## **FLUENT - 2D Transient Diffusion**

This page has been moved to https://courses.ansys.com/index.php/courses/2d-transient-diffusion-using-ansys-fluent/ Click in the link above if you are not automatically redirected in 10 seconds.

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

## 2D Transient Diffusion

Created using ANSYS 2019R2



## Learning Goals

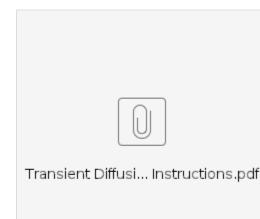
In this tutorial, you will learn to:

- Model a transient 2D axisymmetric diffusion problem
- · Visualize the results and compare to expected results

## **Problem Specification**

See HW statement.

Summary of steps (ANSYS 2019R2):



Go to Step 1: Pre-Analysis & Start-Up

Go to all FLUENT Learning Modules