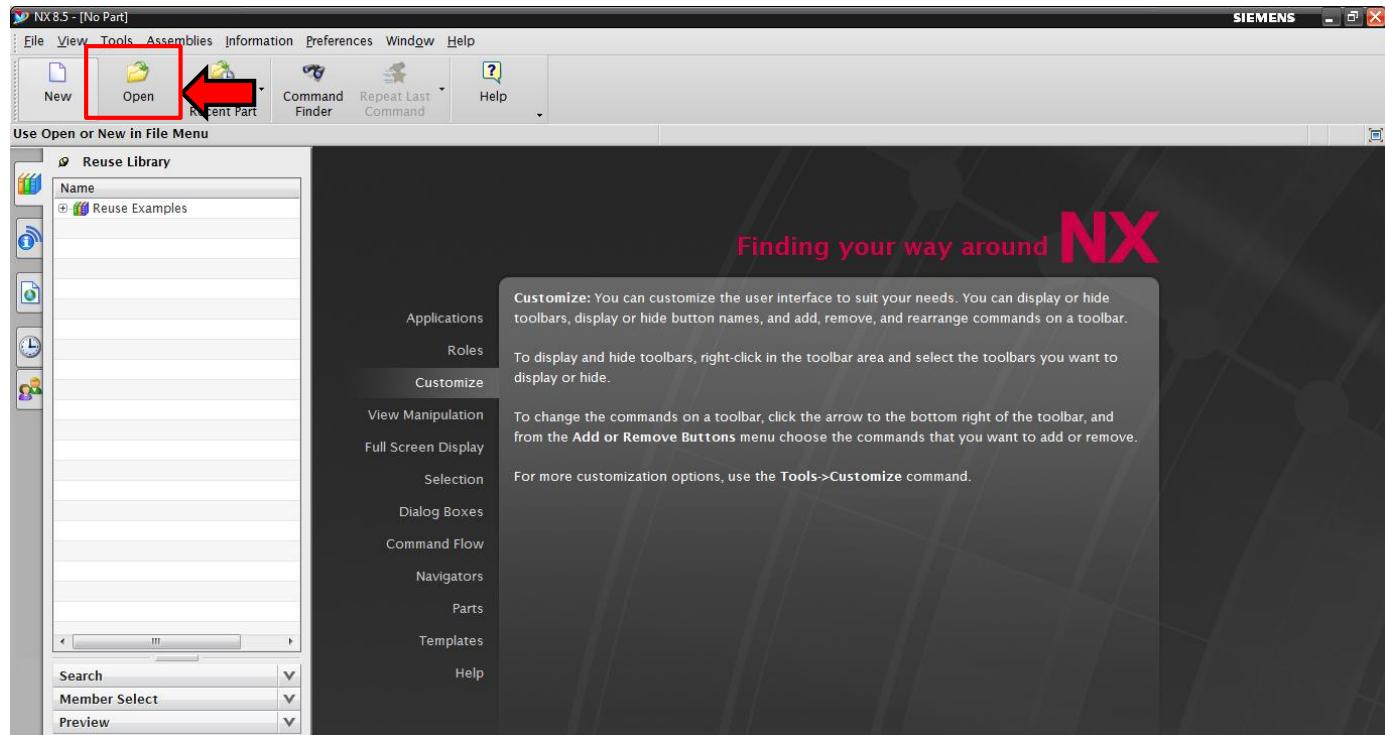
The purpose of this laboratory exercise is to guide the user through some of the basic steps that are required to perform Finite Element Analysis on a simple part and gain experience with:

- Meshing
- FEA (Finite Element Analysis)
- Applying loads/ torques
- Applying constraints
- Creating and understanding simulations
- Altering parameters for different results

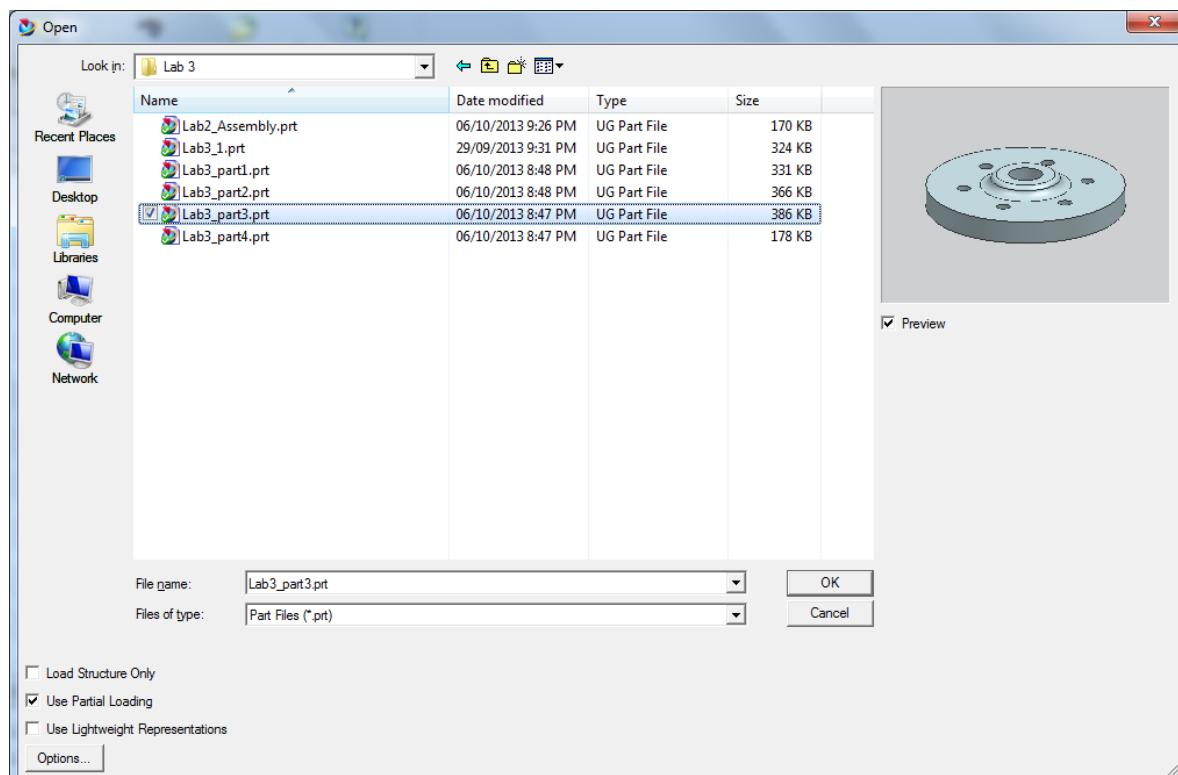
Beginning Module 4

Step 1

Begin by clicking on ‘Open’ in the upper left corner to begin designing your analysis on the disk created from Part III: Disk A from lab manual 3. This previously constructed part will be used for this Finite Element Analysis learning module

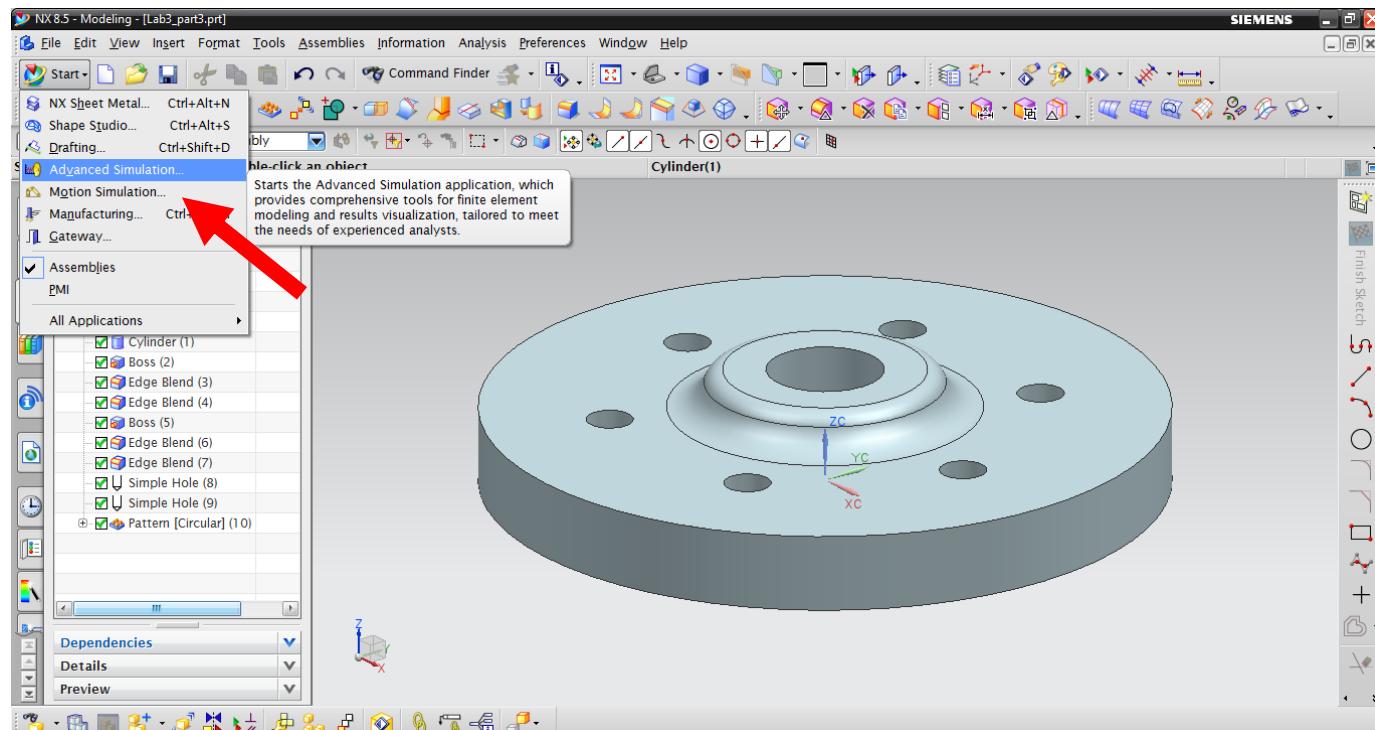


Find the save location and previously saved name of Part III: Disk A (shown below) and open the part



Step 2

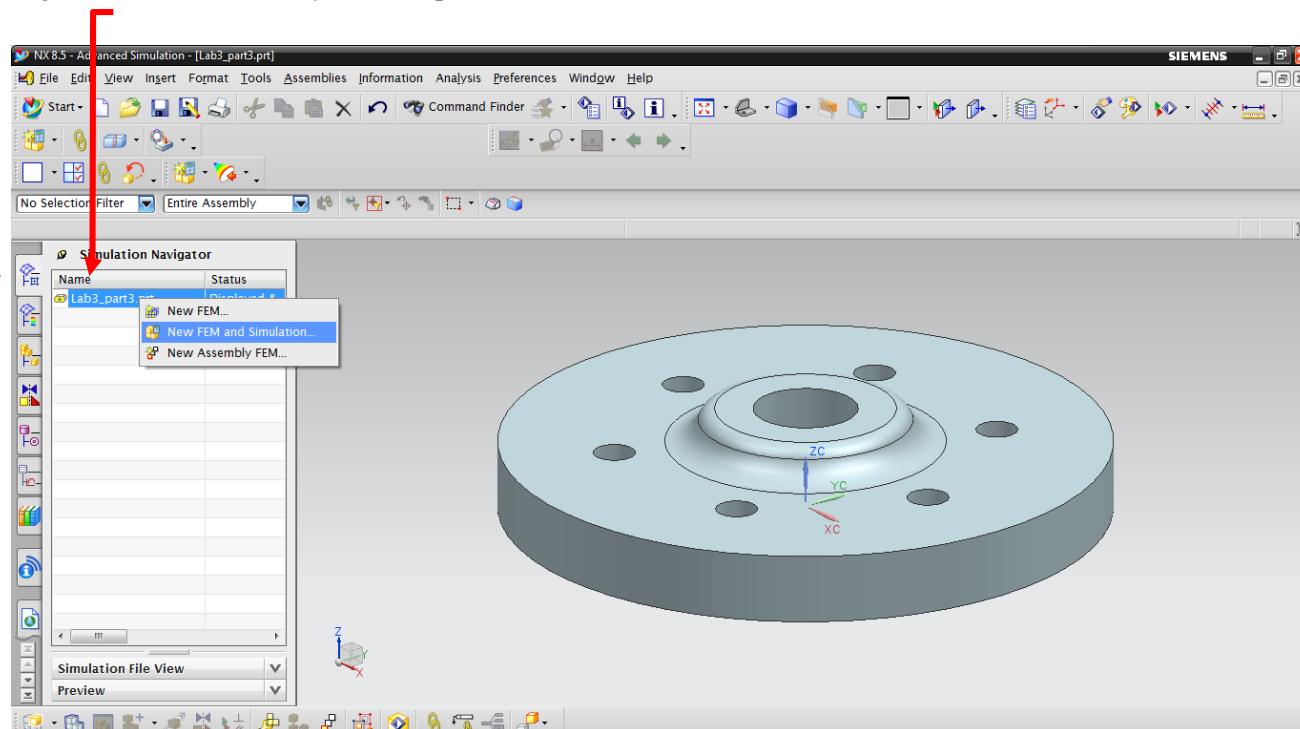
Click on ‘Start’ button and select the ‘Advanced Simulation’ from the drop down list



