

Sphere

Author: Rajesh Bhaskaran, Cornell University

Problem Specification

1. Create Geometry in GAMBIT
 2. Mesh Geometry in GAMBIT
 3. Specify Boundary Types in GAMBIT
 4. Set Up Problem in FLUENT
 5. Solve!
 6. Analyze Results
 7. Refine Mesh
- [Problem 1](#)
[Problem 2](#)

Problem Specification



Consider fluid flowing through a circular pipe of constant cross-section. The pipe diameter $D = 0.2$ m and length $L = 8$ m. The inlet velocity $u_z = 1$ m/s. Consider the velocity to be constant over the inlet cross-section. The fluid exhausts into the ambient atmosphere which is at a pressure of 1 atm. Take density $\rho = 1 \text{ kg/m}^3$ and coefficient of viscosity $\mu = 2 \times 10^{-3} \text{ kg/(ms)}$. The Reynolds number Re based on the pipe diameter is

$$Re = \frac{\rho \bar{u}_z D}{\mu} = 1000$$

where $\bar{u}_z$ is the average velocity at the inlet, which is 1 m/s in this case.

Solve this problem using FLUENT. Plot the centerline velocity, wall skin-friction coefficient, and velocity profile at the outlet. Validate your results.

Note: The values used for the inlet velocity and flow properties are chosen for convenience rather than to reflect reality. The key parameter value to focus on is the Reynolds no.

Preliminary Analysis

We expect the viscous boundary layer to grow along the pipe starting at the inlet. It will eventually grow to fill the pipe completely (provided that the pipe is long enough). When this happens, the flow becomes fully-developed and there is no variation of the velocity profile in the axial direction, x (see figure below). One can obtain a closed-form solution to the governing equations in the fully-developed region. You should have seen this in the *Introduction to Fluid Mechanics* course. We will compare the numerical results in the fully-developed region with the corresponding analytical results. So it's a good idea for you to go back to your textbook in the Intro course and review the fully-developed flow analysis. What are the values of centerline velocity and friction factor you expect in the fully-developed region based on the analytical solution? What is the solution for the velocity profile?



We'll create the geometry and mesh in GAMBIT which is the preprocessor for FLUENT, and then read the mesh into FLUENT and solve for the flow solution.

Go to [Step 1: Create Geometry in GAMBIT](#)

[See and rate the complete Learning Module](#)

[Go to all FLUENT Learning Modules](#)