Session # 2220

Combining Experiments with Numerical Simulations in the Teaching of Computational Fluid Dynamics

Laila Guessous, Radoslav Bozinoski, Russell Kouba and Donald Woodward
Dept. of Mechanical Engineering
Oakland University, Rochester, MI 48309-4478

Introduction

Due to the rapid increase in computing processing power and technology over the past two decades, Computational Fluid Dynamics (CFD) has become an essential tool, in addition to experimental and analytical methods, for the solution and analysis of fluid mechanics and heat transfer problems. The proliferation of commercial CFD software packages, such as Fluent®, Star-CD®, and Flow-3D®, attests to the growing use of CFD in industry. This is in large part due to its usefulness in the design process. CFD analysis can provide insight and foresight into the operation and design of fluid systems, while reducing the "test-and-build" cycle by evaluating multiple designs cost-effectively. In academia, CFD methods have traditionally been taught at the graduate level. However, CFD computer programs and packages are also increasingly being integrated into the undergraduate curriculum, serving as "virtual fluids laboratories" to teach and reinforce concepts from fluid mechanics and heat transfer [1], or incorporated into senior-level engineering course electives [2, 3].

With the prevailing perception of such commercial software packages as *mysterious black boxes*, capable of generating results such as pressure drop, drag, and velocity distributions, it is important for mechanical engineering programs to graduate students with a basic understanding of the underlying concepts, <u>capabilities</u>, *as well as* <u>limitations</u> of CFD, i.e., graduates capable of assessing the validity and accuracy of the underlying numerical techniques used in commercial codes. This is particularly important since the "colorful" results of CFD can often mislead students into trusting all of the results (correct or erroneous) that the computer generates.

With this goal in mind, a senior/beginning graduate level course on Computational Fluid Dynamics (ME 439/539) was introduced in the Mechanical Engineering Department at Oakland University starting in the Fall semester 2001. The aspiration of the course was to strike a balance between 1) the "classical" teaching of CFD, which emphasizes the physical and mathematical foundations of CFD as well as computer code writing [4], and 2) a "hands-on" approach to CFD, which focuses on solving realistic problems using existing commercial software packages [5].

One of the novel features of this course is a class project that combines a laboratory experiment with a CFD flow analysis. A series of labs/projects was developed to enable students to compare and analyze pressure and velocity measurements obtained experimentally in a wind tunnel to those generated using Fluent®, a commercial CFD software package. This paper, describes the setup and write-up of one of these experiments and class assignments, "Flow in a Venturi". Results and lessons learned from this course experience will also be discussed.

Brief Course Description

The CFD course described in this paper was first introduced at Oakland University in the Fall semester 2001. The primary aim of this 4-credit hour course was to present the physical and mathematical foundations of computational fluid Dynamics and to provide students with a working knowledge of CFD. Students are assigned regular homework, as well as several projects of varying length and difficulty. One of the project assignments involves program writing: students implement the MAC method on a staggered grid [7, 8] for a classical fluid mechanics problem. This ensures that students gain an in-depth and detailed knowledge of an algorithm that solves the governing Navier-Stokes as well as the difficulties and limitations of such methods. All other course projects use the finite volume based commercial software Fluent® [9] and its companion grid generation software, Gambit®. This allows for the solution of more realistic flow problems. The homework assignments involve both mathematical/derivation exercises, as well as small "hands-on" computer exercises involving Matlab® programs, Fluent® flow simulations, or Gambit® mesh generation. These exercises allow students to visualize and apply many of the concepts learned in class. A more detailed description of the course contents, objectives, and sample projects may be found in a previous paper [5] and the course website [6].

Experimental Apparatus

A sequence of hands-on labs/projects that combine experimental measurements with CFD simulations was developed for ME 439/539. These assignments were designed and built by the primary author and a group of undergraduate students (the three co-authors) within the framework of a semester-long ME 490 senior design project. All of these projects involv the Oakland University FLOTEK 360 Wind Tunnel apparatus.

The FLOTEK 360 (Figure 1) is a small open-loop laboratory wind tunnel apparatus designed for educational purposes. The tunnel comes with a 6" x 6" x 18" long test section which is visible through the clear acrylic top and side-walls. Laminar flow in the test section is ensured by means of a honeycomb shaped flow straightener. Air is drawn through the wind tunnel and exhausted into the room by 0.5 hp variable speed fan. A control knob allows the air velocity through the test section to be increased up to 65 mph by varying the fan RPM. A standard 16-tube manometer bank allows for pressure measurements throughout the test section. The manometer readings can then be input into a computer using an interface card and Labview® software.

