High Resolution FE Model of Bone - Mesh

Author: Rajesh Bhaskaran, Cornell University

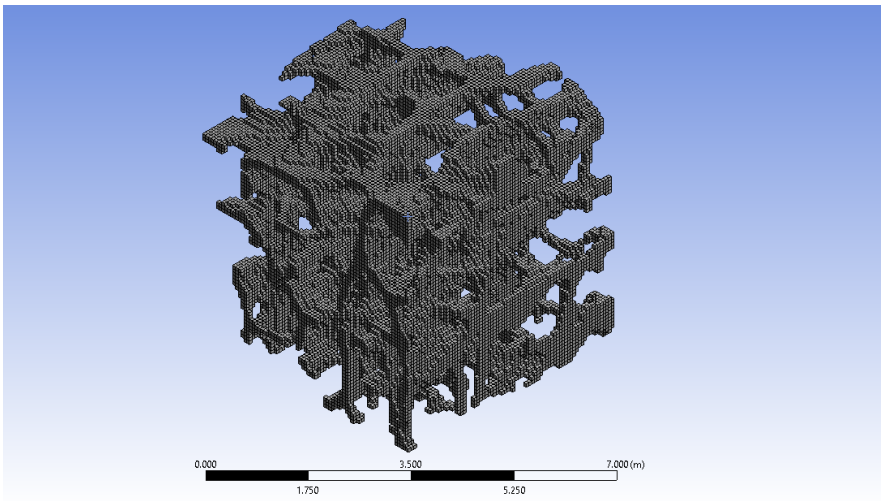
Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

Mesh

Create the mesh

Once the External Model is updated, double click on Model in **Static Structural** to launch ANSYS Mechanical.

By using External Model to import the Abaqus file, a Geometry and Mesh setup are imported into ANSYS Mechanical. To verify the file is imported correctly, click on **Mesh** > **Statistics**, and verify that Nodes and Elements match the results listed below



Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Quality	
Statistics	
Nodes	103654
Elements	60340
Model Assembly	
Read Only	Yes

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules