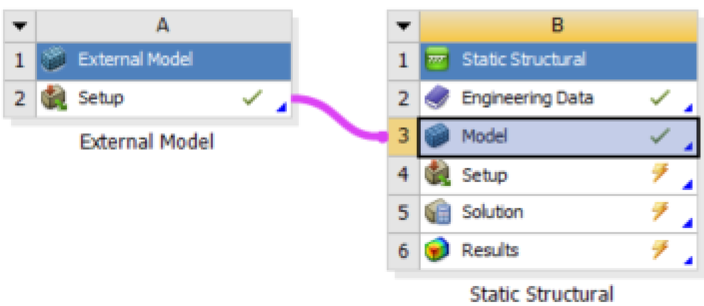
High Resolution FE Model of Bone - Physics Setup

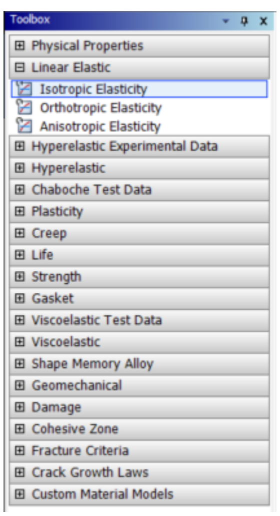
Author: Rajesh Bhaskaran, Cornell University

Physics Setup

Create a material



We have not defined the material for this model so we will do that before we proceed. Double click on ***Engineering Data***. Click on "click here to add a new material" to create a new material. Name the new material "Bone". Expand ***Linear Elastic*** and double click on ***Isotropic Elasticity***.



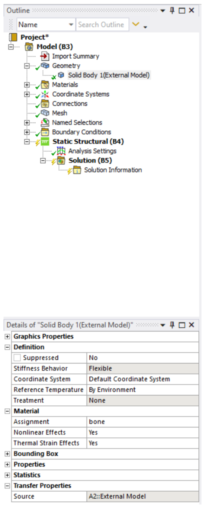
In the properties window, expand Isotropic Elasticity and enter ***1e9 Pa*** for Young's Modulus and ***0.3*** for Poisson's Ratio.

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young's Modulus and Poisson's Ratio			
5	Young's Modulus	1E+09	Pa		
6	Poisson's Ratio	0.3			
7	Bulk Modulus	8.3333E+08	Pa		
8	Shear Modulus	3.8462E+08	Pa		

Click on the Project tab, next to the Engineering Data tab, to return to the Project Schematic page.

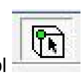
Assign Material

The default material used for Mechanical is structural. We need to change it to the new material we defined earlier (Bone). In the outline window, right click on ***Model***, and select ***Refresh Materials***. Then, expand ***Geometry*** and highlight ***Solid Body 1***. The bone model will now be highlighted in green, meaning it is being selected. Right below the outline window you will see "Details of Solid Body 1". Expand ***Material***, and change the assignment to ***Bone***. The material properties we defined for ***Bone*** will now be assigned to the bone model.

