

FLUENT - 2D Steady Convection

This page has been moved to <https://courses.ansys.com/index.php/courses/steady-state-laminar-convection/>
Click in the link above if you are not automatically redirected in 10 seconds.

Author: Benjamin Mullen, Cornell University

Problem Specification

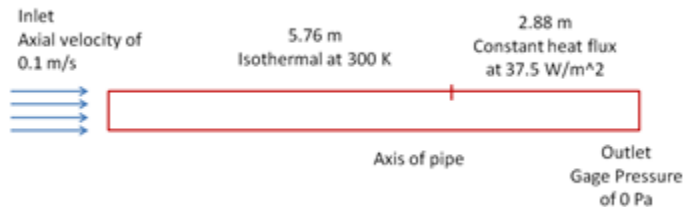
1. Pre-Analysis & Start-Up
 2. Geometry
 3. Mesh
 4. Physics Setup
 5. Numerical Solution
 6. Numerical Results
 7. Verification & Validation
- Exercises
Comments

2D Steady Convection

Created using ANSYS 13.0

Problem Specification

A fluid enters a pipe of radius 0.06 meters at a constant velocity of 0.1 m/s. The fluid has a density of 1.2 kg/m^3 , a thermal conductivity of 0.02 W/mK , a specific heat of 1000 J/kgK , and a viscosity of $1.8\text{e-}5 \text{ kg/ms}$. The first 5.76 meters of the pipe are isothermal, held at 300 K. The remaining 2.88 meters of the pipe have a constant heat flux of 37.5 W/m^2 added at the wall.



[Click here to enlarge](#)

Using ANSYS FLUENT, simulate the above flow. Calculate and plot the velocity, temperature, pressure and Nusselt number variation in the pipe.

[Go to Step 1: Pre-Analysis & Start-Up](#)

[Go to all FLUENT Learning Modules](#)