

# 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

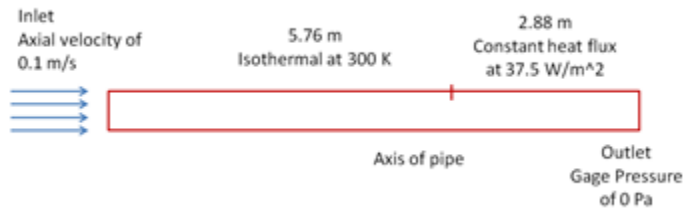
1. Pre-Analysis & Start-Up
  2. Geometry
  3. Mesh
  4. Physics Setup
  5. Numerical Solution
  6. Numerical Results
  7. Verification & Validation
- [Exercises](#)  
[Comments](#)

## 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  $1.2 \text{ kg/m}^3$ , a thermal conductivity of  $0.02 \text{ W/mK}$ , a specific heat of  $1000 \text{ J/kgK}$ , 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  $37.5 \text{ W/m}^2$  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](#)