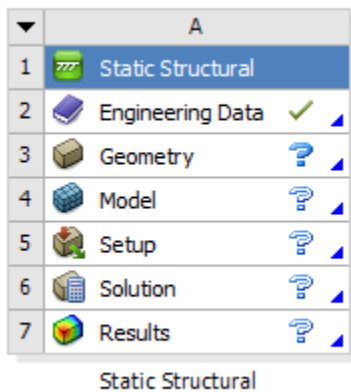


# **Femur Implant Tutorial**

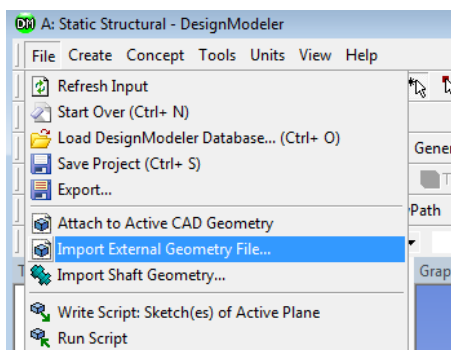
## **Assignment**

1. Follow the tutorial below. Refer to the class notes and use your values for  $W$  and  $h$  as was discussed in class to calculate the joint reaction force.
2. Apply the joint reaction force and find the Von Mises stress and location in the bone, implant, and bone cement.
3. Repeat the exercise this time using the Acetabular cup Component and find the contact stress at the femoral head

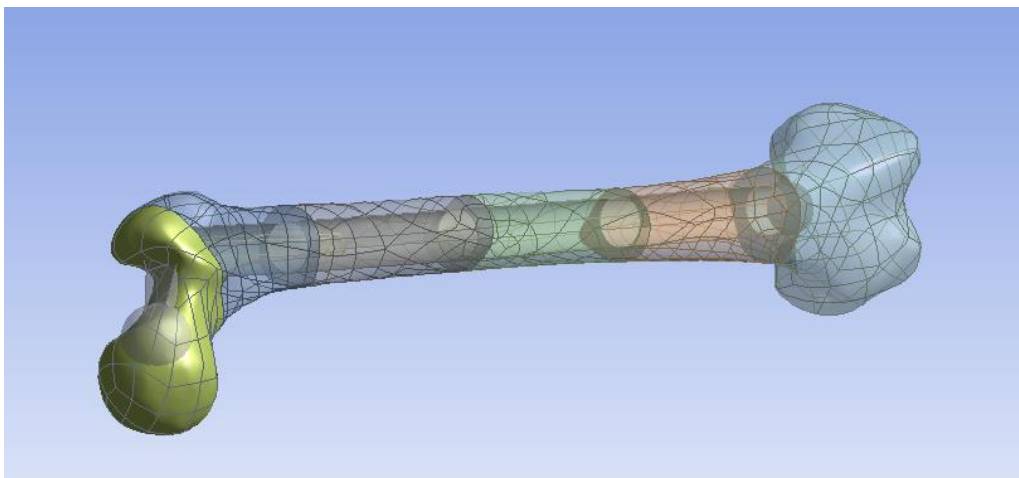
Start workbench and begin a static structural analysis.



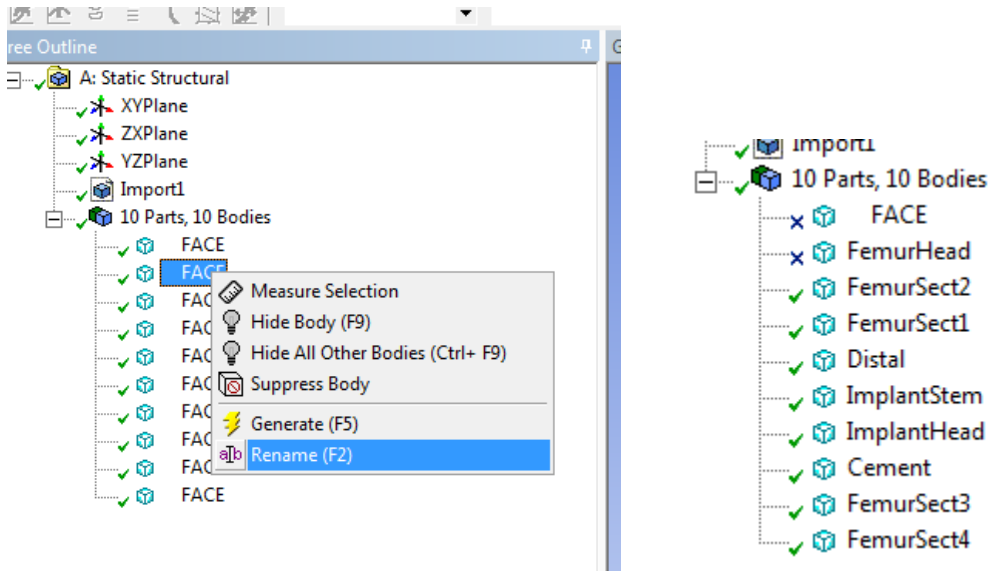
Select Geometry from the tree and import the femur and implant femurimplant.iges file in the canvas module. If prompted, the units are mm.



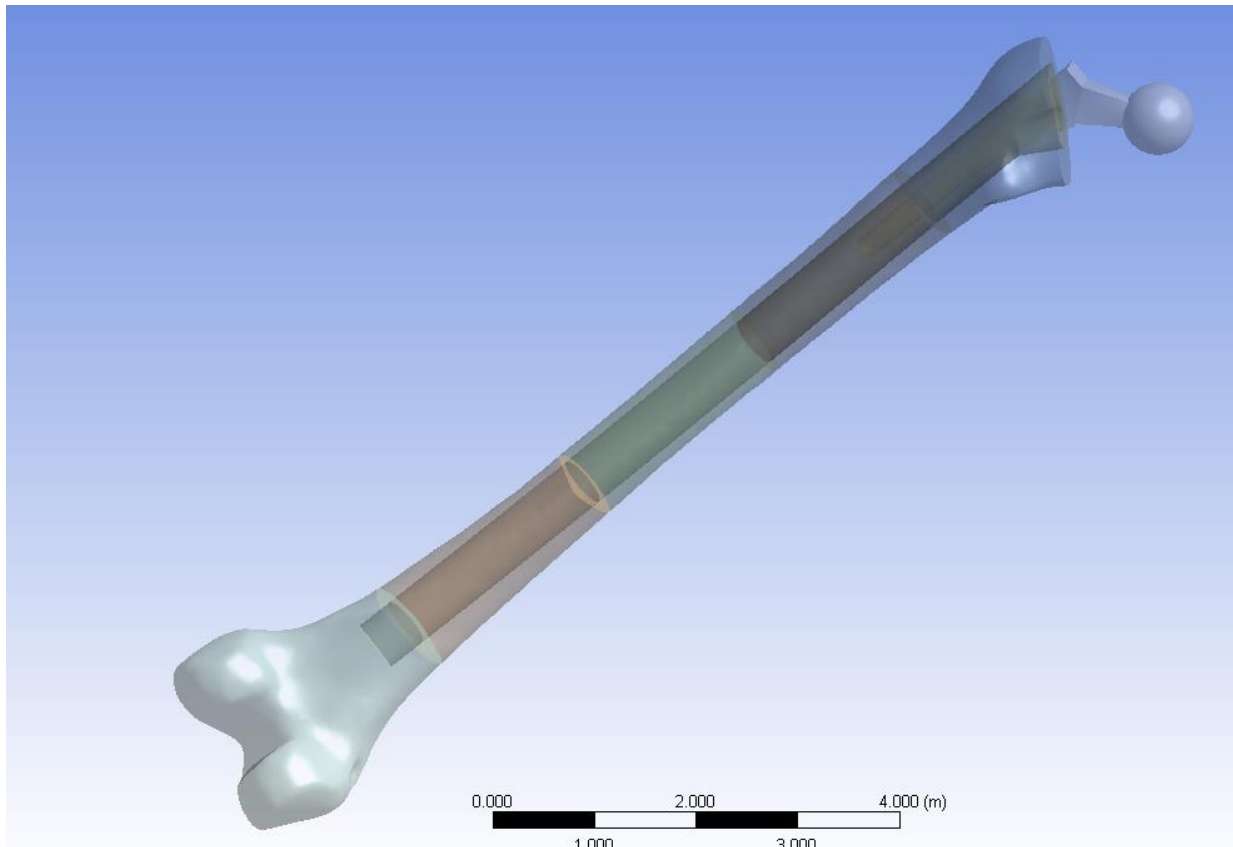
Generate the geometry after importing.



