Session 3666

Using Computational Fluid Dynamics to Excite Undergraduate Students about Fluid Mechanics

David Pines

College of Engineering, Technology, and Architecture
University of Hartford

Abstract

Computational fluid dynamics (CFD) has been included in the junior-level Thermal-Fluids Engineering course at the University of Hartford. The laboratory modules consist of analyzing entrance length region of a pipe, a sudden contraction, and an orifice using Fluent 6.1. Two-dimensional mesh files are given to the students because it is felt that students should concentrate on understanding the fluid flow characteristics and not spend time learning how to create and mesh the models. Students are required to enter input parameters such as viscous model, fluid properties, and boundary conditions. The system's velocity and pressure characteristics are then analyzed using vector, contour, and x-y plots. Feedback from students has indicated that the fluid visualization post processing tools (i.e., vector and contour output plots) gets them interested in the project and motivates them to do a thorough analysis of how changes in Reynolds number affects the fluid characteristics of the system. Furthermore, it is felt that an early introduction to CFD may inspire some students to take more advanced fluid mechanic courses or go to graduate school.

Introduction

At the University of Hartford, the civil, biomedical, and acoustical engineering students take a 4-credit Thermal Fluid Engineering course in their junior year. The fluid mechanics part of the course covers topics such as fluid properties, fluid statics, continuity equation, momentum balance, energy balance, pipe flow, and flow over bodies. Demonstrations supplement the lectures by providing students an opportunity to see first- hand various aspects of fluid flow. However, most undergraduate students are not aware of the power of computational fluid dynamics (CFD) for visualizing, analyzing, and designing fluid/thermal systems. It is these elements, especially visualization of fluid flow, which can get students excited about fluid mechanics.

The basics of computational fluid dynamics are first introduced in a one hour lecture and then students work in-groups in a computer classroom for two class hours learning how to use the CFD software (Fluent 6.1). Students are given a mixing elbow tutorial where they learn how to enter the required inputs to run a simulation, the post processing tools available to view the output, and several techniques that can be used to refine the numerical solution. The CFD laboratory modules are then done outside of class and students are encouraged to discuss their results and any software problems with the instructor. By including CFD in the Thermal-Fluids course, it has meant that less time is now spent on the in-depth analysis of power and refrigeration cycles.

Sample CFD Laboratory Modules

Entrance Length Analysis

The analysis of a horizontal, constant diameter pipe provides the students an opportunity to better understand the fundamentals of fluid flow. A section of the mesh used for the 150 ft length - 1 ft diameter pipe is shown in Figure 1. A finer mesh is used near the wall to better capture boundary layer effects. For this lab module, students input a uniform velocity profile at the inlet of a straight pipe and investigate how the "no-slip condition" and fluid viscosity causes the velocity profile to change until it becomes fully developed. Each student group is given two laminar and two turbulent flow cases to analyze and is required to include the following in their laboratory report.

- Velocity vector plot of the entrance and exit of the pipe
- Velocity profile plots at various cross-sections to show development of fully developed flow
- Comparison of CFD modeled hydrodynamic entry length with approximate hydrodynamic entry length for laminar flow of 0.06ReD and for turbulent flow of $4.4D(Re)^{1/6}$
- Static pressure plot of the pipe
- Comparison of head loss calculated using Darcy friction factor with that predicted by the model
- Comparison of laminar and turbulent flow characteristics
- Comparison of how different Reynolds Number within a flow regime affects the flow characteristics.

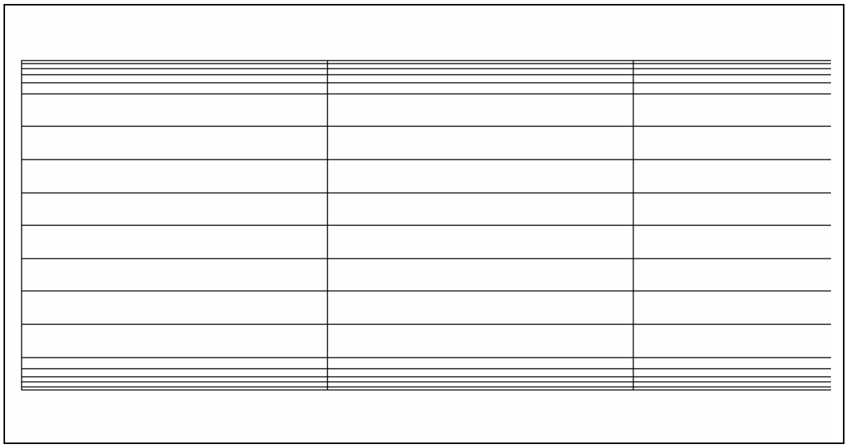

Figure 1. Section of Pipe Mesh for Entrance Length Analysis

Comparison of laminar and turbulent vector plots reemphasizes the difference in velocity profiles in these different flow regimes. Even thought the laminar parabolic profile and “flatter” turbulent profile with sharp drop near the wall are taught in lecture and explained in the textbook, the colorful vector plots appear to have greater impact on the student’s physical understanding of the subject.

