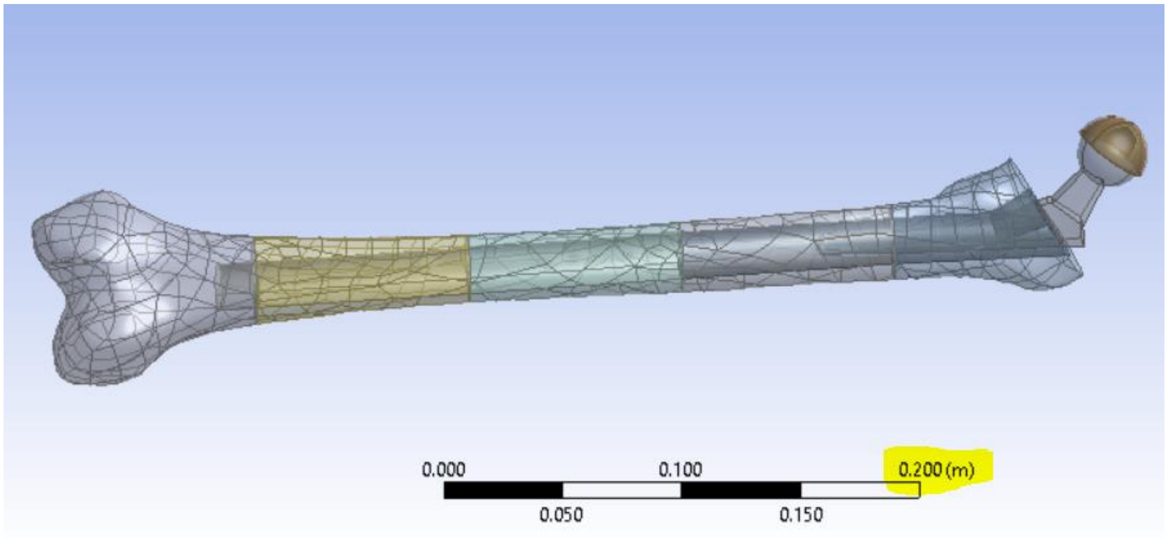
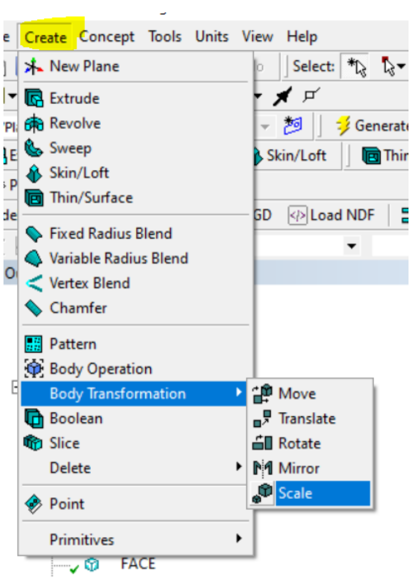
Femur and Implant with Acetabular Cup Contact Analysis

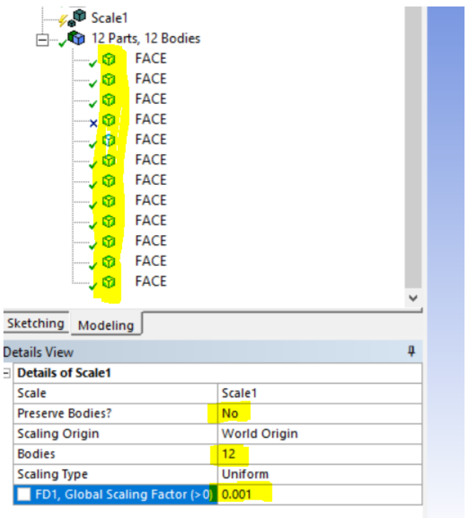
Import the model, verify the scale. The scale should show something less than 1 m.



If the scale is not correct, you will have to change it. The program will read 200 as 200 meters. Changing units will not change the scale of the model, it will only change the units as they are displayed and used for calculating lengths and loads. If you run the analysis without changing the scale it should still be correct, but you will have to scale the results accordingly by 1000. To avoid confusion or misinterpreting the results, it is best to rescale the model to the desired scale. Scaling mismatch with imported models is very common, especially when they are generated from scans such as our femur model. Fortunately, it is very easy to change in Design Modeler. From the top menu bar select Create > Body Transformation > Scale. You can change the scale in Design Modeler and then update the Mechanical model.



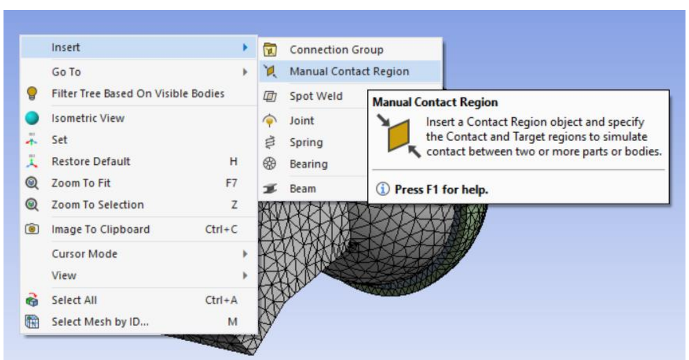
Select all the bodies from the Outline tree using the shift key. In the Details view, select Apply button next to Bodies in the right column. Since the imported model had a scale of 10^2, change the scaling factor to 10^{-3}. This will result in a scale of 10^{-1}. The total height of the femur was about 467 mm. If not scaled, the length would be interpreted as 467 meters. The new scale will be 0.467m.



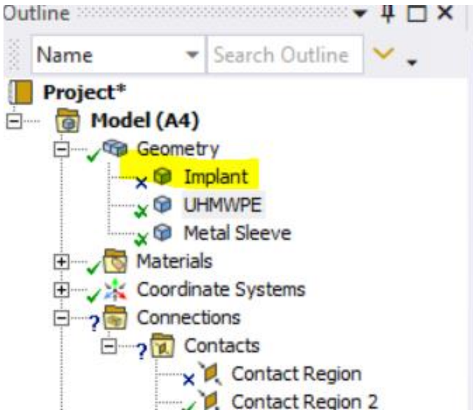
Start Mechanical. Generate the mesh and set up the contact.

Set up the contact between the ball and UHMWPE.

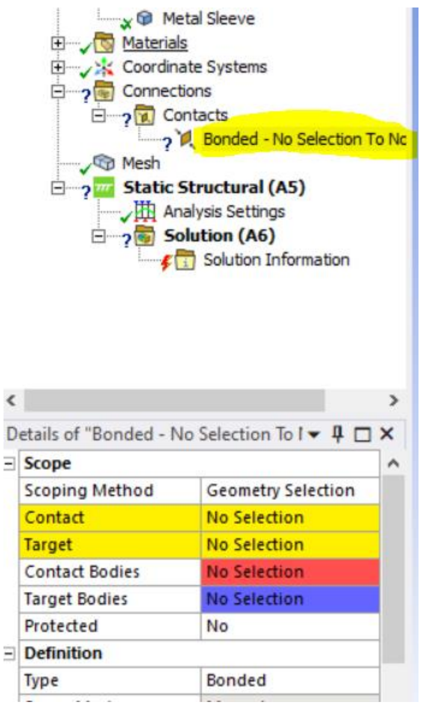
Right click anywhere in the window.