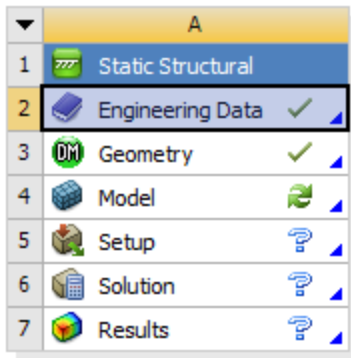
Scroll through the Tree Outline and rename the volumes so that they are easier to identify. Right click on the face and Rename it. You can also Suppress Body to make it easier to toggle through the different volumes.



When finished renaming the sections, suppress the femoral head so that the implant is visible.



Select Engineering Data to add materials.



Static Structural

Unselect Structural Steel by checking the box in the second column. Add new materials by clicking in the line below structural steel. Add materials: bone, bone cement, and stainless steel.

A screenshot of the software interface showing the 'Outline of Schematic A2: Engineering Data' and the 'Properties of Outline Row 4: Bone' dialog box. The 'Outline of Schematic A2: Engineering Data' shows a tree view with 'A' as the root, 'Contents of Engineering Data' as the first row, 'Material' as the second row, 'Structural Steel' as the third row, and 'Bone' as the fourth row. The 'Bone' row is highlighted. Below the tree view, there is a table with columns A, B, C, D, and E. The 'Properties of Outline Row 4: Bone' dialog box shows a table with columns A, B, C, D, and E. The 'Table of Properties Row 2: Isotropic Elasticity' dialog box shows a table with columns A, B, C, D, and E.

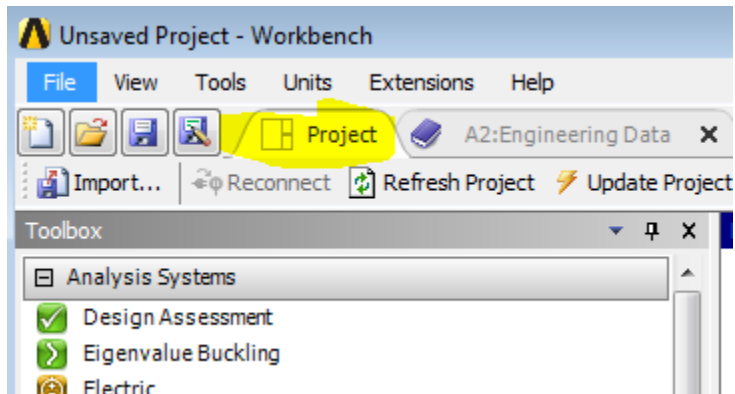
	A	B	C	D	E
1	Property	Value	Unit		
2	Isotropic Elasticity				

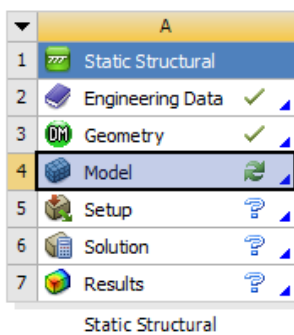
	A	B	C	D	E
1	Temperature (C)	Young's Modulus (Pa)	Poisson's Ratio	Bulk Modulus (Pa)	Shear Modulus (Pa)
2		1.7E+10	0.3	1.4167E+10	6.5385E+09
*					

For each material select the material model on the left by double clicking the elastic behavior. Add the appropriate material properties for elastic modulus and Poisson ratio. Set "linear elastic isotropic". Refer to the paper for material properties of bone and bone cement.

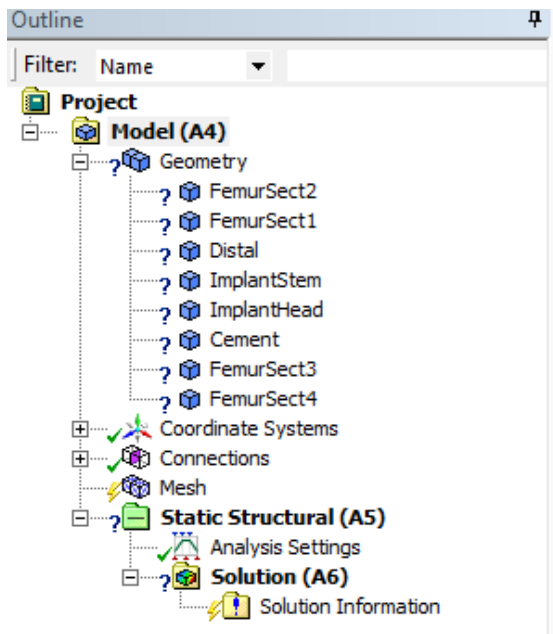
When finished adding the properties select the Project tab in the upper left to return to the project.



