# ANSYS - High Resolution FE Model of Bone

Author: Rajesh Bhaskaran, Cornell University



**This page has been moved to https://courses.ansys.com/index.php/courses/high-resolution-fe-model-of-bone/**
**Click in the link above if you are not automatically redirected in 10 seconds.**

## High Resolution FE Model of Bone

Created using ANSYS 14.0

### Problem Specification

Micro-CT scans can be stacked in layers and converted into a 3 dimensional model for analysis. Previous work done at Cornell has successfully reconstructed the CT images into an Abaqus model. This tutorial will demonstrate how to import the Abaqus model to ANSYS Workbench. The equivalent stiffness will be calculated to validate the model. The CT is provided by Professor Hernandez from Cornell University.

**Acknowledgement**

Special Thanks to Sean Harvey from ANSYS Inc. for his assistance in creating this 3D model.

**Due to the heavy computation involved with this simulation, be sure to switch to a research license and a computer capable of solving with 4 cores to expedite the solver.**

**Go to Step 1: Pre-Analysis & Start-Up**

Go to all ANSYS Learning Modules