

# 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](#)

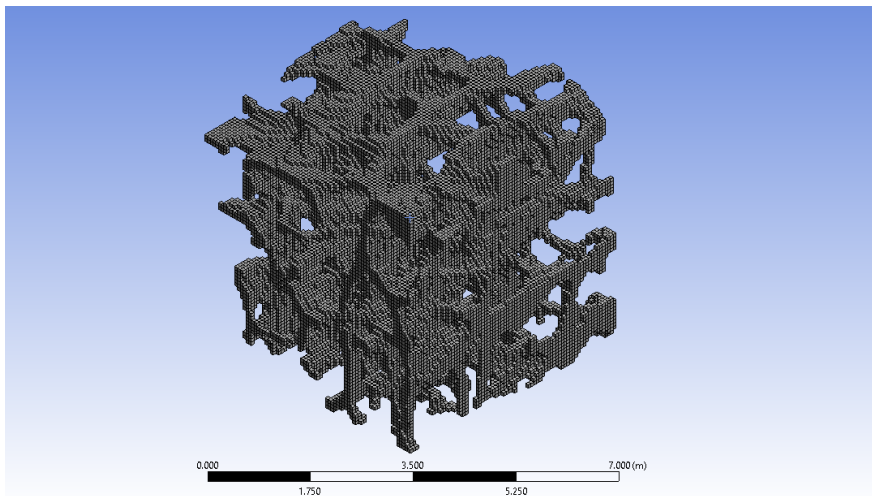
[Comments](#)

## 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	
<input type="checkbox"/> Nodes	103654
<input type="checkbox"/> Elements	60340
Model Assembly	
Read Only	Yes

[Go to Step 4: Physics Setup](#)

[Go to all ANSYS Learning Modules](#)