

2D Transient Diffusion - Pre-Analysis & Start-Up

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

Pre-Analysis & Start-Up

Pre-Analysis

In the *Pre-Analysis* step, we'll review the following:

- **Governing Equations:** We will review the governing equations that need to be solved in this problem.
- **Boundary Conditions:** We will go into more details about the boundary conditions that are applied in this problem.

Governing Equations

Since we are neglecting the convection terms, the diffusion equation we want to use looks nearly identical to the heat equation. Therefore, we can solve the heat equation with the flow terms disabled and then simply reinterpret the results as if we had solved the diffusion equation.

Boundary Conditions

Outer Boundary:

The outer boundary will be set to "wall". Then we will set a zero-flux boundary condition, which corresponds to an impermeable wall when discussing diffusion.

Axis:

The horizontal boundary will be set to "axis" which will impose zero gradients perpendicular to the boundary and will tell Fluent that the full domain is rotated about this axis. Note that Fluent only allows axisymmetric rotation about the x-axis.

Symmetry:

The vertical boundary will be set to "symmetry" which will impose zero gradients perpendicular to the boundary. Note that Fluent allows symmetric boundary conditions along any axis.

Please see the hand out below for more details:



Diffusion_Sim_L...ANSYS_Steps.pdf

[Go to Step 2: Geometry + Mesh](#)

[Go to all FLUENT Learning Modules](#)