High Resolution FE Model of Bone - Pre-Analysis & Start-Up

Author: Rajesh Bhaskaran, Cornell University

Pre-analysis & Start-Up

Pre-analysis

The equivalent stiffness of the model can be determined:

$$E_{equiv} = \frac{\sigma_{equiv}}{\epsilon_{equiv}} = \frac{R/_{Area}}{\delta/_{L}} = \frac{R/_{(L^2)}}{\delta/_{L}}$$

where
R is the reaction force
delta is the deflection
L is the length of the model

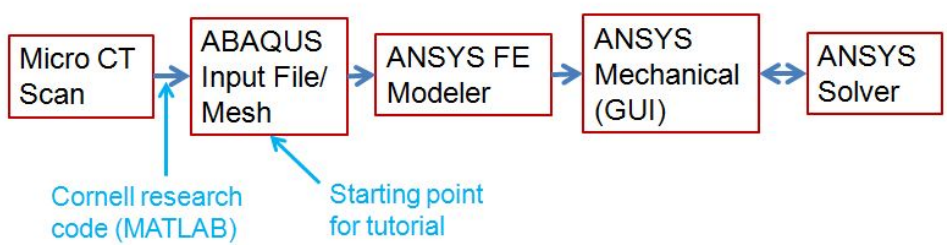
We will find the reaction force and the support and use the above relation to verify the simulation.

Start-Up

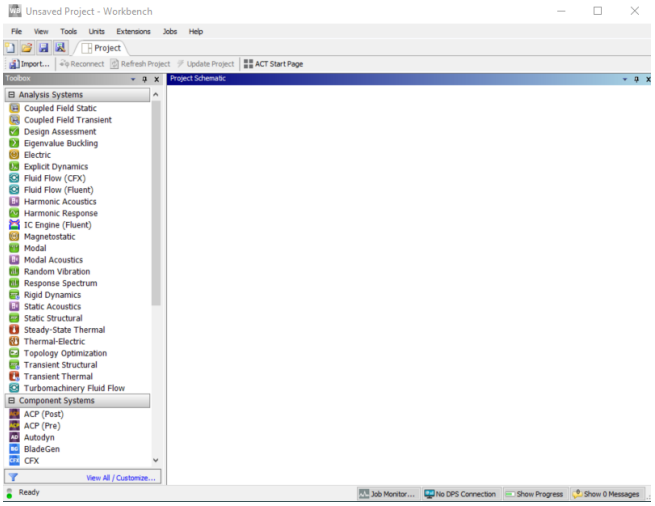
Before you proceed, you need to download the Abaqus input file.

download the Abaqus file here

The following flow chart illustrates the steps to carry out ANSYS simulation starting from Micro CT Scan.

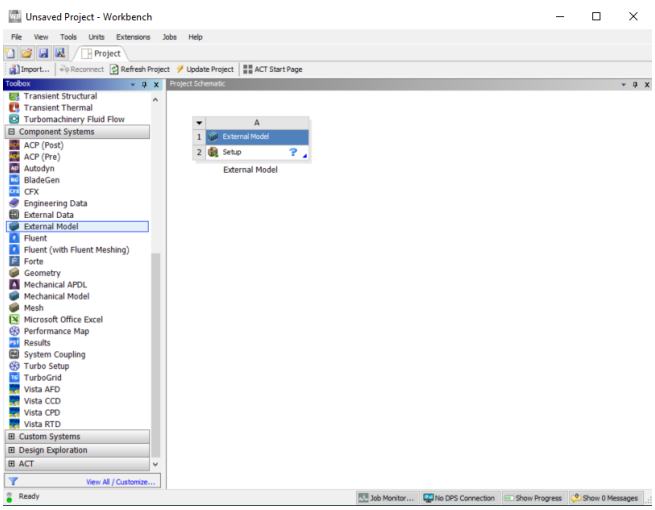


Open ANSYS Workbench:

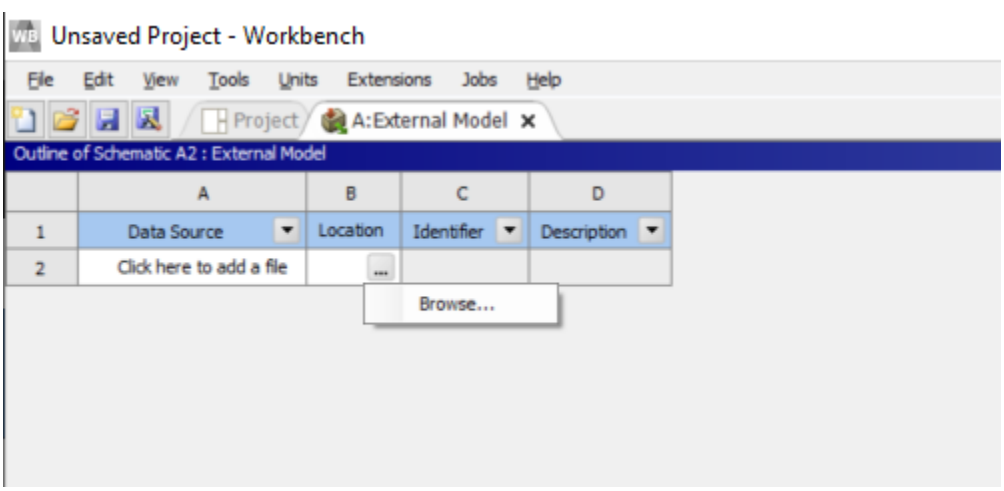


We will import the Abaqus input file into External Model and convert it into a Geometry Setup that ANSYS can simulate.

Expand ***Component Systems*** in the Toolbox. Locate ***External Model*** and drag it into Project Schematic:



Right click on ***Setup*** > ***Edit*** > ***Location*** >***Browse***, and locate the Abaqus input file.



Once the Abaqus file is imported, make sure that the Unit System selected are in mm. Click on the Data Source cell (Cell A2), then check the unit system is as shown below:

Outline of Schematic A2 : External Model

	A	B	C	D
1	Data Source	Location	Identifier	Description
2	C:\Users\aa734\Downloads\C2483o_Coarsen4.inp	...	File1	
3	Click here to add a supporting file	...		
4	Click here to add a file	...		

	A	B	C
1	Property	Value	Unit
2	Description		
3	Application Source	Abaqus	
4	Definition		
5	Unit System	Metric (kg,mm,s,°C,mA,N,mV)	
6	Process Nodal Components	☑	
7	Nodal Component Key		
8	Process Element Components	☑	
9	Element Component Key		
10	Process Face Components	☑	
11	Face Component Key		
12	Process Model Data	☑	
13	Node And Element Renumbering Method	Automatic	
14	Transformation Type	Rotation and translation	
15	Rigid Transformation		
16	Number Of Copies	0	
17	Origin X	0	m
18	Origin Y	0	m
19	Origin Z	0	m
20	Theta XY	0	radian
21	Theta YZ	0	radian
22	Theta ZX	0	radian

Click on the Project Tab to return to Project Schematic. You should see an update symbol 

next to Setup. This means the input file has been loaded and External Model is ready to generate a model.

Go to Step 2: Geometry

Go to all ANSYS Learning Modules