Step 3

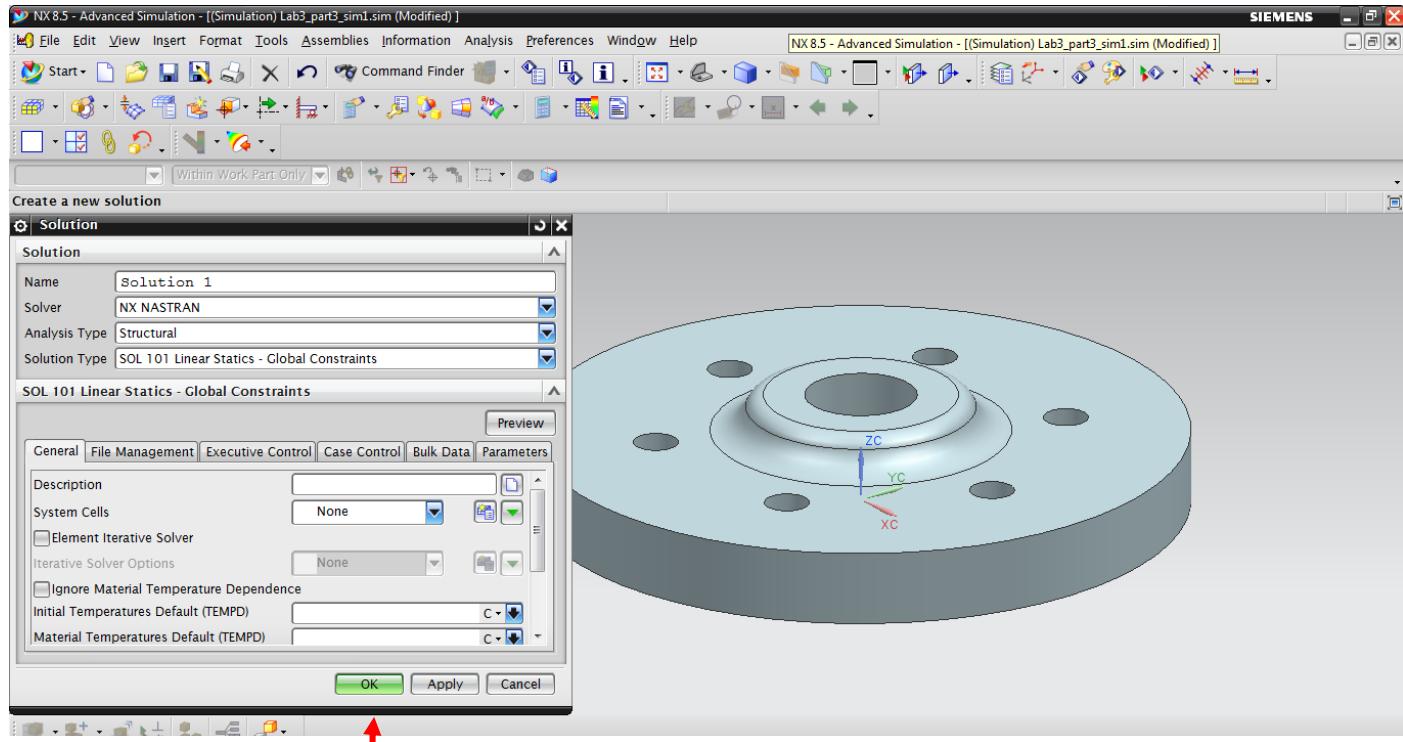
Within the Advanced Simulation mode move the cursor over to the left side of the screen and click on the ‘Simulation Navigator’ tab shown in figure 4.3.

Right click the mouse on your Disk part and select the ‘New FEM and Simulation’ from the list.



Step 4

In the New FEM and Simulation menu that appears, all the default settings are already in place. Click ‘OK’. After selecting ‘OK’, the following dialogue should appear



Step 5

The default settings in the Solution menu are adequate for this lab module, so go ahead and click ‘OK’

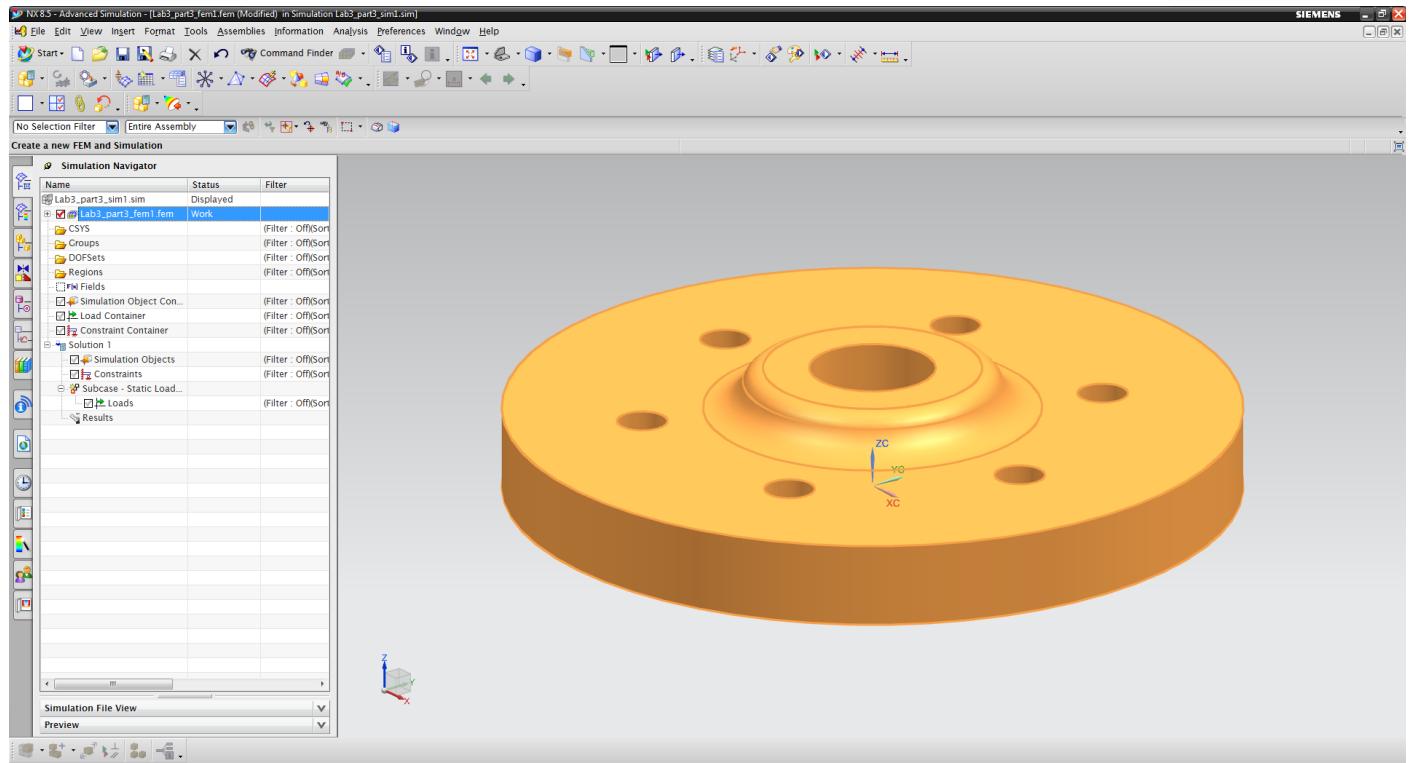
Step 6a

TIP Remember to save your part frequently!

There are several steps that must be performed in order to properly carry the FEA on the disk. The first step that needs to be carried out is the adding of physical properties such as the material type. The properties and constraints are divided into two parts the FEM and the Simulation and execution.

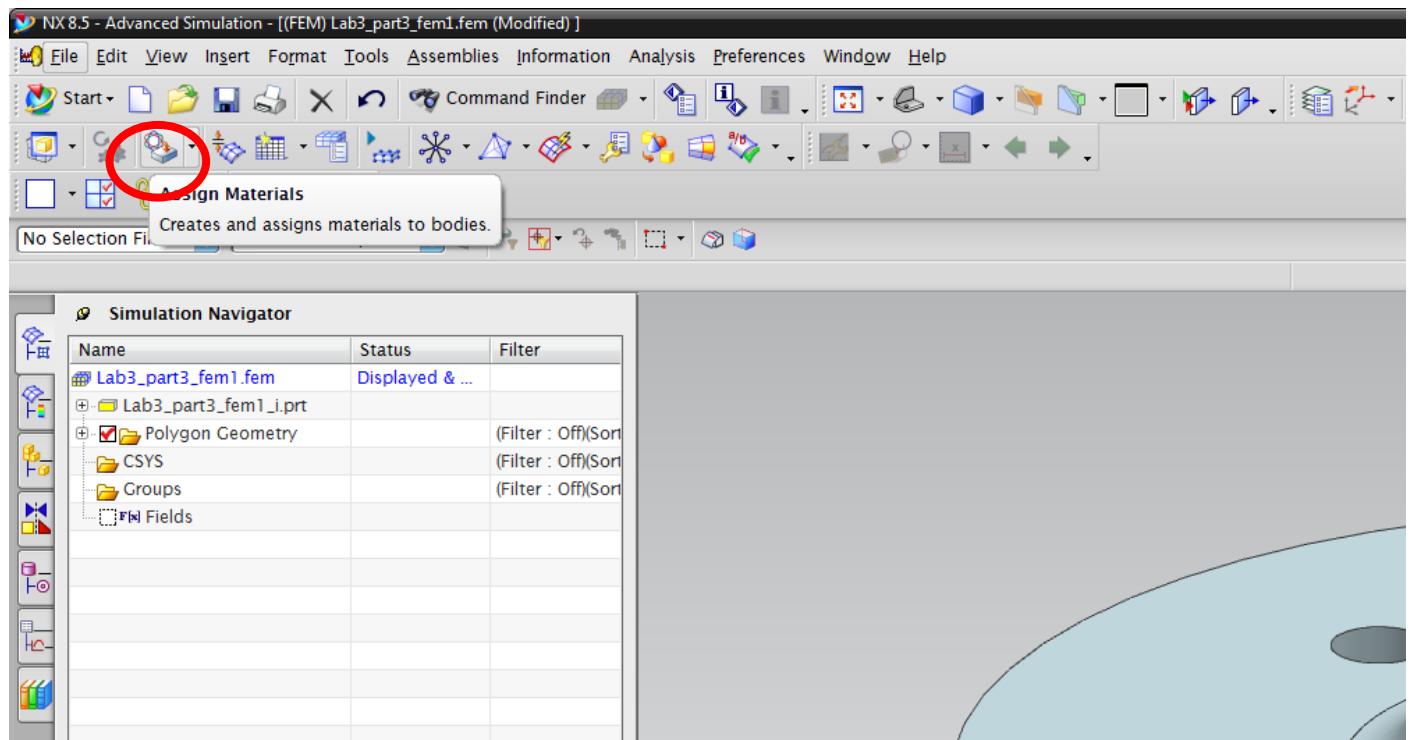
Step 6b

Using the ‘Simulation Navigator’ tab on the right side of the work area, right click on the heading ‘YOURPART_fem1.fem’ and select ‘Make Displayed Part’ from the list.



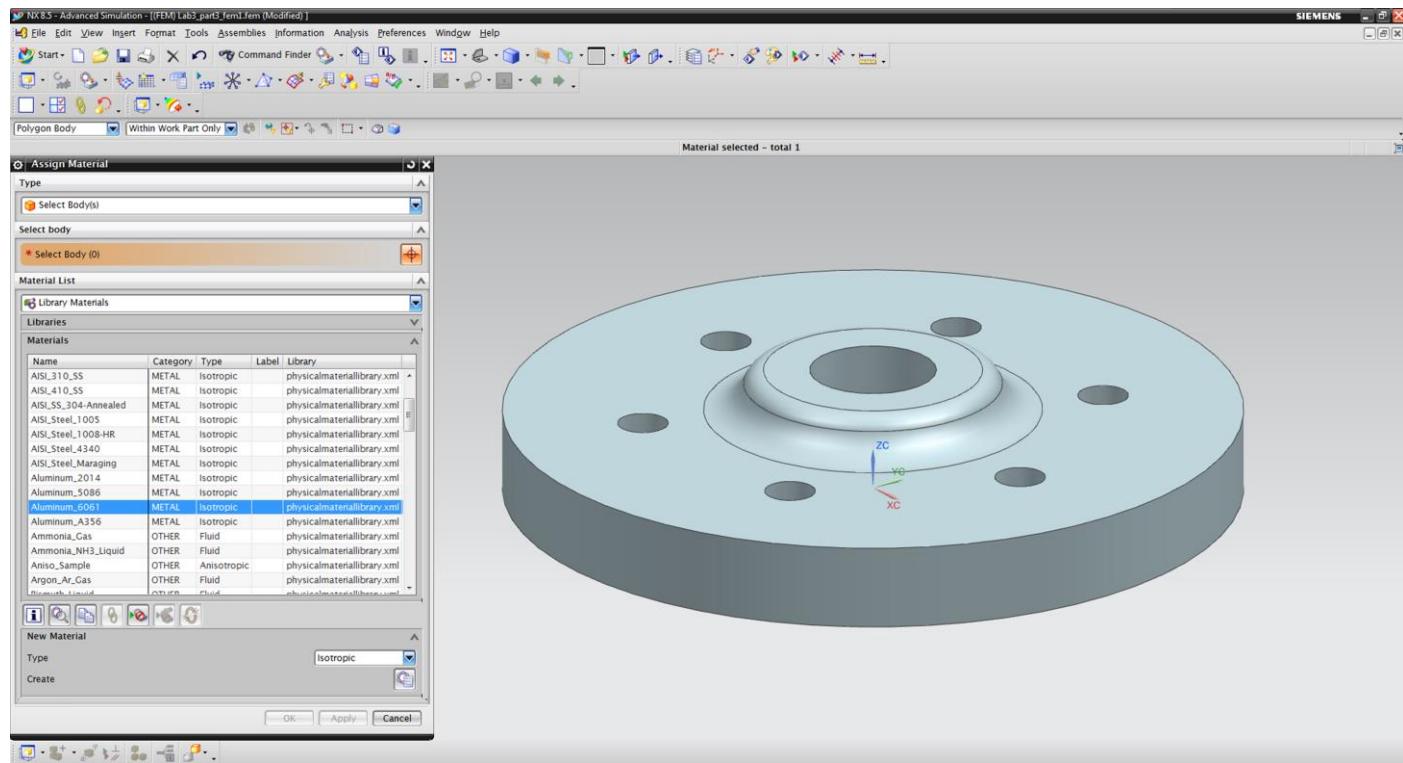
Step 7

To assign the materials properties for the design model click on the ‘Assign Properties’ icon as shown



