

Simulation of Turbulent Pipe Flow

By Michael A. Traetow

Purpose

The purpose of this simulation is use the computational fluid dynamics (CFD) software, Fluent, to simulate turbulent flow through a pipe. Comparisons between the EFD and CFD results will then be made. Specific comparison will include friction factor and velocity profile.

Simulation Design

The simulation will be performed on HP workstations using the commercial CFD code Fluent. The simulation will be modeled after the laboratory set-up of experiment number two (see *Measurement of Flow Rate, Friction Factor, and Velocity Profile in Pipe Flow*) and therefore will use the same pipe geometry as well as the same flow parameters, such as fluid density, viscosity and Reynolds number. By modeling the simulation after the experimental set-up, comparisons between the two test methods can be made.

CFD procedures for smooth pipe turbulent flow

Figure 1 illustrates the physical pipe, velocity profile and the basic grid that will be used for this simulation. The procedure to set-up Fluent to model this flow is very similar to pre-lab #1 (see *Introduction to Fluent and Simulation of Laminar Pipe Flow - CFD procedures for smooth laminar pipe flow*) with the exception of the turbulence inputs and parameters that must be specified. Each of these additional steps has already been introduced to you in the “*Additional Fluent Commands*” section of pre-lab #1.

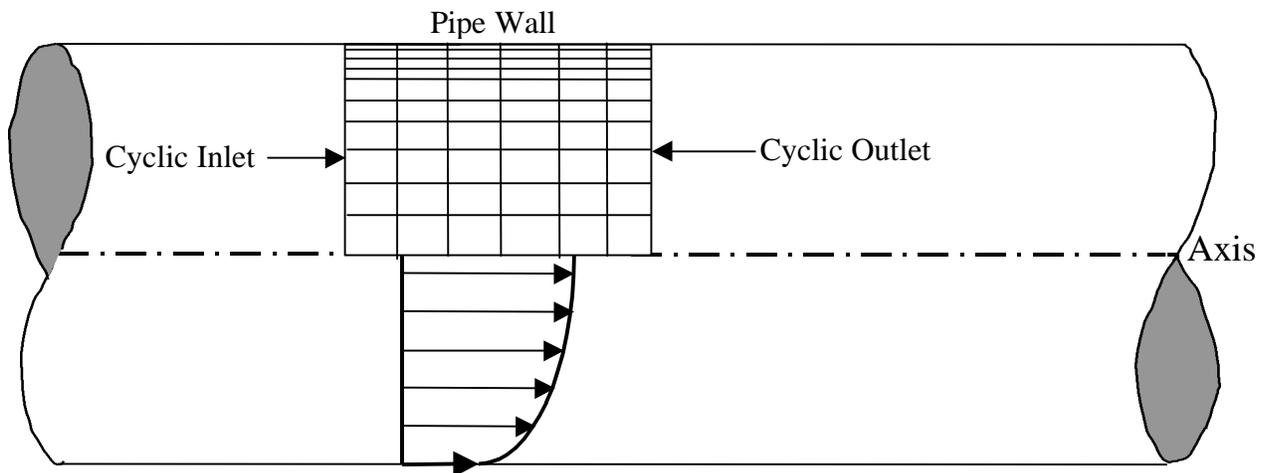


Figure 1: Fully Developed Turbulent Pipe Flow Model with Grid

Step 1: Preliminary Inputs

The inputs, which will be taken from experimental lab #2, that are required for this simulation are the following:

- Fluid viscosity
- Fluid density
- Pipe diameter
- Reynolds number
- Mass flow rate (found by using 2-D mass flow rate equation, (1))
- Dissipation (= $0.001 \text{ m}^2/\text{s}^3$)
- Energy (Turb)(= $0.01 \text{ m}^2/\text{s}^2$)

$$Q = \frac{Re \nu D \rho}{8} \quad (1)$$

Where Re is the Reynolds number, D is the pipe diameter, ρ is the density and ν is the viscosity. The derivation of this equation is available upon request.

Step 2: Grid Generation

- Set pipe length and radius
- Set boundary conditions
- Define turbulence model
- Modify grid such that it will be non-uniform
 - End Point Weighting Factor = 70

Figure illustrates the non-uniform grid generated using an End Point Weighting Factor of 70.

