

3D Steady Diffusion - Geometry + Mesh

Author(s): Keith Works, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry + Mesh
3. Model Setup
4. Numerical Solution
5. Numerical Results
6. Verification & Validation

Geometry + Mesh

The following video shows you how to create the geometry using ANSYS SpaceClaim, and then how to mesh the geometry and create named selections using the ANSYS Mesher.

Make sure to save the project!

[Go to Step 3: Model Setup](#)

[Go to all FLUENT Learning Modules](#)