Integration of Optical Diagnostic Techniques into the Teaching of the Thermal and Fluid Sciences Laboratory Course

Final Report
For the Period of Sept. 1997 to Aug. 1999
Project No. 9750785

by
Chiang Shih
Department of Mechanical Engineering
FAMU-FSU College of Engineering
Florida A&M University and Florida State University

SUMMARY

Visual presentation has always played an important role in teaching thermal and fluid related courses because "seeing is believing". However, traditional visualization techniques do not readily provide quantitative information about the flow field of interest, therefore, these techniques have primarily been used for qualitative demonstrations. In the current project, we integrate two quantitative visualization/image-processing techniques into an undergraduate Thermal and Fluid sciences laboratory (TFSL) course. First, the Particle Image Velocimetry (PIV) technique is used to provide detailed whole-field velocity measurement. Sample projects include the flow-field characterization of a turbulent wake behind a circular cylinder and the droplet injection process of a Hewlett-Packard inkjet printhead. Second, the Laser Speckle Displacement (LSD) method is used to measure the density/temperature variation of selected flow fields, such as the shock-cell structure of a supersonic jet. All laboratory manuals are documented in html format and made available to students before the experiments. Students are required to acquire and process PIV/speckle images and compare the processed data with other theoretical and experimental results in their laboratory reports. In our opinion, by taking advantage of the visual appeal of flow visualization techniques together with the ability to provide quantitative measurements, the integration of the advanced optical diagnostic techniques into the Thermal and Fluid Sciences Laboratory not only enhances students' understanding of the subject but also stimulate their interests in this discipline. A parallel effort has also been made to extend the “seeing is believing” concept to other classes such as engineering dynamics and mechanical vibrations.

INTRODUCTION

"Thermal and Fluid Sciences" are among the most difficult curricula for undergraduate engineering students. A primary reason is the difficulty in conceptually visualizing thermal and fluid behaviors because most fluids are transparent and their motions and the associated thermal processes are invisible to human. Consequently, visualization methods usually play an important role in teaching thermal and fluid related courses because "seeing is believing". However, traditional visualization techniques do not readily provide quantitative information about the flow field of interest, therefore, they have mainly been used for the purpose of demonstration and their results can not effectively be correlated to the available theoretical and analytical solutions. In light of this, we propose to integrate two quantitative visualization/image-processing techniques into the teaching of an undergraduate thermal and fluid science laboratory (TFSL) course. Moreover, the visualization-based courseware produced in the image-processing laboratory will be used extensively in the teaching of a newly designed integrated thermal and fluid curriculum, which will also be described in the following section.

COURSE DESCRIPTION

The TFSL course belongs to a series of integrated thermal courses, which are designed to provide the students with a solid foundation in the thermal and fluid sciences with strong design content and extensive laboratory experience. The first phase of the series starts with a two-semester class sequence that integrates *Thermodynamics*, *Fluid Mechanics* and *Heat Transfer* to provide the theoretical background of the thermal engineering concepts with an emphasis on practical design applications. Innovative project-oriented "just-in-time" delivery methodology has been implemented in the teaching of this class. Frequent introduction of real-world applications and hands-on experiments, which demonstrate the physical principles learned and emphasize the connectivity between heat transfer, fluid mechanics and thermodynamics, is critical to the success of this integrated approach. The use of optical techniques and related instructional courseware provide an efficient teaching tool to achieve this goal. Second phase of the curriculum includes advanced thermal classes such as *Propulsion, Advanced Heat Transfer, Aerodynamics, Thermal-Fluid Design, Senior Design Project,* etc. These courses are intended to further enhance the student's understanding of the concept of design and provide practical experience in a laboratory environment. The TFSL course involves engineering laboratory measurements in fluid and thermal applications, including basic concepts of experiments, measurement devices and their performance characteristics; measurement of fluid and thermal properties, including pressure, velocity, and temperature; calibration procedures; design of engineering experimental systems; laboratory work and report writing. In TFSL, students are required to use these optical techniques in their laboratory assignments and final projects. Basic principles of the optics and image-processing algorithm, including the introduction of the CCD (Charge-Coupled Device) technology, are also provided in lectures. Finally, to complete the learning package, related topics in other engineering fields are integrated to provide students with a broad-based cross-disciplinary understanding of the subject.

PARTICLE IMAGE VELOCIMETRY (PIV)

Particle image velocimetry is a whole flow-field measuring technique, which provides both qualitative and quantitative information on the flow behavior [1,2]. The operation of the technique involves the illumination of the flow, seeded with small tracer particles, with a thin pulsed laser light sheet. The light scattered by the seeding particles, which follow the local fluid motion, generates a moving particle-image pattern. A typical PIV configuration is shown in figure 1. The image pair pattern can be recorded using multiple exposure photographic technique, either on films or in digital formats. The whole-field velocity information can be obtained by evaluating the distance between successive images of particles within a specific interrogation region. Digital image processing technique, using a Fast Fourier Transform algorithm, is used to determine the image separation and convert this information into local velocity data (figure 1). Please refer to Lourenco et al. [3] for a more detailed description of the scheme. Followings are descriptions of two sample projects already been implemented in the laboratory course using the PIV technique. More experiments will be added in the future.

<u>Sample Project 1: turbulent wake behind a circular cylinder in cross flow</u>

Vortex shedding behind a bluff body has been related to many important applications, for example the vortex flow meter can be used to measure flow velocity. Failure to consider this effect can also lead to engineering disasters, such as the collapse of Tacoma Narrows Bridge as a result of the wind-induced vibration. The phenomenon involves the flow separation from the surface, the emergence of shear layer instability and the generation of strong oscillatory loading on the body due to fluid/structure interaction. Traditional techniques based on single point measurement can not fully characterize the behavior of this unsteady event. However, the use of

PIV can capture, at any specific instant, the whole-field velocity and vorticity data and provides a detailed overall view of the flow field. A typical sequence of vortex shedding behind a circular cylinder is shown in figure 2. Color-coded vorticity data is superimposed on the local velocity vector plot to reveal the shedding process. It can be clearly seen that boundary layer from either side of the cylinder separates alternately and vortices are periodically released into the wake from both sides. Corresponding pressure measurements on the cylinder also indicate strong oscillations and are correlated to the PIV data. In addition, time-averaged PIV velocity data are used to describe other phenomena such as flow separation from the cylinder's surface, velocity deficit inside the wake, etc. Finally, lift and drag forces on the cylinder are estimated by performing a momentum balance using the PIV velocity data and the result can be compared to the data obtained by integrating the pressure distribution on the cylinder surface. A water towing tank facility is designed and constructed for this experiment. A detailed description of the facility is given in the laboratory manual attached at the end of the report (Appendix A). Two other experiments are being planned to take advantage of this new facility, which include: flow visualization of the flow over an airfoil at different angles of attack, boundary layer flow development over a flat plate.

