Implementation of Particle Image Velocimetry in the Fluid Mechanics Laboratory

Ricardo Medina[1], Murat Okcay [2], Gustavo Menezes[1,*], Arturo Pacheco-Vega[3,*]

[1]Department of Civil Engineering
California State University-Los Angeles, Los Angeles, CA 90032
[2] Interactive Flow Studies Corp.
P.O. Box 748, Waterloo IA 50704
[3]Department of Mechanical Engineering
California State University-Los Angeles, Los Angeles, CA 90032
[*]gmeneze@calstatela.edu /apacheco@calstatela.edu

Abstract

The study of fluid mechanics is essential to many industrial and commercial applications. Examples include irrigation, sewer collection, water distribution, piping, heating, ventilation and air conditioning systems, aerodynamics, and power generation. Therefore, it is necessary that students have a good understanding of the concepts behind these and other applications. For this reason, the Civil and Mechanical Engineering programs at the College of Engineering at California State University-Los Angeles have two related courses in their curriculum: a theory course named CE/ME 303 Fluid Mechanics I and a corresponding laboratory course named CE/ME 313 Fluid Mechanics Laboratory I. Although the theoretical course has been developed to solve certain types of real-life problems involving fluids, unless one observes what they are, the knowledge is abstract. For this reason the Fluid Mechanics laboratory CE/ME 313, introduces the students through hands-on experiments, to several mechanisms seen in the theory course. Recently, the college of engineering through collaboration between its Center for Energy and Sustainability and Interactive Flow Studies Corporation acquired two educational interactive flow visualization systems, namely FLOWCOACH and ePIV. Flow visualization with these systems provides an excellent opportunity for visual appreciation of the complexity of flow phenomena. Visualization experiments can be used to enhance the learning experience and improve understanding on the following concepts: (i) streamlines, pathlines, timelines and streaklines; (ii) laminar and turbulent flow regimes on a flat plate; (iii) boundary layer development and its associated shear stresses, vorticity and the velocity field; (iv) separation of flows past an object; (v) laminar flow over slender bodies, airfoils, or cylinders; and (vi) the development of vortices behind a moving object, among others. The paper presents some results of visualization experiments and their corresponding computational fluid dynamics (CFD) simulations, which may be used as a basis for the development of innovative teaching modules.

Introduction

Now, more than ever, engineers are required to focus on sustainable designs that lead to more efficient systems with minimal resource consumption and reduced emissions. Required levels of efficiency can only be achieved with a profound understanding of the involved processes.

Optimization of several engineering systems such as irrigation, sewer collection, water distribution, piping, heating, ventilation and air conditioning systems, aerodynamics and power generation can only be achieved with a deep understanding of fluid mechanics (FM). However, FM is often seen as one of the most difficult core subjects encountered by students in engineering and physics. The problem stems from the necessity to visualize complex flow patterns and fluid behavior usually modeled by high level mathematics. In textbooks and classroom lectures, fluid mechanics is treated as abstract, mathematical and conceptual, even though fluid mechanics is a visual subject. Particle Image Velocimetry (PIV) and Computational Fluid Dynamics (CFD) were adopted in a Fluid Mechanics course at California State University Los Angeles to help students better understand concepts such as streamlines, streaklines and pathlines and their implications in the commonly used Bernoulli's Equation.

Particle Image Velocimetry (PIV) is a unique laser based state-of-the-art technology in fluid flow research that enables visual and quantitative analysis of the flow field; i.e., the fluid velocity as a function of both position and time. No other technology now or in the foreseeable future can do what PIV can. It is widely used in research and industry ranging from aircraft aerodynamics to improving heart implant devices. Some applications in which PIV has been used include (a) system design: where wind tunnel velocity experiments for testing aerodynamics of cars, trains, ships, aircraft and buildings have been done; (b) general research: where velocity measurements in water flows for ship hull design, rotating machinery, pipe flows, channel flows, blood flow, hydrodynamics, spray research, combustion research, wave dynamics, coastal engineering and river hydrology have been implemented; and (c) experimental verification of CFD models.