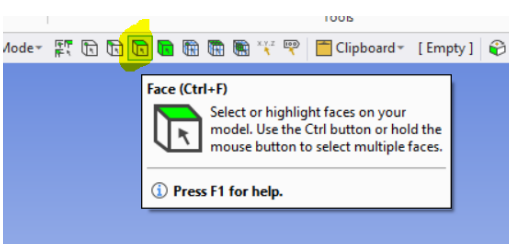
In Geometry, suppress the implant.



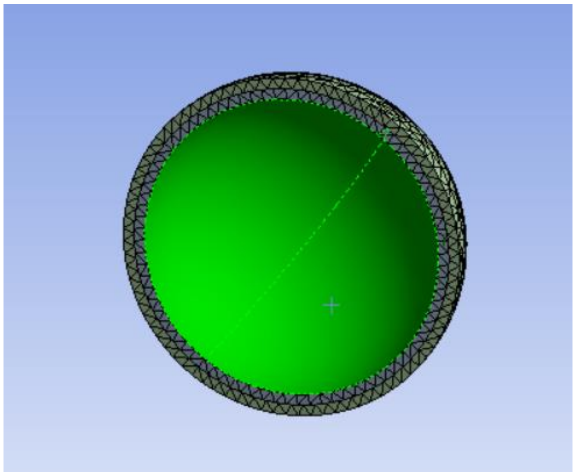
Set the contact region and type. Select Contacts from the outline (yours may look slightly different)



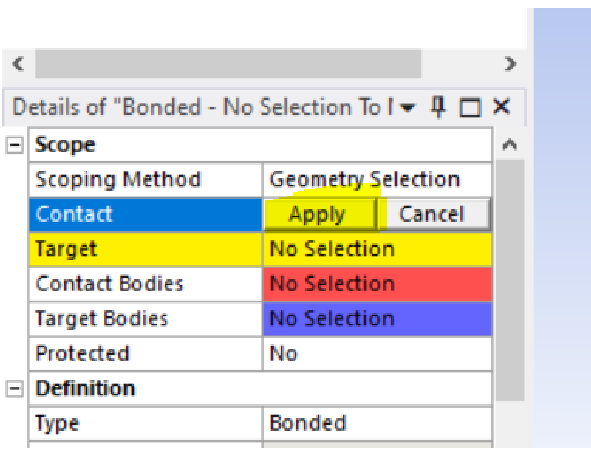
Set the selection type to Face



Select the inner surface of the cup. Select one face, then hold the control key to select the other face.

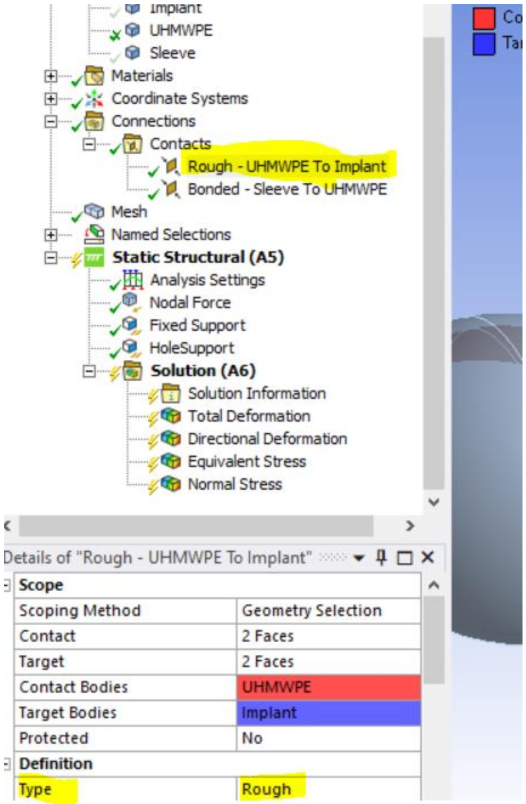


In Details select Contact and Apply

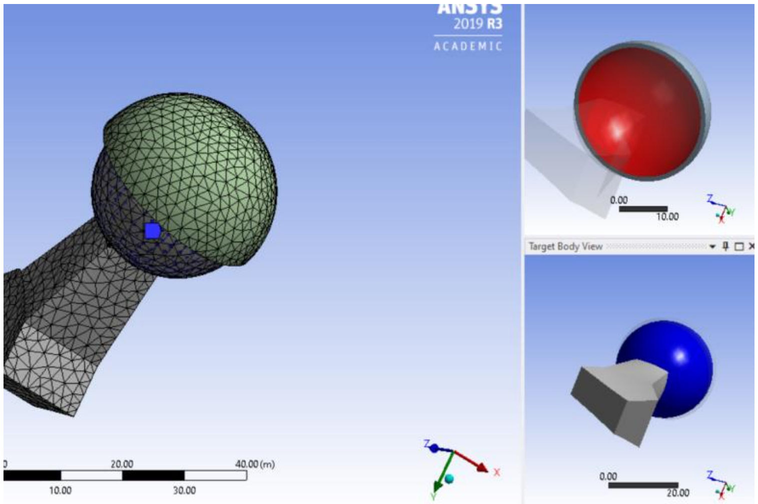


Now unsuppress the implant, and suppress the cup. Select the ball face of the implant, Select Target and Apply.

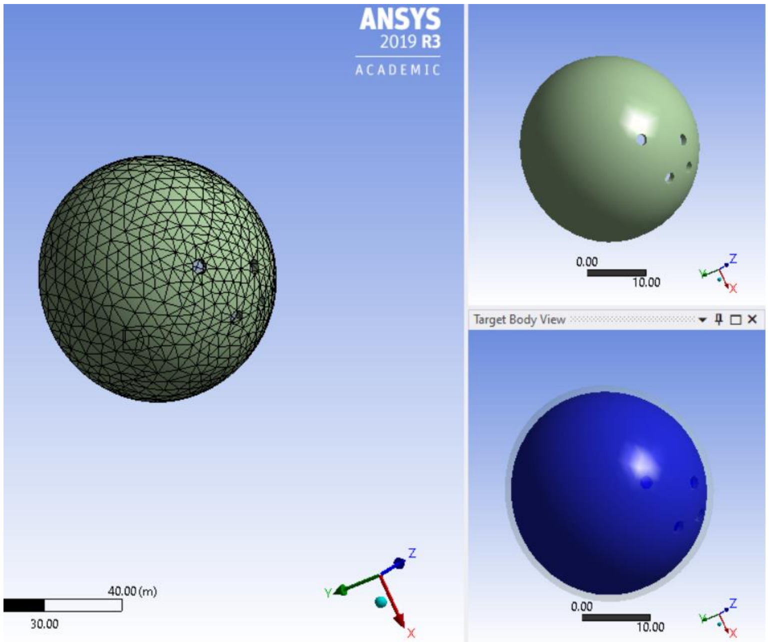
Change the definition to Frictionless. Note that in Scope the contact and target bodies have been set.



