## **High Resolution FE Model of Bone - Geometry**

Author: Rajesh Bhaskaran, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

## Geometry

Drag Static Structural into the Project Schematic page, right next to External Model, like so;

•		А	
1	۵	External Model	
2	٩	Setup	× .
		External Model	

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.

Properti	es of Schematic B3: Model		- <b></b>	×
	A	В	С	
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	<b>V</b>		
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