Figure 1 - FLOTEK 360 Wind-tunnel (courtesy www.gdjinc.com)

The wind-tunnel manufacturer provided a number of generic objects for testing in the apparatus. These include a smooth and dimpled golf ball, an airfoil, a "racing" car, and a venturi. The venturi shown in Figure 2 comes with a set of 2 plates, each equipped with 10 pressure taps equally spaced along the centerline. The pressure taps can be connected to the manometer bank, hence providing static pressure measurements along the surface. In the case of the golf ball and car models, pressure measurements can only be made in the wake region using a separate vertical pressure rake attachment. It was decided early on that at least two CFD labs would be developed using the FLOTEK 360 experimental apparatus: one involving *internal* flow (flow in a venturi), and the other involving *external* flow (flow over an automobile). For each experiment, two different geometries were desired (i.e., 2 venturi shapes and 2 car shapes); this would provide greater flexibility, allowing the instructor to easily vary assignments from one semester to the next. In this paper, we focus on the internal flow (venturi) projects.

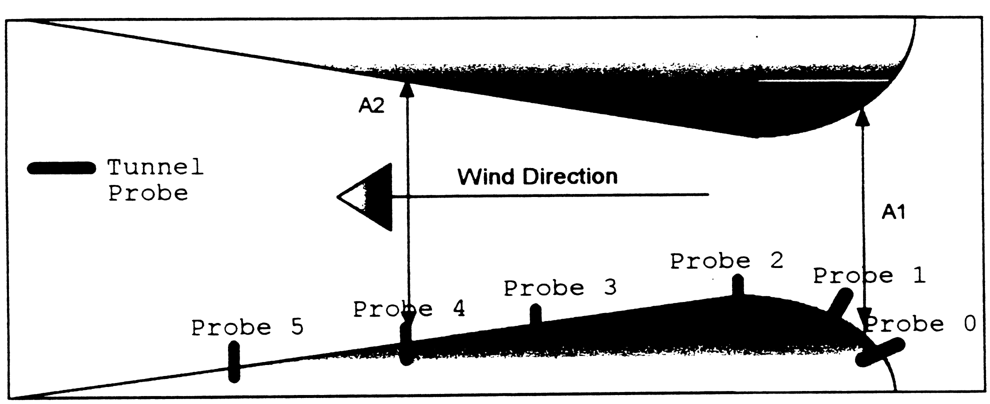

Figure 2 - Cross-Section of the FLOTEK Venturi showing the position of some of the pressure taps

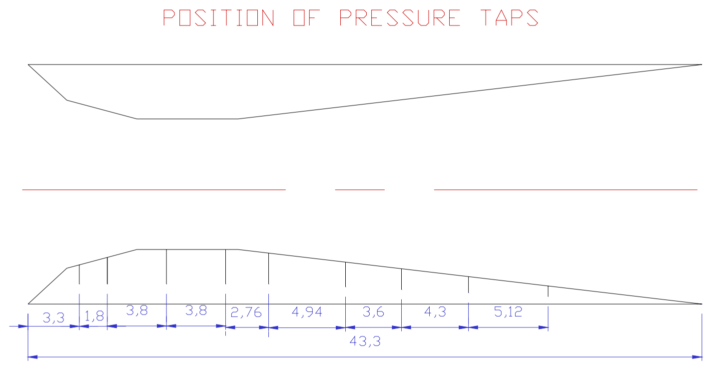

Figure 3 - AutoCAD drawing of the new internal flow apparatus, showing the location of the pressure taps

A new internal flow apparatus (Figure 3) was constructed by the undergraduate student team. This apparatus is similar to the original venturi with a few a design modifications. Instead of a smooth curve, this insert is composed of four straight edged surfaces. The points connecting these edges were intended to provide the same test-section height as the original venturi. The reasons for these choices are two-fold: first, by providing the same cross-sections at discrete locations as the original venturi, comparison of the results between the two geometries would be more meaningful; second, the simpler geometry of the new device would ease the fabrication of the apparatus both in the *real-world* (no complicated curves) and in the *virtual-world* (easier to draw on computer for novice CFD students). The pressure taps on the new apparatus were placed at similar locations as the original. The venturi insert was constructed of wood and was covered with a 1-mm thick sheet of galvanized steel. The pressure taps were constructed of 1/16" copper piping and were sanded flush with the surface. Tygon tubing was used to connect the pressure taps to the water manometers. A lab manual detailing the installation and operation of the two sets of inserts was developed and provided to ME 439/539 students.

