2D Transient Diffusion - Geometry + Mesh

Author(s): Keith Works, Cornell University

Problem Specification

- 1. Pre-Analysis & Start-Up
- Geometry + Mesh
 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.



If you don't see the "Properties" window in Workbench, select View > Properties and it should appear

Make sure to save the project!

Go to Step 3: Model Setup

Go to all FLUENT Learning Modules