One of the more thought provoking analyses was to determine the model’s prediction of entry length. The project teams took several different approaches. One approach was to create a centerline and plot magnitude of velocity as a function of pipe length (Figure 2). The entrance length was determined when the velocity remained constant. Another innovative approach was to plot the radial velocity component (y-component in the pipe’s 2-D coordinate system) as a function of pipe length (Figure 3). The point at which the radial velocity component went to zero was then used to estimate the entrance length.

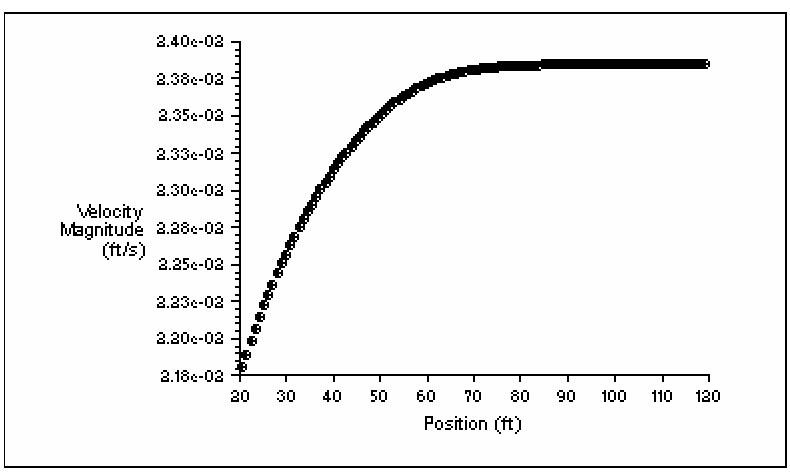

Figure 2. Determining Entrance Length by Analyzing Change in Velocity Magnitude

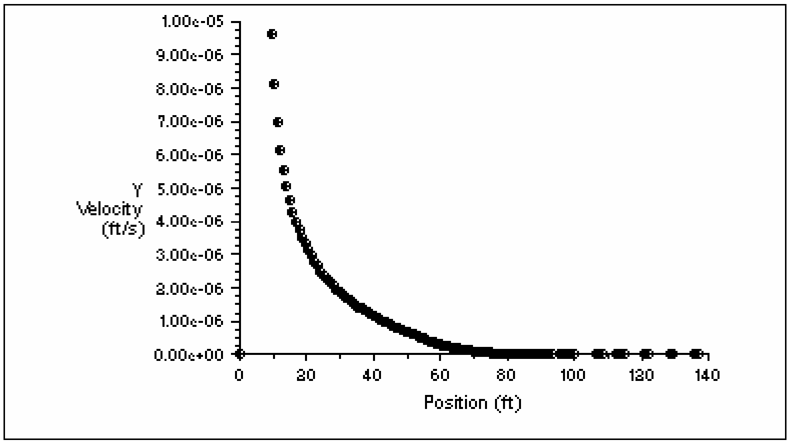

Figure 3. Determining Entrance Length by Analyzing Radial Velocity

The plot of wall static pressure as a function of pipe length for laminar flow case provided a good visual example of the relationship between head loss, shear stress, and radial velocity. Students learn in lecture that head loss is directly proportional to pipe length for fully developed flow. However, the static pressure plot as shown in Figure 4 shows a non-linearity at the inlet of the pipe. In their lab reports, the students are expected to explain this non-linearity by analyzing the velocity cross section profiles throughout the pipe. Students should observe that the velocity gradient at the wall of the pipe is greater at the inlet of the pipe then when the flow becomes fully developed. They should then conclude that this results in the higher initial head loss.

