FLUENT - 3D Steady Diffusion

This page has been moved to https://courses.ansys.com/index.php/courses/3d-steady-diffusion/
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

3D Steady Diffusion

Created using ANSYS 2019R2

⚠ This tutorial has videos. If you are in a computer lab, make sure to have head phones.

Learning Goals

In this tutorial, you will learn to:

- Create a model of a three dimensional diffusion problem
- Visualize the results and compare to expected results and analytical solutions

Problem Specification

See HW statement.

Summary of steps (ANSYS 2019R2):



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

Go to all FLUENT Learning Modules