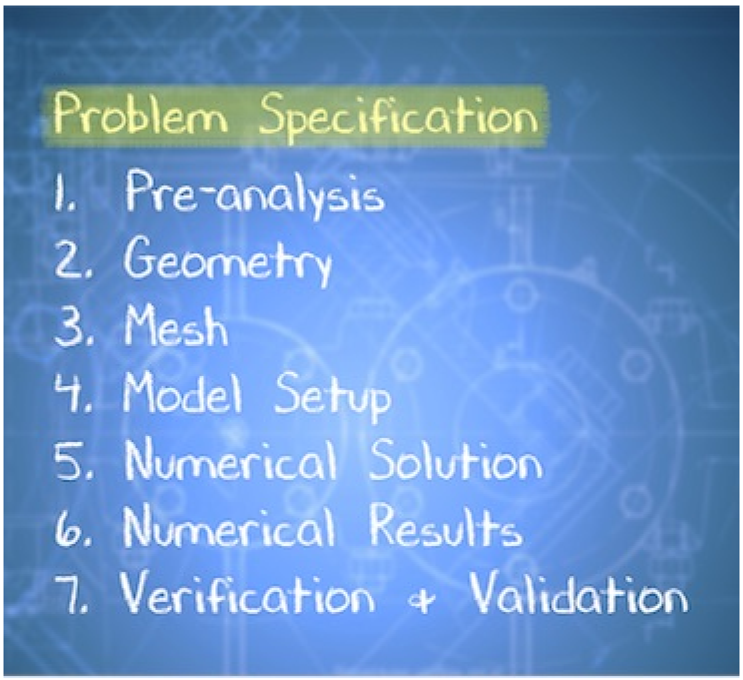
FLUENT - Laminar Pipe Flow

Author: Rajesh Bhaskaran, Cornell University

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 have moved the module there.

Laminar Pipe Flow

Created using ANSYS 16.2



Learning Goals

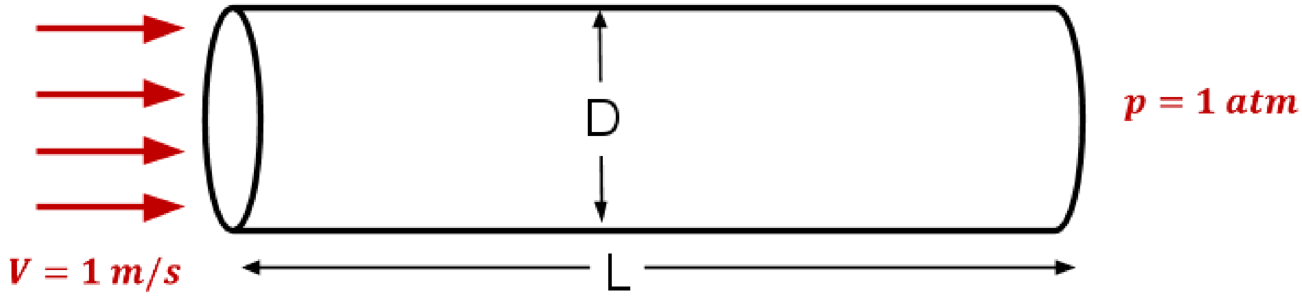
In this module, you'll learn to:

- 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 $= 1\ kg/m^3$ and coefficient of viscosity $\mu = 2 \times 10^{-3}\ kg/(m*s)$. 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 the EdX course

Go to all FLUENT Learning Modules