Selecting the contact in the tree outline will show the contacts in the display window. Verify that the contacts are properly selected.

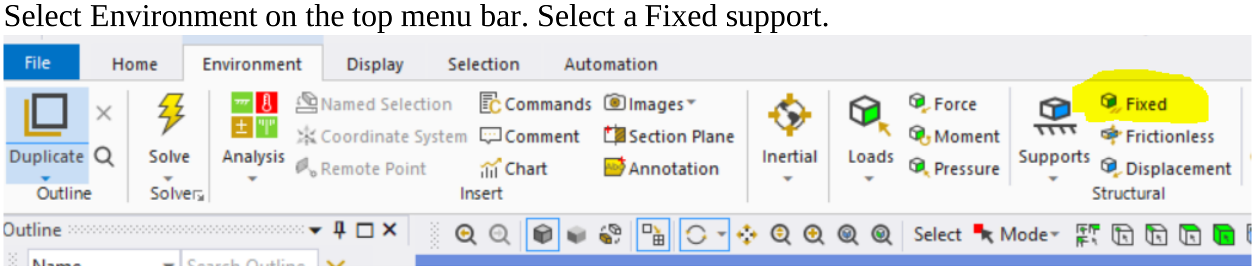


Repeat the above steps to set up an additional contact between the UHMWPE and metal sleeve. Make sure to pick the upper part of the UHMWPE this time and the inside of the metal sleeve. This time set the contact type to bonded. This contact can also be set to frictional contact. The contact pair is shown in the figure below.

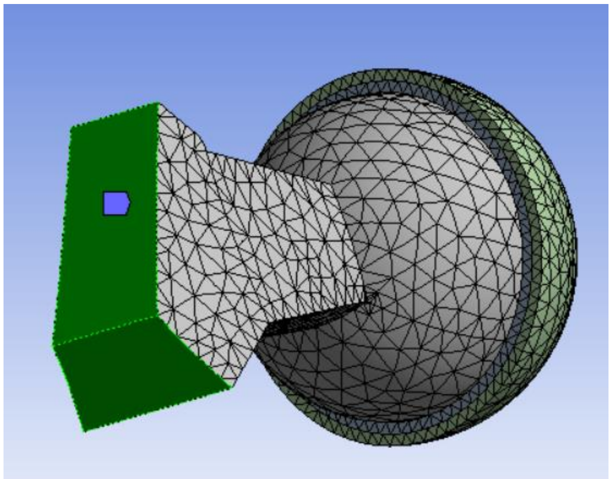


Unsuppress all bodies and proceed to set up the support on the implant and apply the load at the top of the acetabular cup sleeve.

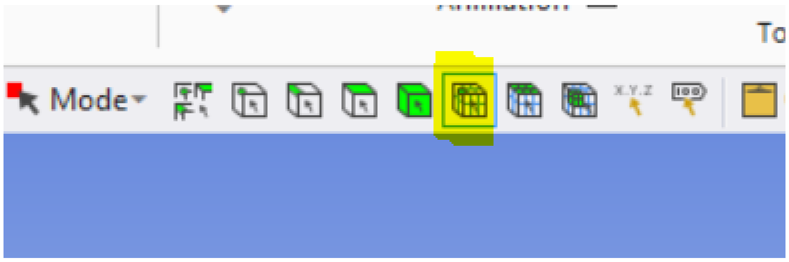
Note: this step is for demonstrating how to apply a fixed support, you are working on the full femur, so you can disregard this step.
Select Environment on the top menu bar. Select a Fixed support.



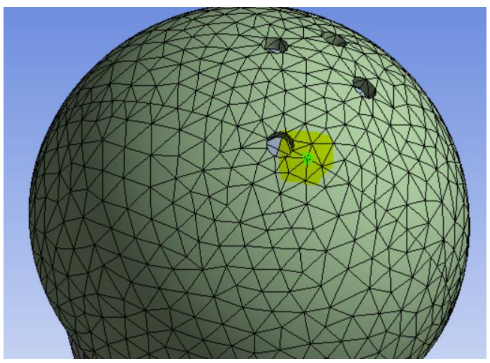
Select the 2 faces of the implant and apply the geometry in the details view



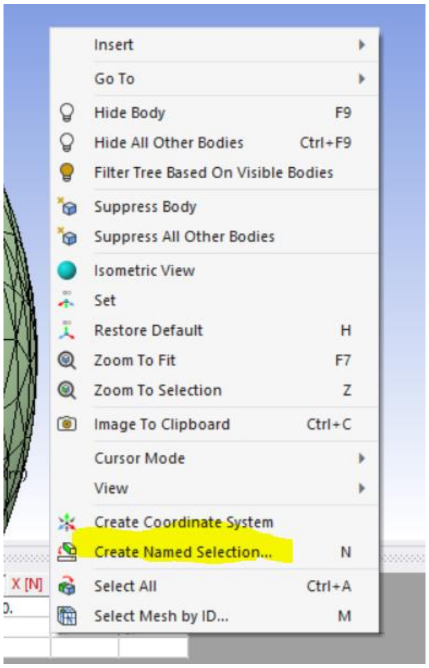
Change the select mode to node.

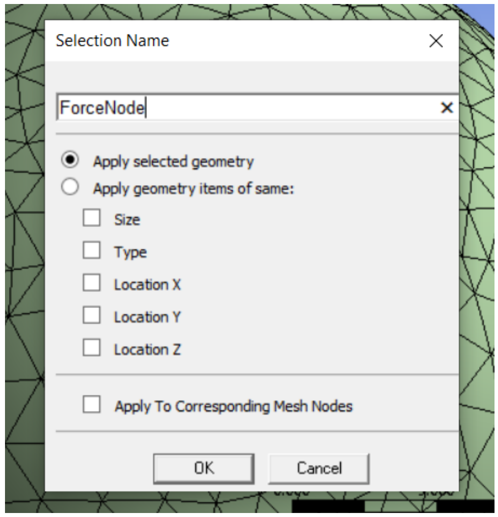


Select a node near the top center of the cup near the center hole.

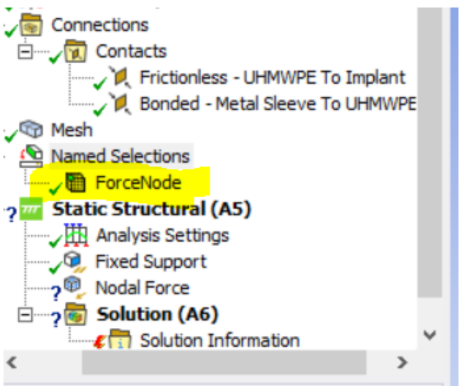


After selecting the node, right click anywhere in the geometry window. Select the "create named selection" option. Set a name for the selection.