Sample project 2: droplet injection process of an inkjet printhead

A new experiment concerning the droplet injection process of a HP inkjet cartridge was developed. Using stroboscopic illumination and a microscope, the near-field ink droplet injection process is visualized and the droplet velocity is measured using the PIV system as shown in figure 3. The emergence of a primary droplet and several satellite droplets can be clearly seen. The far-field droplet trajectory can be visualized using a CCD camera and back illuminating the flow using a strobelight (figure 4). The visualized results can then be compared to the theoretical data predicted by using the droplet injection velocity and the associated aerodynamic drag force acting on the droplet. This experiment is interesting since it uses advanced diagnostics techniques (PIV & image processing) on a commonly used commercial product (an inkjet printer) and illustrates several fundamental thermal principles (e.g., surface tension driven stability, aerodynamic drag, and boiling heat transfer of the thermal bubble formation).

LASER SPECKLE DISPLACEMENT LSD TECHNIQUE

The basic principle of the LSD technique to measure density has been described by Kopf [4]. When a collimated laser source is used to illuminate a piece of ground glass, a random speckle pattern is produced as a result of the interference of light scattered by the fine grain in ground glass (see figure 5). When viewing the ground glass through a flow field of interest involving density variation, the speckle pattern is displaced due to the change of refractive index inside the flow. Keeping in mind that, the change of the refractive index is directly related to the change of the fluid density by the Gladstone-Dale constant, the speckle displacements of a flow medium can be measured with reference to the quiescent and isothermal ambient background. The whole-field speckle displacement field can therefore be directly related to the variation of the density field due to the presence of the flow. The technique can thus be used to obtain density and temperature fields of compressible flows and thermally driven convective flows. A sample project using this technique is described in the following.

Sample Project 3: periodic shock-cell structure of an under-expanded supersonic jet

The density field of an axisymmetric jet, with a design Mach number of 2 operating at an under-expanded condition, is shown in figure 6. The density field is obtained by integrating the measured speckle displacement (density gradient) field for the entire flow field of interest (figure 6(a)). The presence of a periodic shock cell structure can be clearly seen in the figure. Using the LSD field data, the density variation can also be digitally reconstructed to obtain first order spatial density gradients in the entire flow field therefore producing images analogous to those obatined using traditional schlieren methods, (figures 6(b) & 6(c)). Similarly, the second order spatial density gradients analogous to tradition shadowgraph (figure 6(d)) can also be obtained via reconstruction of LSD data. The existence of a periodic shock cell structure in a supersonic jet is important since it can interact with the jet turbulence and generates intense screeching noises (screech tones). These high decibel tones can produce premature material fatigue and significant structural damage to neighboring mechanical devices. Also, the propulsion efficiency of a jet decreases significantly because of the loss of thrust due to the presence of these shocks. A homemade optical bench with an integrated optical system is set up for the operation of this experiment. The design of the optical system is flexible so that it can be used not only for the LSD method but also other traditional techniques, such as the schlieren and shadowgraph. Real-time images from the visualization results are directly relayed to students on a TV monitor for demonstration purpose. The optical bench arrangement is also included at the end of the report for reference (Appendix B).

EVALUATION AND DISSEMINATION

All experiments, accompanied with comprehensive operation manuals, are made available in electronic form either via Internet WWW access for any groups of interest and in CD-ROM format for internal usage and upon request. Demonstration laboratories will not only be presented in the laboratory class but also in all related thermal and fluids classes. Demonstration courseware with detailed descriptions of the experiments will be archived into digital formats and included in the course Web pages, which will be available not only to our students but also to the entire teaching community. A complete evaluation of the effectiveness of the program will be conducted at the end of each class through student evaluation surveys. Specific questions and comments will be solicited to continually improve the program in the future. The tabulated evaluation data from students will be collected at the end of this semester and the next semester and the results will be submitted to the NSF data base at a later date. Preliminary results from the project have already been submitted to SUCCEED (The **S**outheastern **U**niversity and **C**ollege **C**oalition for **E**ngineering **Ed**ucation) and have been disseminated through their CD ROM "Greatest Bits" project (see attached *Qualitative Flow Visualization* project as Appendix C). Currently, continued development of the visualization-based courseware is being supported by SUCCEED funds and the results will be included in the SUCCEED CD-ROM project again. I intend to submit the final package of the courseware to the NSF-sponsored NEEDS (the **N**ational **E**ngineering **E**ducation **D**elivery **S**ystem) program to be included in their national dissemination database in year 2000. Internally, a workshop will be arranged both in the College of Engineering and, later through the SUCCEED, to assist other faculty members from this College and other member institutions in getting into visualization-based teaching.

CONCLUSION

Optical diagnostic techniques, based on digital image-processing algorithm, have been integrated into the teaching of an undergraduate Thermal and Fluid sciences laboratory (TFSL) course. First, the Particle Image Velocimetry (PIV) technique is used to provide the detailed whole-field velocity measurement. Sample projects using PIV include the flow-field characterization of a turbulent wake and the droplet injection process of a Hewlett-Packard inkjet printhead. Second, the Laser Speckle Displacement (LSD) method is used to measure the density/temperature variation of selected flow fields, such as the shock-cell structure of a supersonic jet. Sample projects involving the use of these two techniques are described in the paper. Students are required to acquire and process PIV/speckle images and compare the processed data with other theoretical and experimental results in their laboratory reports. In our opinion, the integration of the advanced optical diagnostic techniques in the Thermal and Fluid Sciences Laboratory not only enhances students' understanding of the subject but also stimulate their interests in participation. Finally, demonstration courseware with detailed descriptions of the experiments will be archived into digital formats and be published on the Web, which will be made available to the entire engineering educational community.