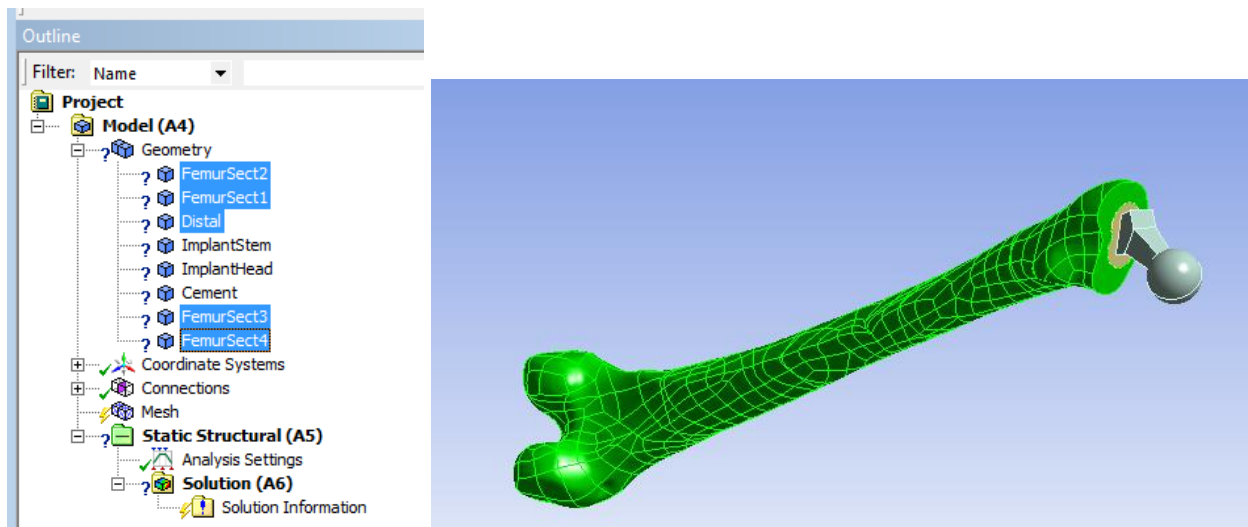
Select Model from the project tree



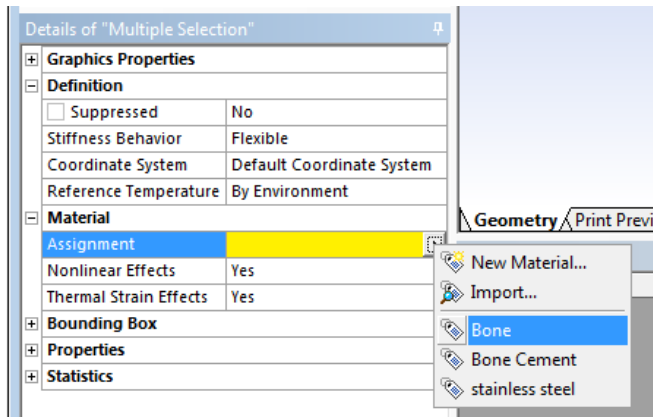
After the model is finished loading expand Geometry from the outline



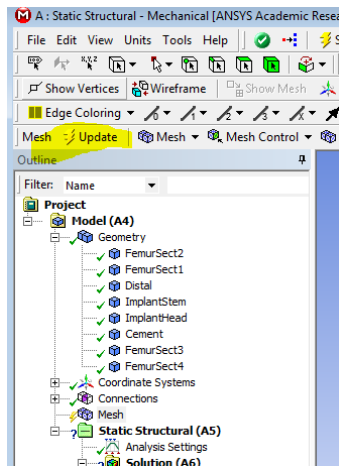
Holding the Control key, select all bone geometries in the Outline tree. The corresponding volumes will highlight

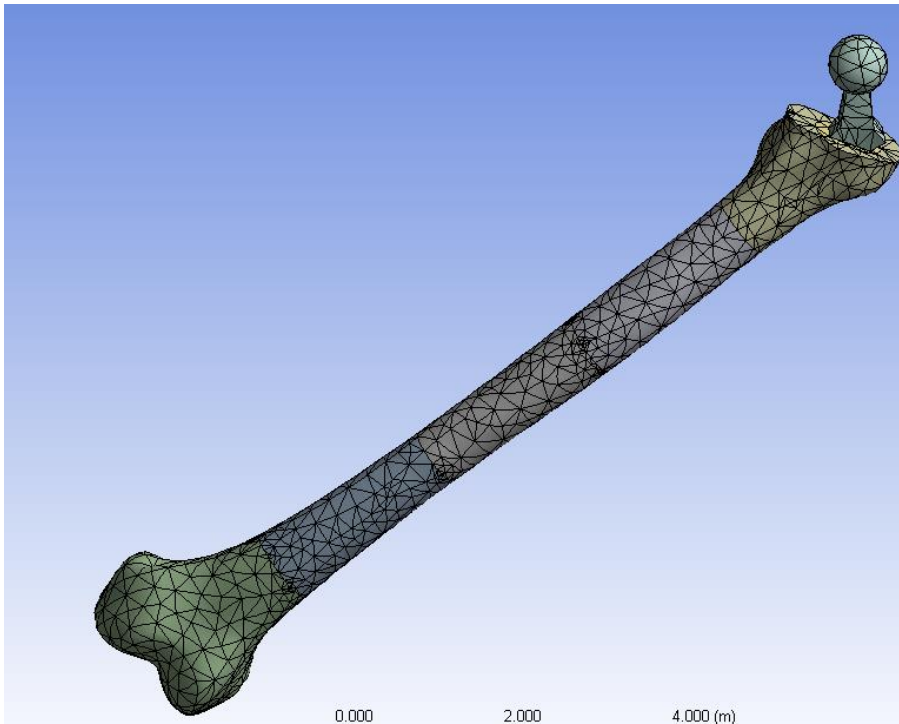


In Details view select the material Bone from the list

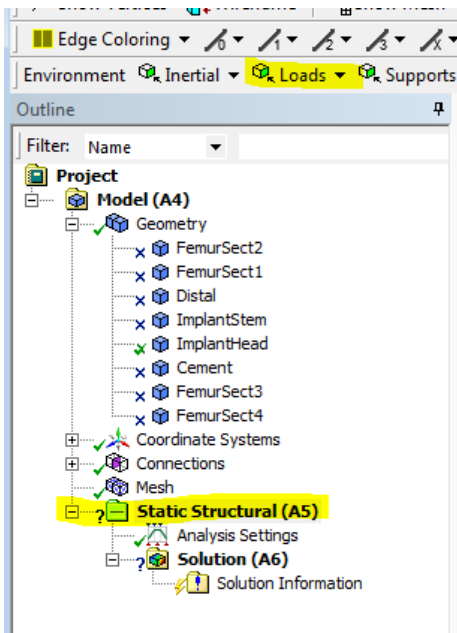


Repeat the above steps to set bone cement and stainless steel properties. Note that in the Outline Tree the question symbol will become a check when the properties have been set. When all the material properties have been defined generate the mesh.





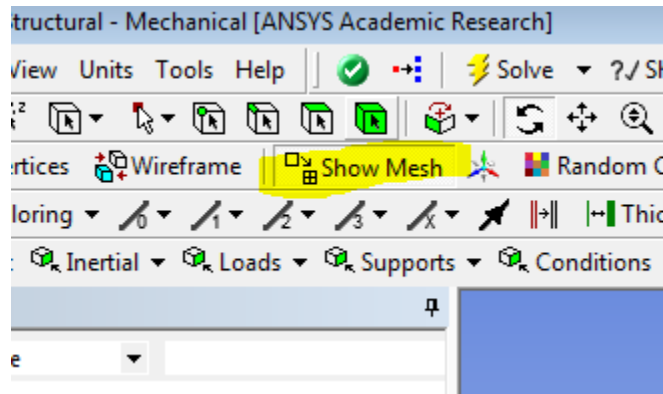
Select Static Structural from the Outline tree and pull down the Loads tab.