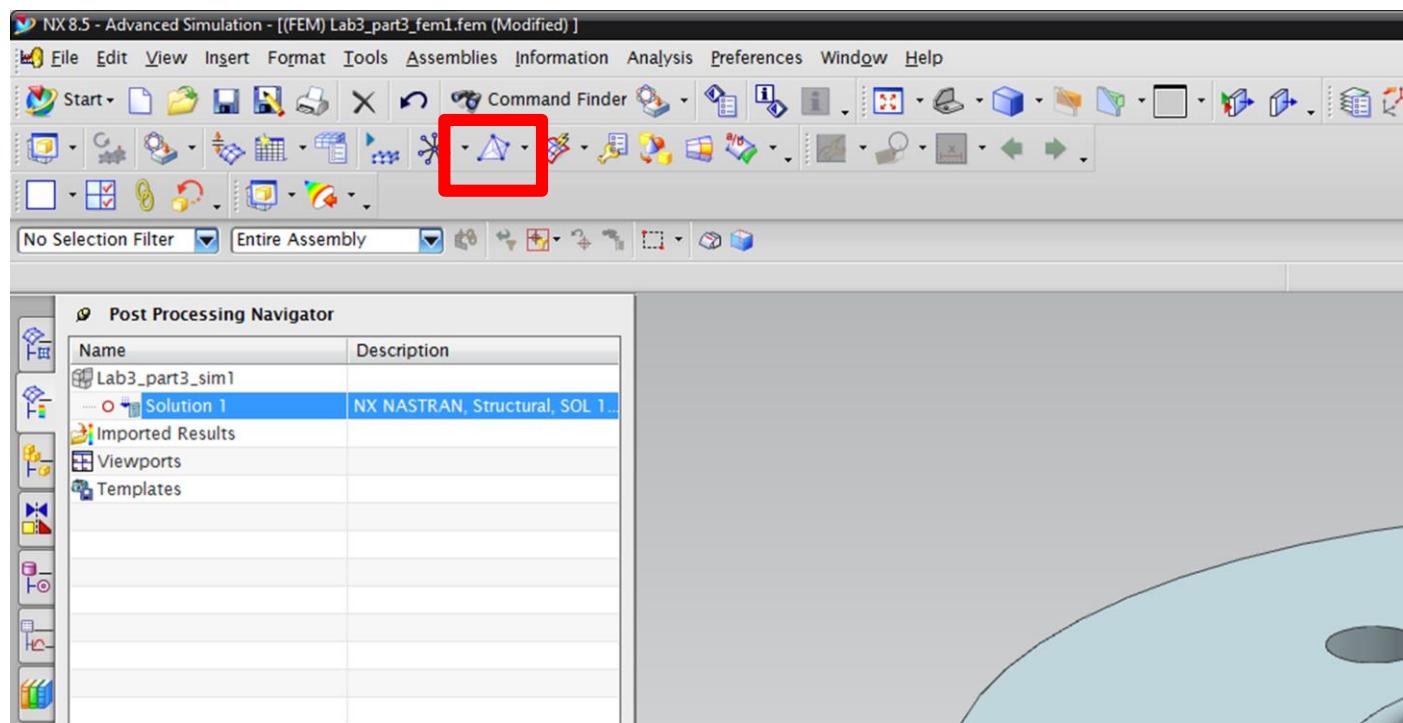
Step 8

If you click the assign properties icon a full list of available materials will appear. Highlight the ‘Aluminum_6061’ material under the Materials section (above), choose the disk in the work area and Click ‘OK’.



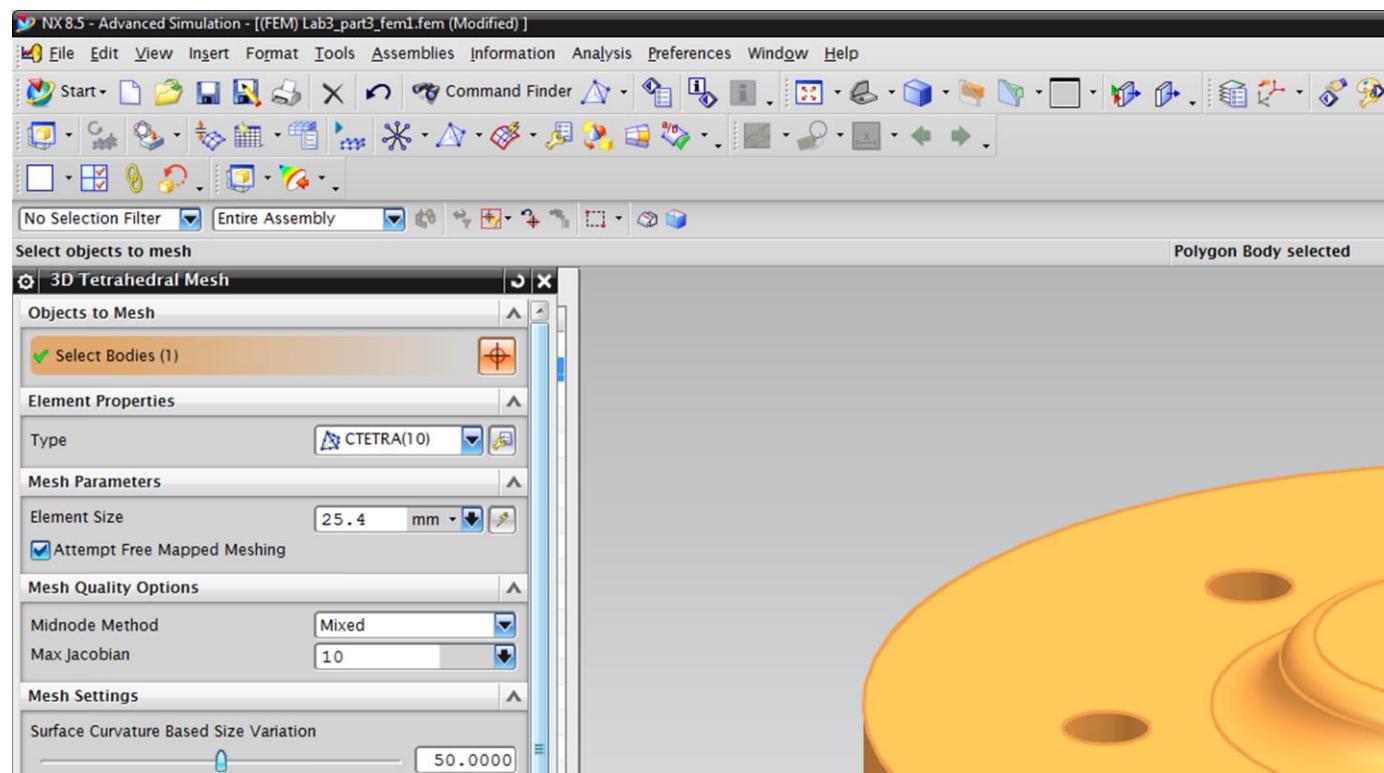
Step 9

Click on the ‘3D Tetrahedral’ icon  located in the task bar.



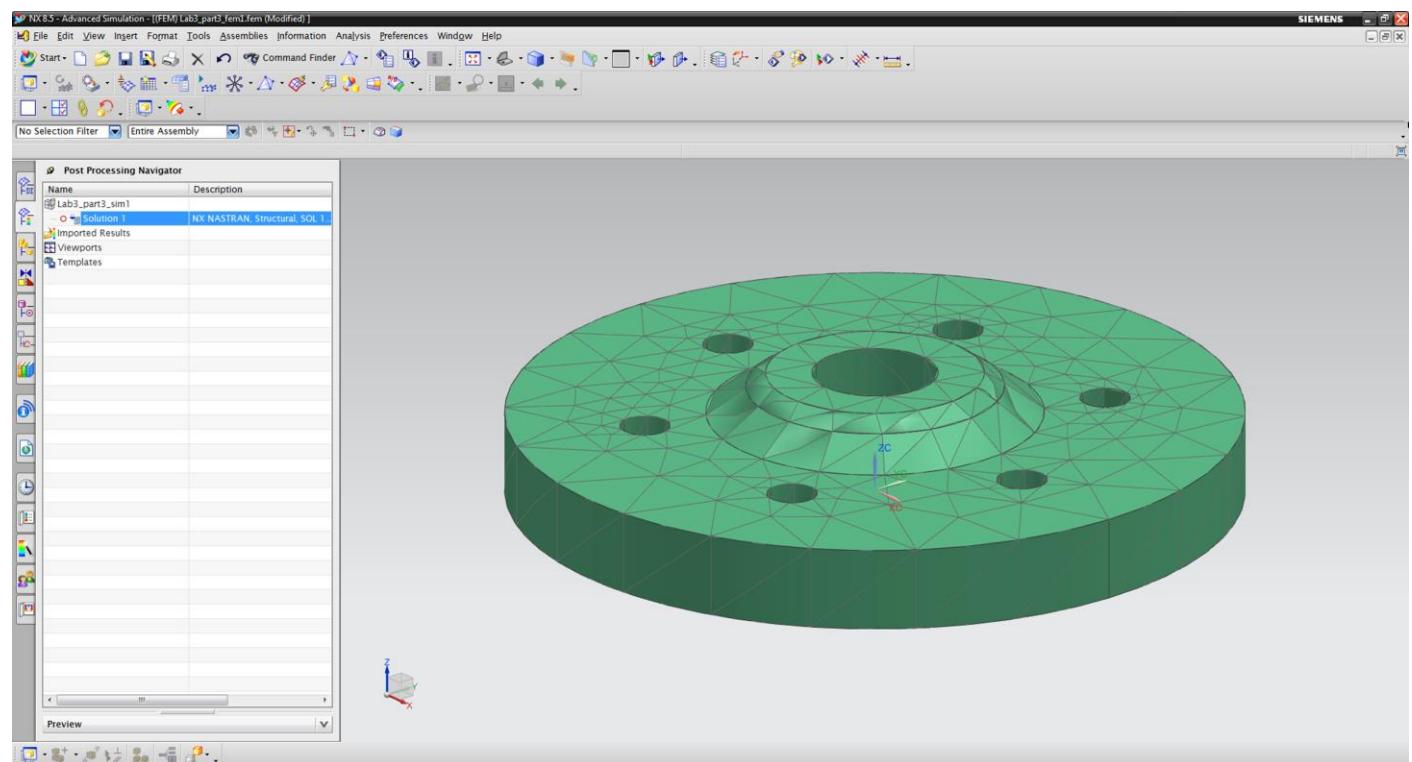
Step 10

Select the disk as the body you wish to apply the mesh to. Change the Element Size to 25.4 and click ‘OK’



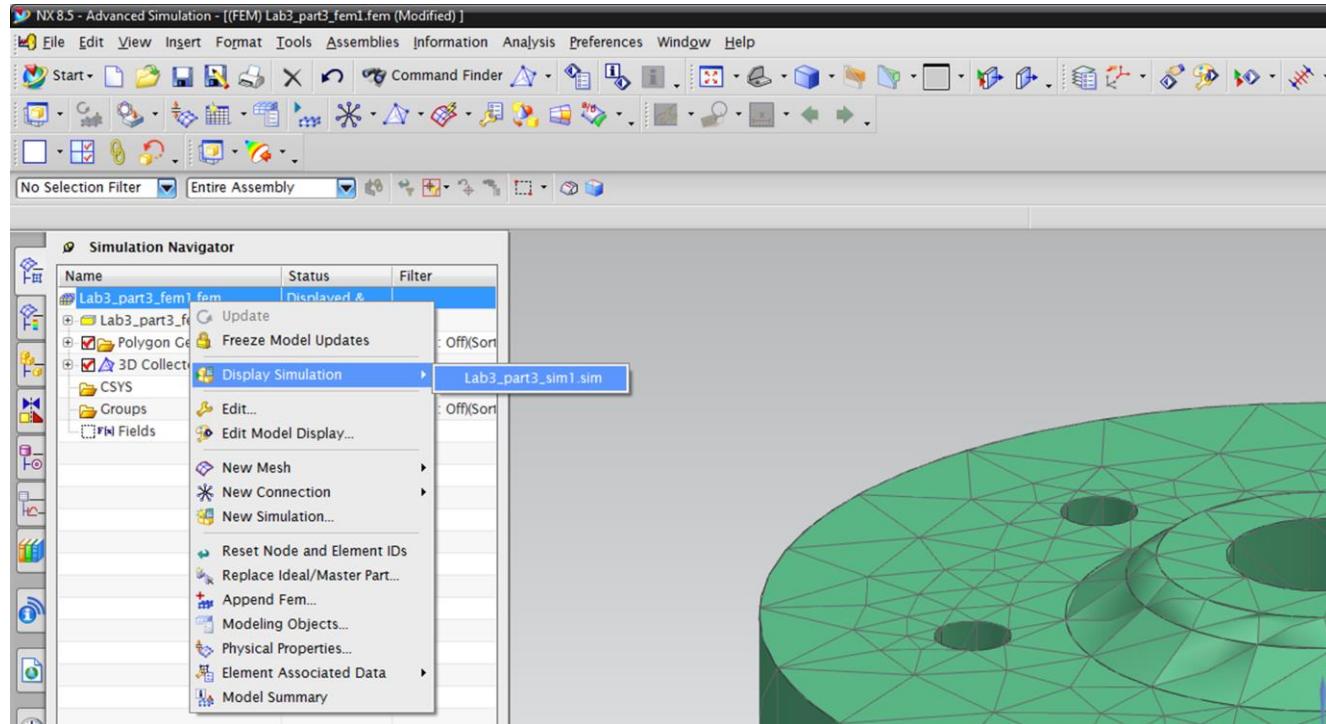
Step 11

After completing the previous step your part should now resemble a series of meshed triangles as seen below.