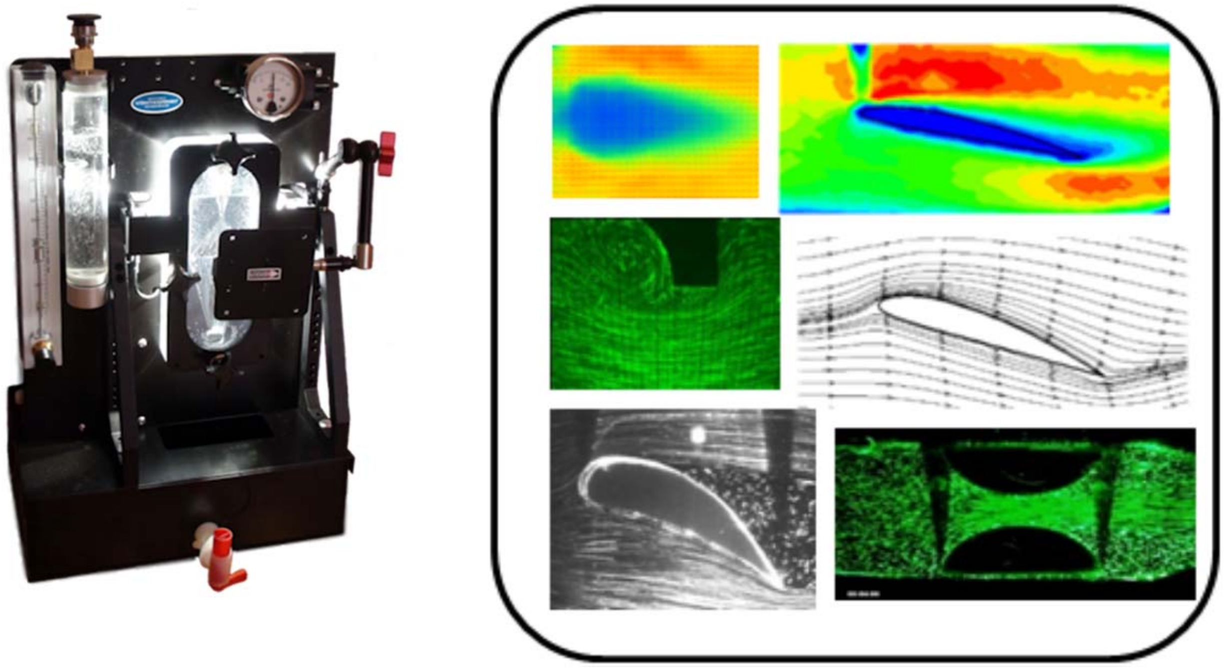
CFD (Tannehill et al., 1997) is a sub-field in fluid mechanics which attempts to solve the detailed governing equations associated with the interaction between the fluid and the body (system), and its corresponding forces via numerical methods. Though the fluid flow can be described mathematically by a set of nonlinear partial differential equations; the resulting system of equations is usually very complex and can seldom be solved analytically. This impediment is handled by means of computers and efficient algorithms that enable quantitative solutions (sometimes in a parametric form) of the fluid flow in a system without the necessity of expensive experimental equipment.

Although PIV is not a novel technology (Hopkinson, 1987; Wernet and Edwards, 1990; and Towers et al., 1991), its elevated cost has made its use for education purposes prohibitive. Recently, Interactive Flow Studies LLC (Interactive Flows Inc.) developed two educational instruments whose aim is to provide support in the quantitative and qualitative analysis of flow around objects, or flow through channels of different sizes and shapes. The instruments developed by Interactive Flows, named FlowCoach (shown in Figure 1) and ePIV, are able to capture images of neutrally-buoyant particles, which reflect light and travel with the flow, allowing for qualitative analysis of the flow field. The data analysis is carried out by means of a linux-based software known as FlowEx. The FlowEx environment uses PIV data to compute parameters of the flow, such as velocity and pressure. FlowEx also provides the option for CFD analysis of flow using Gerris, an open-source framework to solve the governing equations (Popinet, 2003). The FlowEx interface allows for straight-forward CFD analysis of computer-aided-design (CAD) models to estimate velocity and pressure vector fields. These devices enable the comparison of experimental (PIV) and computational (CFD) data.

The potential use of FlowCoach to enhance teaching of fluid mechanics is investigated in this paper. In a laboratory experiment, students used FlowCoach to acquire velocity data for water flowing around a square-shaped obstruction and then computed the pressure change along streamlines using Bernoulli's equation. The experimental data were then compared to the corresponding CFD results obtained for the same conditions.