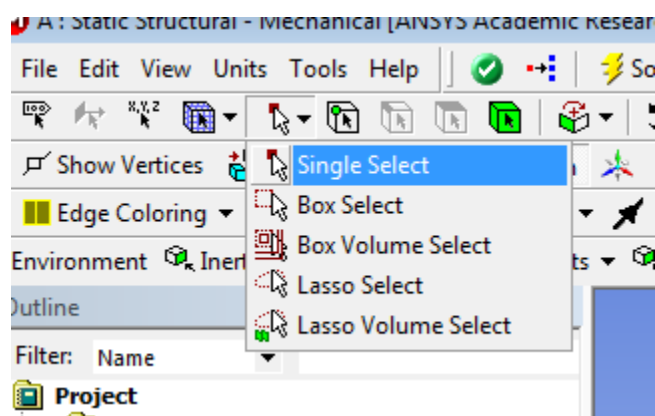
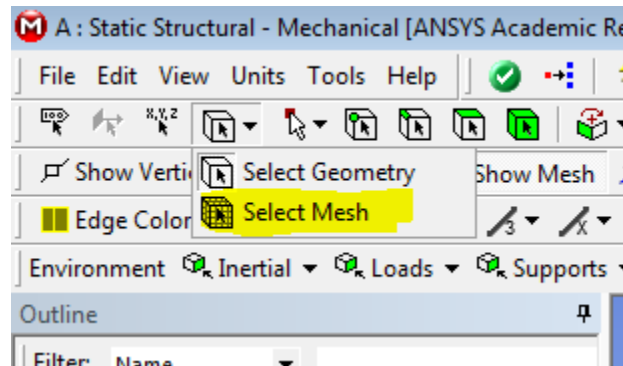
The joint reaction force will be applied on a node at the top of the femur. Since we will be applying the force at a node, we need to show the mesh and then select the location (node) where the load will be applied. This is equivalent to the “select entities” commands in mechanical



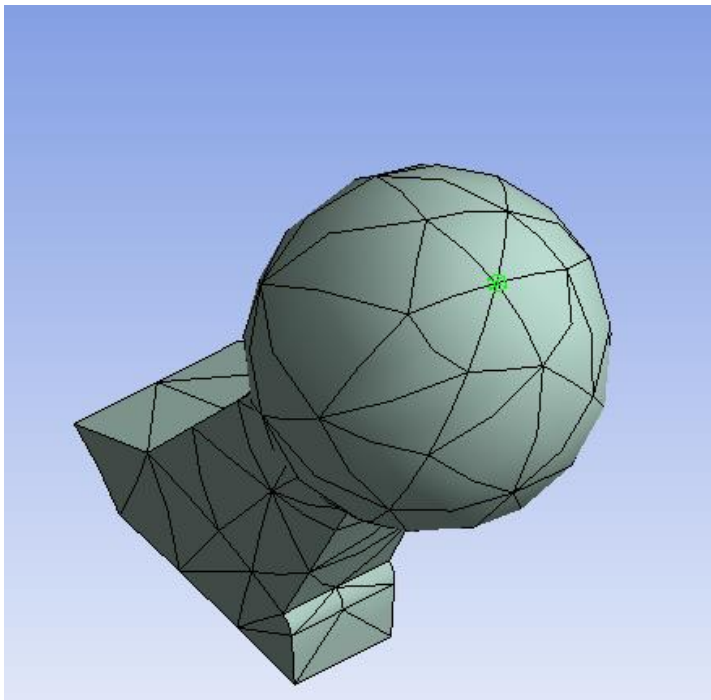
APDL. Show the Mesh. Note, all bodies except the femoral head component have been suppressed in the Geometry Outline for clarity.



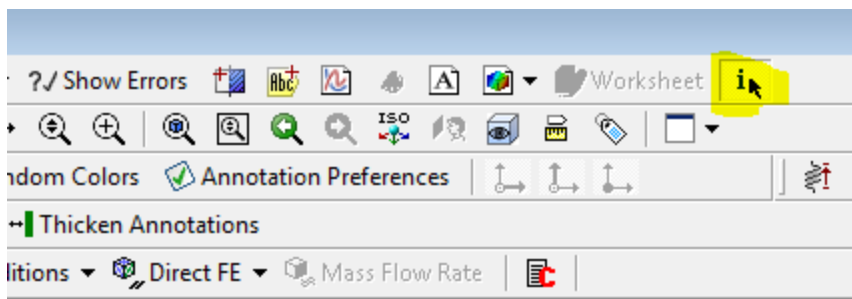
Select Mesh and Single Select to select the node.



Select a node near the top of the femoral head



Information, such as location, can be viewed by selecting the Information button.

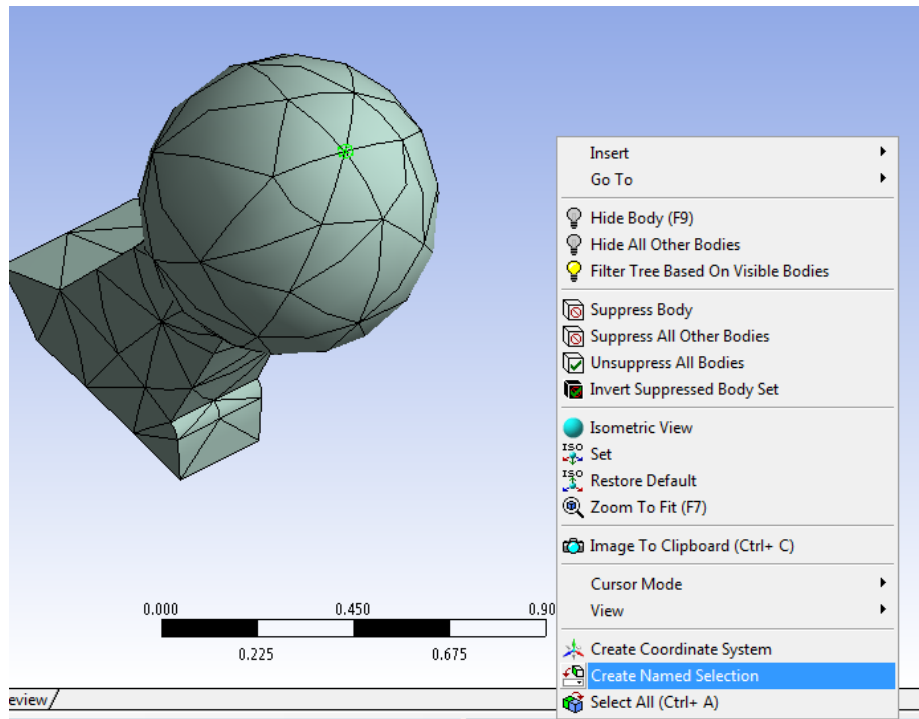


Selection Information

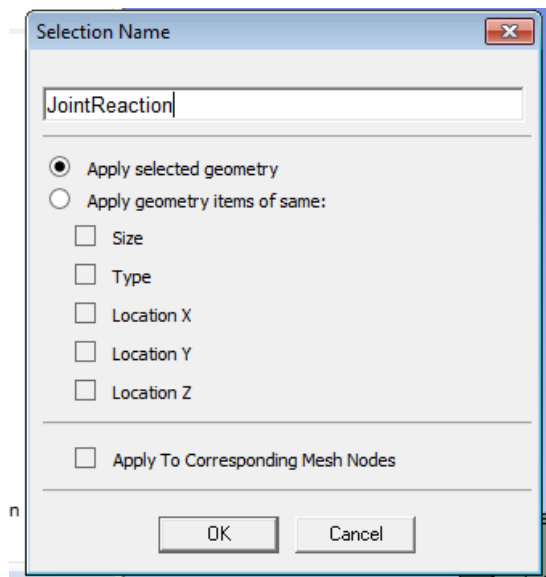
Coordinate System: Global Coordinate System

Entity	X(m)	Y(m)	Z(m)
1 Node, Summary			
Mesh Node 1	-2.8618e-002	-0.75114	-0.54677

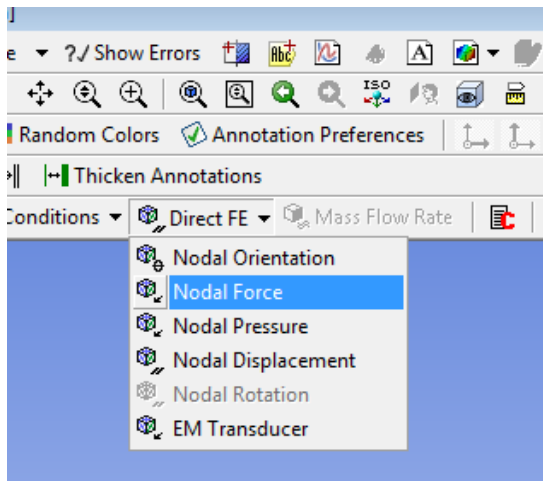
To apply the load at the node a Named selection must first be created. Right click in the graphics window and select “Create Named Section”. Make sure that before you do this that the desired node is highlighted.



