High Resolution FE Model of Bone - Geometry

Author: Rajesh Bhaskaran, Cornell University
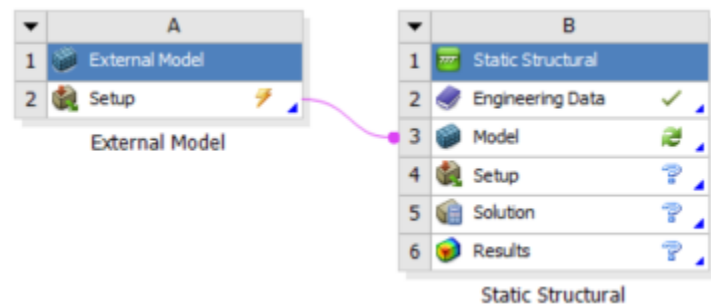
Geometry

Drag *Static Structural* into the Project Schematic page, right next to External Model, like so;
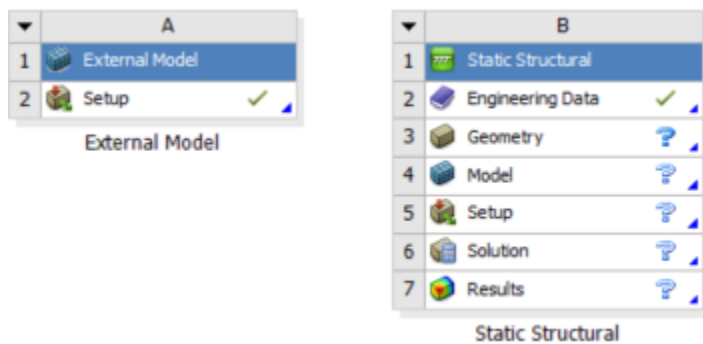
In the Project Schematic page, click and drag *Setup* from *External Model* and drop it onto *Model* in *Static Structural*.


This will load the External Model into ANSYS Mechanical. When External Model is linked with *Model* in *Static Structural*, the geometry and mesh in the External Model is transferred to Mechanical and the user can proceed straight to setting up the physics.

Notice there is a lighting bolt symbol on the External Model. Right click on Setup and select *Update*.

Under **Static Structural**, right click on **Model** > **Properties**. The *Tolerance Angle* is set to *45* . The Mesh Conversion process computes the angle between the normals of two adjacent elements in the mesh. If this angle is less than or equal to the Tolerance Angle then the two elements are in the same component, otherwise, they are separated. We set the tolerance angle to 15 degrees. For this particular mesh, the changing the tolerance angle does not affect solution results. For larger scale meshes, changing the tolerance angle of the mesh will have an affect on solution results.

	A	B	C
1	Property	Value	Unit
2	□ General		
3	Component ID	Model	
4	Directory Name	SYS-2	
5	□ Notes		
6	Notes		
7	□ Used Licenses		
8	Last Update Used Licenses		
9	□ System Information		
10	Physics	Structural	
11	Analysis	Static Structural	
12	Solver	Mechanical APDL	
13	□ Mesh		
14	Save Mesh Data In Separate File	☐	
15	□ General Model Assembly Properties		
16	Length Unit	m	
17	Object Renaming	Based on System Name	
18	□ Mesh Conversion Options		
19	Analysis Type	3D	
20	Create Geometry	☑	
21	Tolerance Angle	45	degree
22	Vertex Insertion Angle	120	degree
23	Create Geometry Face Components	☐	
24	Create Geometry Edge Components	☐	
25	Create Geometry Vertex Components	☐	
26	Process Line Bodies	☑	
27	Body Grouping	Material Number	
28	□ Licenses		
29	License	Use License Preference	
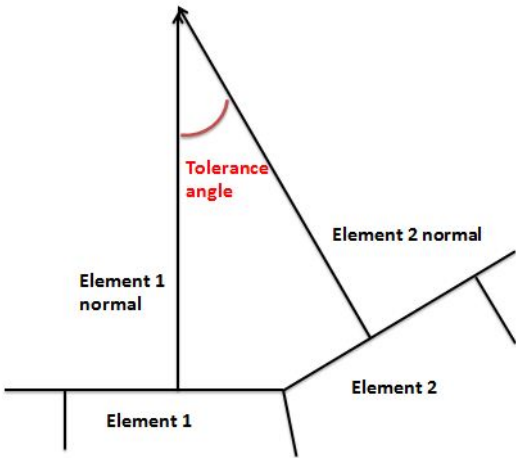
You may now move on to the next step

Go to Step 3: Mesh

Go to all ANSYS Learning Modules