

# 2D Transient 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.



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](#)