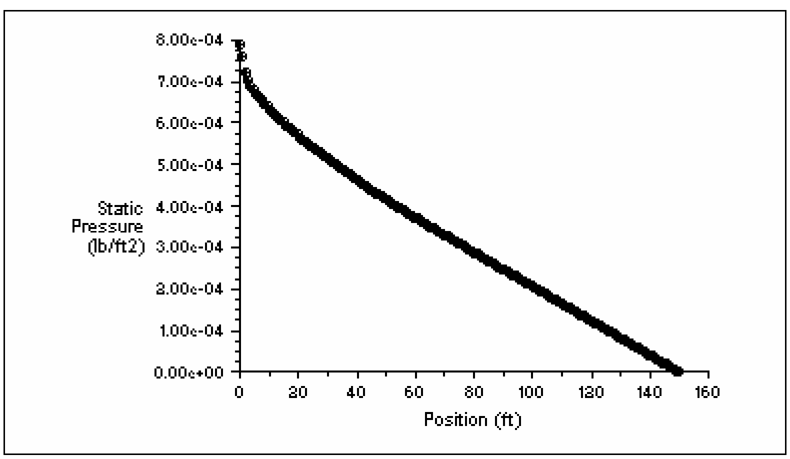

Figure 4. Static Pressure Profile along the Pipe Wall

Orifice Plate Analysis

The indirect measurement of volumetric flow rate using an orifice plate is covered in lecture and in the laboratory course. A CFD laboratory module was added so students could better visualize the flow patterns upstream and downstream of the orifice. The mesh around the orifice plate that was used for this analysis is shown in Figure 5. Each group is given two velocities to analyze and is required to include the following in their laboratory report.

- Velocity vector profile before and after the orifice plate
- Static pressure plot at the wall of the pipe
- Static pressure plot on the front and backside of the orifice plate
- Static pressure contour plot around the orifice plate

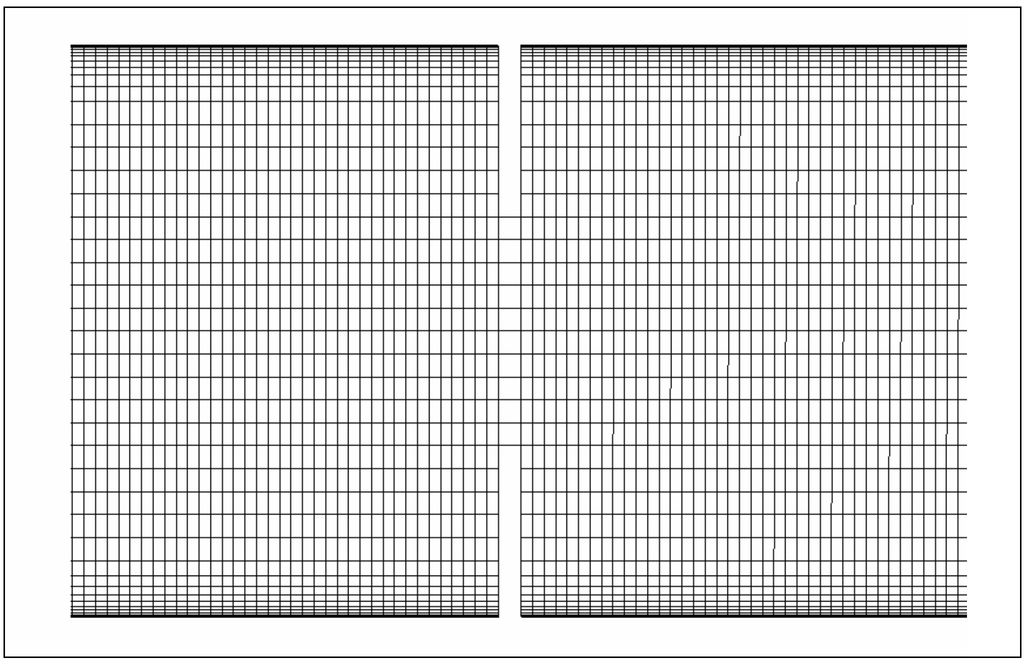

Figure 5. Mesh used for Orifice Analysis

The students were most intrigued by the vector plot showing the water flowing through the orifice plate and the recirculation pattern downstream of the orifice plate (Figures 6 and 7). The plots illustrate how the flow separates at the corners and is forced into the vena contracta region in the center of the orifice plate. This level of insight about the fluid flow patterns can not be achieved in lecture or by doing an orifice calibration experiment.

