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

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

Pre-analysis & Start-Up

Pre-analysis

The equivalent stiffness of the model can be determined:

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

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

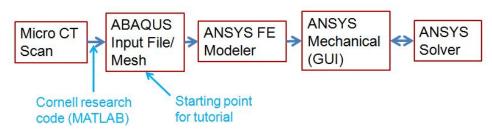
We will find the reaction force and the support and use the above relation to verify the simulation.

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.

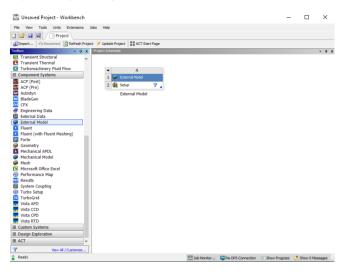


Open ANSYS Workbench:

Wa Unsaved Project - Workbench			_		\times
File View Tools Units Extensions Jobs Help					
🗋 🧉 📓 🧸 🗇 Project					
Import Refresh Project / Update Project CT Start Page					
					- 4 X
Analysis Systems					
E Coupled Field Static					
🖳 Coupled Field Transient					
🗹 Design Assessment					
Eigenvalue Buckling					
e Electric					
12 Explicit Dynamics					
S Fluid Flow (CFX)					
S Fluid Flow (Fluent)					
Harmonic Acoustics					
2 Harmonic Response					
IC Engine (Fluent)					
Magnetostatic					
1 Modal					
Modal Acoustics					
Random Vibration					
tesponse Spectrum					
Rigid Dynamics					
Static Acoustics					
Static Structural					
1 Steady-State Thermal					
1 Thermal-Electric					
Topology Optimization					
Transient Structural					
🔃 Transient Thermal					
C Turbomachinery Fluid Flow					
E Component Systems					
ACP (Post)					
ACP (Pre)					
Autodyn					
BladeGen					
CFX Y					
View All / Customize					
Ready	Job Monitor	No DPS Connection	 Show Progress 	Show 0 M	ssages

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.

we Ur	saved Project -	Work	bench				
Ele Edit View Tools Units Extensions Jobs Help							
1	🚹 🚰 🛃 🕢 Project 🍓 A:External Model 🗙						
Outline	of Schematic A2 : Exte	rnal Mo	del				
	A		в	С		D	
1	Data Source	•	Location	Identifier	•	Description	•
2	Click here to add	a file					
Browse							
			_			_	

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							
	А	в	С	D			
1	Data Source 💌	Location	Identifier 💌	Description 💌			
2	C:\Users\aa734\Downloads\C2483o_Coarsen4.inp						
3	Click here to add a supporting file						
4	Click here to add a file						

	А	В	С	
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 2 🔮 Setup input file has been loaded and External Model is ready to generate a model.

next to Setup. This means the

External Model

1

Go to Step 2: Geometry

Go to all ANSYS Learning Modules