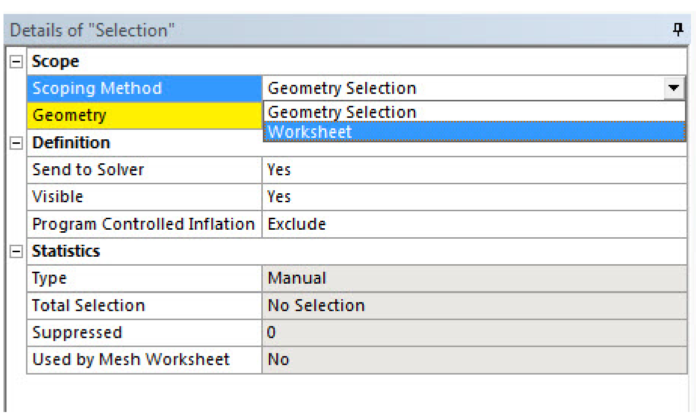


Named Selection

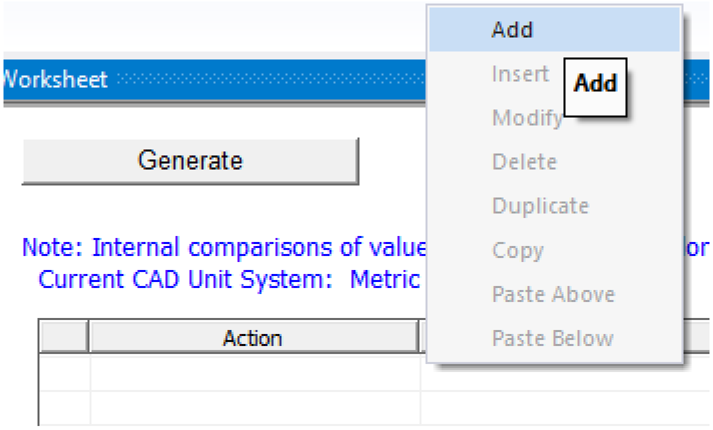
We will assign ***deformation*** to the ***positive y*** face and ***roller support*** to the ***negative y face***. The geometry of the model makes it difficult to select all the faces in a given plane. However, we can easily assign the BC using name selection.

Click on the z axis in the coordinate axis viewer to view the XY plane. Use the vertex selection tool  and select a point on the top edge. Right below the model you will see the coordinates of this point. The y position is 4.98 mm.

Right click on Named Selections and insert a new named selection. Change the Scoping Method to ***Worksheet***.



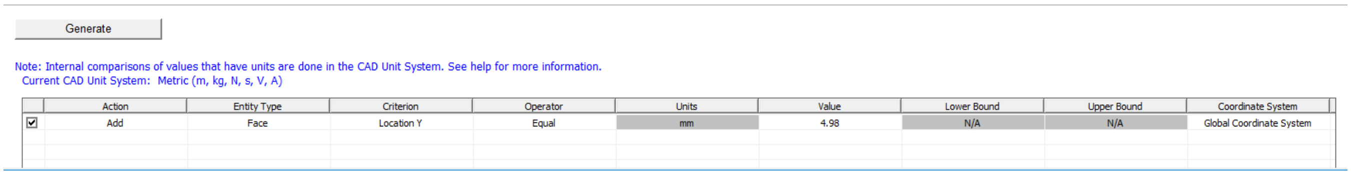
In the main window you will see an empty table. Right click on the empty table and click on ***Add***.



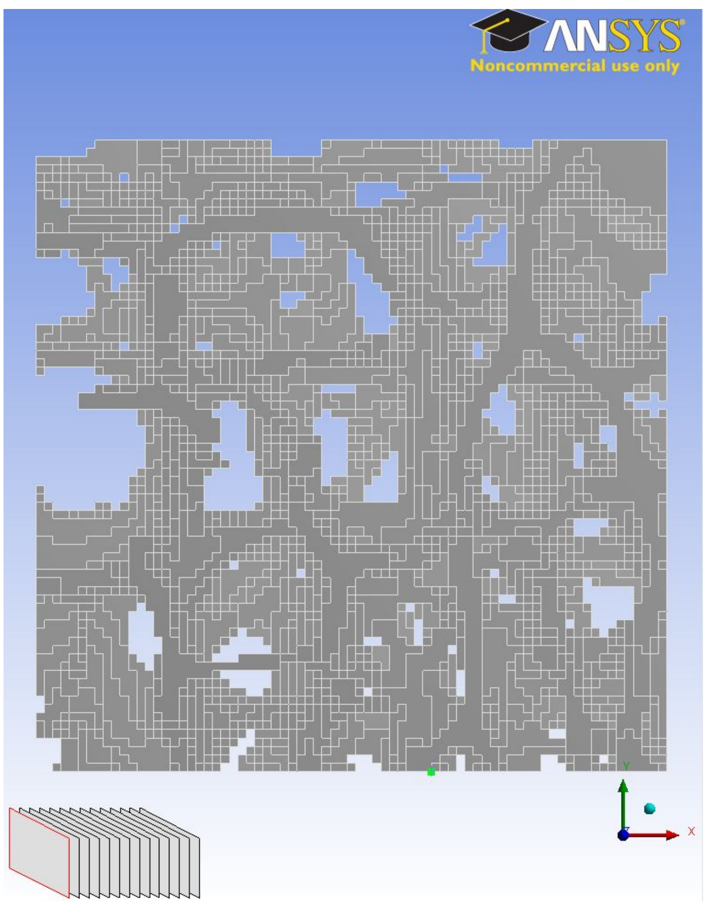
We will now enter the appropriate information that will help us create a named selection that includes all the faces in the top edge of the model viewed from the XY plane.