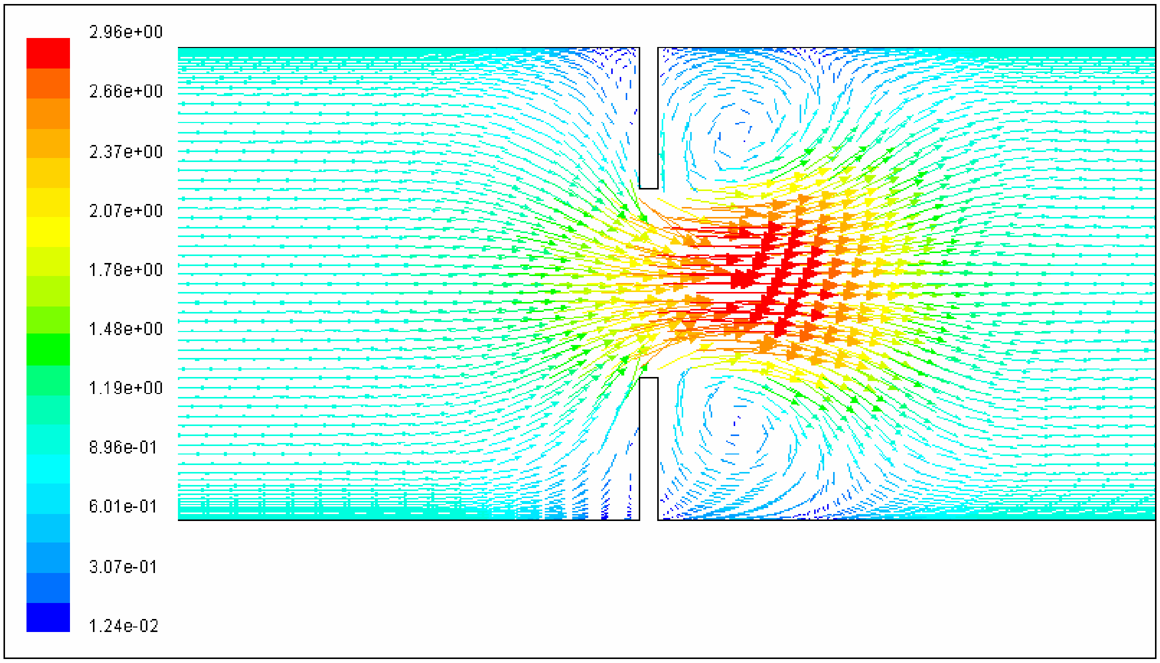

Figure 6. Velocity Vector Plot Upstream and Downstream of Orifice Plate

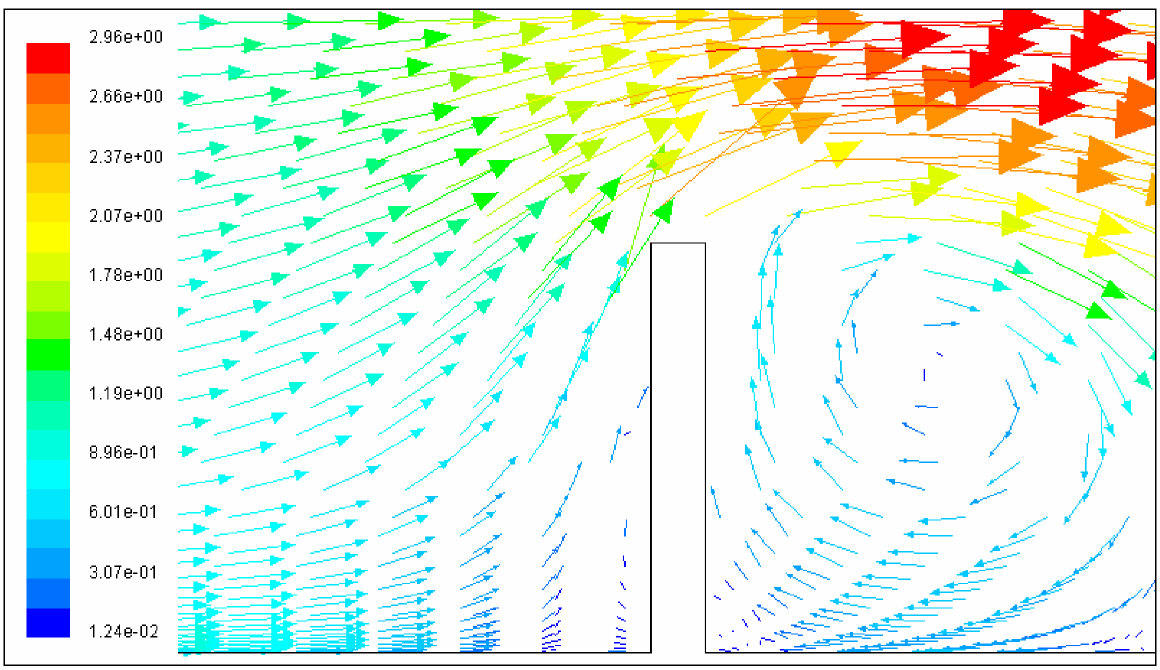

Figure 7. Magnified View of Orifice Plate Flow Characteristics

An example of the static pressure plot at the wall of the pipe is shown in Figure 8. As part of the laboratory module, students were required to investigate design guidelines for location of pressure taps. This analysis was designed to emphasize how the orifice coefficient can change based on the location of the orifice taps.

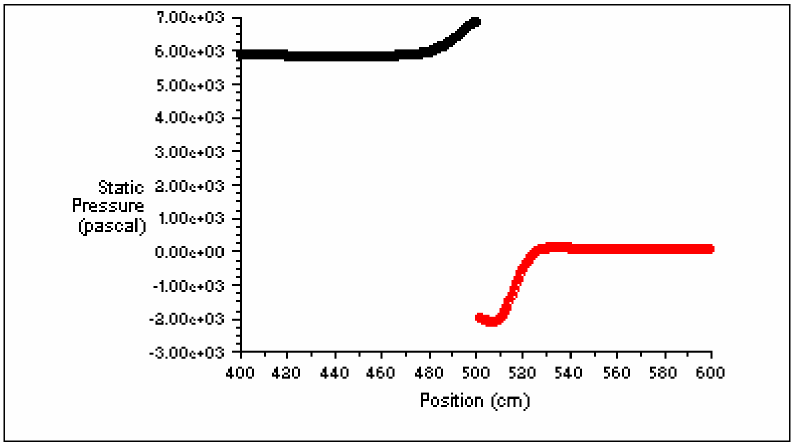

Figure 8. Static Pressure at Wall of Pipe Upstream and Downstream of Orifice Plate

Use of CFD in Senior Level Design Projects

Introducing CFD modeling in the junior year gives the students an opportunity to use Fluent for design and research projects in their senior year. Also, the students are more apt to take the finite element course as a professional elective where they will learn about all aspects of CFD modeling. One project performed by a group of Civil Engineering students for their environmental engineering project was to analyze the fluid flow characteristics of two different sedimentation basins. Students were primarily interested in how the inlet flow structures (diffusion wall and sluice gates) and outlet flow structures (weir and orifice) affected the desired plug flow characteristics. Because students already had a basic understanding of Fluent, they were then able to go one-step further by including a discrete phase model where different diameter and density particles were injected into the calculated fluid flow field.

Assessment of Including CFD in the Introductory Fluid Mechanics Course

The tools used to assess the CFD laboratory modules were the Instructor and Course Appraisal questionnaire given to the students at the end of the semester, informal student feedback, and the quality of the student's lab reports. The questionnaire was not specific to the CFD laboratory modules so only general