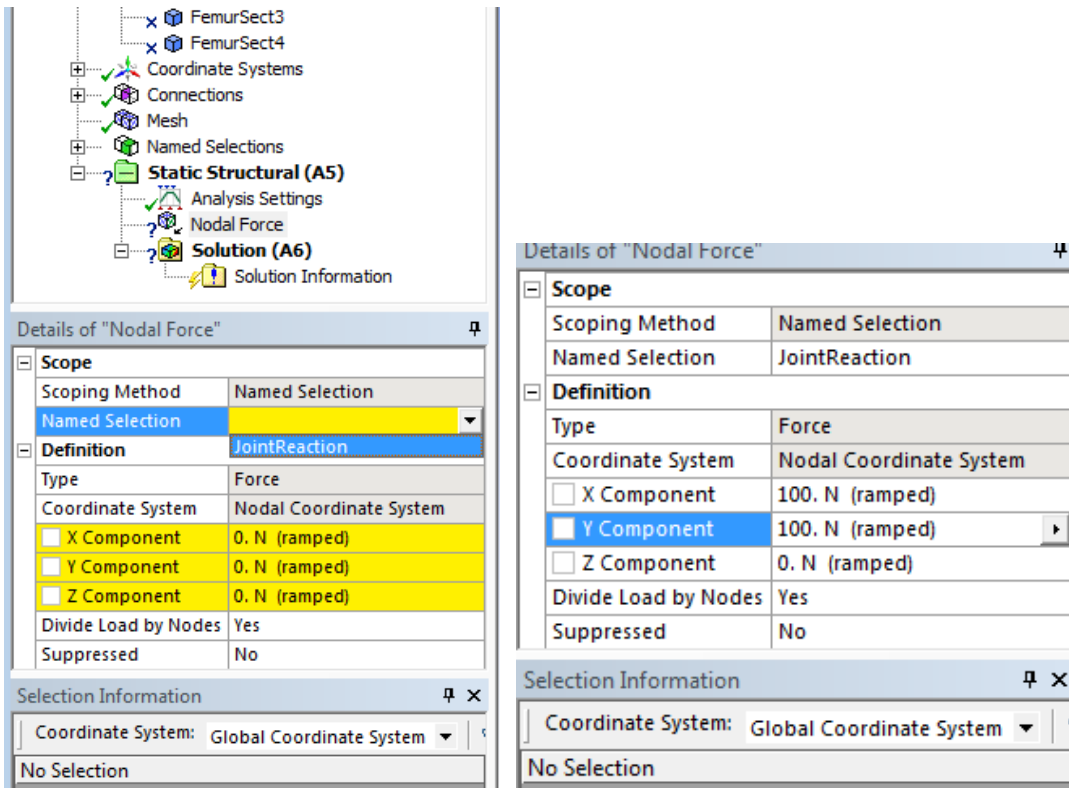
Name the Section



Select “Direct FE” and Nodal Force shortcuts.

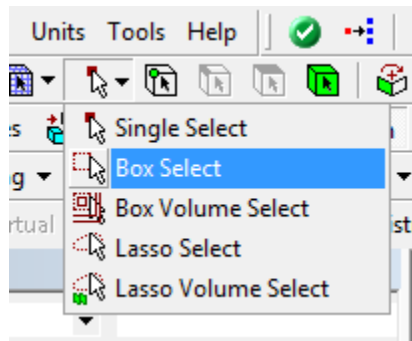


Apply the joint reaction force. Refer to the class notes and use your values for  $W$  and  $h$  as was discussed in class to calculate the joint reaction force. Do not use the 100N forces shown below. That is for demonstrative purposes only. Input the force in the right column next to the force component.

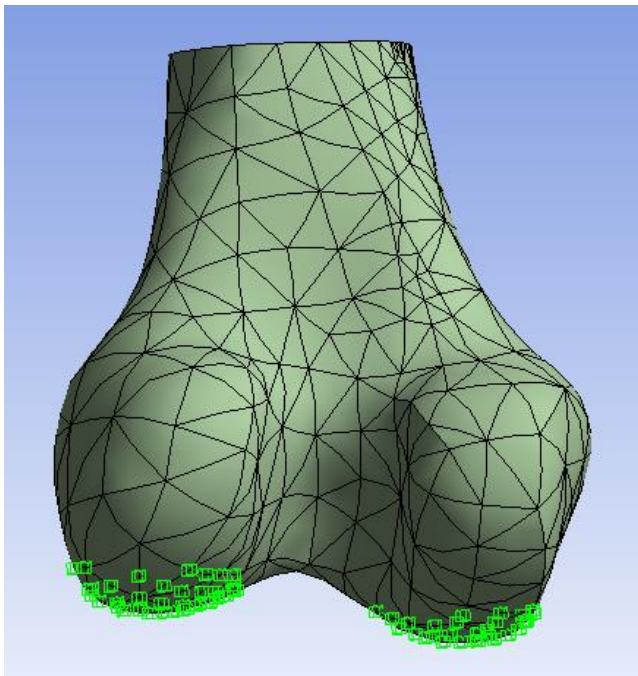


Unsuppress all bodies.

Working on the distal end, repeat the above step to select nodes. However, this time use the “Box Select” instead of the “Single Select”.