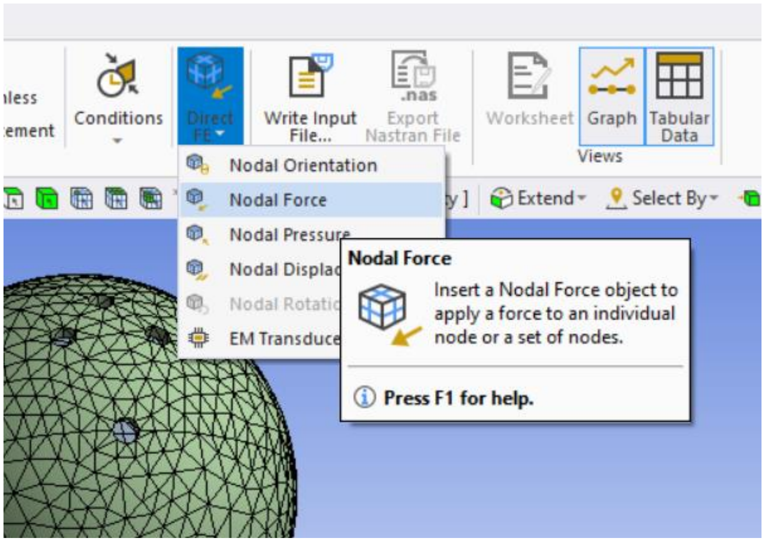




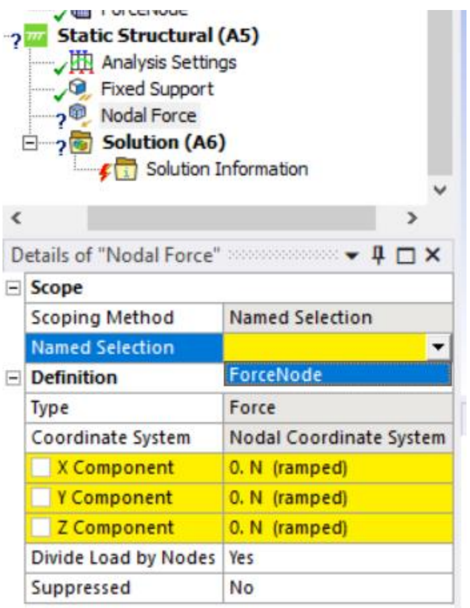
Verify in the outline that the selection has been created.



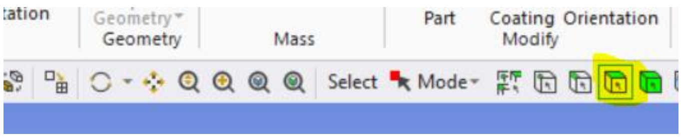
In the environment tab, select Direct FE option and Nodal force

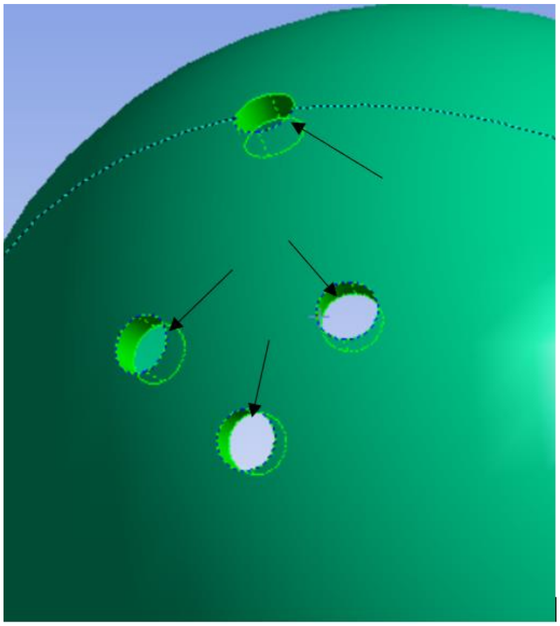


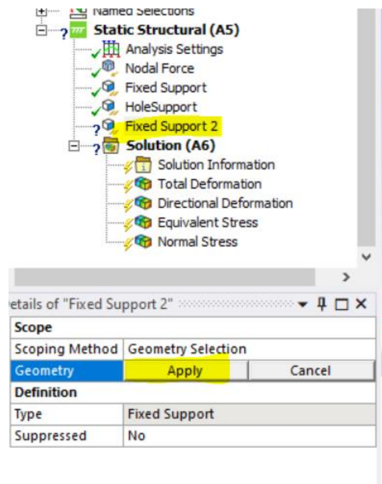
Select Nodal Force from the Outline. In Named Selections select ForceNode. Set the appropriate x,y,z components of force.



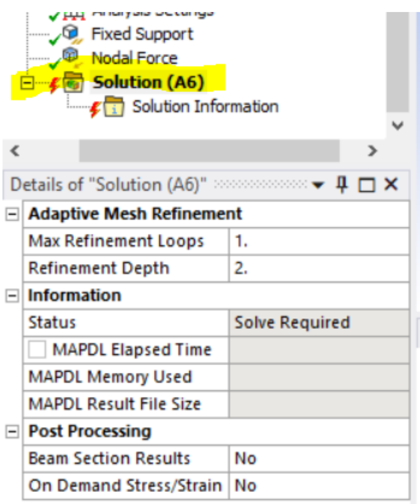
NOTE: To run the analysis with a frictionless support between the femoral head (implant) and the UHMWPE, create a fixed support at the screw holes of the sleeve. Select each of the holes, then select apply for the fixed support in details.