Teaching Fluid Mechanics

The study of fluid mechanics has benefited from substantial contributions of various well-known scientists such as Archimedes, Leonardo da Vinci, Isaac Newton and Blaise Pascal. An important milestone in the advancement of fluid mechanics was the publication of "Hydrodynamica" in 1738 by Daniel Bernoulli, which led to further investigations by past and current mathematicians, physicists and engineers. As fluid mechanics continues to evolve, a question still remains. How can the body of knowledge that has been built over centuries be compiled and transferred to junior students over 4-quarter hours. Approaches that vary from regular lectures to more dynamic enthusiastic classes (Blanks, 1979) have been tried. Lately, the use of fluid visualization has been widely proposed and used, especially



(a) (b)

Figure 1. FlowCoach by Interactive Flow Studies: (a) Instrumentation; (b) Sample of results.

through the use of CFD approximations (Curtis et al., 2004; Cimbala et al., 2004; Pines, 2004; Sert and Nakiboglu, 2007; and Hu et al., 2008).

The advantage of using CFD is that students are able to visualize velocity fields and construct streamlines, streaklines and pathlines from the detail solution of the governing equations. However, in this case students are only dealing with mathematical models and often it is not easy to relate the theoretical model to the real phenomenon. Therefore, the use of PIV in conjunction with CFD is herein investigated as an approach to enhance teaching of most abstract concepts of fluid mechanics.

Experimental Setup

The FlowCoach equipment helps visualize fluid flow and can aid in the understanding of fluid mechanics concepts such as pathlines, streamlines, and streaklines. The experimental setup consists of a water/fluid reservoir, a flow model manufactured in acrylic, a camera, a set of neutrally-buoyant particles, and a PC containing the FlowEx environment (software) which processes the data. The neutrally-buoyant particles are mixed with the fluid, and travel with it as the fluid is pumped from the reservoir into the flow model section. As the fluid passes through the model section, a camera captures the fluid motion, which is visible due to the reflecting characteristics of the particles. Once the images about the flow patterns have been captured by the camera, they are transferred to FlowEx for processing. FlowCoach uses light-emitting diodes (LED) as a light source whereas the source in the case of the ePIV is a laser. In either case, the flow patterns that are observed provide a true qualitative representation of the fluid flow, as it passes through section where the model is located. The idea behind this type of experiments is that students are able to relate the theory seen in the lectures to the actual visualization of the water flowing around an obstruction. PIV, however, is also able to provide quantitative results.

The motion of the particles is recorded by the camera and divided into a sequence of frames. The current camera records at a maximum speed of 30 frames per second, thus setting the maximum number of sequential frames at 30. It is important to note that the maximum flow rates and velocities that can be resolved depend on the frame rate capability of camera used. After the set of frames has been stored, FlowEx uses the frames to compute the velocity vector field for the flow at hand. The vector field is obtained by comparing a set of consecutive frames and determining the average distance that the particles have traveled. Once the distance has been determined, it is divided by the time interval between each frame, thus producing the velocity output (velocity = distance/time). Since this process is carried out between two consecutive frames, it is called a picture pair. It is important to note that, as more picture pairs are considered, the averaged results provide a better estimate of the actual velocity of the fluid. This technique is called particle image velocimetry (PIV) and produces the average instantaneous velocity of the fluid at a given location. The PIV-velocities provide quantitative data which can be analyzed and compared to the data produced by the CFD. A computer aided design (CAD) representation of the system is integrated into the numerical scheme that solves the mathematical equations describing the fluid motion. CFD outputs velocity and pressure vector fields at each node of domain under consideration.

The advantage of using FlowCoach is that it enables qualitative (visual) analysis without the need of any computer or data processing; it also allows for flow control. The disadvantages of using FlowCoach, since it is an experimental system, are that at (1) high flow-rates (i.e., high velocities) the PIV analysis produces inaccurate results, (2) we are limited to using 30 frames/second (though the camera can be replaced for one with higher resolution), and (3) the analysis depends on the positioning of the camera which makes direct comparison between PIV and CFD more complicated. It should also be noted that the analysis is limited to steady flows.

Preliminary Tests

The experiments were conducted by a class 20 students. The students were divided into five groups of four. Each group was instructed on how to setup the experiment, perform the analysis and collect their own set of data. On average, each group spent approximately 1 hour to complete the lab. The PIV analysis run time is dependent on the level of discretization, and can range from 5 minutes to more than 1 hour. Parameters that had proven to offer good results while maintaining a relatively short computing time (15 minutes) were provided to the students. While students waited for the PIV results, Flowex was used in different computer to run the CFD analysis. Students were then asked to compare the numerical results against the experimental data.