ACKNOWLEDGEMENTS

The PI would like to thank Dr. Farrukh Alvi for his assistance in arranging some of the experiments and disseminating laboratory manuals on the web. He would also like to thank Dr. Luiz Lourenco and Mr. Bahadir Alkislar for providing their LSD supersonic jet data. Finaly, he would like to thank Dr. Fang-Gang Tseng who kindly allowed us to use his inkjet droplet photographs in the project.

BIBLIOGRAPHY

1. Lourenco, L., Krothapalli, A. and Smith, C.R., "Particle Image Velocimetry," Lecture Noted in Engineering, 45, Advances in Fluid Mechanics Measurements, pp. 128-199, ed. Gad-el-Hak, Springer-Verlag, 1989.
2. Adrian, R.J., "Particle Imaging Techniques for Experimental Fluid Mechanics," Annual Review of Fluid Mechanics, 23, pp. 261-304, 1991.
3. Lourenco, L., Gogineni, S. and LaSalle, R., "On-Line Particle-Image Velocimeter: an Integrated Approach," Applied Optics, Vol. 33, No. 13, pp. 2465-2470, 1994.
4. Kopf, U., "Application of Speckling for Measuring the Deflection of Laser Light by Phase Objects," Optics Communications, n.5, pp. 347-350, 1972.