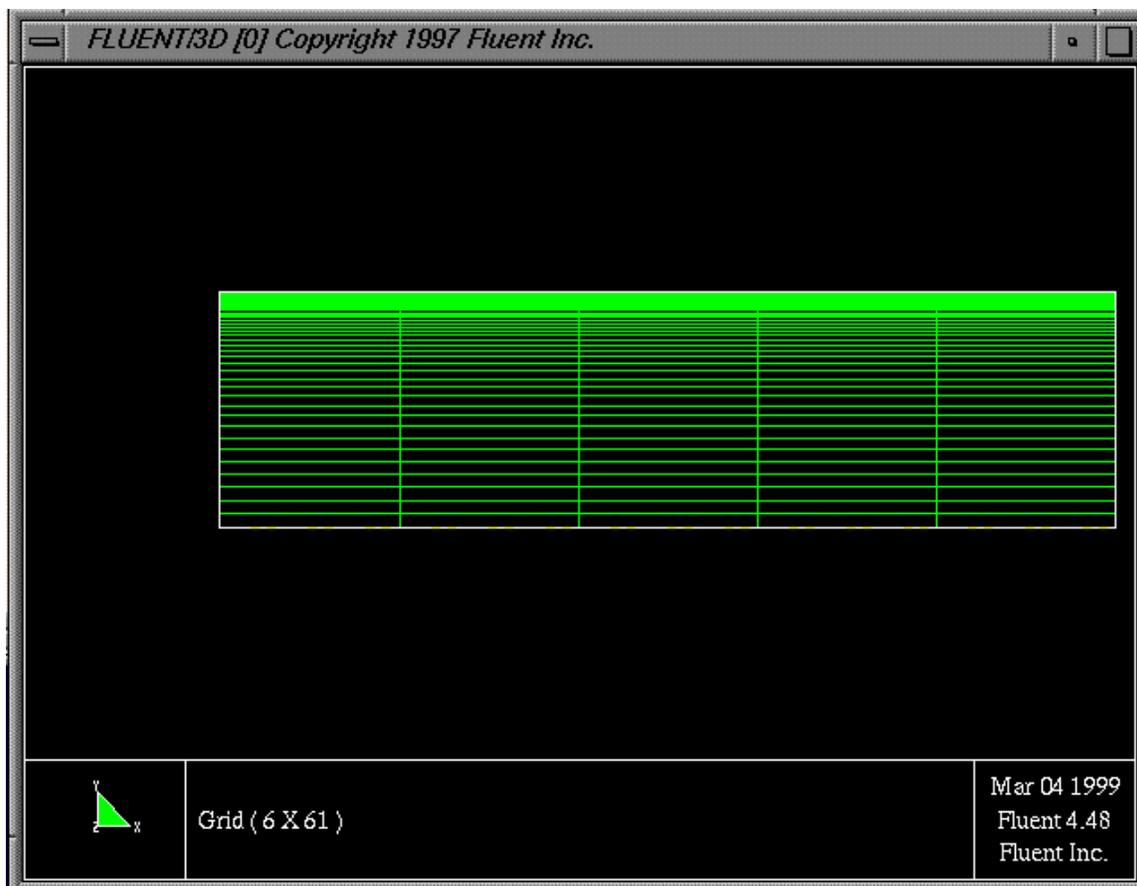


Figure 2: Non-uniform grid generated using Fluent

Step 3: Flow Parameters

The following inputs will be set at this time:

- Density
- Viscosity
- Mass flow rate
- Dissipation
- Energy (Turb)

