FLUENT - Turbulent Pipe Flow

This page has been moved to https://courses.ansys.com/index.php/courses/turbulent-pipe-flow-rans/
Click in the link above if you are not automatically redirected in 10 seconds.

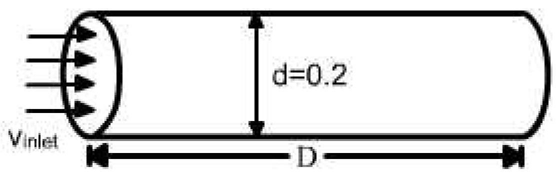
Author: Rajesh Bhaskaran, Cornell University

Turbulent Pipe Flow

Created using ANSYS 13.0

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

Problem Specification



Let's revisit the pipe flow example considered in the previous exercise. As before, the inlet velocity is 1 m/s, the fluid exhausts into the ambient atmosphere and density is 1 *kg/m*^{3}. For $\mu = 2 \times 10^{-5}$ *kg/(ms)*, the Reynolds no. based on the pipe diameter and average velocity at the inlet is

$$Re = \frac{\rho V D}{\mu} = 10,000$$

This change of viscosity has taken us from a Reynolds number of 100 to 10,000. At this Reynolds number, the flow is usually completely turbulent.

We'll solve this problem numerically using ANSYS FLUENT. Among the results we'll look at are centerline velocity, skin friction coefficient and the axial velocity profile at the outlet.

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

Go to all FLUENT Learning Modules