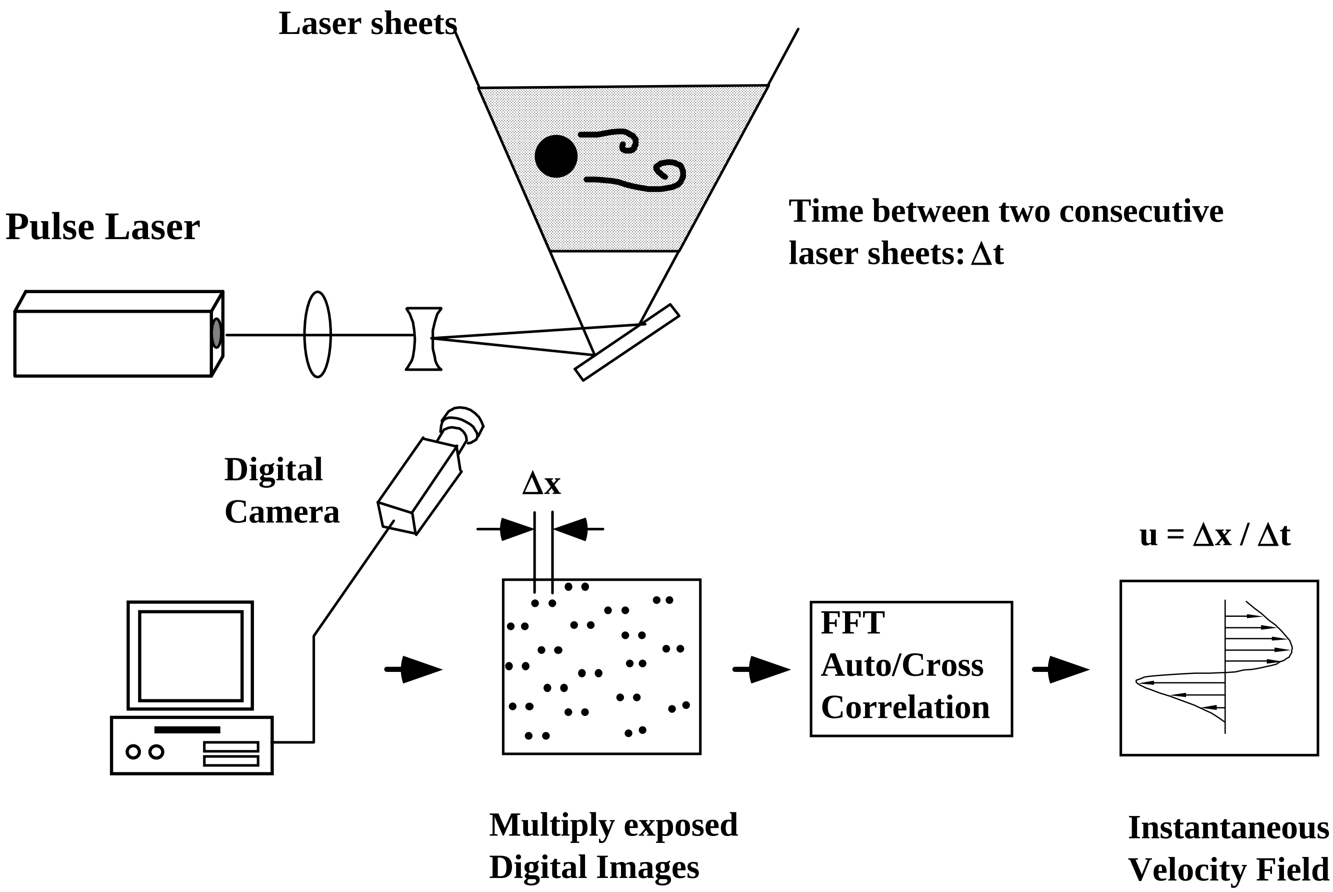


Figure 1 Schematic arrangement of the PIV system

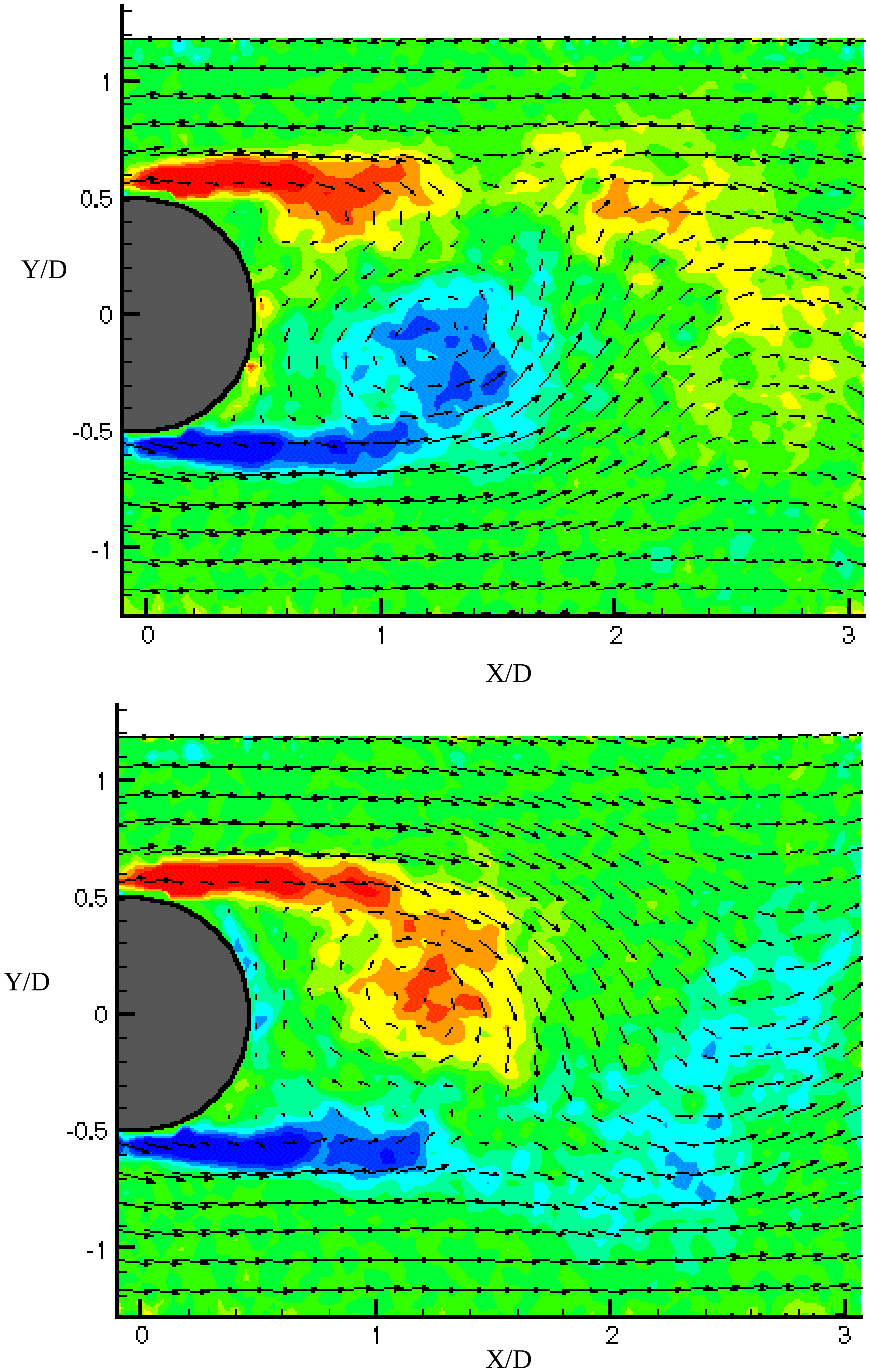


Figure 2 Vortex shedding behind a circular cylinder, color-coded vorticity plus velocity vector fields, depicting the vortex shedding process of the flow behind a circular cylinder during two different instants of its development.

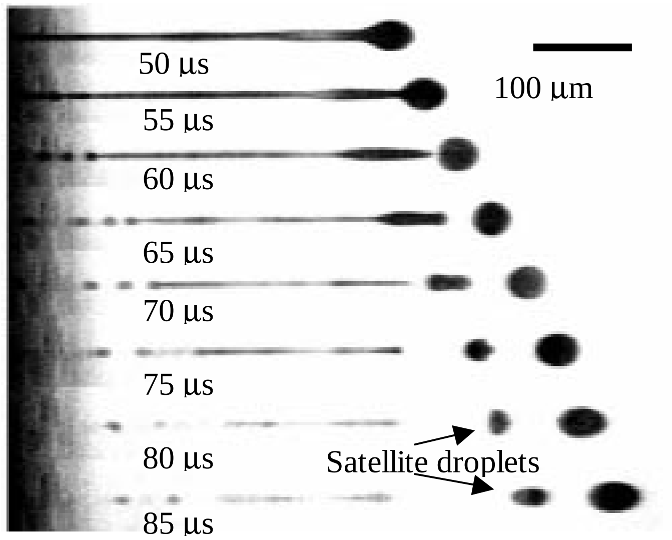


Figure 3 Near field visualization of the inkjet droplet injection process, showing the emergence of primary and satellite droplets.

HP Inkjet Printhead

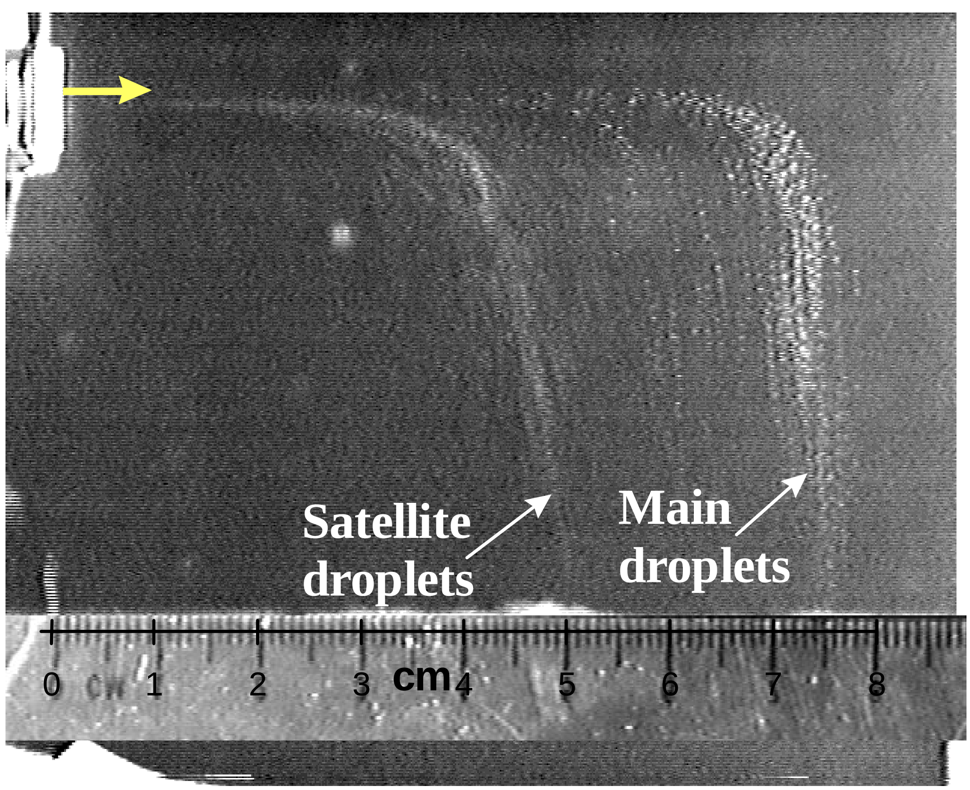


Figure 4 Far field visualization of the inkjet droplet injection process. Due to their different sizes and initial velocities, the trajectories of the main droplets are drastically different from that from the satellite droplets. (Figures 3 & 4 are photographs courtesy of Dr. Fang-Gang Tseng)