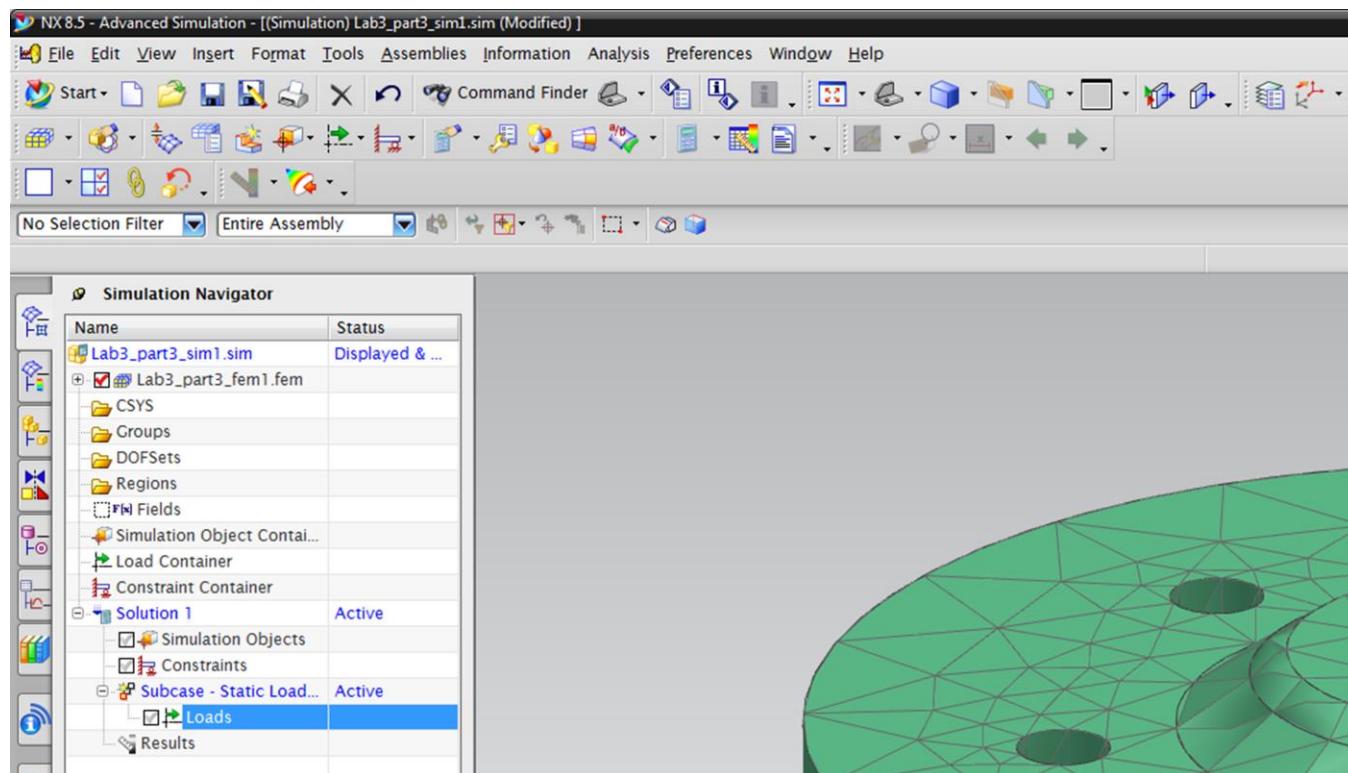
Step 12

Now that you have completed the basic steps for adding the FEM properties to the disk, the simulation environment can now be created. By right clicking on ‘YOURPART_fem1.fem’ in the ‘Simulation Navigator’ dialogue, hovering the mouse over ‘Display Simulation’ and clicking on ‘YOURPART_sim1.sim’



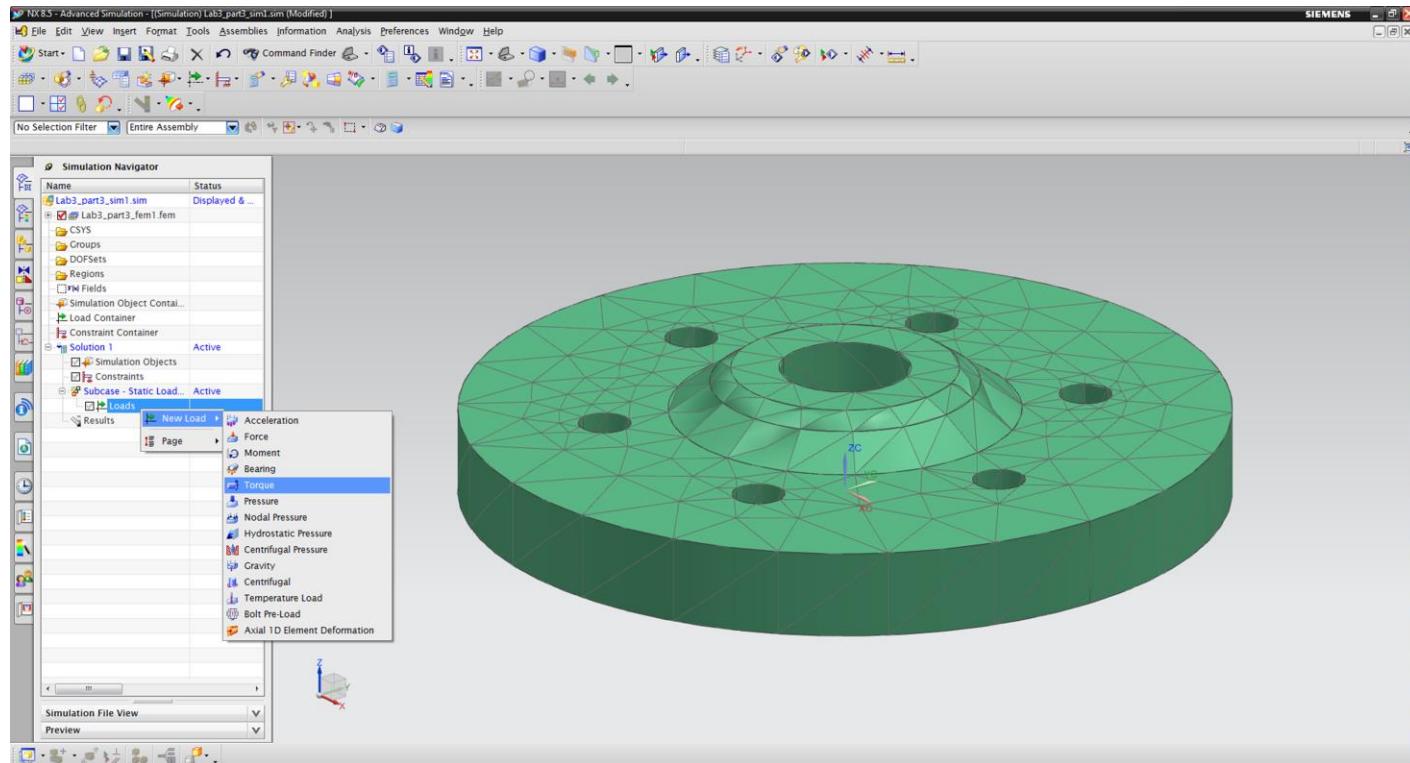
Step 13

When the simulation is made the displayed part, the Simulation Navigator will now display a different set of directories. This can be shown below:



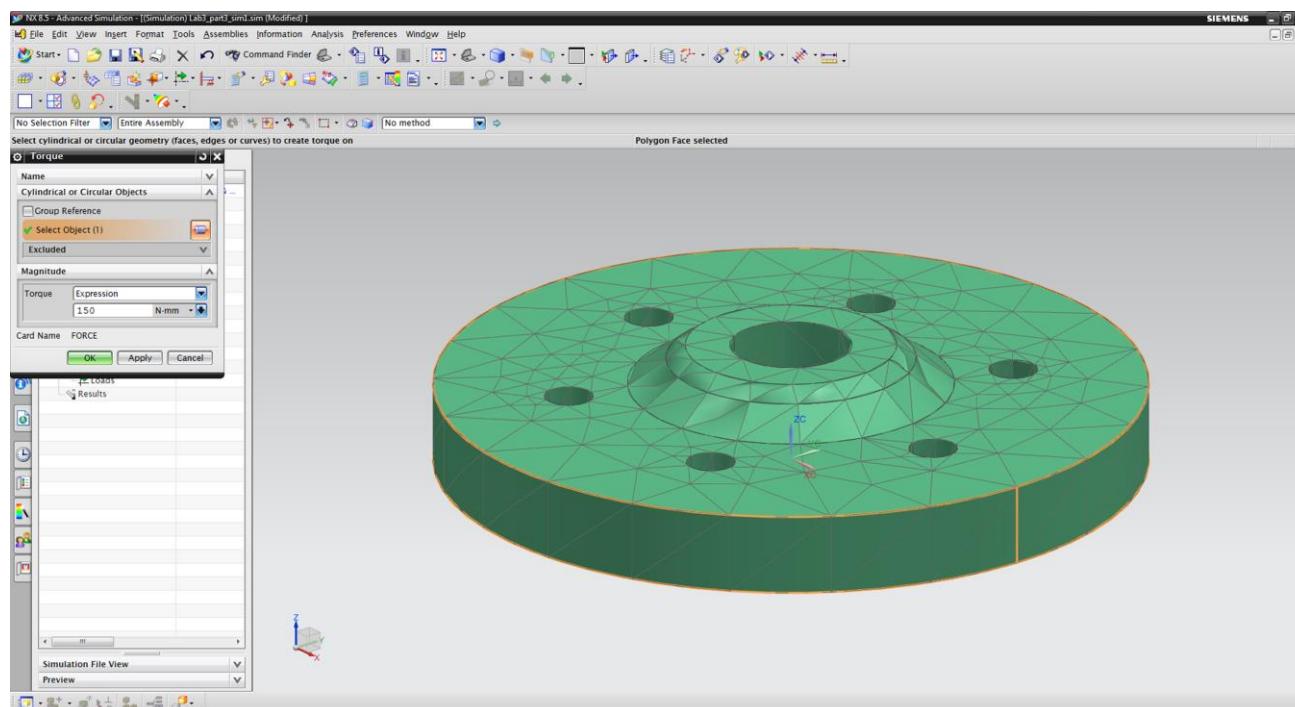
Step 14

When applying a load to your part there are a variety of methods to choose from. In this module you will study the **Torque** load since it can be assumed that the disk you developed for a head stock assembly would experience a torque load. Right click on ‘Loads’, located near the bottom of the ‘Simulation Navigator’, hover the mouse over ‘New Load’, and select ‘Torque’. This is how to apply a torque to the part.



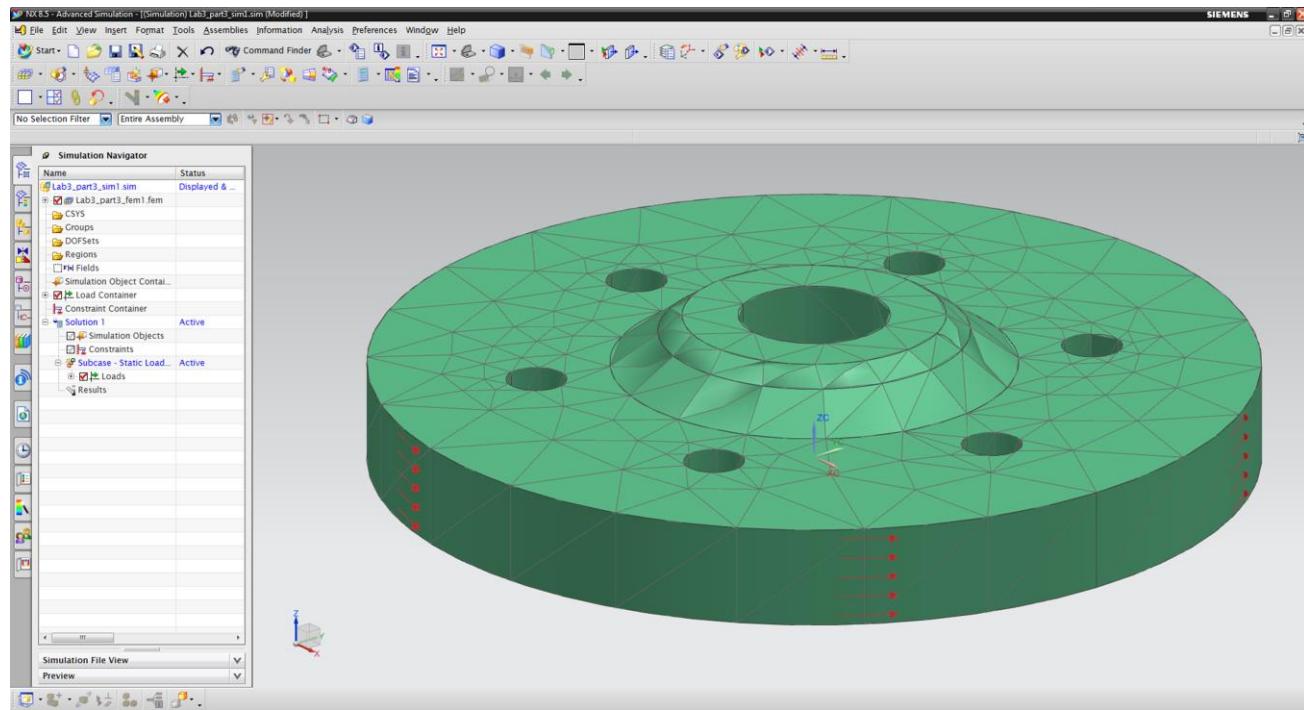
Step 15

In the Torque menu under the Magnitude section enter a testing torque value of 150 N-m. Use the drop down menu to the right of the Torque value to select the units you wish to use. Pick the outside circumference of the disk as shown below and click ‘OK’.