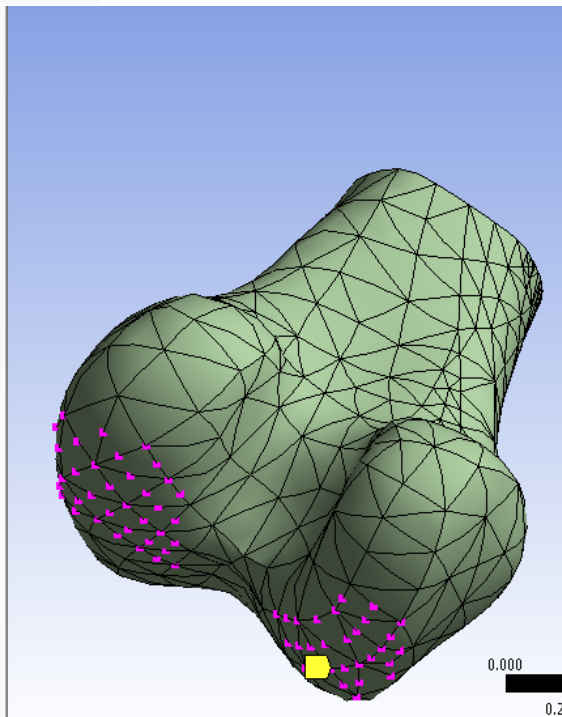
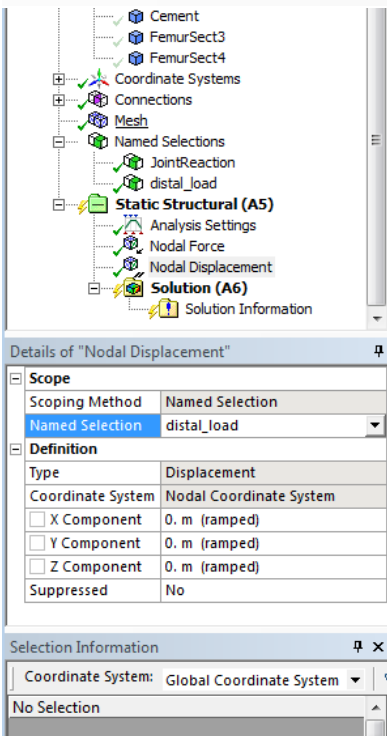
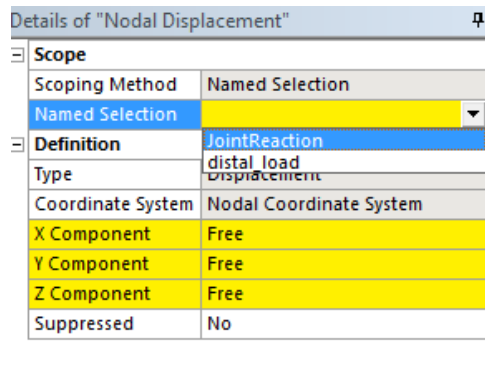
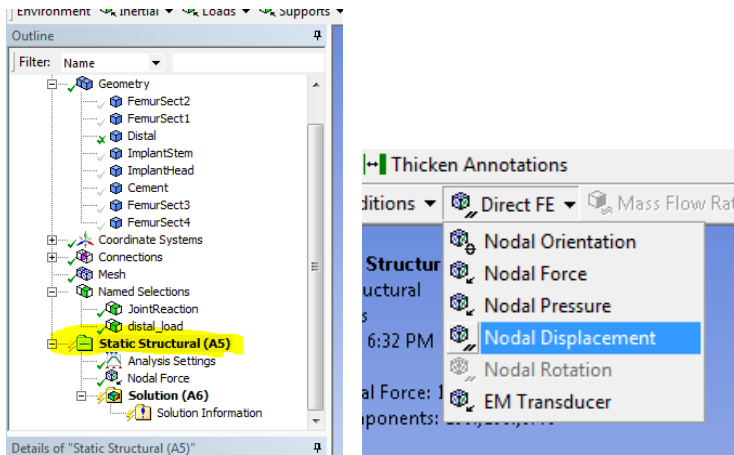
Holding the Control key, draw boxes around the left and right sides of the distal end. Draw the box such that it captures the nodes on the round portion of the distal end. This is the part that would be in contact with the tibia.



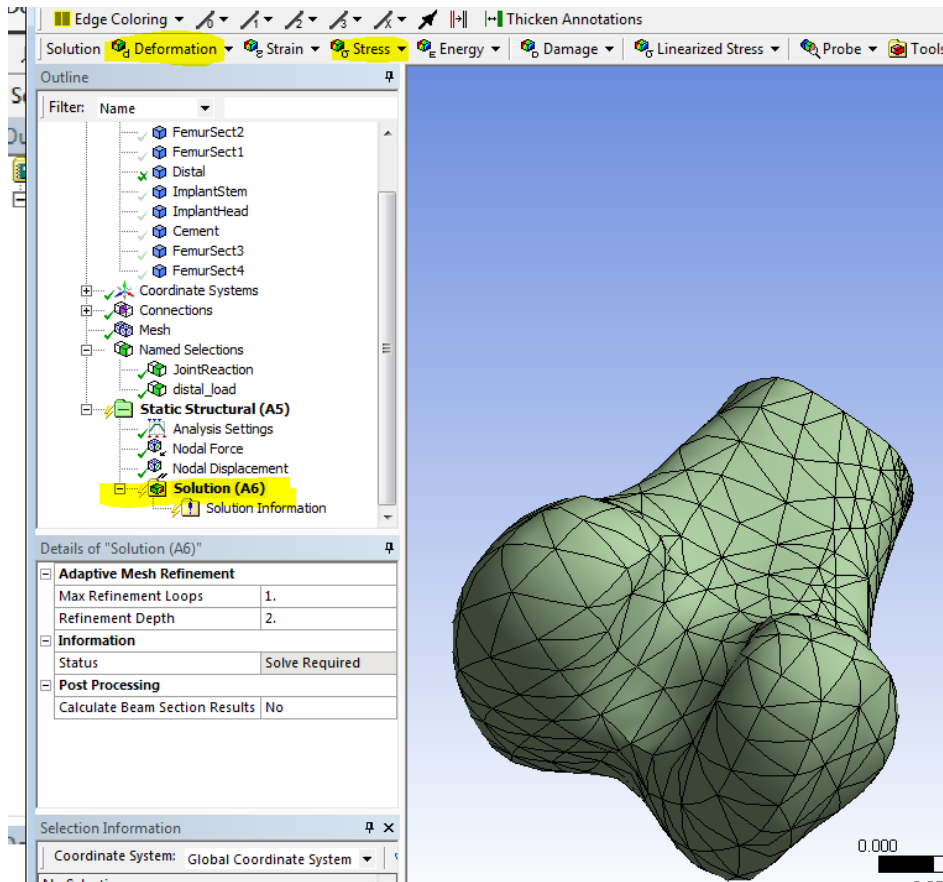
Follow the previous procedure to create a named selection

Select Static Structural from the Outline and then select Direct FE > Nodal Displacement

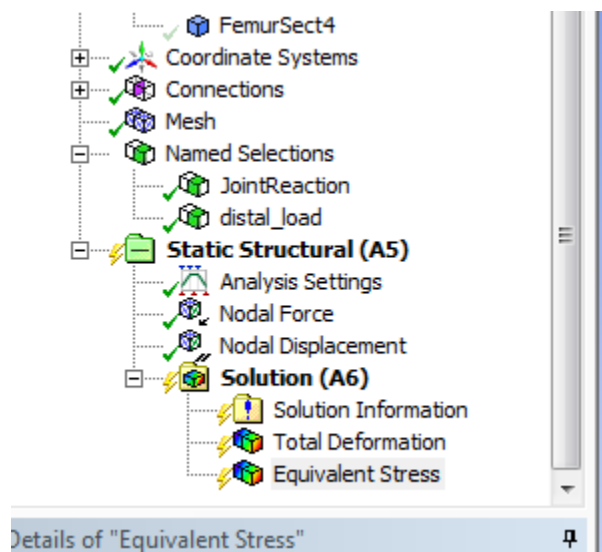
Fix the distal end by setting the displacement in all directions to zero.