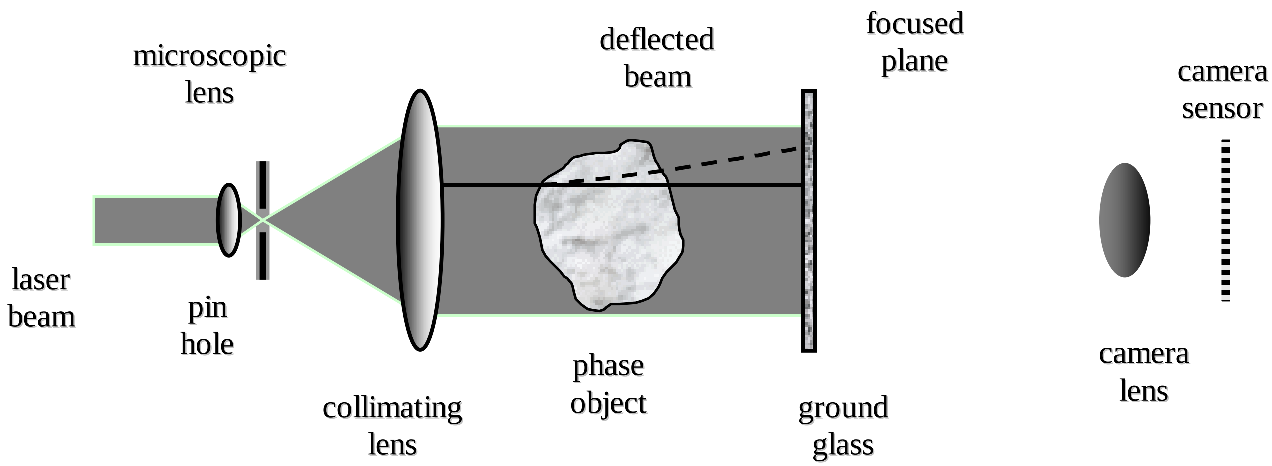


Figure 5 Schematic arrangement of the LSD system

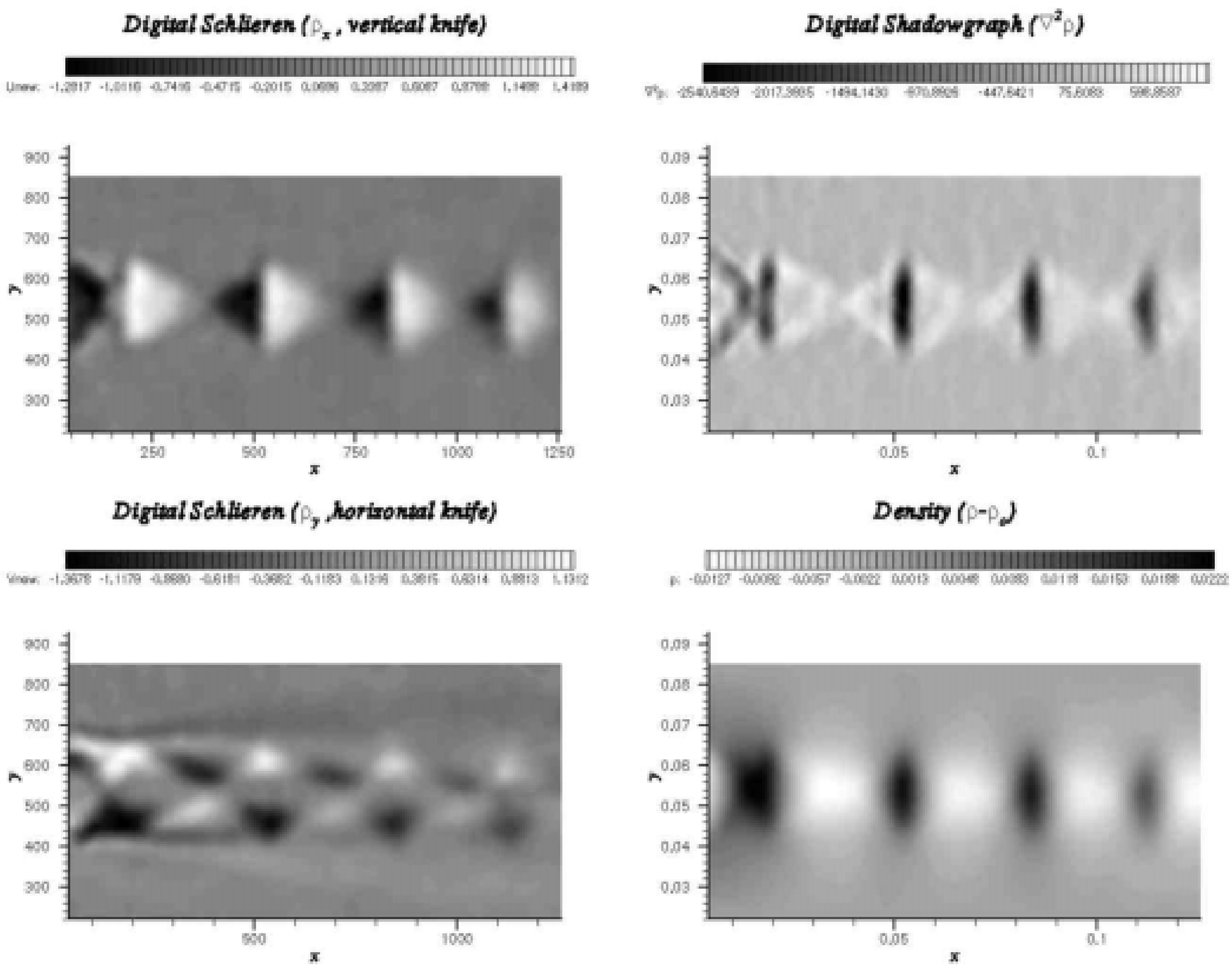


Figure 6 Periodic shock-cell structure of an under-expanded supersonic jet, showing the presence of periodic shock cells in a supersonic jet by the LSD data. It contains a sequence of four figures: (a) density field, (b) density gradient along the stream-wise directions, (c) density gradient along the cross-stream directions, (d) the second derivative of the density field.

