

# 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](#)

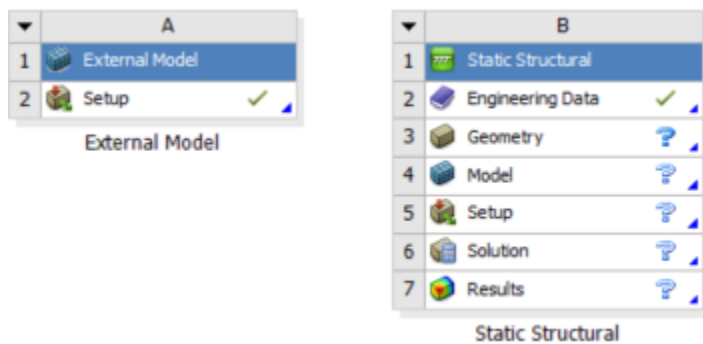
[7. Verification & Validation](#)

[Exercises](#)

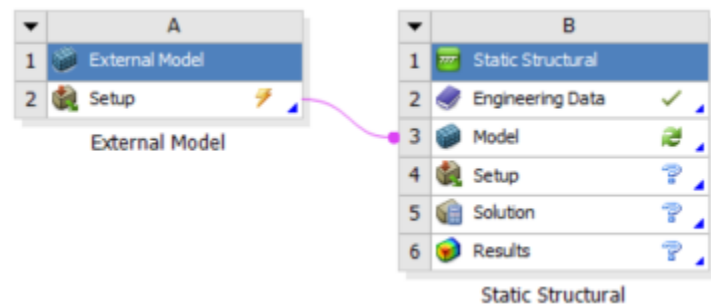
[Comments](#)

## Geometry

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



In the Project Schematic page, click and drag **Setup** from **External Model** and drop it onto **Model** in **Static Structural**.

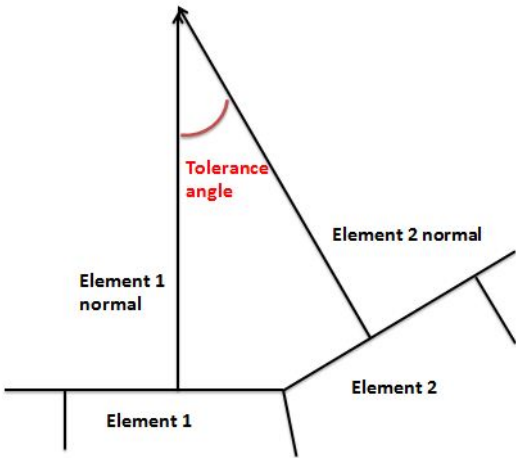


This will load the External Model into ANSYS Mechanical. When External Model is linked with **Model** in **Static Structural**, the geometry and mesh in the External Model is transferred to Mechanical and the user can proceed straight to setting up the physics.

Notice there is a lightning 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.

Properties of Schematic B3: Model			
	A	B	C
1	Property	Value	Unit
2	General		
3	Component ID	Model	
4	Directory Name	SYS-2	
5	Notes		
6	Used Licenses		
7	Last Update Used Licenses		
8	System Information		
9	Physics	Structural	
10	Analysis	Static Structural	
11	Solver	Mechanical APDL	
12	Mesh		
13	Save Mesh Data In Separate File	<input type="checkbox"/>	
14	General Model/Assembly Properties		
15	Length Unit	m	
16	Object Renaming	Based on System Name	
17	Mesh Conversion Options		
18	Analysis Type	3D	
19	Create Geometry	<input checked="" type="checkbox"/>	
20	Tolerance Angle	45	degree
21	Vertex Insertion Angle	120	degree
22	Create Geometry Face Components	<input type="checkbox"/>	
23	Create Geometry Edge Components	<input type="checkbox"/>	
24	Create Geometry Vertex Components	<input type="checkbox"/>	
25	Process Line Bodies	<input checked="" type="checkbox"/>	
26	Body Grouping	Material Number	
27	Licenses		
28	License	Use License Preference	
29			



You may now move on to the next step

[Go to Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)