

# 3D Conduction

Author: Samuel H. Smith, Cornell University

**3D Conduction: Tips and Tricks**

## 2. Geometry

### Tips for importing a Geometry from a CAD Package:

- Divide the geometry into suitable components. This will be an "assembly" in a standard CAD package.
- Save the CAD geometry in STEP format (.stp) and then import the .stp file into ANSYS. This is the recommended format.
- Double-click on Geometry in the project page in ANSYS, select all the parts in the tree using Ctrl-click. Then right-click and select "Form new part".
- After opening the model in ANSYS Mechanical, you will see a "Connections" folder in the tree. Make sure that the "contact" (i.e. interface) regions are all set to "bonded." This is the default for interfaces between flush entities.

## 3. Mesh

## 4. Setup (Physics)

## 7. Verification and Validation

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)