CFD Simulation Overview

In addition to constructing the wind-tunnel inserts, the undergraduate senior design team constructed *virtual* venturi models to be used by the CFD class. Two-dimensional CAD drawings and meshes of the two internal flow apparatuses were generated using Gambit®. Examples of these meshes are provided in Figures 4 and 5. These files are provided to the ME 439/539 students with the project assignment. Alternatively, if sufficiently proficient in the use of Gambit®, the students may be asked to generate their own drawings and meshes based on provided geometrical dimensions. Meshes with either triangular or quadrilateral elements are available, allowing for mesh comparison studies. In all cases, boundary layer elements (as shown in Figure 5) are attached to the solid surfaces to more accurately capture the larger gradients in the wall region. These elements allow the use of finer meshes where locally necessary and the use of coarser meshes elsewhere in the domain. *Virtual Pressure Taps* at the same locations as those in the experimental apparatus are also provided with the mesh files. These are created by inserting virtual edges in the Gambit® geometry at discrete axial locations.

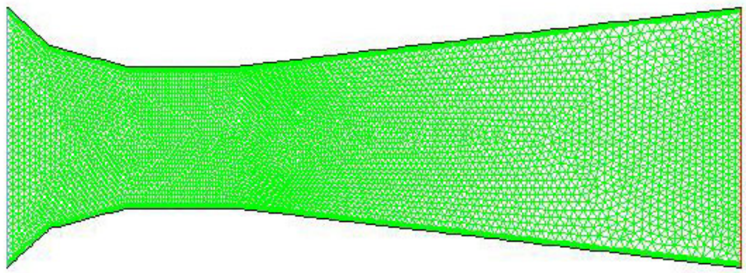

Figure 4 - Sample 2-D Gambit mesh of the Venturi model; either triangular or quadrilateral elements can be used.

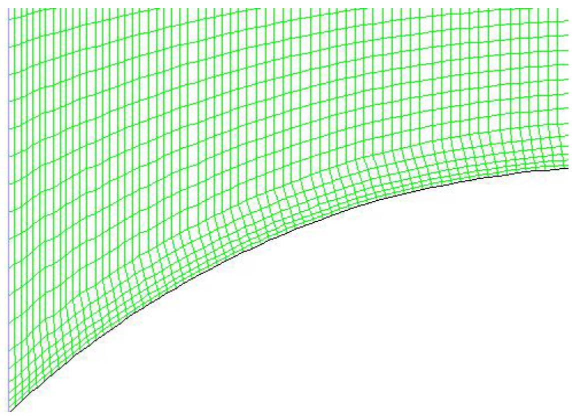

Figure 5 - Close-up of Gambit mesh showing boundary layer in the near-wall region

The mesh file is next imported into Fluent® and rescaled to reflect the actual measurement units used in Gambit® (Fluent assumes that length dimensions are given in meters by default). The fluid material is defined as air at standard conditions. Per the lab manual, the inlet boundary condition is set as *Velocity Inlet* with a value corresponding to that measured experimentally. Since the exit flow cannot be assumed to be fully-developed, a weak constraint, *Pressure Outlet*, with a zero gage pressure value, is used for the outlet boundary. Students are asked to run the simulations with different viscous models (inviscid, laminar, turbulent, etc.) and different numerical schemes for comparison. Average velocity and pressure values along the virtual pressure taps are recorded upon convergence and compared to experimental values.

Theoretical Analysis

Prior to assigning the project, the following topics are reviewed in class: 1) Hydrostatic manometer equation, 2) Bernoulli equation, and 3) Conservation of mass. These concepts are used to compute and analyze the experimental pressures and velocities.

The wind tunnel apparatus is set up to measure static pressure at various points along the venturi. The static gage pressure at a point is measured using water manometers,

$$p_{gage} = \rho g h \tag{1}$$

where ρ is the density of water, g is the gravitational constant, and h is the height of the water in the manometer.