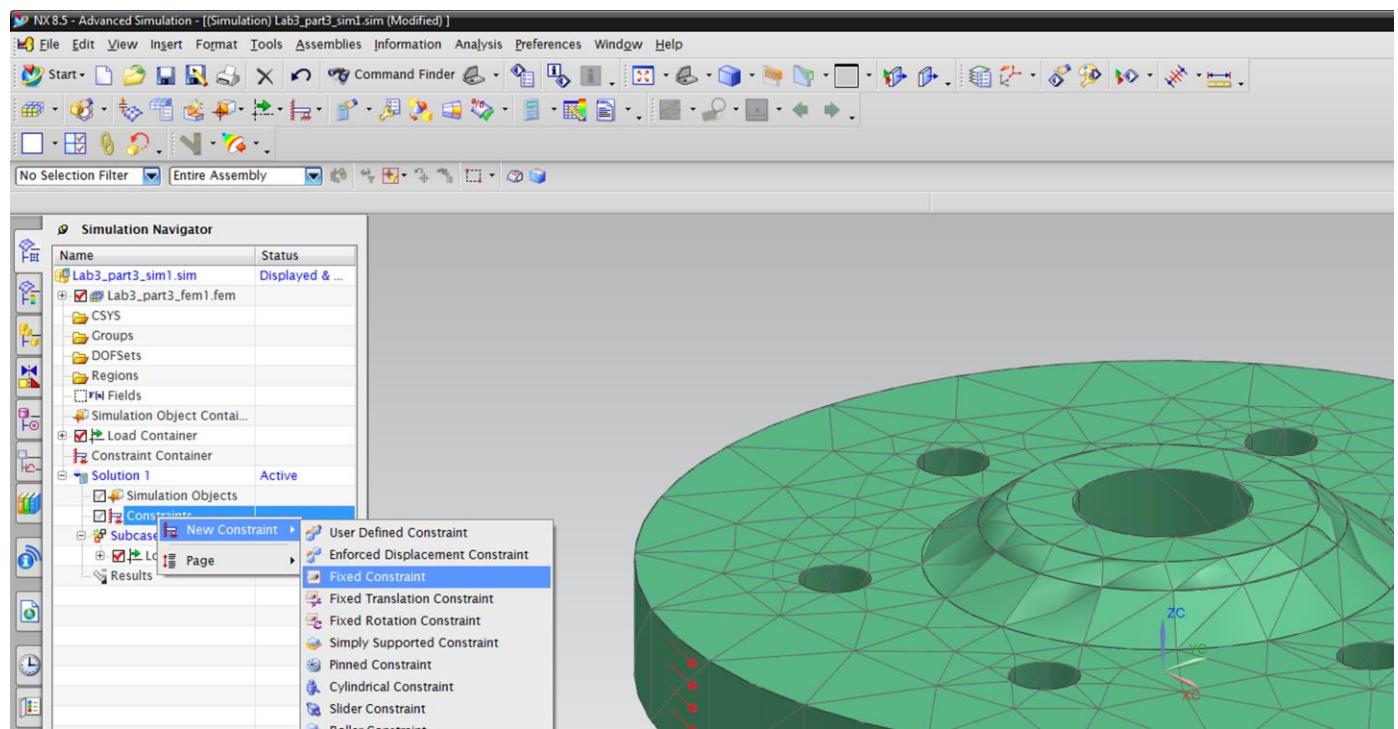
Step 16

Once the Torque load has been added to the surface edge of the disk, indication arrows will appear showing the way that the load is being applied



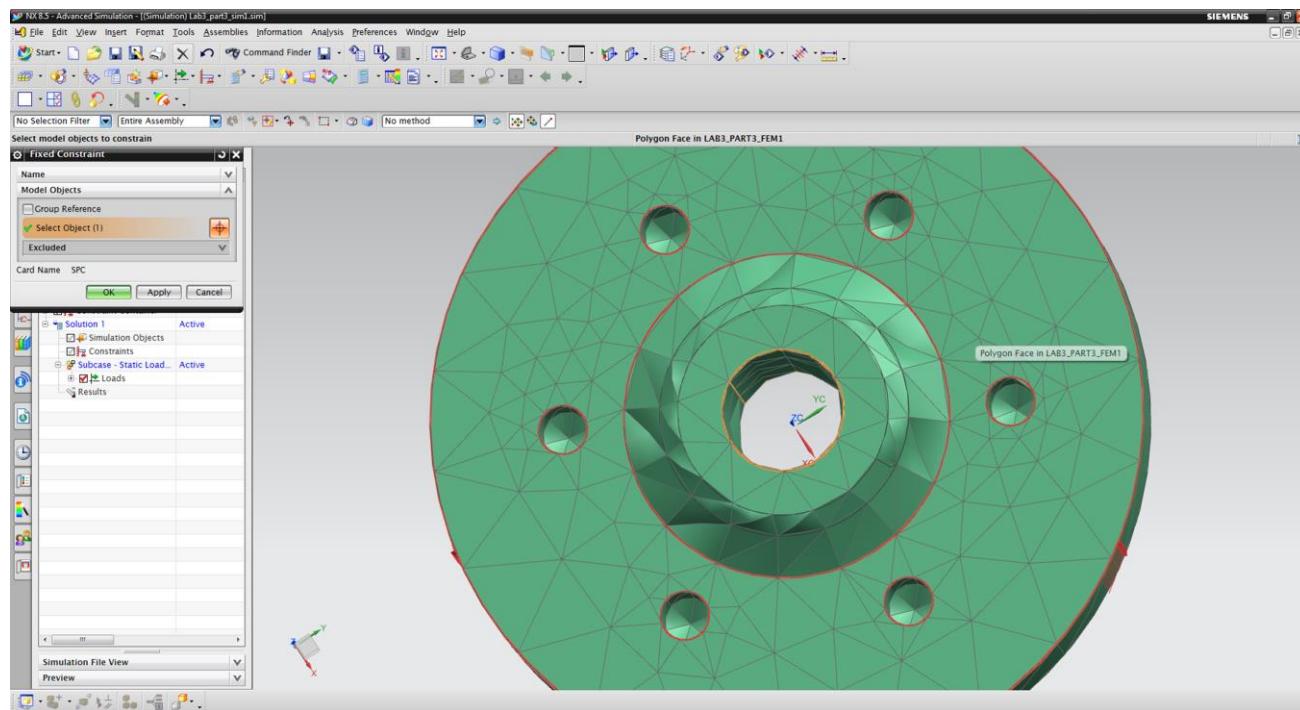
Step 17

When applying constraints to the disk, there are many ways in which to restrict the movement of your part. Since this disk would in reality be ideally constrained in the middle, the application of a fully fixed center hole will be used. In the case of adding components to the assembly that would be bolted to the holes located around the outside parameter of the disk. Right click on ‘Constraint’, hover over ‘New Constraint’, and click on ‘Fixed Constraint’.



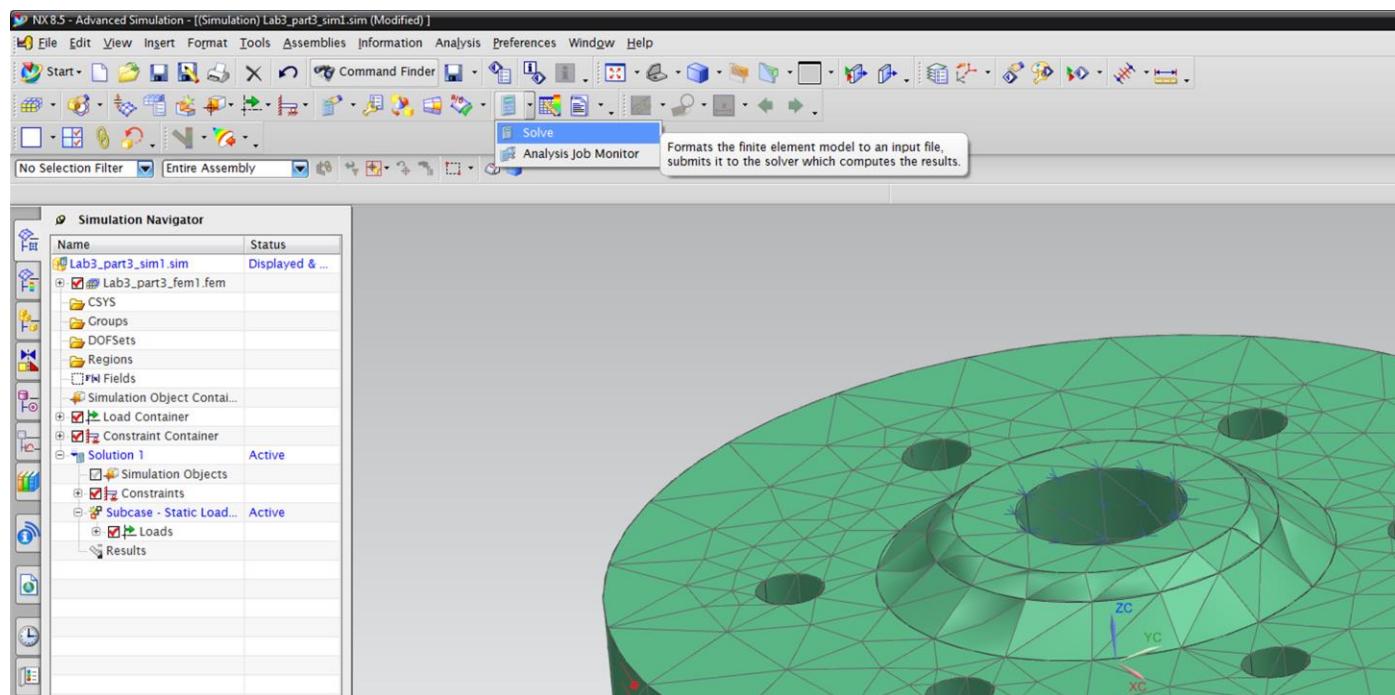
Step 18

Select the center hole in the middle of the disk. Make sure to select the face, not just the edge.



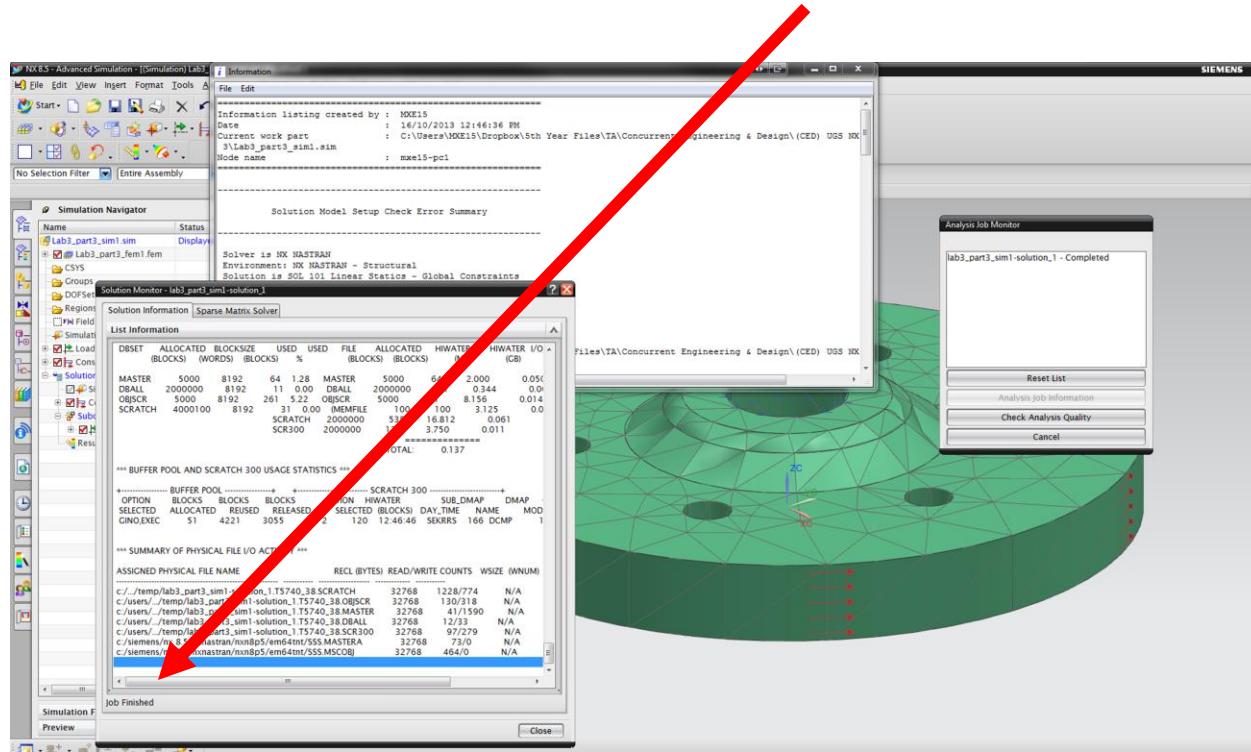
Step 19

Click on the drop down arrow if it is not in the existing visible menu and select the ‘Solve’ calculator. This formats the information you have input into a readable graphics file allowing you to view the stress and displacement of the part with the given input variables.