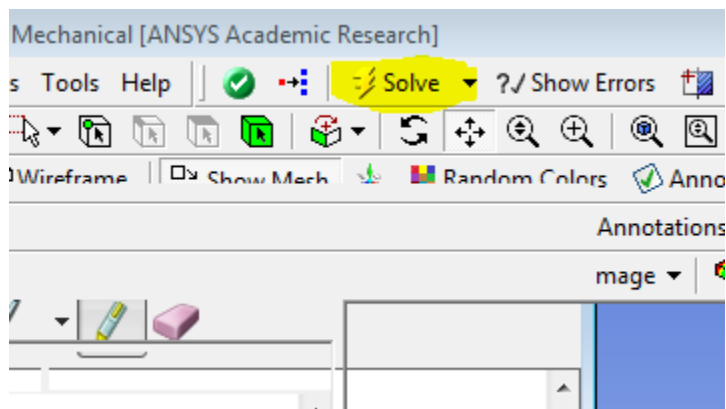
Select Solution from the Outline. Set the Solution outputs



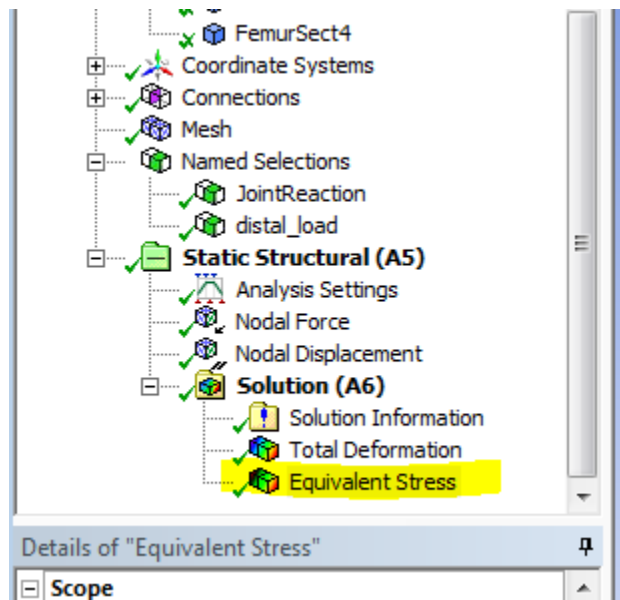
They will be added to the Outline Tree in Solution.



Show all bodies if any were unsuppressed previously and Solve.

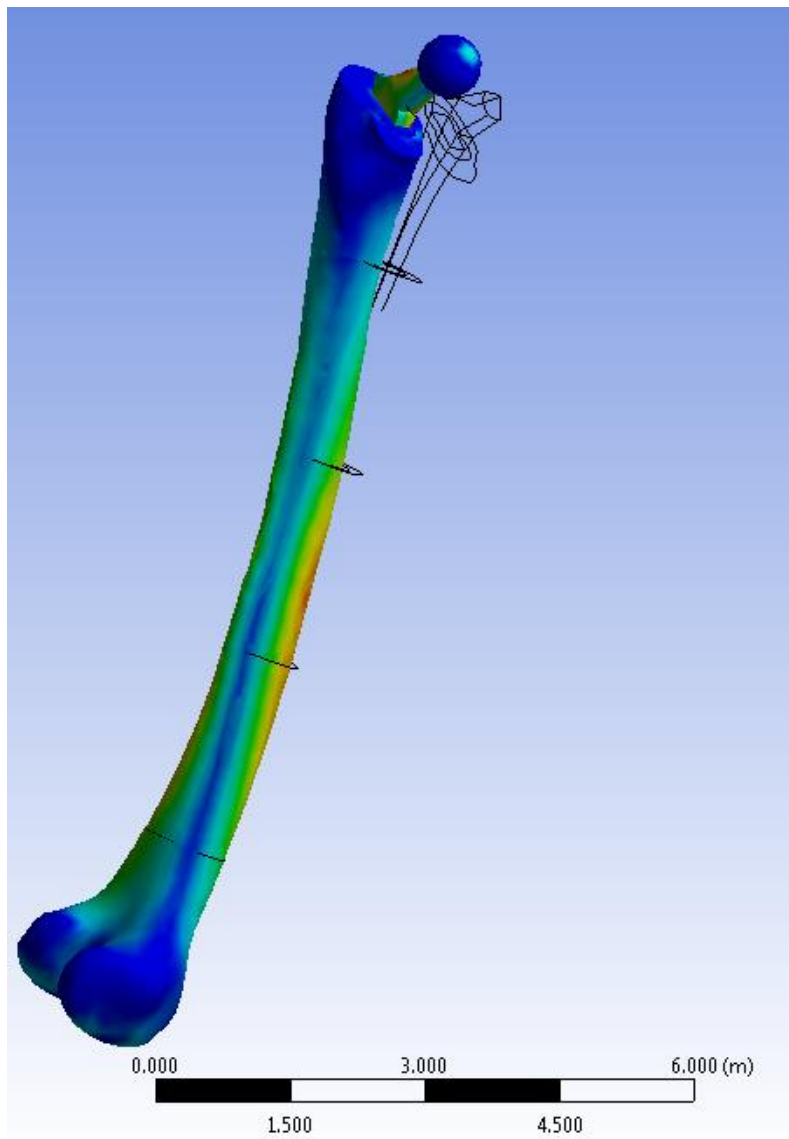
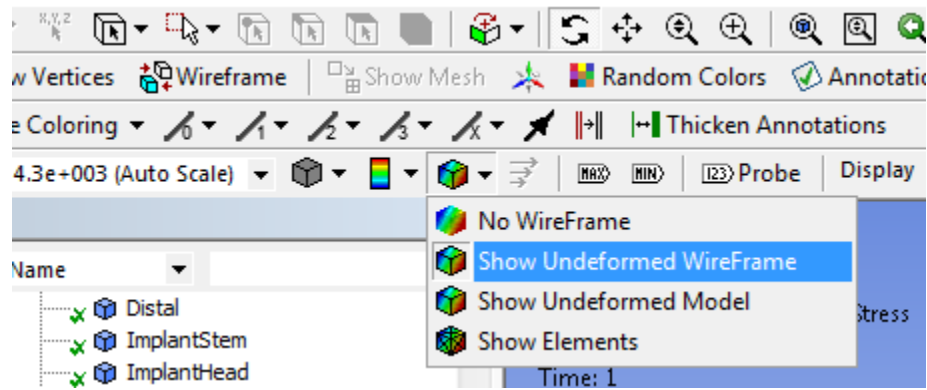