A Pitot-static probe at the inlet of the venturi allows for the experimental determination of the inlet velocity. Neglecting elevation differences, Bernoulli's equation can be used to relate the stagnation and static pressures at a cross-section:

$$p_o = p + \frac{1}{2}\rho V^2 \tag{2}$$

where p_o is the stagnation pressure, p is the thermodynamic or static pressure, ρ is the density of the fluid, and V is the velocity of the fluid. Therefore,

$$V = \sqrt{\frac{2 * \Delta p}{\rho}} \tag{3}$$

where Δp is the pressure difference between the stagnation and static pressures.

The steady average air velocities at the other cross-sections are evaluated using the conservation of mass equation. Assuming the air to be incompressible (students are asked to evaluate this assumption by computing a maximum Mach number for the wind tunnel), the velocities at two cross-sections 1 and 2 can be related as:

$$V_2 = V_1 \times \frac{A_1}{A_2} \tag{4}$$

where A_1 and A_2 represent the corresponding cross-sectional areas.

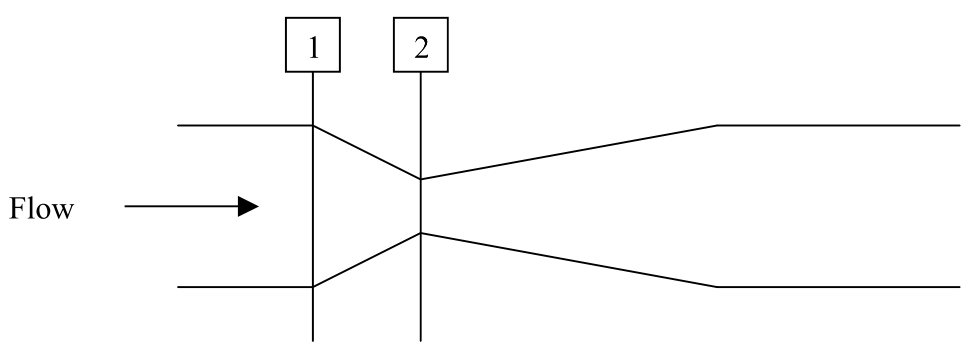

Figure 6 - Example control volume used for conservation of mass analysis

Results and Discussion

The laboratory/CFD project outlined above is conducted by students in teams of two. A number of different variations on this assignment are possible. For example, students may be asked to collect and simulate data for one insert geometry and to focus on a comparison of the experimental and CFD results, as well as on the performance of different numerical schemes. Alternatively, the emphasis could be on comparing (experimentally and computationally) the performance of the two inserts. In all cases, experimental and computational pressure and velocity data is collected and analyzed.

Table 1 lists sample velocity data collected for the original venturi insert at an inlet velocity of 28.45 ft/s. A pitot-static probe allowed the calculation of the inlet velocity using equation (3). The remainder of the experimental velocity results was computed using the continuity equation (4). These same results are shown graphically in Figure 7. Both show very good agreement between the experimental and CFD results, with errors not exceeding 3.25%. Similar results are observed at other velocities.

Figure 8 compares experimental and CFD gage pressure values. The experimental values were calculated using the manometer height readings listed in Table 1. While the results seem to agree qualitatively, the agreement is not as good quantitatively. Students learn an important lesson from this comparison. As illustrated by the large error bars, the manometers supplied with the wind tunnel and used for the pressure measurements are quite poor. An uncertainty analysis shows this error to be on the order of ±0.0045 psi. However, accounting for this uncertainty, the CFD results fall within the error bars and hence appear to be quite satisfactory for most data points.

Table 1: Experimental and Computational Velocity Results – Original venturi; V_o = 28.45 ft/s.