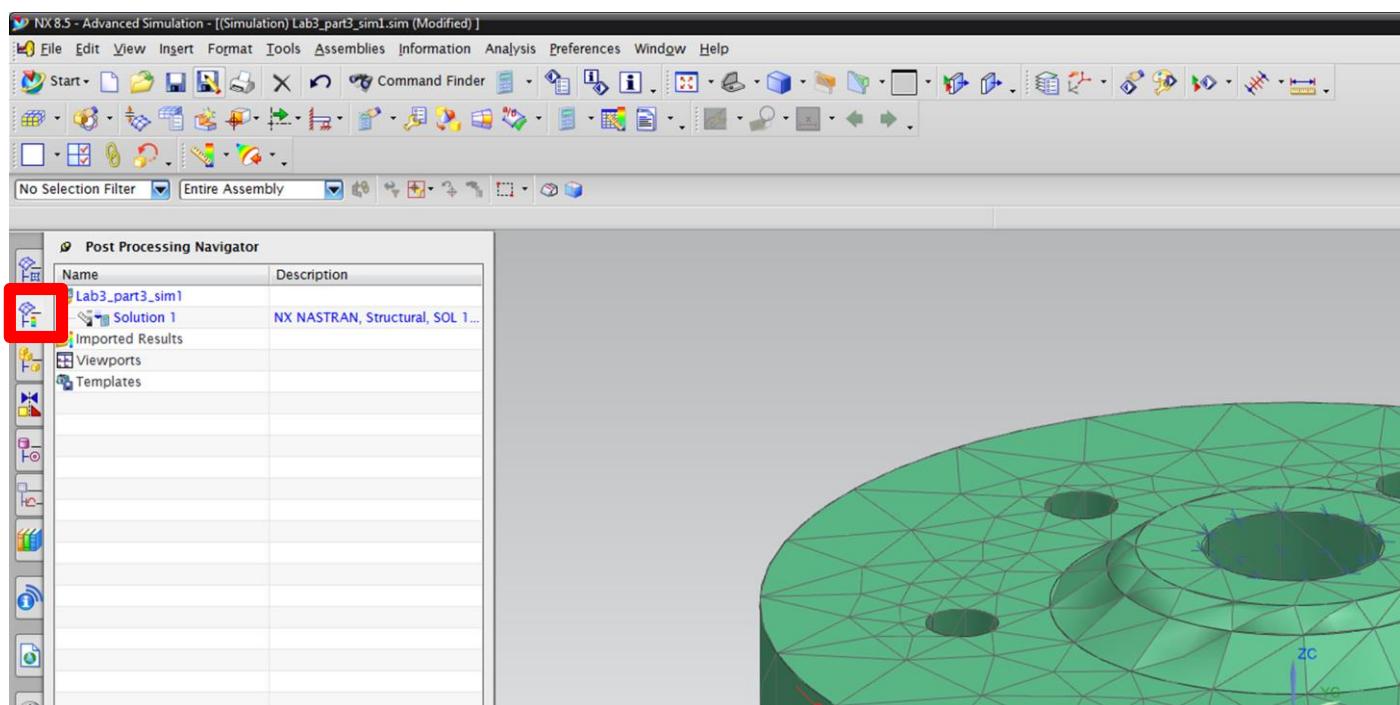
Step 20

You will most likely be bombarded by pop-up windows after solving. Be patient, and leave these windows alone until you see ‘Job Finished’ at the bottom of the ‘Solution Monitor’ dialogue. Then, feel free to close all.



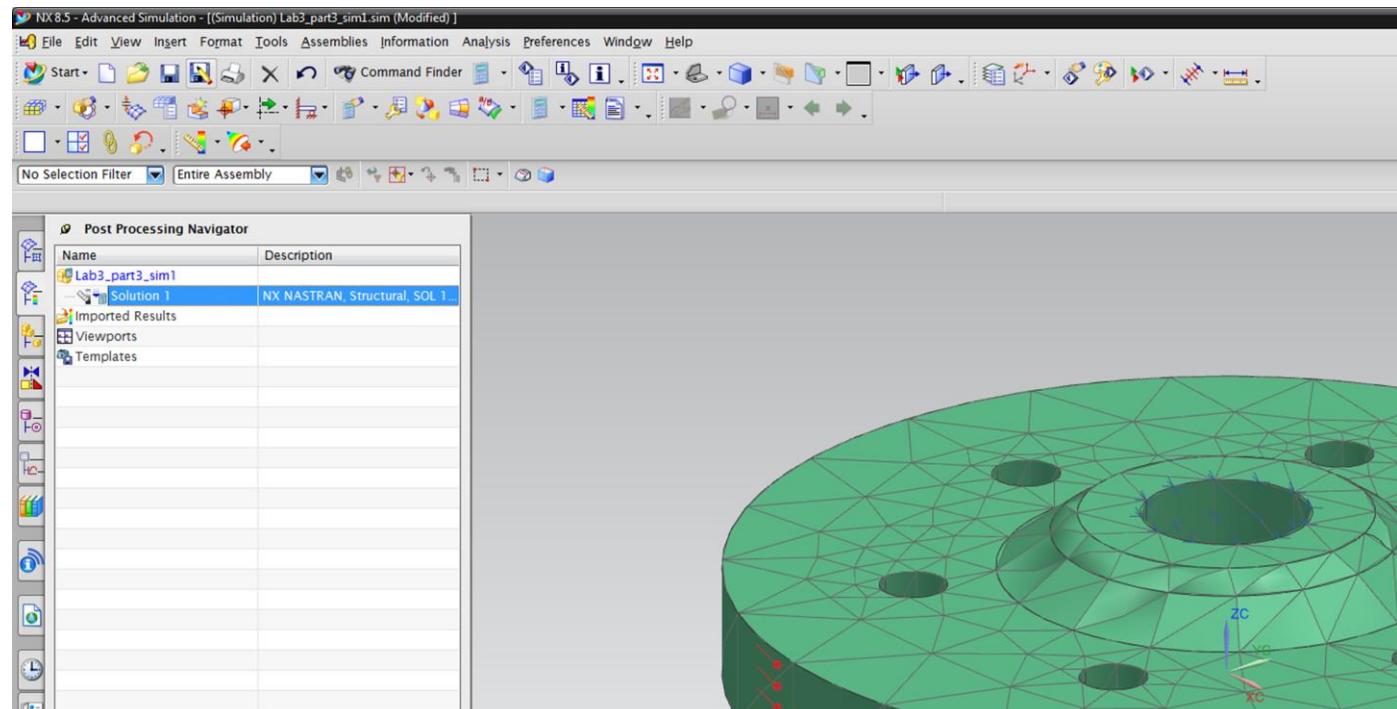
Step 21

Once the Nastran solver has completed calculating the parameters you specified move the cursor over to the tabs located on the left side of the screen and open the second tab from the top menu called Post-Processing Navigator.



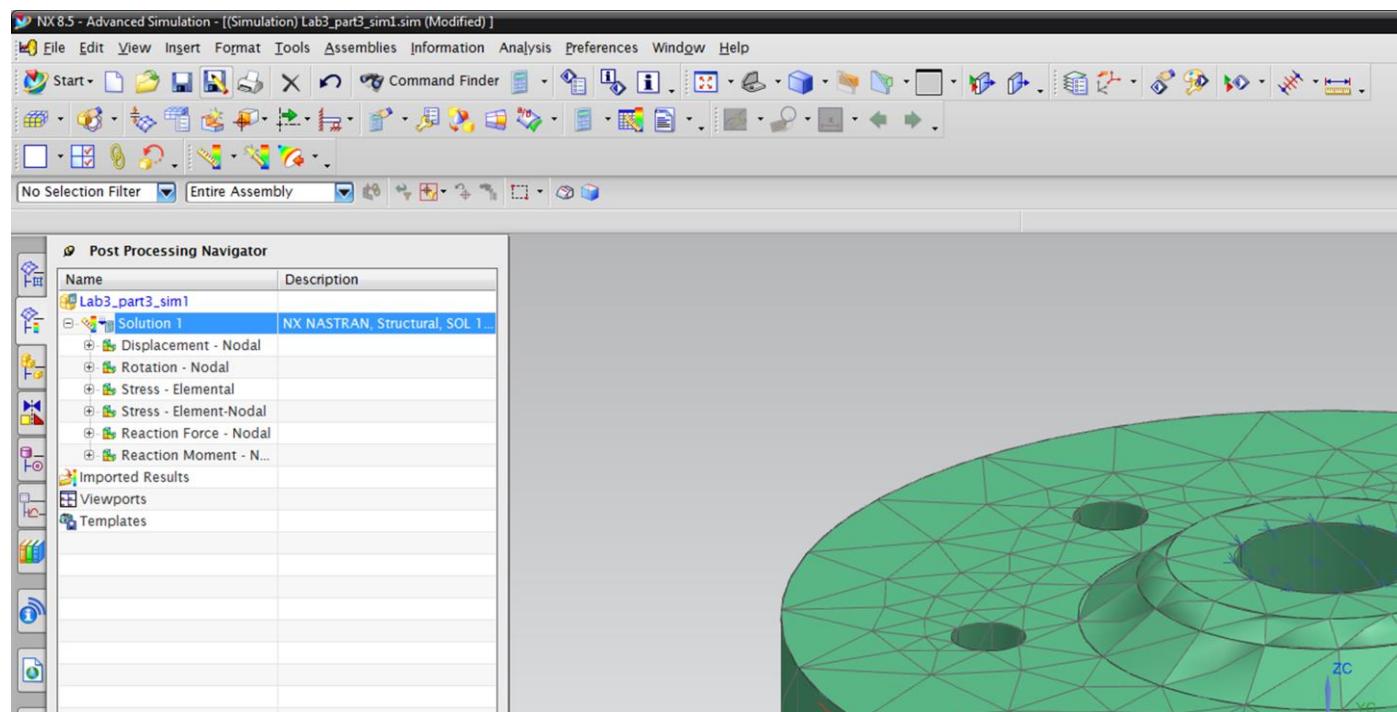
Step 22

Double click on ‘Solution 1’



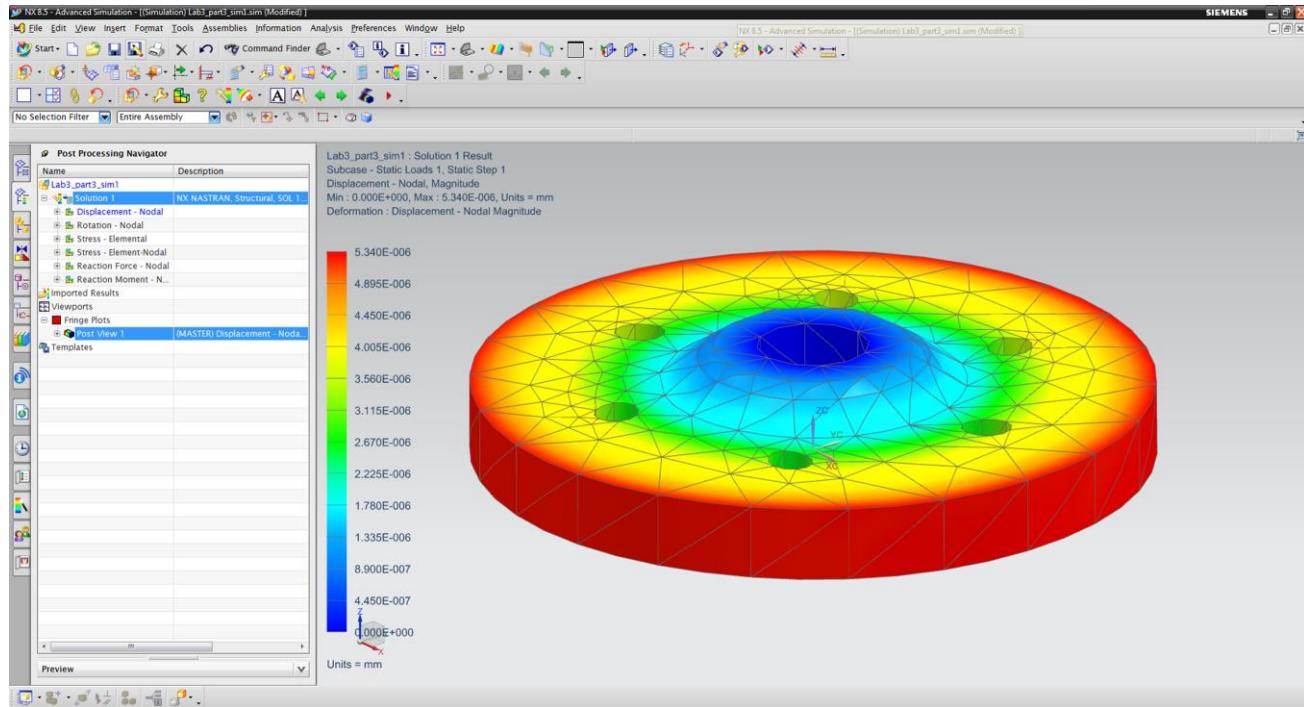
Step 23

After double-clicking the ‘Solution 1’ line use the plus sign to expand the menu and view all the different stress and displacement analysis