Appendix A

Sample Laboratory Manual:

Flow Over a Circular Cylinder

Introduction

(a) Background

Flow separation

Wake

Vortex shedding

Vortex-induced vibrations

Lift and drag/momentum balance

(b) PIV technique

(c) CCD technology

(d) Other links

Other Pages

Laboratory Setup

Lab. Procedures

Report Writing

Further Readings

FAQ

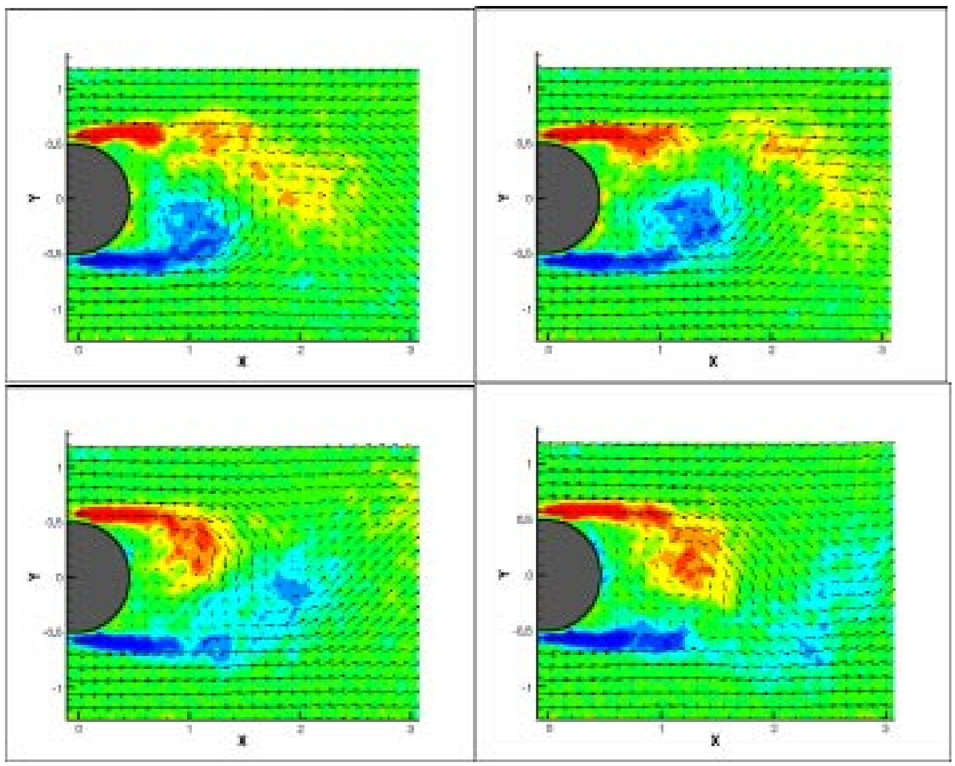


Figure 1. A sequence of four PIV velocity/vorticity fields depiciting different stages of development within a vortex shedding cycle. Vortices are shed alternatively from the upper and lower surfaces of the cylinder, creating a periodic flow pattern. Color-code is used to indicate the strength of the flow vorticity. Red color represents the clockwise vorticiity; blue color represents the counterclockwise vorticity. PIV velocity field is represented by the velocity vector data.

Introduction

External flows past objects have been studied extensively because of their many practical applications. For example, airfoils are made into streamlined shapes in order to increase the lift while at the same time, reducing the aerodynamic drag exerted on the wings. On the other hand, flow past blunt body, such as a circular cylinder, usually experiences boundary layer separation and very strong flow oscillations in the wake region behind the body. In certain Reynolds number range, a periodic flow motion will develop in the wake as a result of boundary layer vortices being shed alternately from either side of the body. This regular pattern of vortices in the wake is called a Karman vortex street. It creates an oscillating flow at a discrete frequency that is related to the Reynolds number of the flow. The periodic nature of the vortex shedding phenomenon can sometimes lead to unwanted structural vibrations, especially when the shedding frequency matches one of the resonant frequencies of the structure. One example is the famous Tacoma Narrow bridge incident and it will be discussed I more details later. In this laboratory, we are going to investigate the flow past a circular cylinder inside a towing tank facility. You are asked to make measurements of the cylinder wake flow field using Particle Image Velocimetry (PIV) technique. Based on your PIV velocity field measurements and the cylinder surface pressure distribution data taken from an earlier experiment, you are required to write a laboratory report which will be due two weeks from today. In your report, you should include comprehensive discussion about boundary layer flow separation, wake flow, vortex shedding, vortex-induced oscillations, the concept of momentum balance and how it relates to the lift and drag forces on the cylinder. A more detailed discussion of these topics is given in the following section.

Background

Flow Separation: The presence of the fluid viscosity slows down the fluid particles very close to the solid surface and forms a thin slow-moving fluid layer called a boundary layer. The flow velocity is zero at the surface to satisfy the no-slip boundary condition. Inside the boundary layer, flow momentum is quite low since it experiences a strong viscous flow resistance. Therefore, the boundary layer flow is sensitive to the external pressure gradient (as the form of a pressure force acting upon fluid particles). If the pressure decreases in the direction of the flow, the pressure gradient is said to be favorable. In this case, the pressure force can assist the fluid movement and there is no flow retardation. However, if the pressure is increasing in the direction of the flow, an adverse pressure gradient condition is said to exist. Under this condition, in addition to the presence of a strong viscous force, the fluid particles now also have to move against the increasing pressure force. Therefore, the fluid particles can be stopped and their direction of travel can even be reversed, causing the neighboring particles to move away from the surface. This phenomenon is called the boundary layer separation.