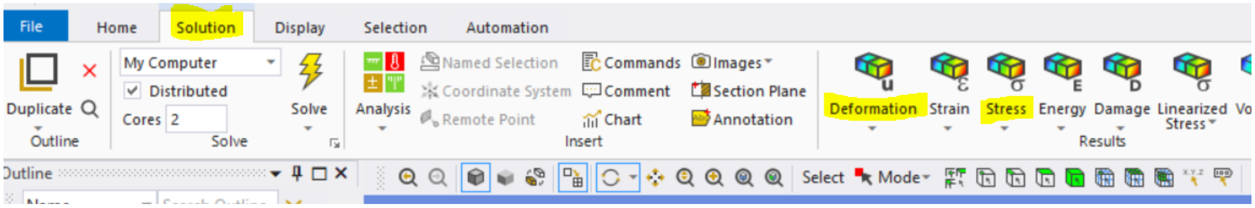




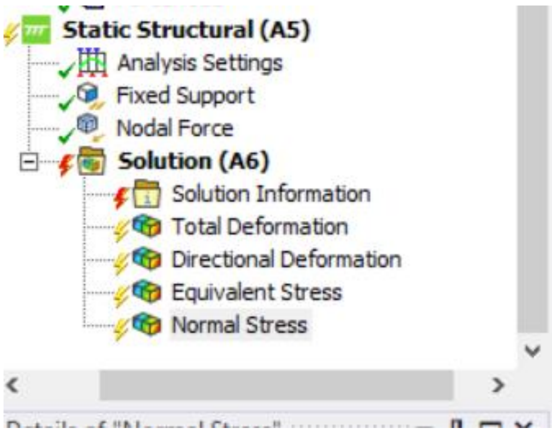
Select Solution from the outline



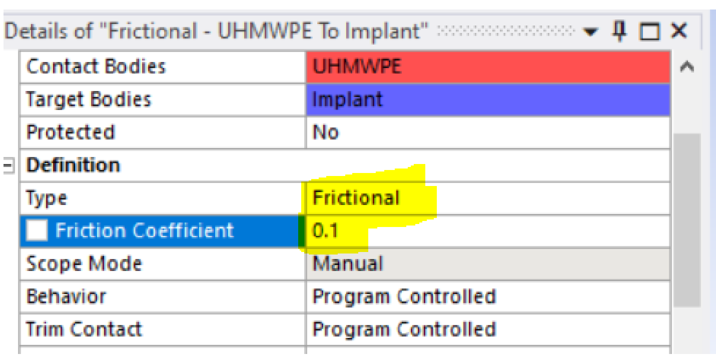
Select the Solution tab at the top menu bar. Select Deformation: Total and Directional. Select Stress: VonMises and Normal

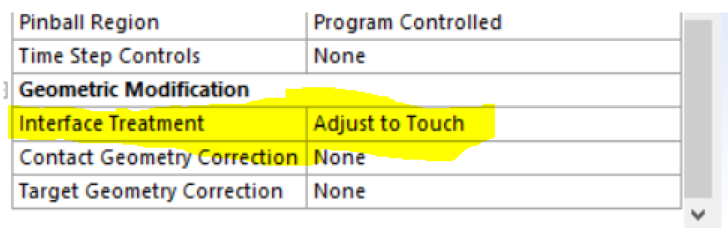


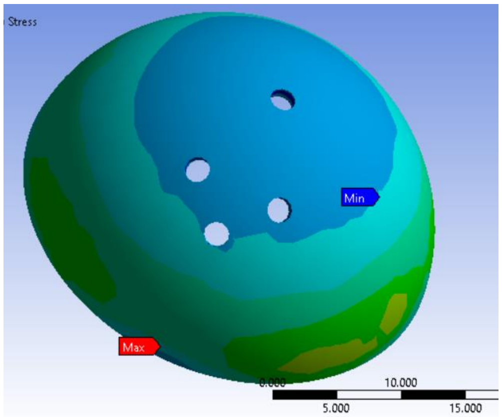
The selected solutions should appear on the tree

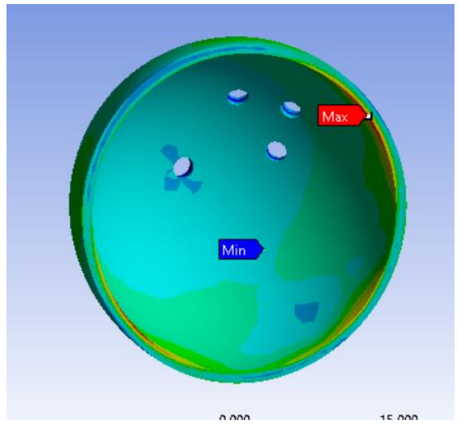


Save your work and then solve.





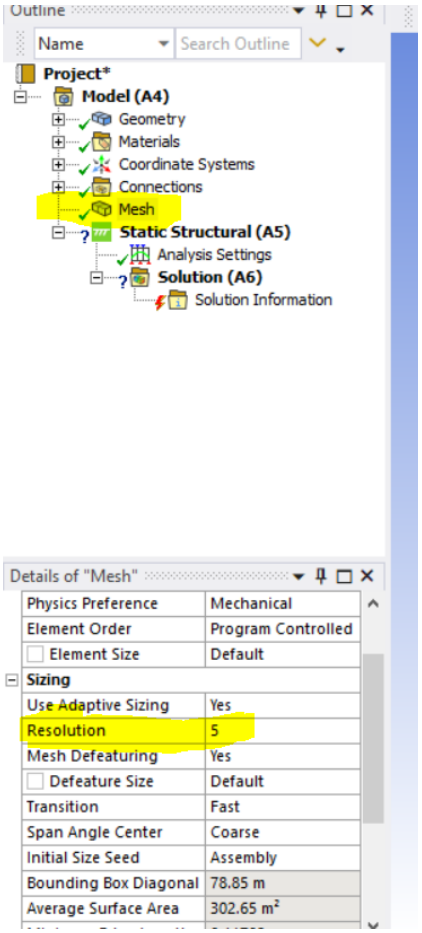


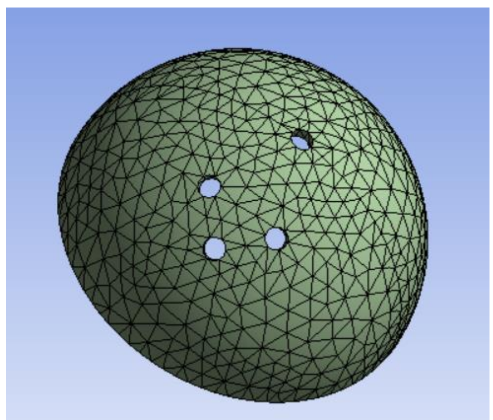


Mesh Refinement

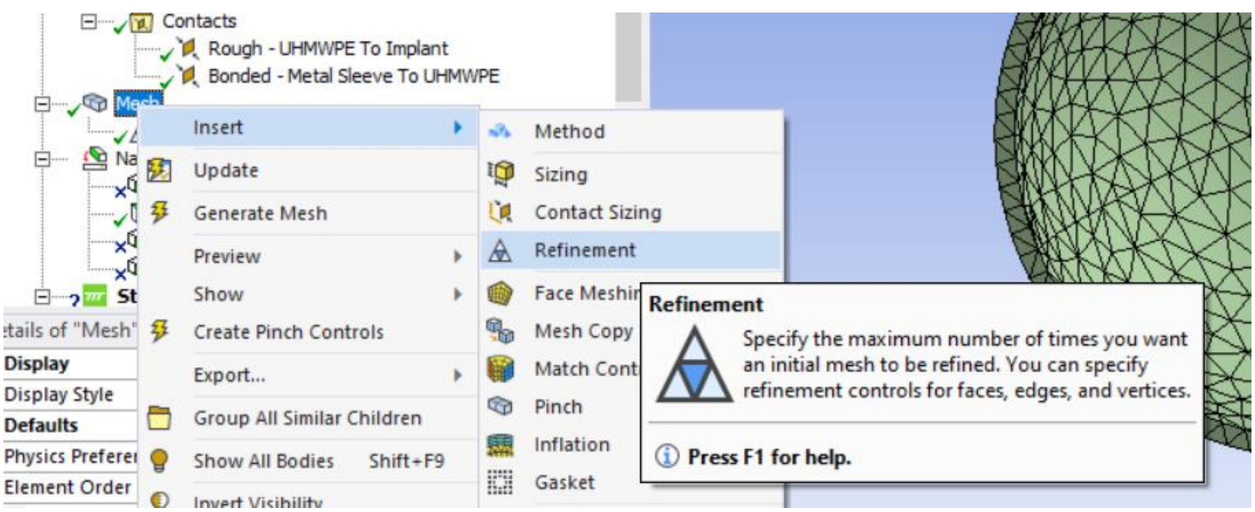
When setting up a conact analysis it is important that the mesh be sufficiently fine in the region where contact is expected. The accuracy of the solution is directly dependent on the quality of the mesh, which is a function of the number of elements in that region. A full discussion of meshing and finite element analysis is beyond the scope of this class. However, the steps below will demonstrate how to refine the mesh.

