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