Select ***Face*** for ***Entity Type***.
Select ***Location y*** for ***Criterion***.
Select ***Equal*** for ***Operator***.
Enter ***4.98*** for ***Value***.

Click on ***Generate*** to generate the named selection. Rename it ***top y***.



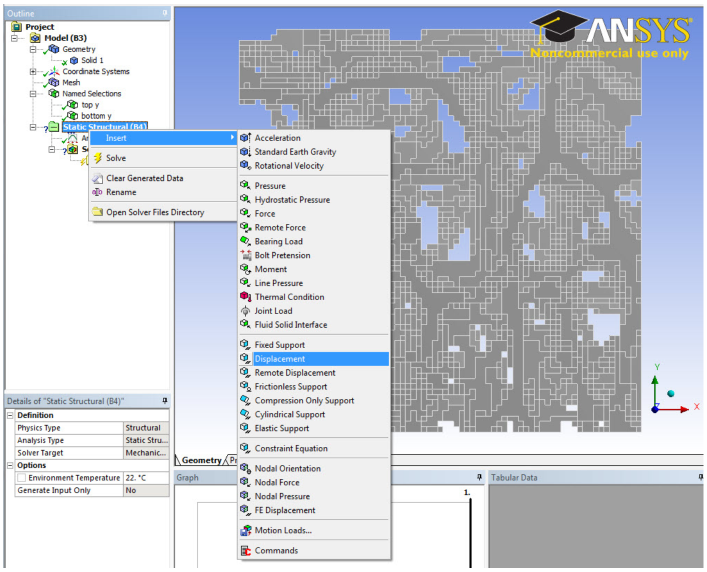
We will repeat the same steps to create a named selection for all the faces on the bottom edge of the model viewed from the XY plane.



The y coordinate for the bottom edge is 3.3e-2 mm. Rename the second named selection ***bottom y***

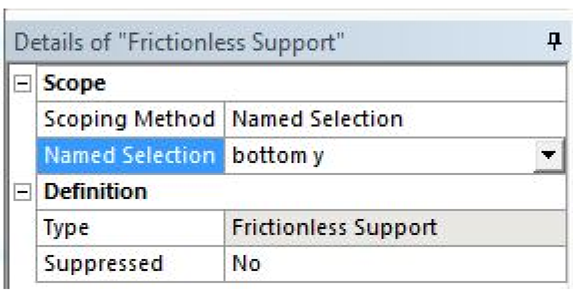
Boundary Conditions

Right click on Static Structural (B4) and insert a displacement.



Change the Scoping Method to ***Named Selection*** and select ***top y***. Enter ***-0.5 mm*** for the ***Y Component***. This will assign a 0.5 mm displacement to the model in the -y direction on the faces selected for ***top y***.

We can model the roller support as frictionless support in Mechanical. Right click on Static Structural (B4) and insert a frictionless support. Similarly, change the scoping method to Named selection but select bottom y.



This will constraint the displacement in the y direction but the model is allowed to displace in the x direction. This is very similar to the conditions of a roller support.

The setup is finished. You may move on to setting up the solution.

Go to Step 5: Numerical Solution

Go to all ANSYS Learning Modules