Start by setting the resolution. Select Mesh from the Outline. In Details of Mesh, expand Sizing. Increase the resolution from the default of 2 to 5 or higher. The maximum resolution is 7. Regenerate the mesh.

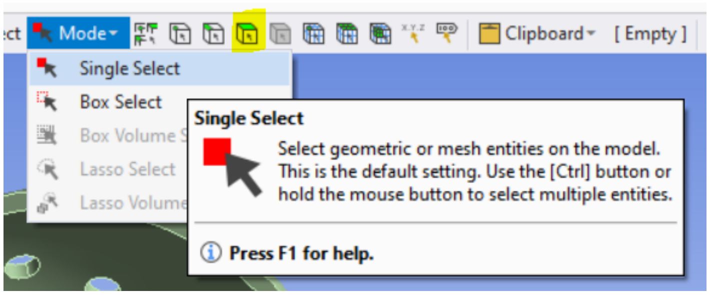


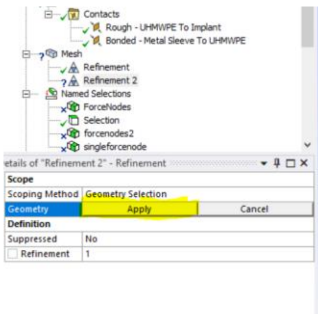


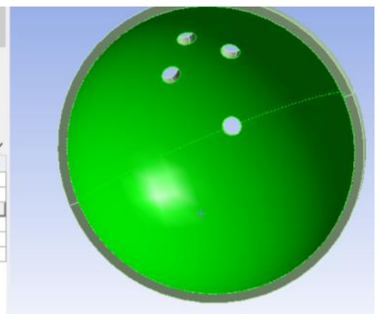
We can now further improve the mesh on the contact faces. Show only the metal sleeve. Select Mesh from the outline. Right click and insert and refinement.



Verify that SingleSelect is selected. Then select the Face Select tool. Select the inside faces of the sleeve. Hold the control key to select multiple faces. Then select Apply in details.



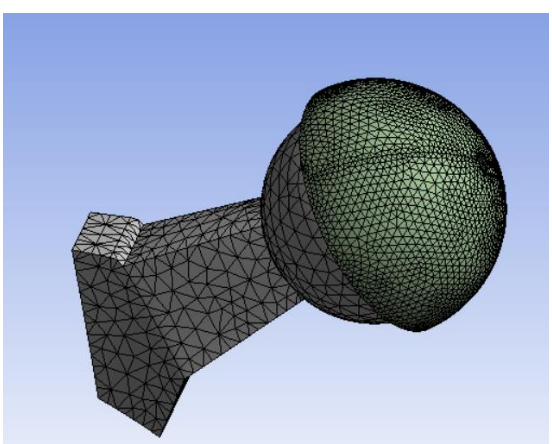


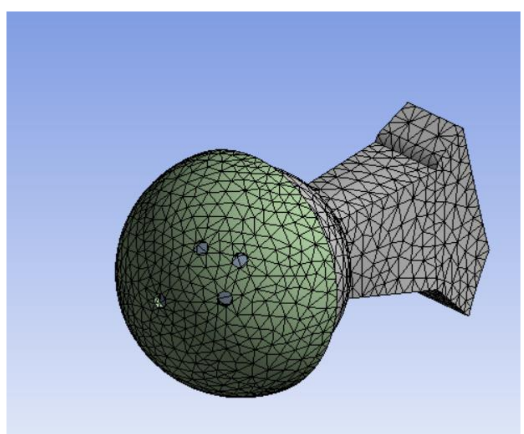


Repeat for the top face of the UHMWPE sleeve that is in contact with the metal sleeve.

Update the mesh. There should now be more elements. In Selections in the outline tree, select the named selection that was previously created and pick a new node and apply. Since there is now a new mesh the previously selected node does not exist and a new one needs to be selected. Repeat on the femoral head (sphere face) of the implant.

Note the mesh refinement has resulted in a finer mesh of the cup on the left compared to the original in the right figure below.



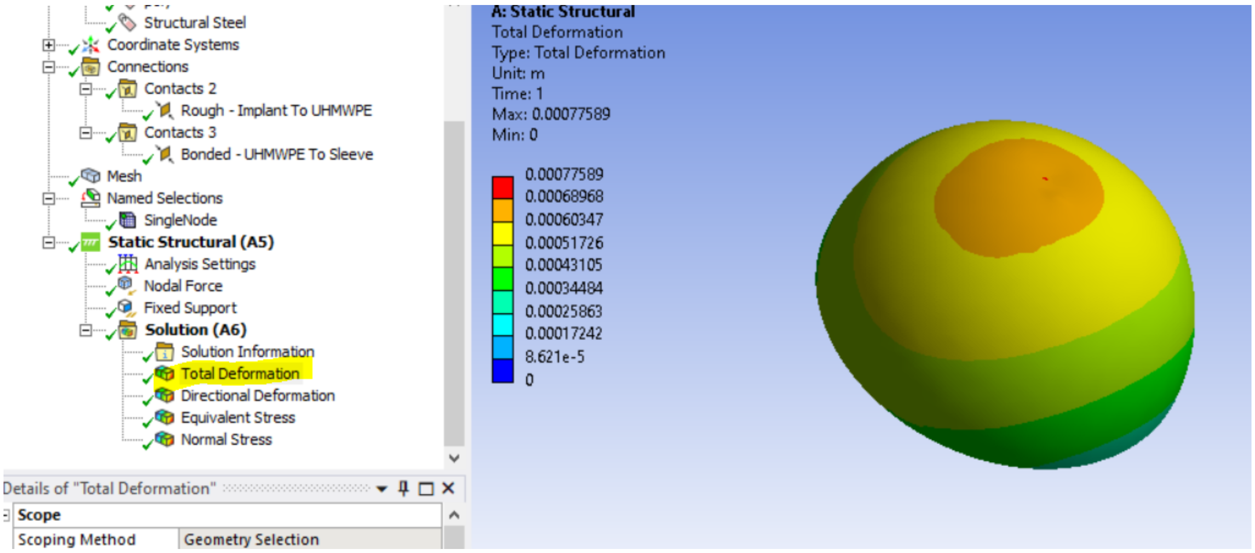


Repeat on the femoral head (sphere face) of the implant.