Data Points	Pressure [in H2O]	Experimental Velocity [ft/s]	Velocity Found using CFD [ft/s]	Error Between experimental and CFD velocities [%]
Input	0.185	28.45	28.45	0
1	0.250	41.53	42.88	3.250662
2	0.400	47.97	48.16	0.396081
3	0.450	51.38	52.11	1.420786
4	0.400	47.97	47.88	0.187617
5	0.300	44.98	44.21	1.711872
6	0.250	41.53	41.05	1.155791
7	0.240	38.58	38.41	0.440643
8	0.230	36.01	35.99	0.05554
9	0.220	34.31	33.82	1.428155
10	0.210	32.26	31.94	0.99194

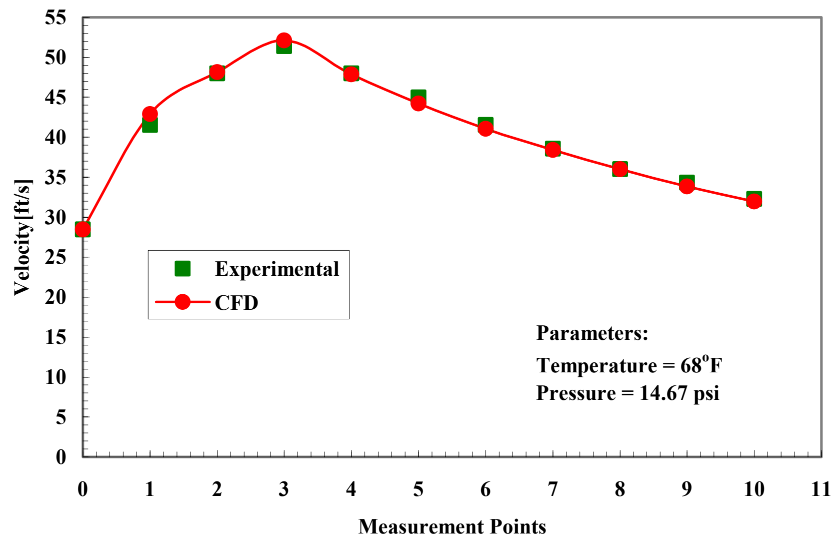

Figure 7 - Comparison of experimental (continuity) and CFD velocity results for an inlet velocity of 28.45 ft/s; original venturi

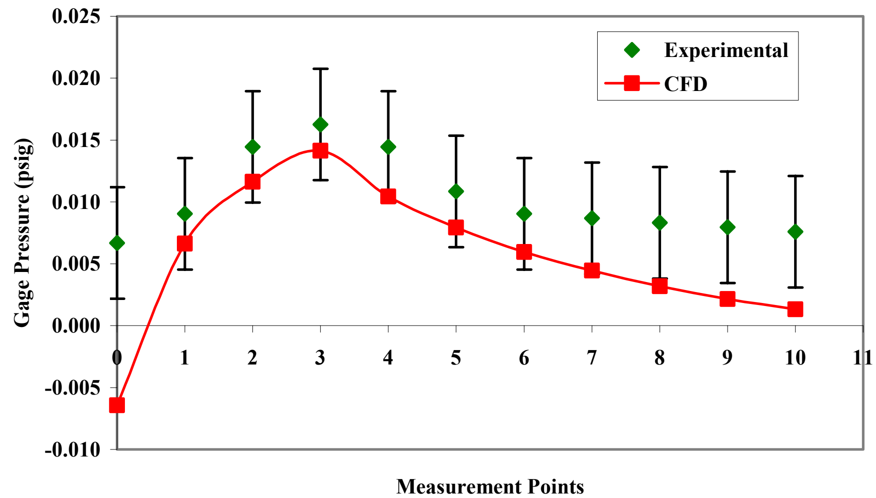

Figure 8 - Comparison of experimental and CFD gage pressure results for an inlet velocity of 28.45 ft/s; original venturi

Figure 9 compares the experimental and CFD results for the sharper-edged venturi insert at an inlet velocity of 28.45 ft/s. While the results agree to within 10%, this agreement is not as good as the previous geometry. This apparent discrepancy was quickly explained upon closer inspection of the experimental apparatus. It was determined that the actual cross sectional area dimensions for the modified venturi were slightly less than the design dimensions (which were used to generate the mesh), hence the lower CFD values.