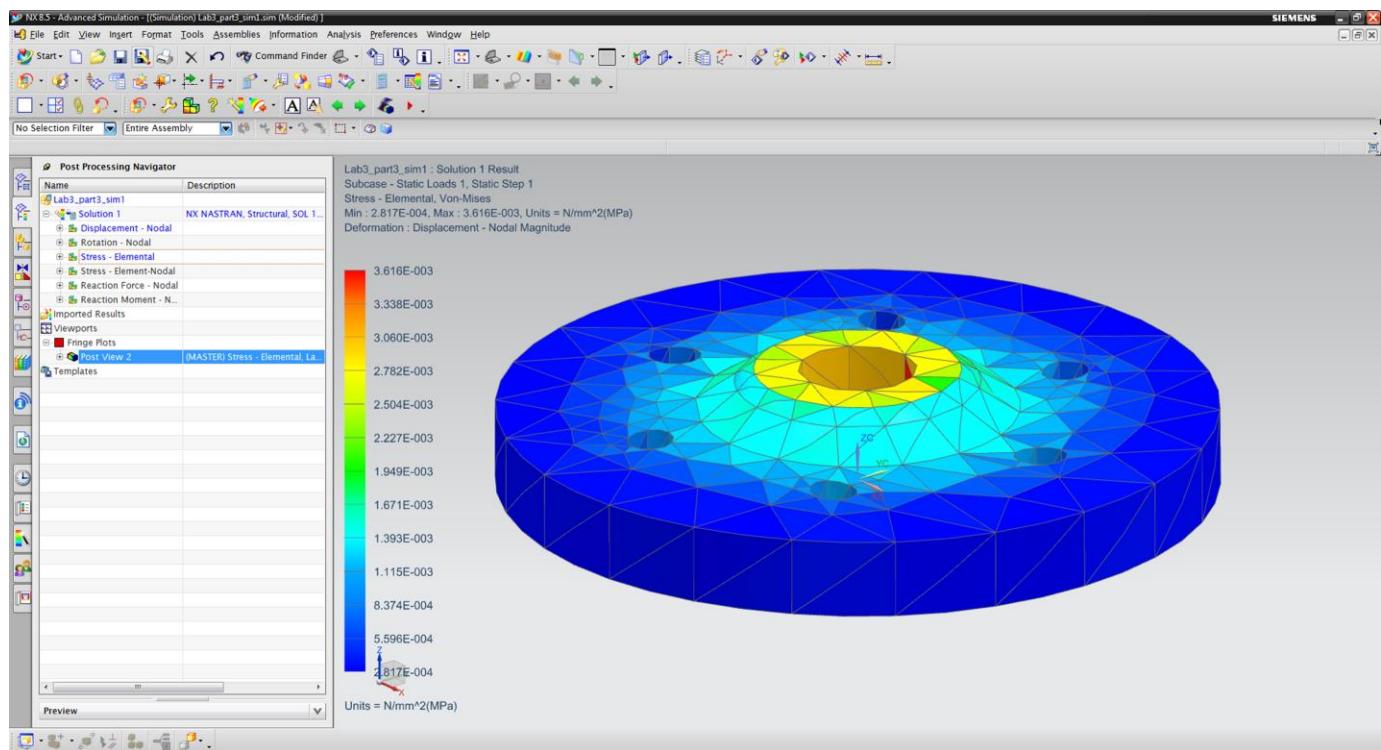
Step 24

Right click on the first analysis named ‘Displacement’ and select ‘Plot’ from the list. The plot displays the amount of displacement the disk component receives given the initial parameters. The bar on the left side of the work area shows the degree of displacement that the colors represent on the disk.



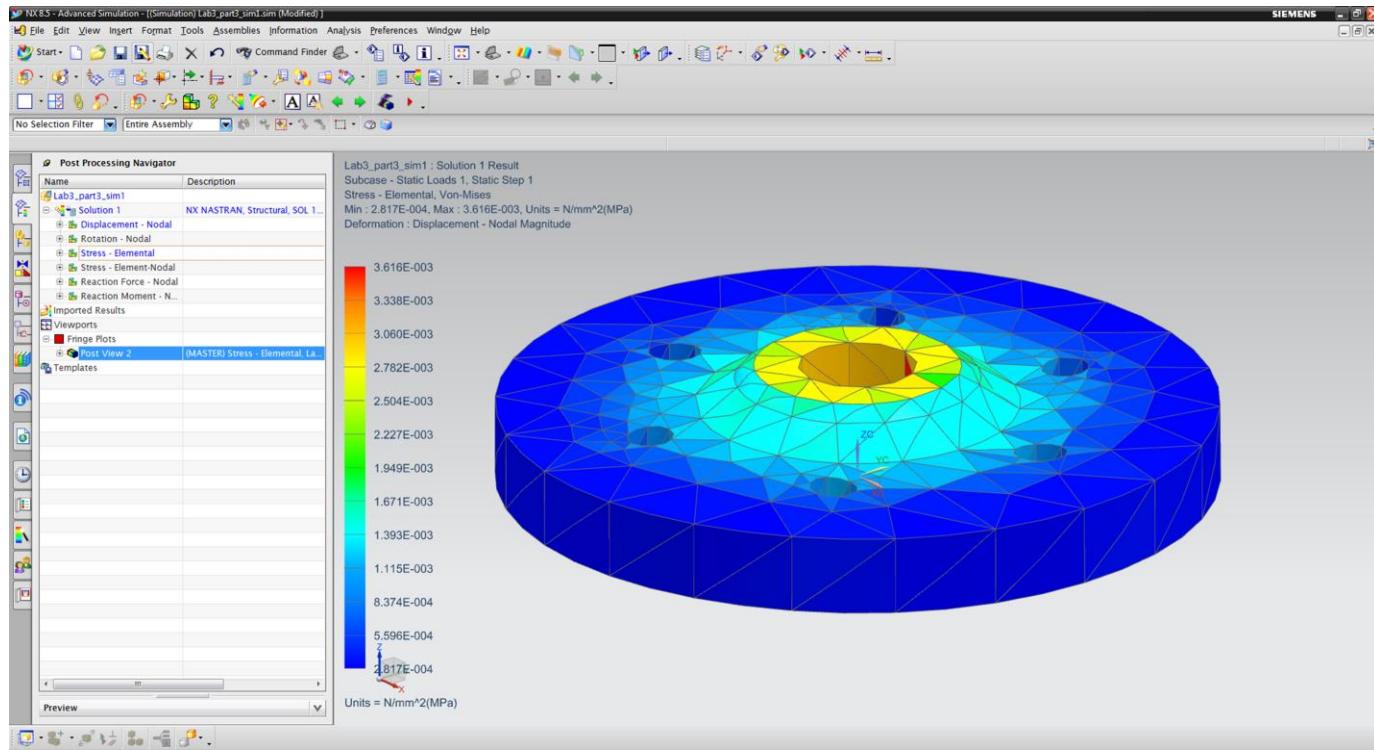
Step 25

Repeat step 24 and open the ‘Stress-Element’ plot



Step 25

Repeat step 24 and open the ‘Stress-Element’ plot



Step 26

We can now repeat this process for a different mesh size, material, load component and constraint. Click on the ‘Simulation Navigator’ tab. Right click the mouse on your Disk part and select the ‘Make Displayed Part’ from the list. Then right click on "YOURPART.part" in your simulation navigator and select "New FEM and Simulation". In the New FEM and Simulation menu that appears, Click ‘OK’. Click ‘OK’ in the Create Solution menu that follows the New FEM and Simulation menu.

Step 27

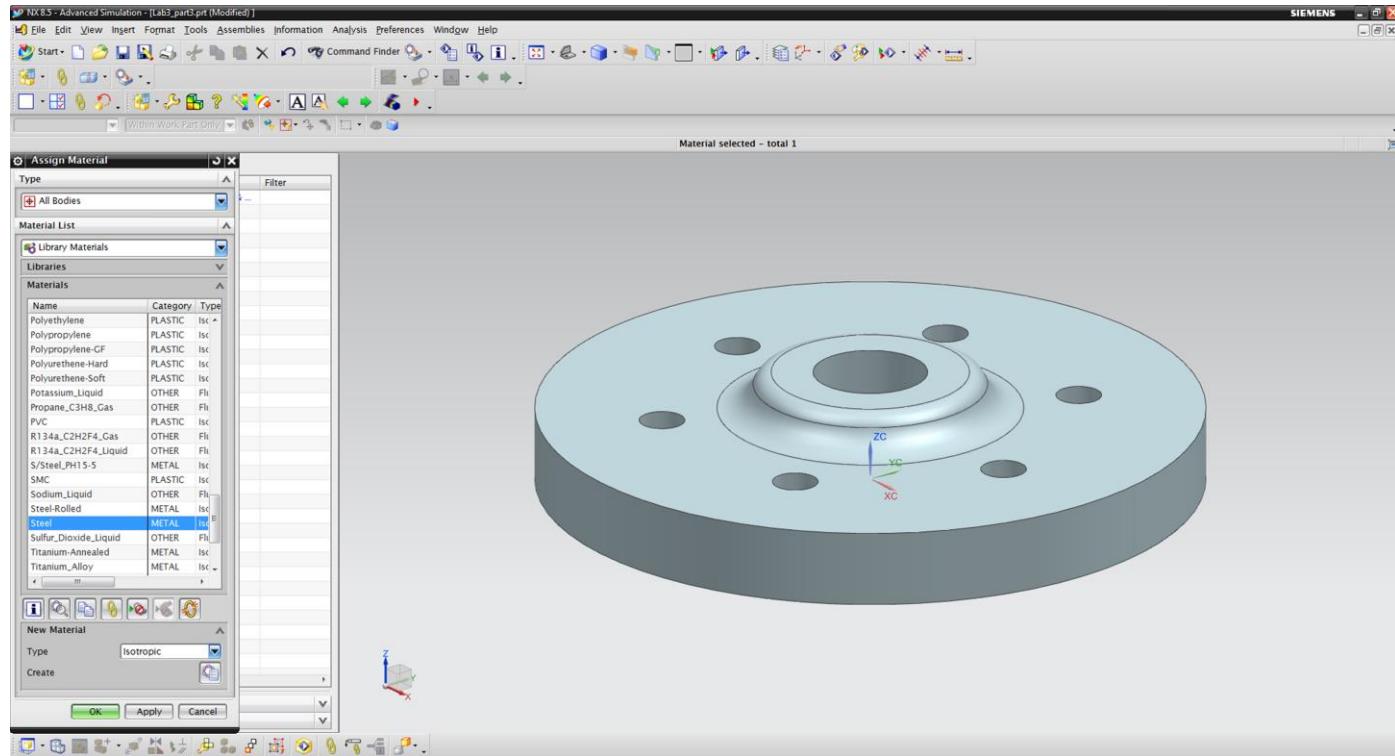
Using the ‘Simulation Navigator’ tab on the left side of the work area as seen in step 12, right click on the heading ‘YOURPART_fem1_i.prt’ and select ‘Make Displayed Part’ from the list.

Step 28

Click on the ‘Material Properties’ icon. If you select the drop down menu on materials a full list of available materials will appear. Note: you will have to select the "aluminum_6061" material and select "remove selected material assignments" in order to assign your part a new material.

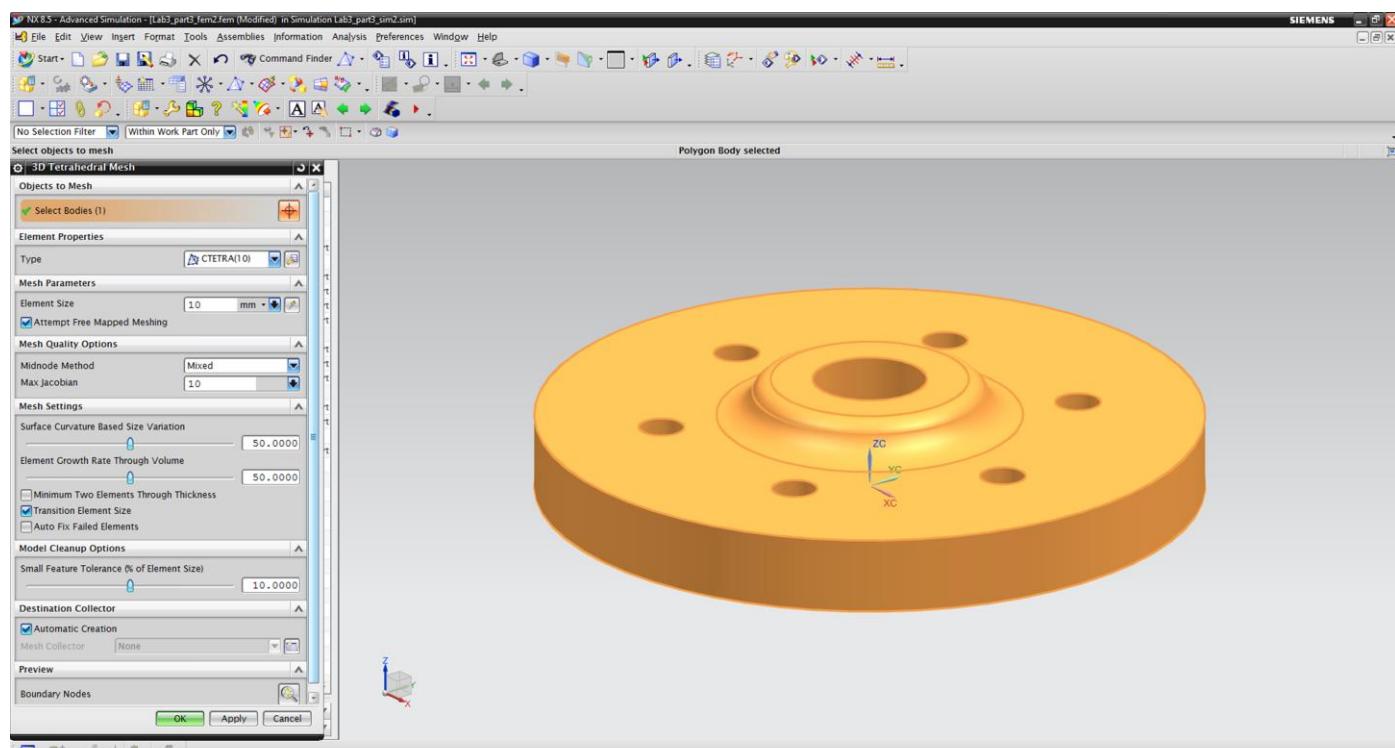
Step 29

Highlight the 'Steel' material under the Materials section, choose the disk in the work area and Click 'OK'. Then double click "YOURPART_fem2" from your simulation file view.