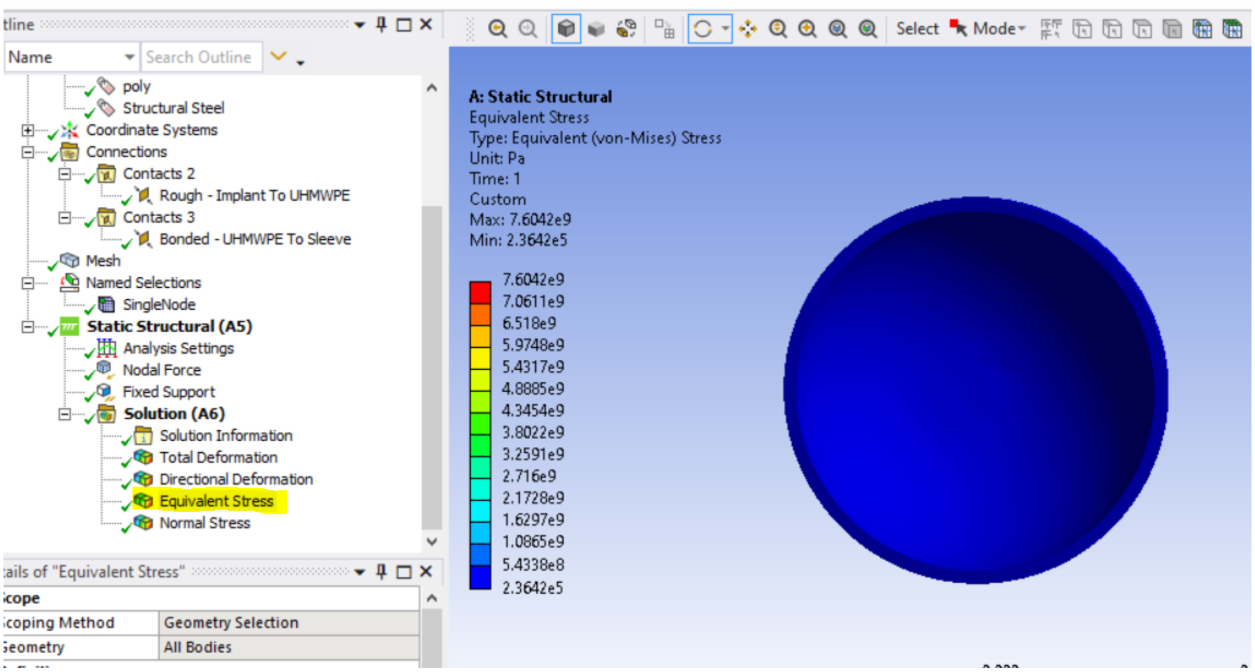
It will take some time to solve it with the refined mesh.

Displaying the Results

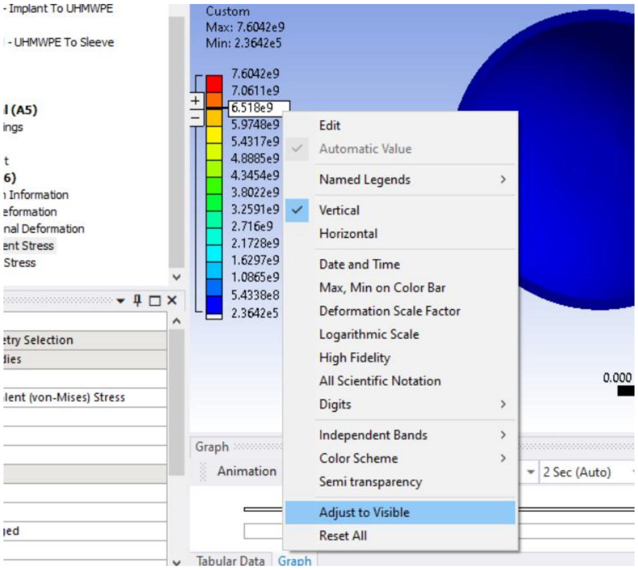
Show only the polyethylene sleeve. Plot the total deformation.



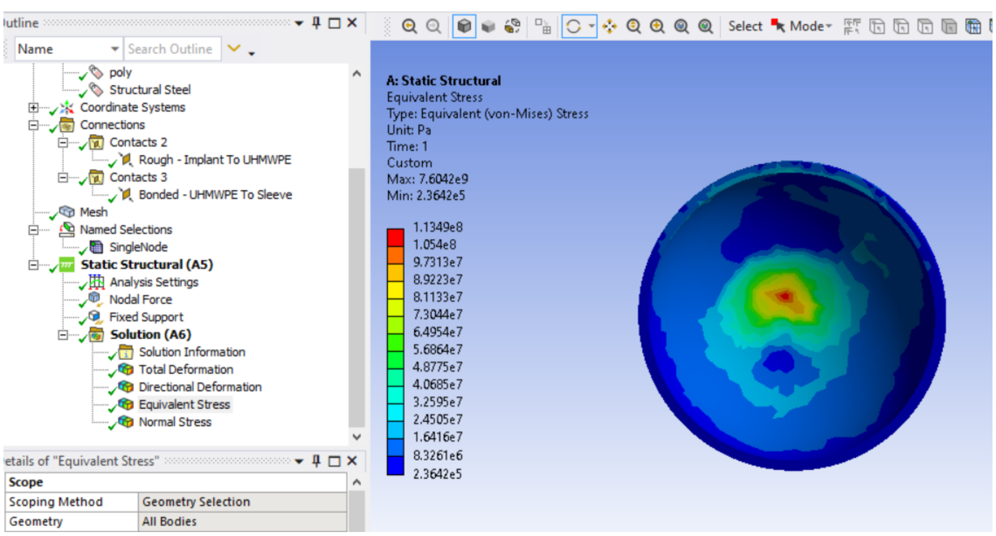
To plot the contact stress between the polyethylene and the, rotate the poly so that the inside of the cup is visible. Plot the VonMises stress. The cup is entirely blue and it would appear that there is no stress gradient. This is due to the scale.



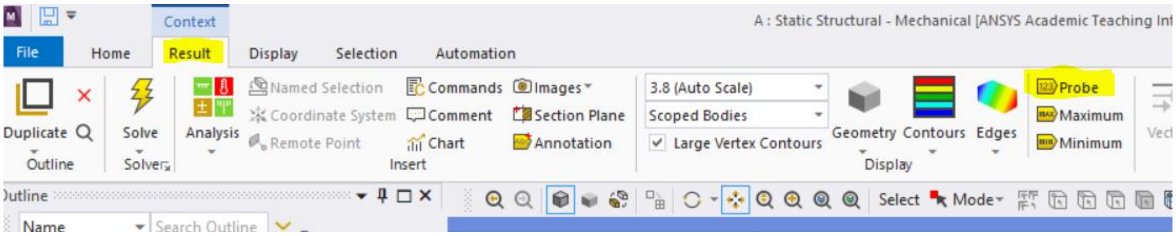
Move the mouse over the vertical contour bar. The numerical value will be replaced by "Automatic" Right mouse click on any of the values. In the menu options list that appears, select "Adjust to Visible".