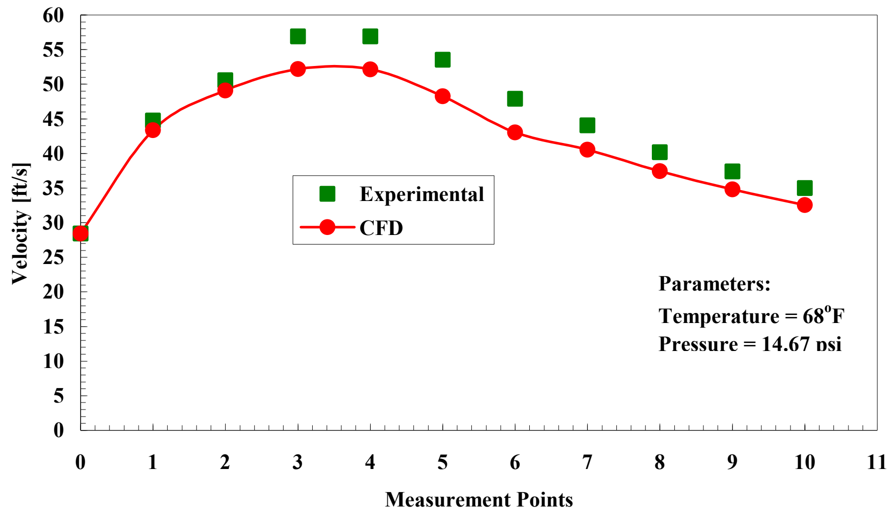

Figure 9 - Comparison of experimental (continuity) and CFD velocity results for an inlet velocity of 28.45 ft/s; modified venturi.

In addition to comparing experimental and computational results, students have the opportunity in this project to investigate the pressure distribution in the test section in more detail (Figure 10). As expected, pressure is seen decreasing as the fluid accelerates in the throat region, and increasing as the fluid decelerates in the diffuser section, leading to the development of an adverse pressure gradient.

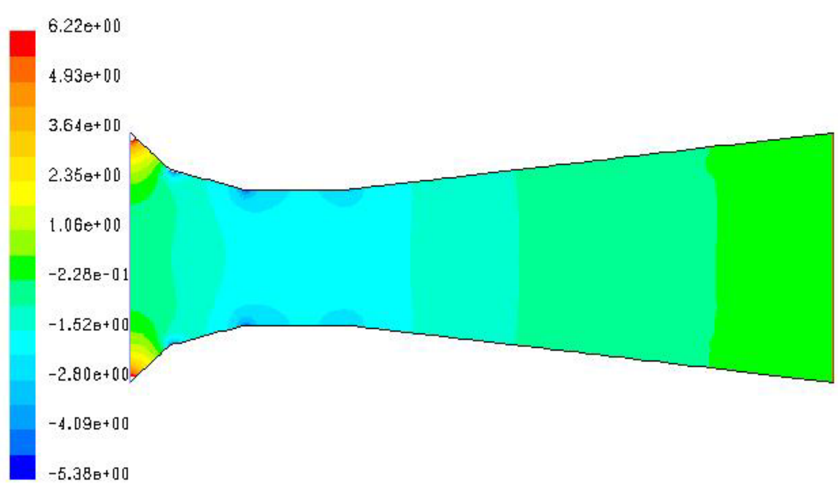

Figure 10 – Static Pressure Contour plot for an inlet Velocity of 28.45 ft/s; modified venturi. Flow is from left to right.

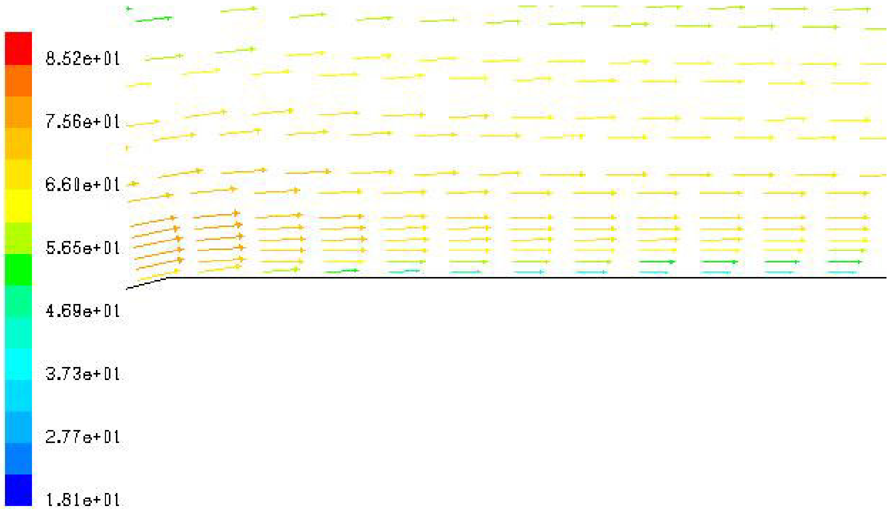

Figure 11: Velocity Vector Profile of Boundary Layer Near Throat of Modified Venturi with Inlet Velocity 28.45 ft/s.

