Tutorial Week 3

Importing your ScanIP model into Ansys Workbench

This guide follows the Chapter 8 tutorial from the ScanIP tutorial guide (Tutorial Week 3 – CAD Import in ScanIP).

Importing your model

After you have exported your mesh from ScanIP, you will need to import it into Ansys Workbench. To do this, open Ansys Workbench from the Start menu.

```
Start > All Programs > ANSYS 16.1 > Workbench 16.1
```

When the main window is open, drag an External Model module from the Component Systems set in the Toolbox panel into the Project Schematic. Double click on Setup to open the External Model tab.
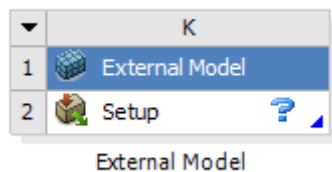
In the Outline panel, click on the button (…) under **Location** to search for your model. Select your .cdb file.

In the Properties panel, you need to change the length unit to mm so that it is the same as in ScanIP. Close the External Model tab.


Right click on Setup, and click on Update (lightning bolt) to load the model.

Linking your model to a Static Structural module

In this course, we will be mainly focussed on structural analyses. To do this for your femur model, drag a Static Structural module from the Analysis Systems set in the Toolbox panel into the Project Schematic. Link your imported model to this new

module by dragging Setup from the External Model module to Model of the Static Structural module.
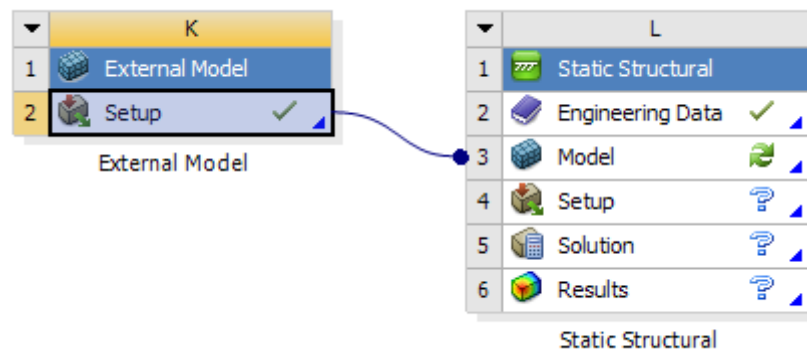
From this point on, your model should be ready for you to set up for a structural analysis. Recall from MECH3361 that you will need to define the material properties (Engineering Data), define your boundary conditions (Setup), and the choose the results you want to obtain (Solution) before solving. Please refer to the MECH3361 guide (also on the AMME4981/9981 website) if you need to become familiar with these steps again. Note that the MECH3361 guide was written up for a completely different problem. You will need to perform similar steps to achieve the same outcome for this new model.