Plot the VonMises Equivalent Stress and Displacement

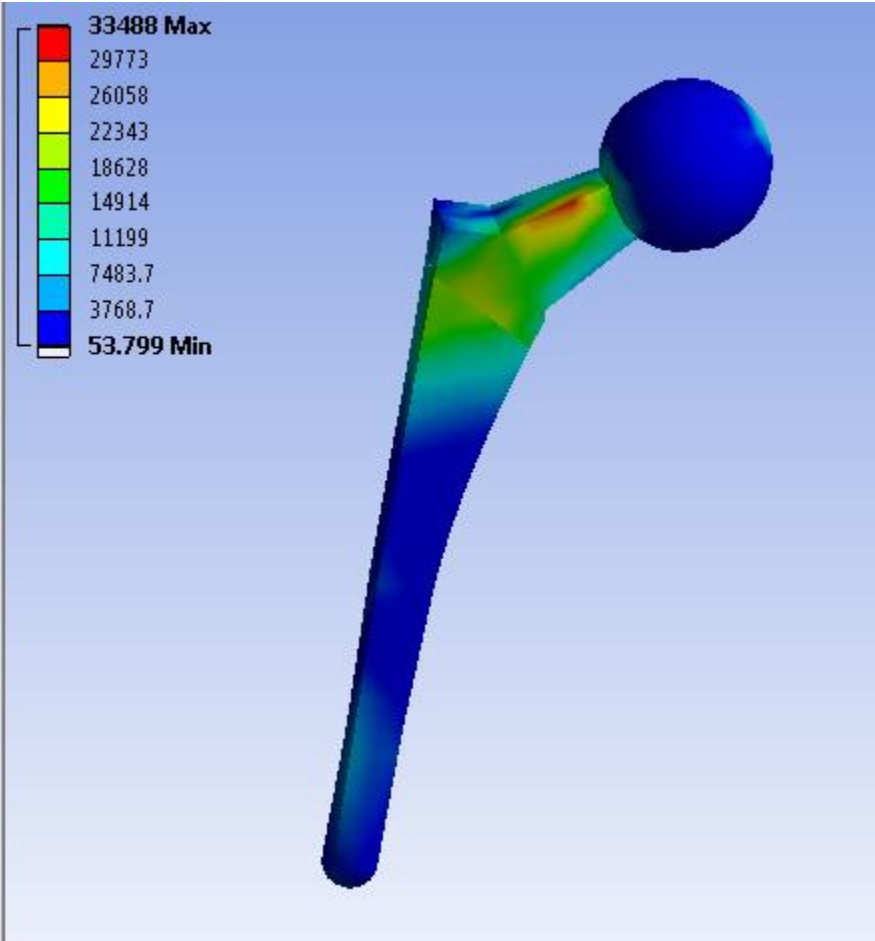




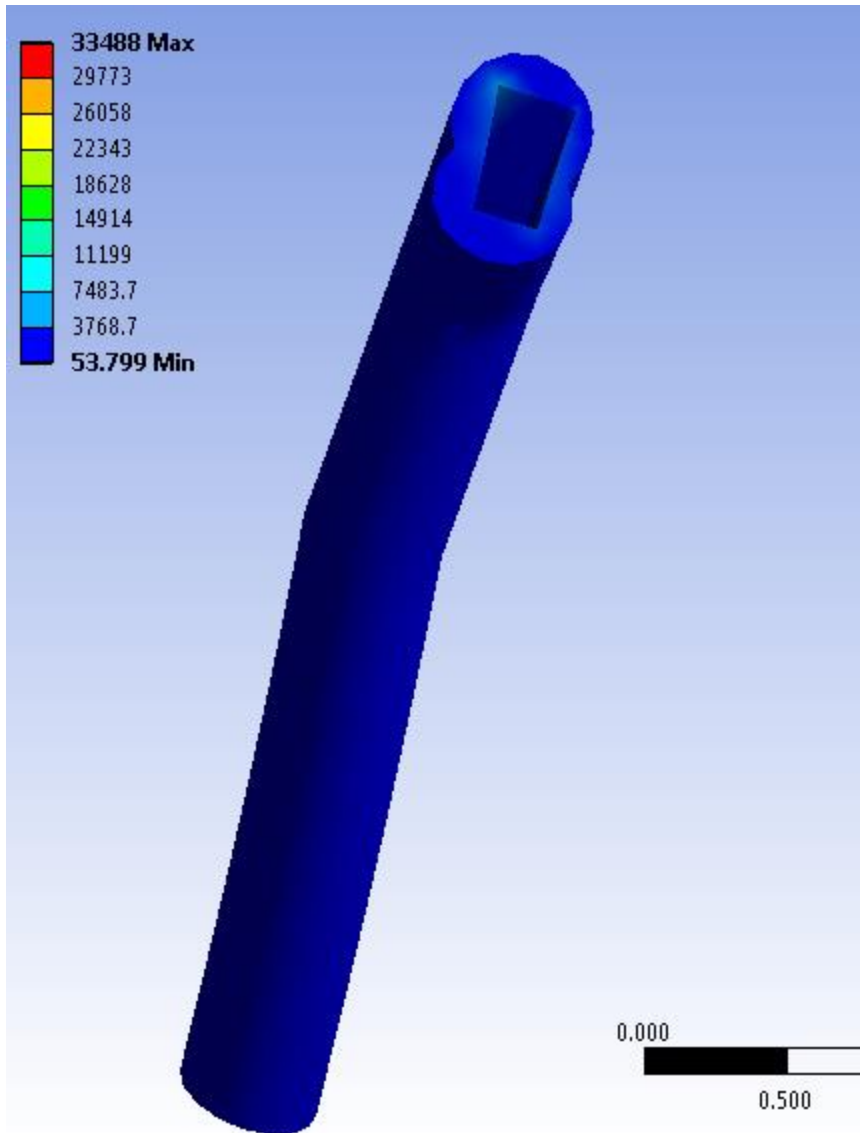
The view can be changed by selecting the edge view



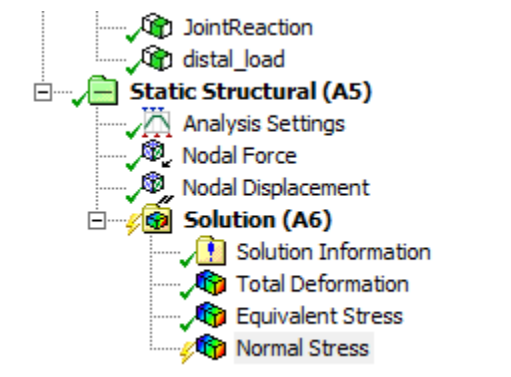
Hide all bodies except for the implant



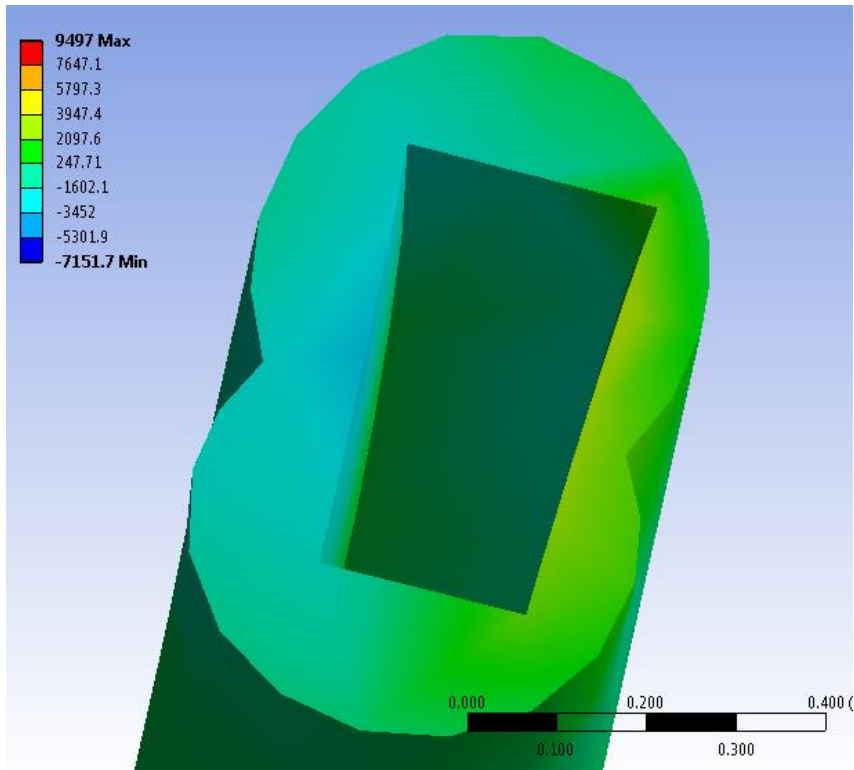
Repeat for Bone Cement and Bone volumes.



Add normal stress and directional stresses and solve.



Plot the normal stresses in the cement



Lower/Raise the bone cement property in engineering data and rerun the analysis.

Update the model (right click in Model) and solve. Compare the results.