Wake: Consider a fluid particle flows within the boundary layer around the circular cylinder surface. From the pressure distribution measured in an earlier experiment, the pressure is maximum at the stagnation point and gradually decreases along the front half of the cylinder. The flow stays attached in this favorable pressure region as expected. However, the pressure starts to increase in the rear half of the cylinder and the fluid particle now experiences an adverse pressure gradient. Consequently, the flow separates from the surface and creating a highly turbulent region behind the cylinder called the wake. Inside the wake region, the flow is dominated by the regular vortex shedding process and the emergence of a highly irregular turbulent motion. The pressure inside the wake region remains low and a net pressure force (pressure drag) is produced. Right behind the body in the near wake region, the time-averaged flow field shows a closed recirculating pattern with strong flow reversal (see PIV velocity profiles). Further downstream, the mean velocity profiles still shown a significant slowing-down near the center and the exist of this momentum deficit region can be directly related to the generation of drag force on the body.

Vortex Shedding: The boundary layer which separates from the surface forms a free shear layer which it is highly unstable. This shear layer will eventually roll into a discrete vortex and detach from the surface (a phenomenon called vortex shedding). Another type of flow instability emerges as the shear layer vortices shed from both the top and bottom surfaces interact with one another. They shed alternatively from the cylinder and generate a regular vortex pattern (the Karaman vortex street) in the wake (figure 1). The vortex shedding occurs at a discrete frequency and is a function of the Reynolds number. The dimensionlese frequency of the vortex shedding, the shedding Strouhal number, St = f D/V, is approximately equal to 0.21 when the Reynolds number is greater than 1,000.

Vortex-Induced Vibrations: When vortices shed from the cylinder, uneven pressure distribution develops between the upper and lower surfaces of the cylinder, generating an oscillatory aerodynamic loading (lift) on the cylinder. This unsteady force can induce significant vibrations on a structure, especially if the "resonance" condition is met. That is when the frequency of an external forcing signal matches that of the natural frequency of the mechanical system. The most famous example is the collapse of Tacoma Narrow bridge in 1940 under the action of wind-induced vibrations. It is believed that natural vortex shedding frequency behind the bridge matched one of the resonant modes of the bridge and eventually lead to a catastrophic vibration that eventually destroys the bridge. Please refer to the other links section for a more comprehensive account of the incident.

Lift and Drag/Momentum Balance: According to the Newton's second law, time rate change of the linear momentum is equal to the sum of all external forces acting on a system. Therefore, an integration of the linear momentum inside a control volume surrounding the circular cylinder can provide information of the aerodynamic forces (lift and drag) acting on the cylinder. In general, there is a momentum deficit in the wake along the streamwise direction, therefore, according to the Newton's law there must be a net drag force acting on the cylinder. Also, from figure 1, there are alternative upward and downward flows in the wake as the result of vortex shedding. Consequently, there must be also an oscillatory lift force acting periodically on the cylinder in order to produce this alternative change of flow directions inside the wake. This periodic forcing is responsible for the vortex-induced vibrations as described earlier.

PIV Technique

The PIV is a quantitative flow visualization technique, which can be used to determine the instantaneous whole-field fluid velocities by recording and processing the multiply exposed particle image pattern of small traces suspended in the fluid. The PIV particle image is first obtained by illuminating the seeded flow field with a thin laser sheet. The light scattered by the seed particles generates a particle image pattern. This pattern is recorded using a digital CCD video technique (see figure 2). The local fluid velocity is then obtained using digital signal processing techniques.

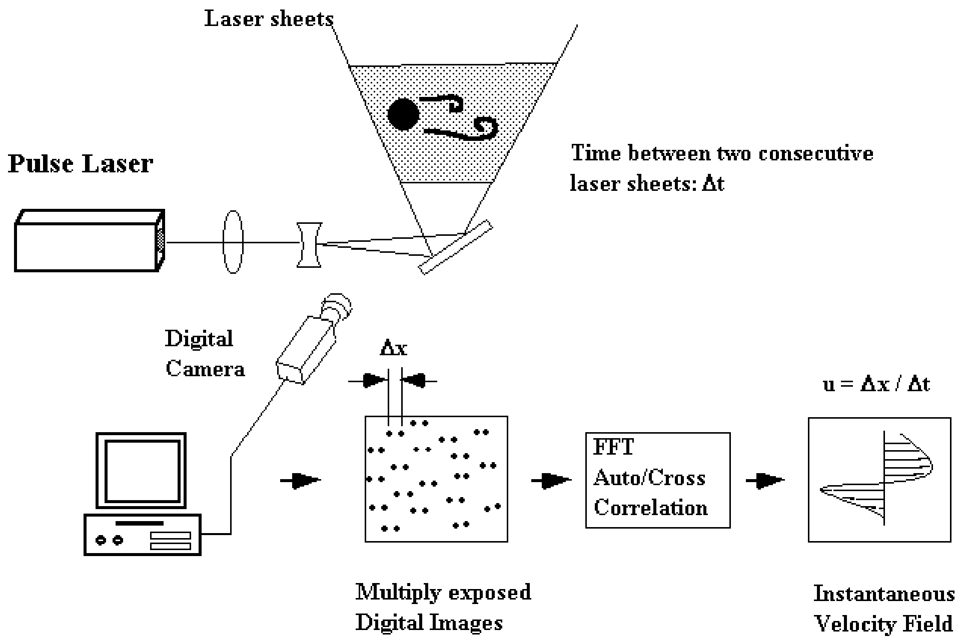


Figure 2. Schematic arrangement of the PIV system

Figure 3 is a sample PIV image pattern of the flow past a circular cylinder. The multiple illumination is provided using a rotating multi-faceted mirror. A total of four images are recorded for each particle in successive instants. Fast Fourier Transform (FFT) algorithm and an autocorrelation scheme are used to determine the separation between particle pairs. The whole-field velocity vectors can be determined by time differentiation and the result is shown in figure 3.

