This module is from our free online simulations course at edX.org (sign up here). The edX interface provides a better user experience, so we recommend that you go through the module there rather than here. Also, you will be able to see answers to the questions embedded with the module there.

Laminar Pipe Flow

Created using ANSYS 16.2

Learning Goals

In this module, you'll learn to:

- Develop the numerical solution to a laminar pipe flow problem in ANSYS Fluent
- Verify the numerical results from ANSYS Fluent
- Connect the ANSYS steps to concepts covered in the Computational Fluid Dynamics section
Problem Specification

This module is drawn from MAE 4230/5230 Intermediate Fluid Dynamics at Cornell University.

Consider fluid flowing through a circular pipe of constant radius as illustrated below. The figure is not to scale. The pipe diameter $D = 0.2$ m and length $L = 3$ m. Consider the inlet velocity to be constant over the cross-section and equal to $1$ m/s. The pressure at the pipe outlet is $1$ atm. Take density $\rho = 1 \text{ kg/m}^3$ and coefficient of viscosity $\mu = 2 \times 10^{-3} \text{ kg/(m*s)}$. These parameters have been chosen to get a desired Reynolds number of 100 and don't correspond to any real fluid.

We'll solve this problem numerically using ANSYS Fluent. We'll look at the following results:

- Velocity vectors
- Velocity magnitude contours
- Pressure contours
- Velocity profile at the outlet

We'll verify the results by following a systematic process which includes comparing the results with the analytical solution in the full-developed region.

Go to Step 1: Pre-Analysis

Go to all FLUENT Learning Modules