comments were obtained. For example, students stated that the CFD laboratory modules were a major strength of the course and that they thought that being introduced to Fluent contributed to their professional development. Informal student feedback was very positive. "Wow, that's cool" type of comments were made by the students during the tutorial when students first learned how to use the software. This enthusiasm appears to have carried through to the lab modules where students asked good, probing questions and submitted lab reports that were very well done.

Conclusions

Post processing capabilities of CFD software packages provide good fluid flow visualization tools that catch student's interest. Even for simple flow systems (e.g., straight pipe), students are intrigued by the output and are more apt to go back to their lecture notes or textbook to better understand the phenomenon. Also, the analysis of a sudden contraction and orifice plate provided the students an opportunity to observe how the fluid flow field is affected by changes in geometry. With this experience, students can easily see the power and usefulness of CFD for designing fluid/thermal systems.

The current CFD laboratory modules consist of internall fluid flow systems. These modules will be refined so that they converge faster and more accurately model the physical system. A CFD project consisting of flow over streamline and blunt objects will also be added to complement the external fluid flow part of the course.

Acknowledgements

The author wishes to thank the Connecticut Space Grant Program, which provided funding for the Fluent software.

Biographies
DAVID PINES
David Pines is an Assistant Professor of Civil and Environmental Engineering at the University of Hartford. He completed his Ph.D. studies in the Department of Civil and Environmental Engineering at the University of Massachusetts, Amherst in 2000. He is actively involved with undergraduate student projects on the hydraulic and chemical aspects of environmental systems.