

FLUENT - Laminar Pipe Flow

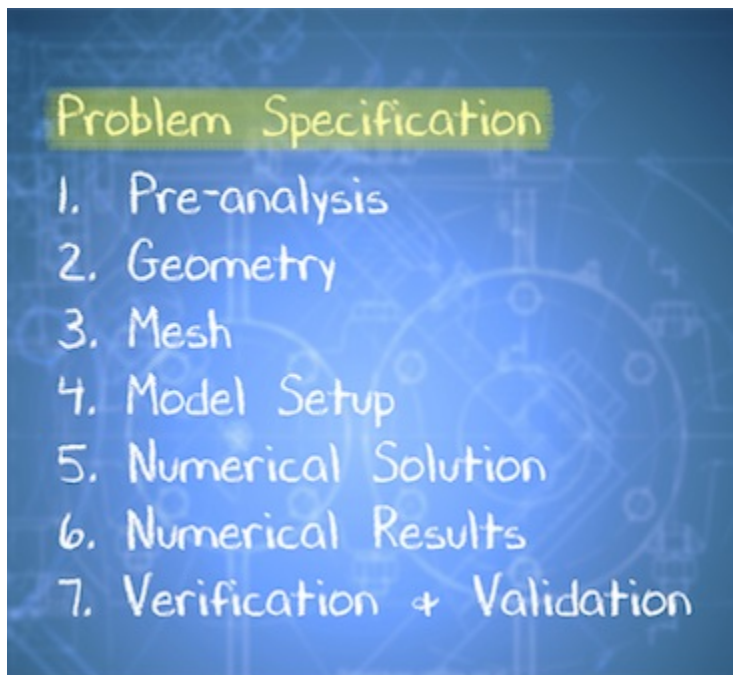
Author: Rajesh Bhaskaran, Cornell University

[Problem Specification](#)
[Exercises](#) (OLD)

i 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

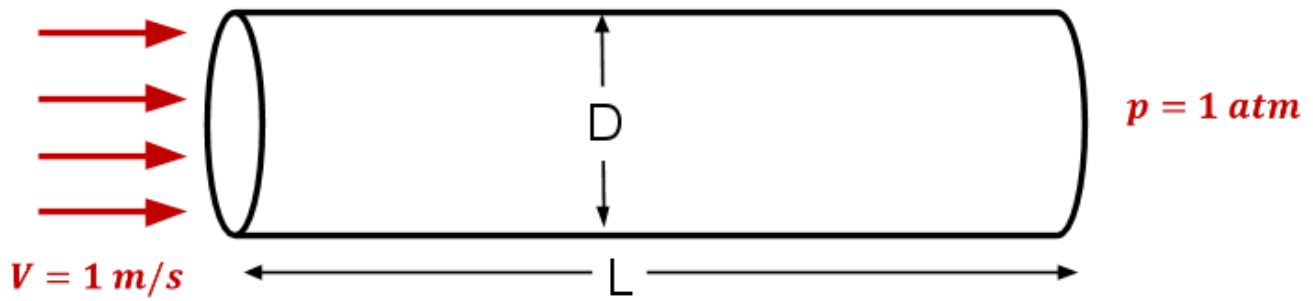
In this module, you'll learn to:

- 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 $\rho = 1$ kg/m³ 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](#)