Step 30

Click on the '3D Tetrahedral' icon located in the task bar and select the disk as the body you wish to apply the mesh to. Change the Element Size to 10 and click 'OK'



Step 31

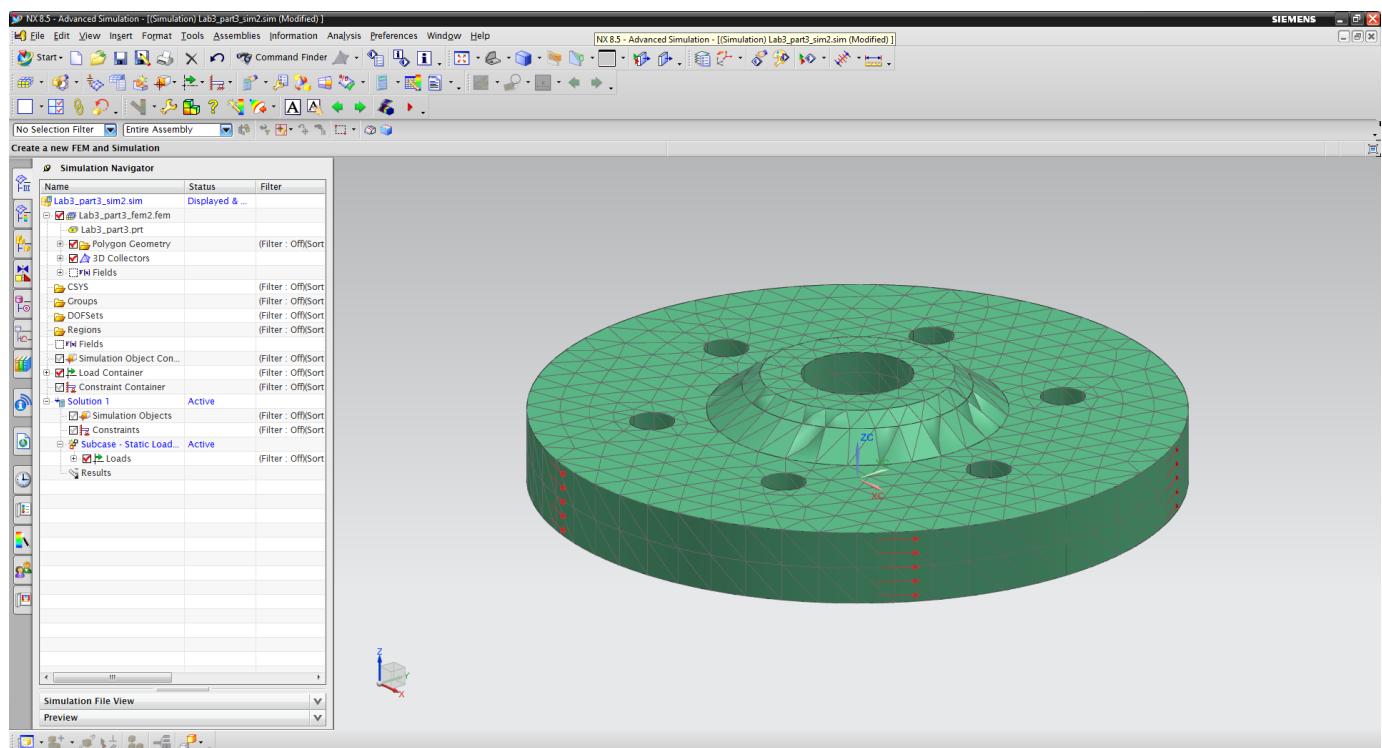
Use the ‘Simulation view file’ to select the Simulation mode. Click on the drop down arrow and Right click on ‘YOURPART_sim2’ to select ‘Make Work Part’ from the list.

Step 32

Click on the ‘Load Type’ and select the ‘Torque’ button.

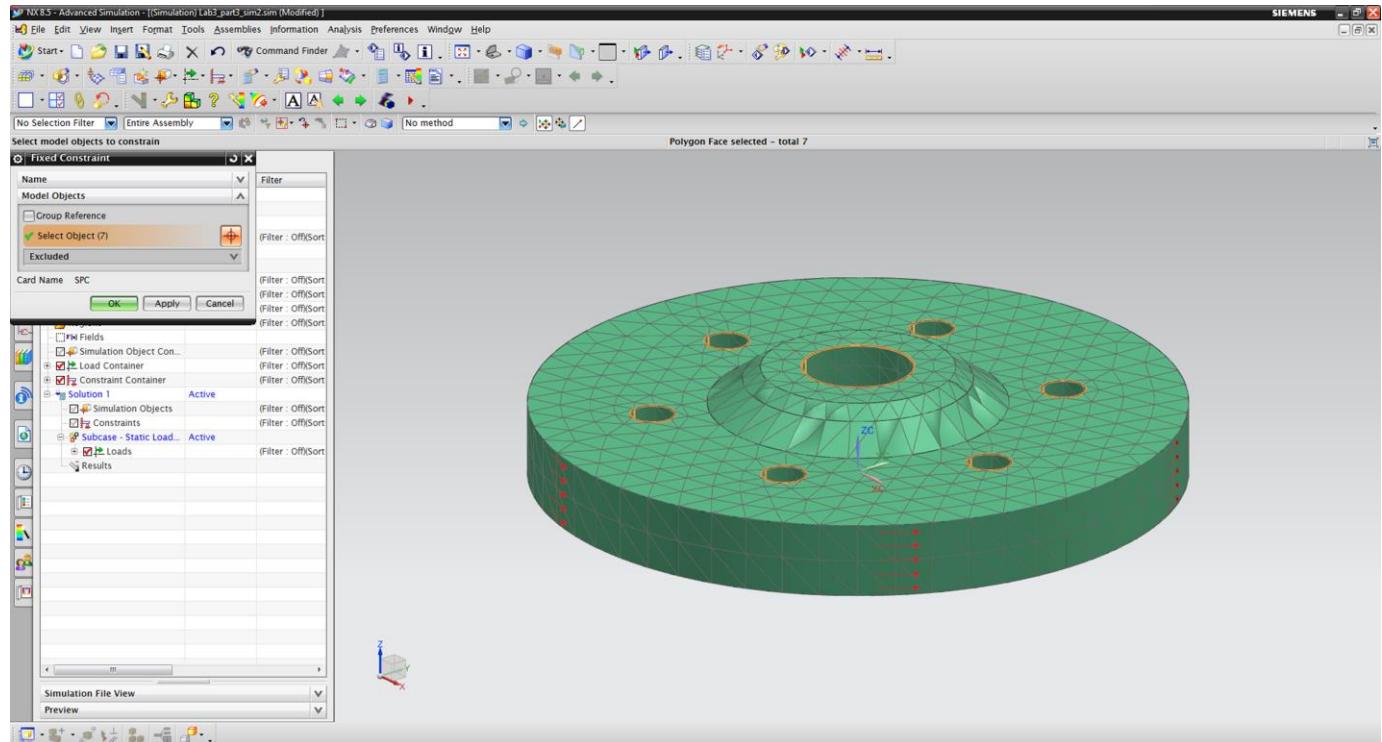
Step 33

In the Torque menu under the Magnitude section enter a testing torque value of 200 N-m. Use the drop down menu to the right of the Torque value to select the units you wish to use. Pick the outside circumference and click ‘OK’.



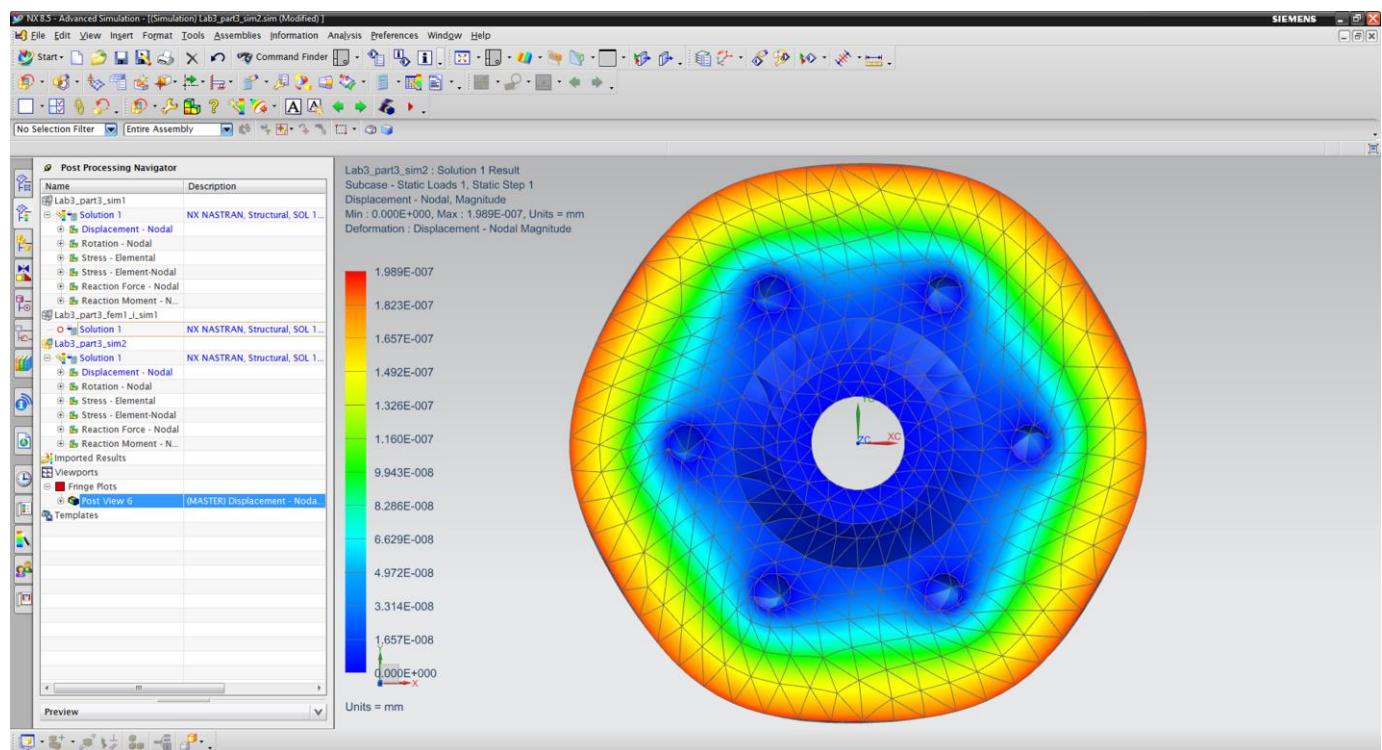
Step 34

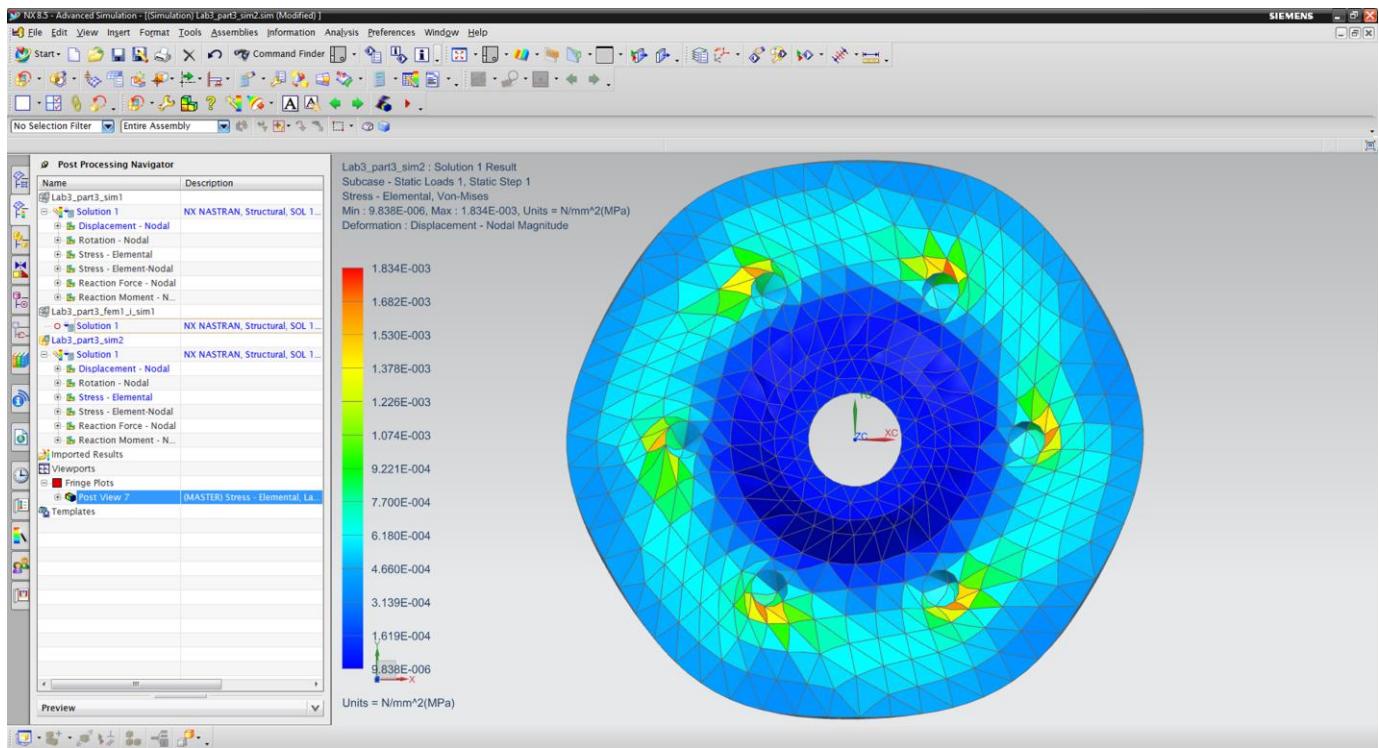
Click on the ‘Fixed Constraint’ icon and select the center hole in the middle of the disk as well as the 6 holes about the perimeter of the disk shown. Click ‘OK’.



Step 35

Performing the same viewing techniques as done before, view the Displacement and stress element plots. Presented in figures below





Practice adjusting these values and constraints to see what type of output plots you receive. Test another part from the assembly.

End of Module Four