

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

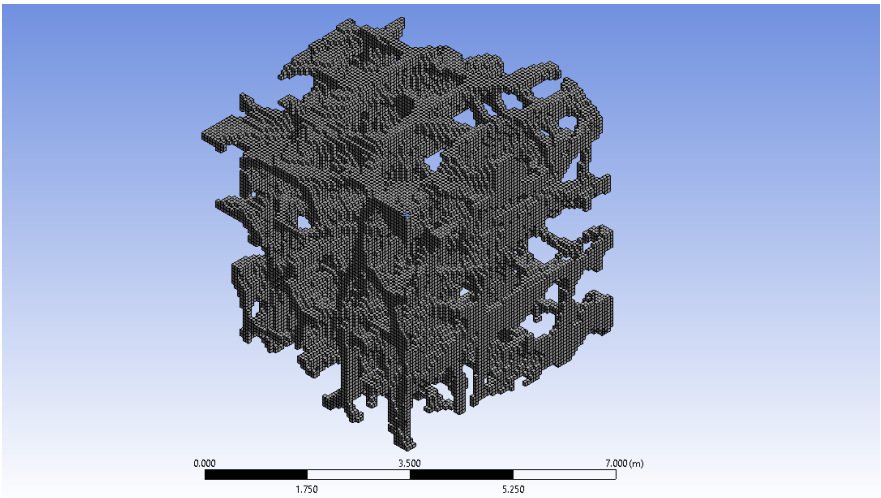
Comments

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](#)