The experimental test consists of flow visualization around an obstruction with shape of a square, and determination of the velocity field using particle image velocimetry (PIV) analysis of the flow around such an object. The numerical test, on the other hand, consists of performing a computational fluid dynamics (CFD) study of the flow under same conditions of experimental setup. Although direct comparison of velocity fields provided by both PIV and CFD is possible, one of the objectives of the class was to reinforce the learning of fluid dynamics concepts such as streamlines, pathlines and streaklines. Thus, the validation is carried out by calculating the change in pressure along a streamline using Bernoulli's equation.

Bernoulli's equation can be derived from the application of Newton's law to an inviscid fluid particle moving in a steady flow along a streamline. If the flow is also incompressible, this equation can be written as

$$\frac{p}{\gamma} + \frac{v^2}{2g} + z = \text{Constant}$$

(1)

where p is the thermodynamic pressure (also commonly known as static pressure), vv is the fluid velocity and z its position (elevation) with respect to a fixed coordinate system; γ is the fluid specific weight and g the well-known gravitational constant. During the lab session, students are encouraged to discuss on how the assumptions used in the theoretical development of Bernoulli's equation may affect the results obtained by experimental data, in which such assumptions are often not valid.

Equation (1) relates pressure changes to changes in velocity and elevation. In general, the value of the constant in the Bernoulli equation is different on different streamlines. The streamlines in the experiment can be observed qualitatively by tracking the particles flowing with the fluid (water). Figure 2 shows a 3-image sequence captured 1/30 s apart. In this sequence it is possible to notice the movement of particles, as highlighted in Figure 1. PIV uses the position at different times to develop the velocity field vectors (Figure 3), allowing for quantitative analysis of the flow.

Velocity fields obtained from CFD and PIV are compared, quantitatively, by computing the pressure differential between points along particular streamlines. Reference points in a streamline are identified through visual inspection of the CFD-based velocity field and verified using Bernoulli's constant shown in Equation (1), using output values of velocity and pressure from the numerical solver. The reference points are then located on the PIV velocity vector field as seen in Figure 4. Assuming that the points have the same location with respect to the square-shaped obstruction, the pressure difference between these two points should be the same in both PIV and CFD results. The pressure differential between these two points (positions x_1 and x_2) is calculated with a modified version of Equation (1), for relative small changes in z position [see Equation (2)], for 5 different streamlines, as depicted in Figure 5.

$$\Delta p = \frac{\rho(v_2^2 - v_1^2)}{2} \tag{2}$$

where ρ is the fluid density. The corresponding results are shown in Figure 6. The results show a reasonable agreement between PIV and CFD, with the standard deviation varying 0.89 to 0.99.

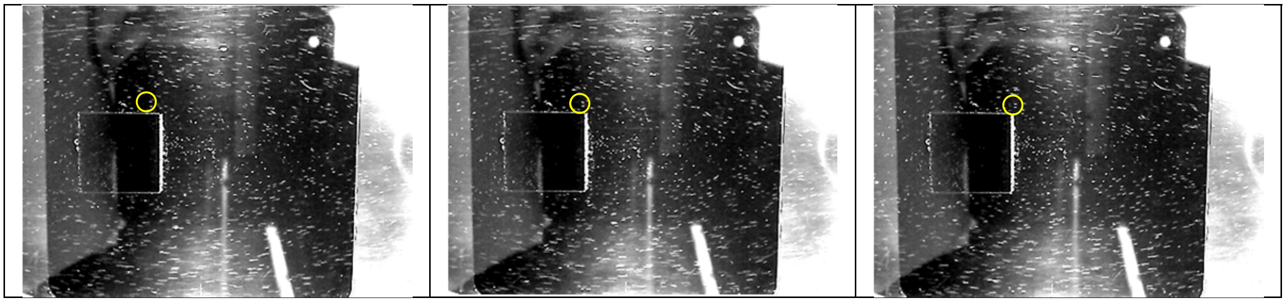


Figure 2. Sequence of images obtained by FlowCoach for PIV analysis (note: the yellow circle in the images depict the region where tracking of two particles, as they move around the obstruction, is done).