Students are also asked to focus on the wall boundary regions (see Figure 11). To ensure that all information near the boundary layer was gathered, a finer mesh was generated in this region using Gambit's *boundary layer elements* feature. As a rule of thumb, students are instructed to use the equation for boundary layer thickness over a flat plate to approximate the expected boundary layer thickness at a given location. In general, mesh boundary layers should include a minimum of 6-7 grid points or elements to yield satisfactory results. With this estimated thickness, students are then instructed to locally refine the mesh as necessary.

Project deliverables include a report detailing the experimental and computational data collection procedure, velocity and pressure measurements/calculations for three different inlet velocities, a comparison of the two sets results, an investigation of the effect of mesh/solver/BC, etc. on the CFD solution, and a discussion of the results. Students are left to choose on their own which plots to include in the report.

Conclusions

The project described in this paper provides an opportunity for students to explore first-hand the various capabilities of CFD as an analysis tool. This project addresses multiple objectives of the course: a) students are introduced to fundamental concepts of fluid mechanics, such as the Bernoulli equation, and conservation of mass b) the use of CFD as a flow prediction and analysis tool, c) the accuracy of numerical solutions and experimental measurements, and d) mesh generation, and e) the commercial software, Fluent®. It is expected that this and other similar projects will continue to be included in the CFD course at Oakland University.

Bibliography

1. LaRoche, R.D., Hutchings, B.J., and Muralikrishnan, R., "FlowLab: Computational Fluid Dynamics (CFD) Framework for Undergraduate Education," *Proceedings of the 2002 ASEE/SEFI/TUB Colloquium*.
2. Hailey, C.E., Spall, R.E., and Snyder, D.O., "Computational Fluid Dynamics Presented in the Undergraduate Engineering Curriculum," *Computers in Engineering Journal*, Vol. XI (4), pp.2-8 (2001)
3. Navaz, H.K., Henderson, B.S., and R.G. Mukkilmarudhur, "Bringing Research and New Technology into the Undergraduate Curriculum: A Course in Computational Fluid Dynamics," Proceeding of the 1998 ASEE Annual Conference and Exposition, Seattle,WA.
4. Guessous, L., "Matlab: A Tool in the Teaching of Computational Fluid Dynamics," Proceedings of 2002 ASEE North Central Spring Conference, Rochester, MI.
5. Guessous, L., """Using Practical Projects In Teaching Computational Fluid Dynamics," Proceedings of 2002 ASEE North Central Spring Conference, Rochester, MI.
6. http://www.oakland.edu/~guessous/me539
7. Hoffmann, K.A. and S.T. Chiang, Computational Fluid Dynamics, Volumes I and II, 4th edition, Engineering Education System, 2000.
8. Ferziger, J.H. and M. Peric, Computational Methods for Fluid Dynamics, 2nd edition, Springer, 1999.
9. http://www.fluent.com

Biography

LAILA GUESSOUS (Guessous@oakland.edu) is an assistant professor in the department of Mechanical Engineering at Oakland University. She received her M.S. (1994) and Ph.D. (1999) from the University of Michigan and joined OU in August 2000. Her research and teaching interests lie in the areas of fluid mechanics and heat transfer, with an emphasis on computational methods. For more information, see http://www.oakland.edu/~guessous

RADOSLAV BOZINOSKI is an undergraduate student in the School of Engineering and Computer Science at Oakland University. He will be graduating in April 2004 with dual B.S. degrees in Mechanical Engineering (Fluids Option) and Electrical Engineering. He assisted in the development of the lab/CFD experiment as part of his ME490 senior design project.

RUSSELL KOUBA received his B.S. from the Department of Mechanical Engineering at Oakland University in December 2002 and is currently working as a Mechanical Engineer in the Tank-automotive & Armaments Research Development and Engineering Center at TACOM. He assisted in the development of the lab/CFD experiment as part of his ME490 senior design project.

DONALD WOODWARD received his B.S. from the Department of Mechanical Engineering at Oakland University in December 2002 and is currently working as a Quality Engineer with Irvin Automotive. He assisted in the development of the lab/CFD experiment as part of his ME490 senior design project.