

# Laminar Pipe Flow - Exercises

Author: Rajesh Bhaskaran, Cornell University

[Problem Specification Exercises](#) (OLD)

## Exercises

### Exercise 1: Vertical Channel Flow

[Problem Specification \(pdf file\)](#)

### Exercise 2: Laminar Flow within Two Rotating Concentric Cylinders

**Contributed by Prof. John Cimbal and Matthew Erdman, The Pennsylvania State University**

[Problem Specification \(pdf file\)](#)

The video below shows how to use ANSYS Fluent to set up and solve a problem like this.

### Exercise 3: Laminar Pipe Flow

Consider developing flow in a pipe of length  $L = 8$  m, diameter  $D = 0.2$  m,  $\rho = 1$  kg/m<sup>3</sup>,  $\mu = 2 \times 10^{-3}$  kg/m s, and entrance velocity  $u_{in} = 1$  m/s (the conditions specified in the [Problem Specification](#) section). Use FLUENT with the "second-order upwind" scheme for momentum to solve for the flow on meshes of  $100 \times 5$ ,  $100 \times 10$  and  $100 \times 20$  (axial divisions  $\times$  radial divisions).

- Plot the axial velocity profiles at the exit obtained from the three meshes. Also, plot the corresponding velocity profile obtained from fully-developed pipe analysis. Indicate the equation you used to generate this profile. In all, you should have four curves in a single plot. Use a legend to identify the various curves. Axial velocity  $u$  should be on the abscissa and  $r$  on the ordinate.
- Calculate the shear stress  $\tau_{xy}$  at the wall in the fully-developed region for the three meshes. Calculate the corresponding value from fully-developed pipe analysis. For each mesh, calculate the % error relative to the analytical value. Include your results as a table:

Mesh	$\tau_{xy}$	% error

- At the exit of the pipe where the flow is fully-developed, we can define the error in the centerline velocity as

$$\epsilon = \frac{|u_c - u_{exact}|}{u_{exact}}$$

where  $u_c$  is the centerline value from FLUENT and  $u_{exact}$  is the corresponding exact (analytical) value. We expect the error to take the form

$$\epsilon = K \Delta r^p$$

where the coefficient  $K$  and power  $p$  depend upon the order of accuracy of the discretization. Using MATLAB, perform a linear least squares fit of

$$\ln \epsilon = \ln K + p \ln \Delta r$$

to obtain the coefficients  $p$  and  $K$ . Plot  $\ln \epsilon$  vs.  $\ln \Delta r$  (using symbols) on a log-log plot. Add a line corresponding to the least-squares fit to this plot.

Hint: In FLUENT, you can write out the data in any "XY" plot to a file by selecting the "Write to File" option in the Solution XY Plot menu. Then click on Write and enter a filename. You can strip the headers and footers in this file and read this into MATLAB as column data using the load function in MATLAB.

4. Let's see how  $p$  changes when using a first-order accurate discretization. In FLUENT, use "first-order upwind" scheme for momentum to solve for the flow on the three meshes. Repeat the calculation of coefficients  $p$  and  $K$  as above. Add this  $p$  vs.  $r$  data (using symbols) to the above log-log plot. Add a line corresponding to the least-squares fit to this plot. In all, you should have four curves on this plot (two each for second- and first-order discretization). Make sure you include an appropriate legend in the figure.

Contrast the value of  $p$  obtained in the two cases and briefly explain your results (2-3 sentences).

Hint: To interpret your results, you should keep in mind that the first or second-order upwind discretization applies only to the inertia terms in the momentum equation. The discretization of the viscous terms is always second-order accurate.

[Go to Comments](#)

[Go to all FLUENT Learning Modules](#)