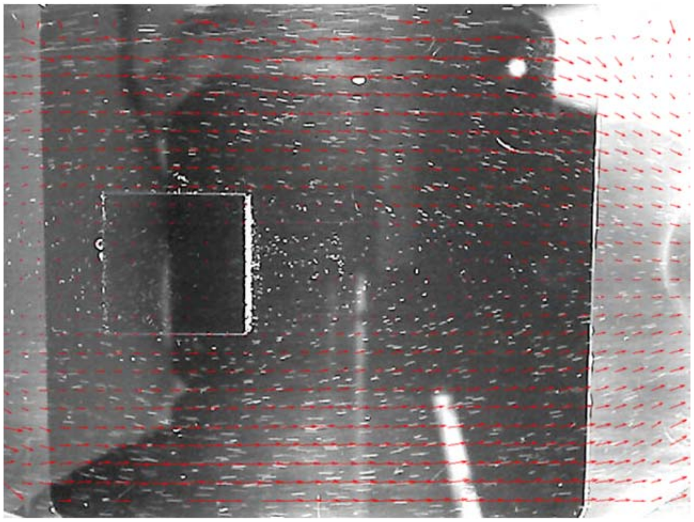


Figure 3. Velocity field obtained using FlowCoach particle image velocimetry (PIV).

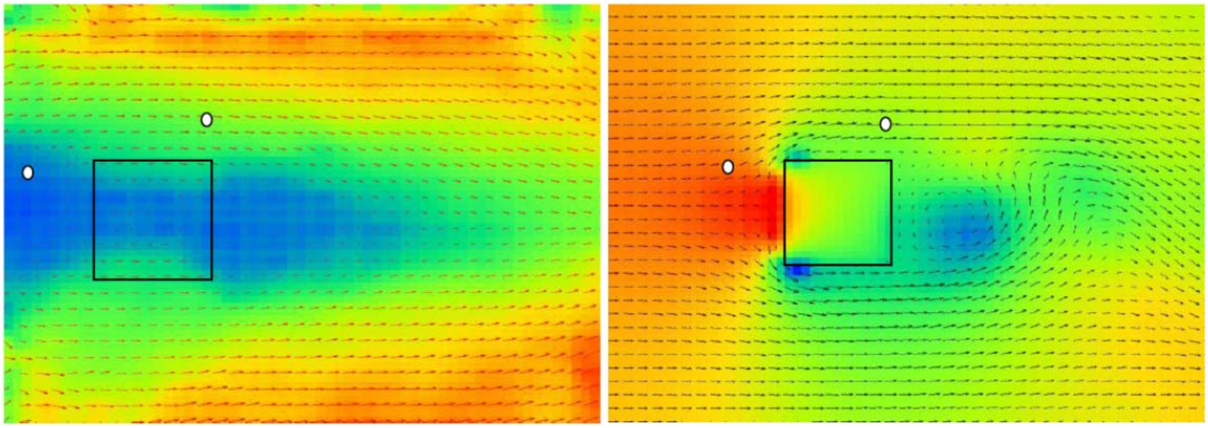


Figure 4. Velocity vector field from (a) Experimental data by PIV, and (b) Computational data by CFD.

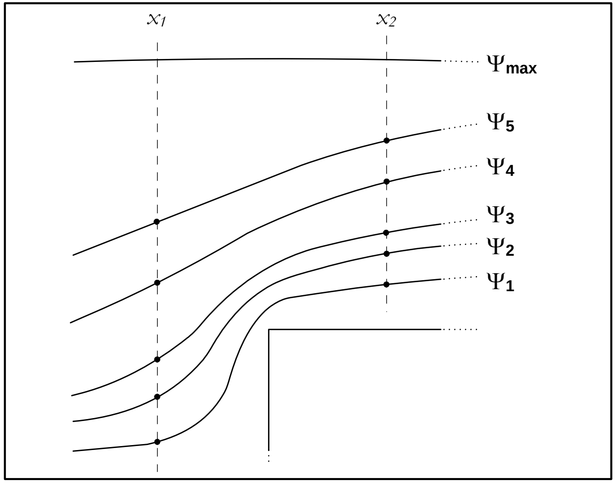


Figure 5. Sketch of streamlines of a flow through a square-shaped obstruction and location of reference points used in pressure change calculation