Step 4: Solve

- Set underrelaxation
- Plot residuals
- Perform 2500+ iterations

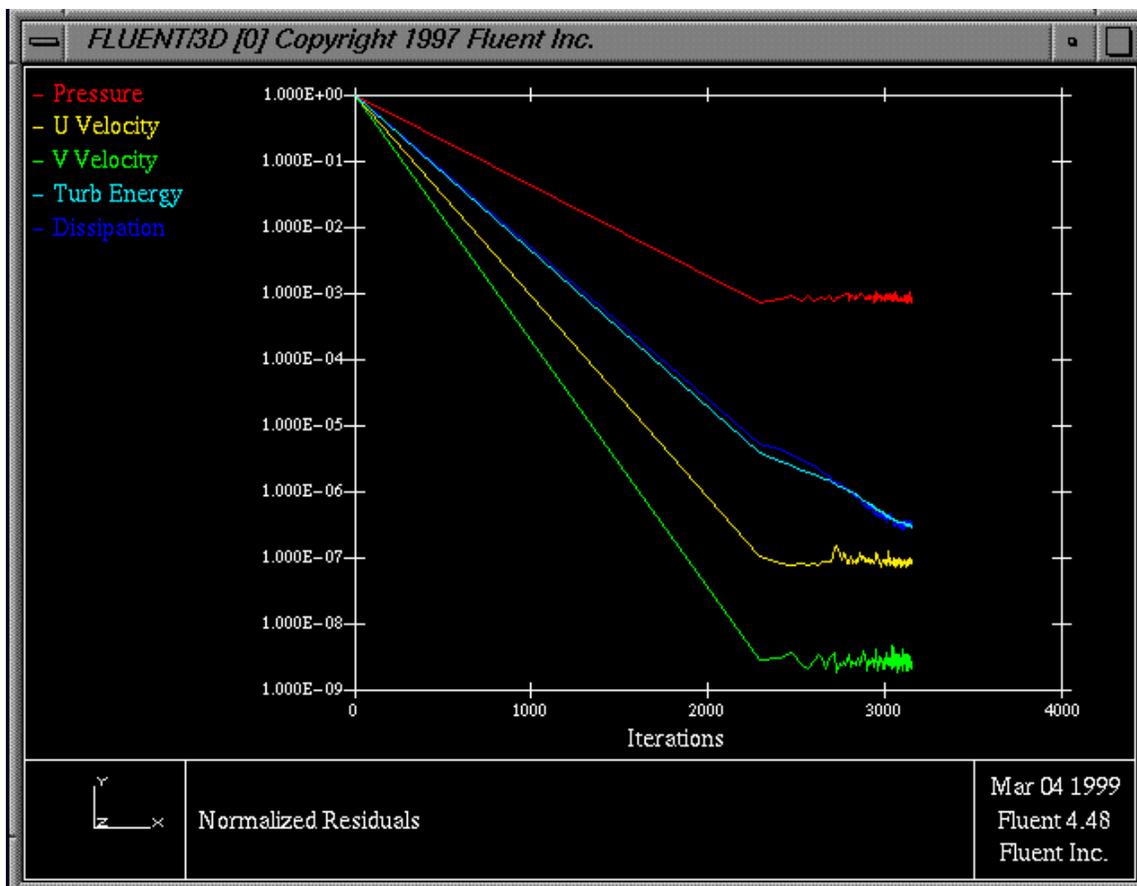


Figure 3: Plot of residuals after 3200 iterations

Step 5: Post Processing

- Display various plots/profiles
 - Save plot of residuals (as shown in figure 3)
 - Save plot of velocity profile (figure 4)
- Extract the wall shear stress from the X-dir-wall-shear-stress file