The stress contours are now visible. The Maximum stress is approximately 100MPa.



Additionally, to see the value at a specific point the Probe can be used. Select "Probe" from the menu and move the probe/cursor over the desired region to display the stress.

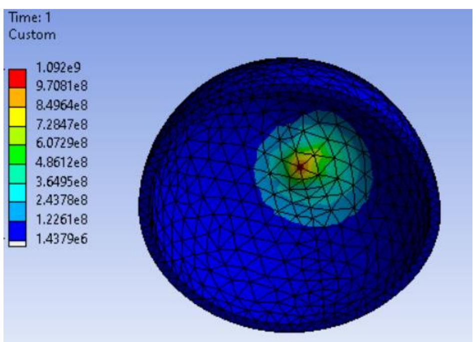


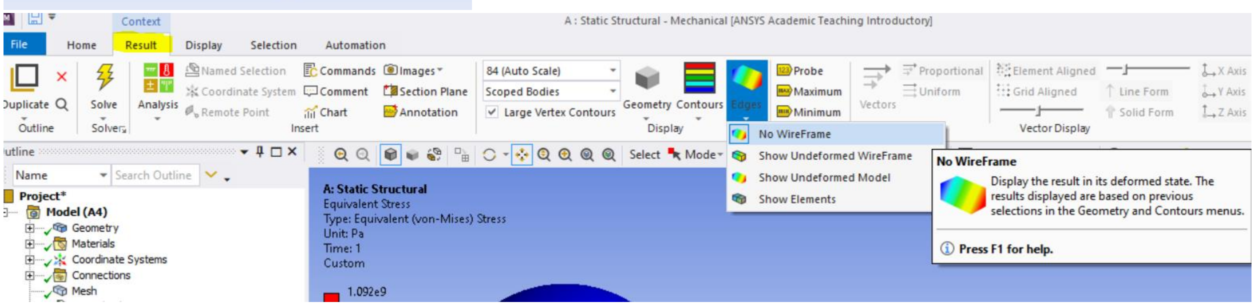
Note that the contour plot should show a smooth border. The rough, jagged borders is due to the coarse mesh. The mesh should be refined and the simulation run again. However, it may not be possible to further refine the mesh if you are using the student version, which is limited to 32,000 nodes.

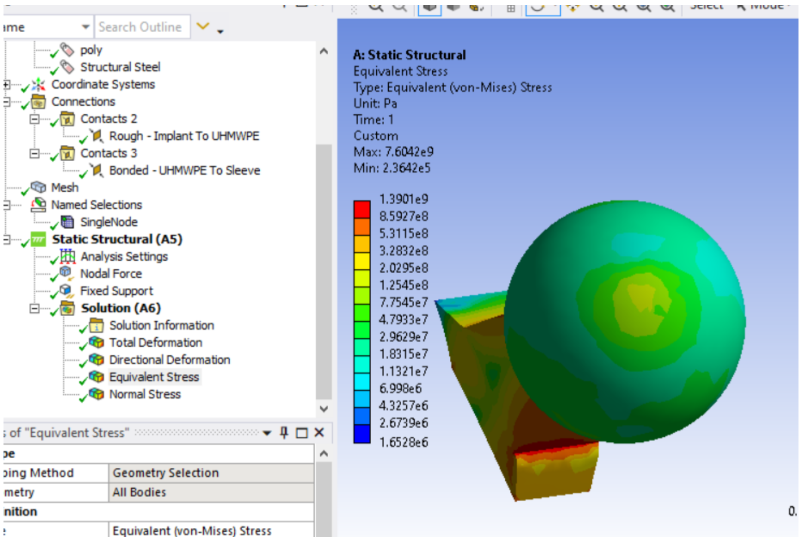
The top side of the cup shows the contact stress between the cup and the sleeve.

The contact stress in the femoral head on the implant and the stress on the sleeve can be similarly plotted.

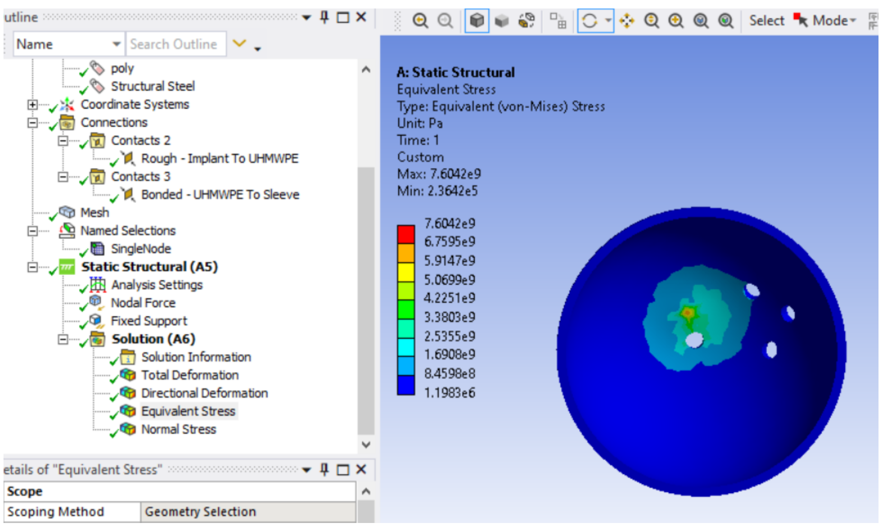
Tip: If your results plot with the elements visible as in the plot below, change the setting in the results tab in Edges to No Wireframe.



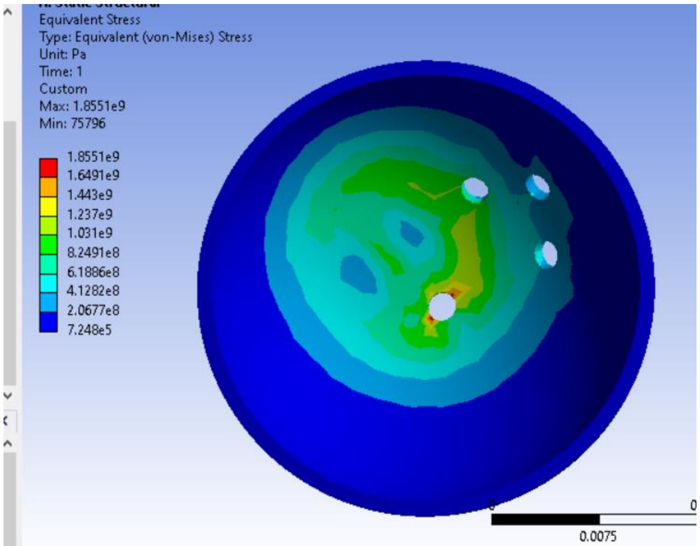




Inside sleeve stresses



The above plots were obtained with a point force load (on a single node) near the top center of the implant. An alternate approach is to apply the load of a region of nodes. Create a new named selection and select a region of nodes.



Compare this analysis to the paper that was shared: Finite Element Study of Acetabular Cup Contact Region for Total Hip Replacement (THR)