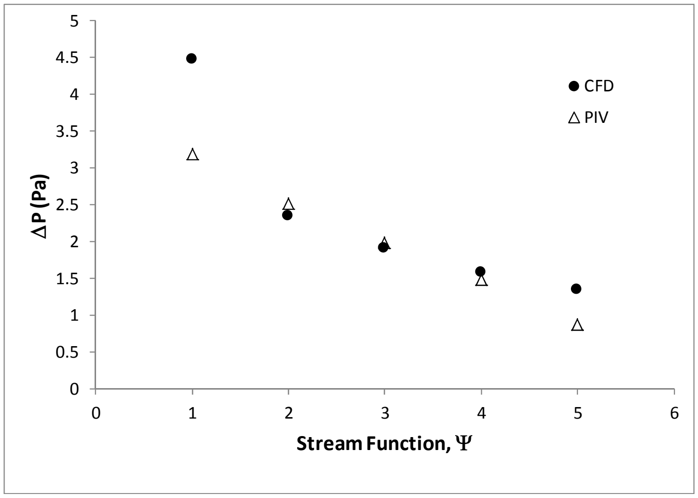


Figure 6. Comparison of local values of ΔP between two *x*-locations along streamlines.

Future Directions

The results of this preliminary study indicate that CFD along with PIV visualization capabilities may have a positive impact on students learning of abstract concepts in fluid mechanics. Thus, the next steps that will be undertaken are directed to the development of additional teaching modules and the corresponding effective assessment/evaluation tools.

Acknowledgements

This material is based upon work supported by the National Science Foundation NSF-SBIR phase II grant (IIP-0844891), and partially supported by NSF-funded CREST center (HRD-0932421). Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation.

References

Blanks, R.F., 1979, "Fluid mechanics can be fun", *Chemical Engineering Education,* vol. 13, no. 1, pp. 14-19.

Cimbala, J., Moeykens, S., Kulkarni, A., and Parihar, A., 2004, "Using flowlab, a computational fluid dynamics tool, to facilitate the teaching of fluid mechanics," *2004 ASME International Mechanical Engineering Congress and Exposition, IMECE 2004, November 13, 2004,* American Society of Mechanical Engineers, Anaheim, CA, United States, pp. 17-27.

Curtis, J.S., Henthorn, K., Moeykens, S., and Krishnan, M., 2004, "Enhancing the teaching of fluid mechanics and transport phenomena via flowlab - A computational fluid dynamics tool," *Proceedings of the ASME Heat Transfer/Fluids Engineering Summer Conference 2004, HT/FED 2004, July 11, 2004,* American Society of Mechanical Engineers, Charlotte, NC, United States, pp. 165-174.

Hopkinson, G.R., 1987, "Analytic modeling of charge diffusion in charge-coupled-device imagers," *Optical Engineering,* vol. 26, no. 8, pp. 766-772.

Hu, J., Zhang, L., and Xiong, X., 2008, "Teaching Computational Fluid Dynamics (CFD) to design engineers," *2008 ASEE Annual Conference and Exposition, June 22, 2008,* American Society for Engineering Education, Pittsburg, PA, United States.

Pines, D., 2004, "Using computational fluid dynamics to excite undergraduate students about fluid mechanics," *ASEE 2004 Annual Conference and Exposition, "Engineering Research New Heights," June 20, 2004,* American Society for Engineering Education, Salt Lake City, UT, United States.

Popinet, S., 2003, "Gerris: a tree-based adaptive solver for the incompressible Euler equations in complex geometries," *Journal of Computational Physics,* vol. 190, no. 2, pp. 572-600.

Sert, C., and Nakiboglu, G., 2007, "Use of computational fluid dynamics (CFD) in teaching fluid mechanics," *114th Annual ASEE Conference and Exposition, 2007, June 24, 2007,* American Society for Engineering Education, Honolulu, HI, United States

Tannehill, J.C., Anderson, D.A. and Pletcher, R.H., 1997, *Computational Fluid Mechanics and Heat Transfer*, Second Ed., Taylor and Francis, Philadelphia, PA.

Towers, C.E., Bryanston-Cross, P.J., and Judge, T.R., 1991, "Application of particle image velocimetry to large-scale transonic wind tunnels," *Optics and Laser Technology,* vol. 23, no. 5, pp. 289-295.

Wernet, M.P., and Edwards, R.V., 1990, "New space domain processing technique for pulsed laser velocimetry," *Applied Optics*, vol. 29, no. 23, pp. 3399-3417.