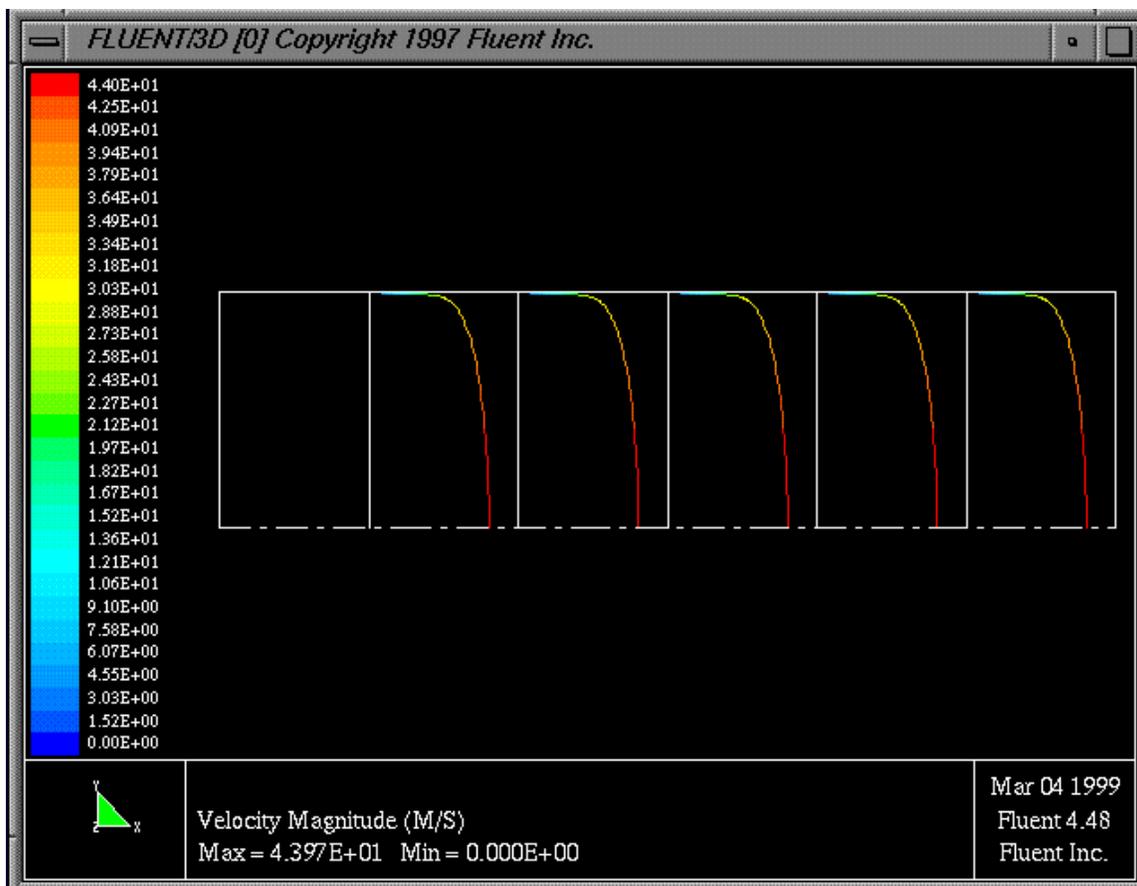


Figure 4: Velocity profile for fully developed turbulent flow in pipes

Analysis

- 1) View and save the velocity profile displayed using Fluent.
- 2) View and save a plot of the residuals.
- 3) Construct a figure comparing the velocity profile of EFD Lab #2 to the profile obtained by using CFD (*hint, save the CFD numerical velocity profile data to an excel file*). Include error bars on the experimental data.
- 4) Create a table comparing the friction factor of the CFD simulation (see equation 1 below), EFD (including the error bar) and theoretical value (as determined by the Moody diagram). Calculate the percent error between analytical and CFD, and analytical and EFD. The friction factor using the Fluent output can be calculated as follows:

$$f = \frac{8\tau_w}{r U_{ave}^2} \quad (1)$$

where τ_w is the wall shear stress, ρ is the fluid density and U_{ave} is the average fluid velocity.

- 5) Include in the above table the centerline velocity values measured/predicted through EFD/CFD.

Discussion

In the text of the report please discuss the following issues:

- 1) Velocity profiles – how does CFD compare to EFD?
- 2) Friction factors – how does CFD compare to EFD?
- 3) Discuss reasons for differences in the above values (e.g., effect of grid, iterative non-convergence, turbulence modeling errors, etc.)

Report Format

The lab write-up should be brief yet describe fully what you feel are the main points of the lab. The report should be organized such that the purpose is stated, the procedure followed is highlighted and the results are presented, compared and discussed accounting for any differences in the results of the data acquisition methods (EFD & CFD). Finally the report should end with your conclusion and recommendations. The point break down is listed below.

Introduction	10 points
Procedure (prelim. inputs, grids, flow parameters)	10
Presentation and Discussion	
CFD velocity profile	5
CFD plot of residuals	5
Velocity profile comparison	10
Table of friction factors	15
Discussion	20
Conclusion and Recommendations	15
Presentation of paper (including spelling, grammar, and appearance)	10