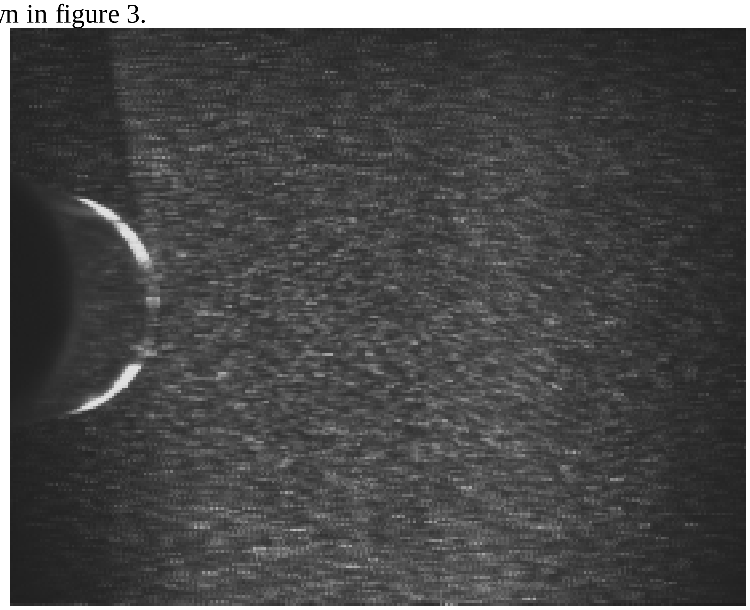


Figure 3 Multiply exposed PIV particle image field of the flow past a circular cylinder

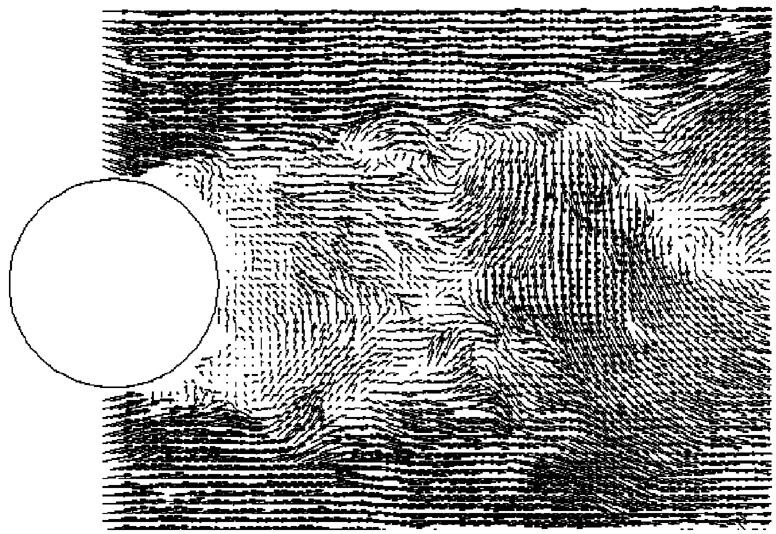


Figure 4. PIV velocity vector field processed using the image pattern in figure 3

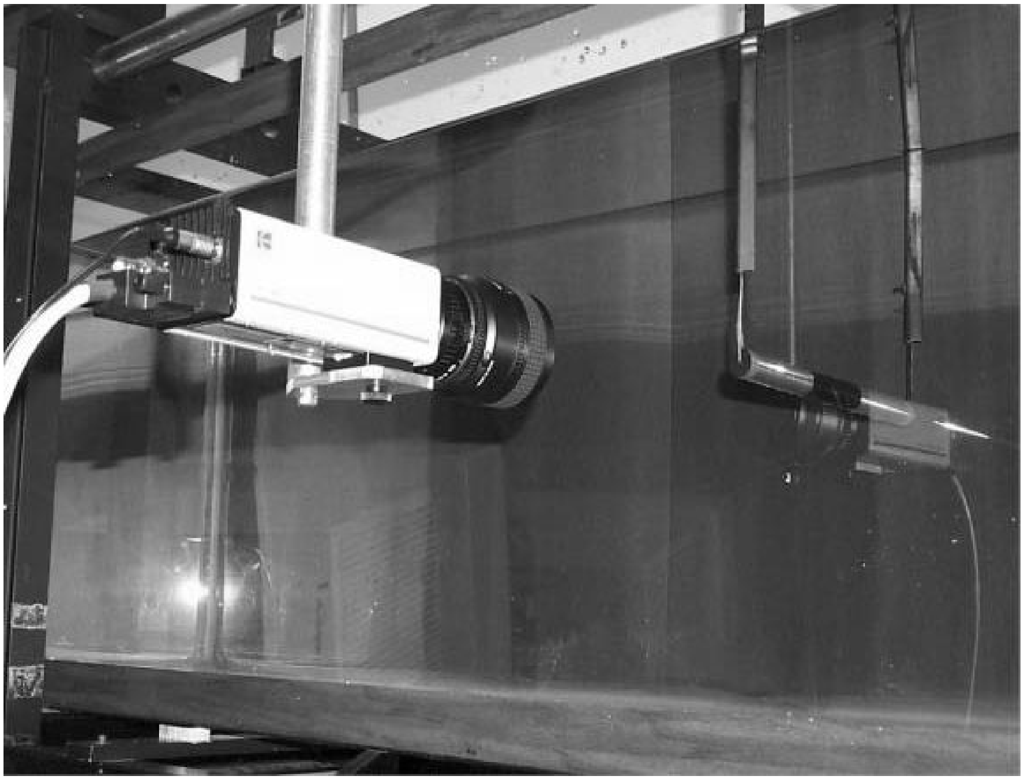
CCD Technology

CCD (Charge-couple device) technology has been used commonly for digital video recording for many years. A CCD imager consists of an array of sensing elements, called pixels, connected to a set of Metal-Oxide Semiconductor (MOS) Capacitors. When a CCD sensor receives light, each pixel absorbs photons and generates electric charge to be stored into the MOS capacitor. The sensitivity of a CCD depends on the capacity (potential well) of the amount of charge it can hold. These charges will need to be transferred to attached electrodes and be amplified to convert into digital signal before the pixels can be exposed to another "picture". Full frame transfer is more sensitive since all sensing array is light sensitive. However, it is usually much slower and not suitable for PIV measurement. Interline transfer mode allows the charges to be moved from the MOS capacitors to neighboring masked area which serves as a buffer area between the readout elements and the photosensing elements. Continuous image acquisition at video rate (30 frame/sec) is therefore possible. One added advantage of this mode is that it enables the double exposure mode, which allows two consecutive images to be recorded within a very short time period (in the order of μsec.) This feature is important for the PIV system as being described in the PIV section.

Other Useful Links

- **PIV Animation**
- **PIV Velocity and Vorticity Fields**
- **CFD Animations**
- **Tacoma Narrows Bridge Incident**

Laboratory Setup and Procedures



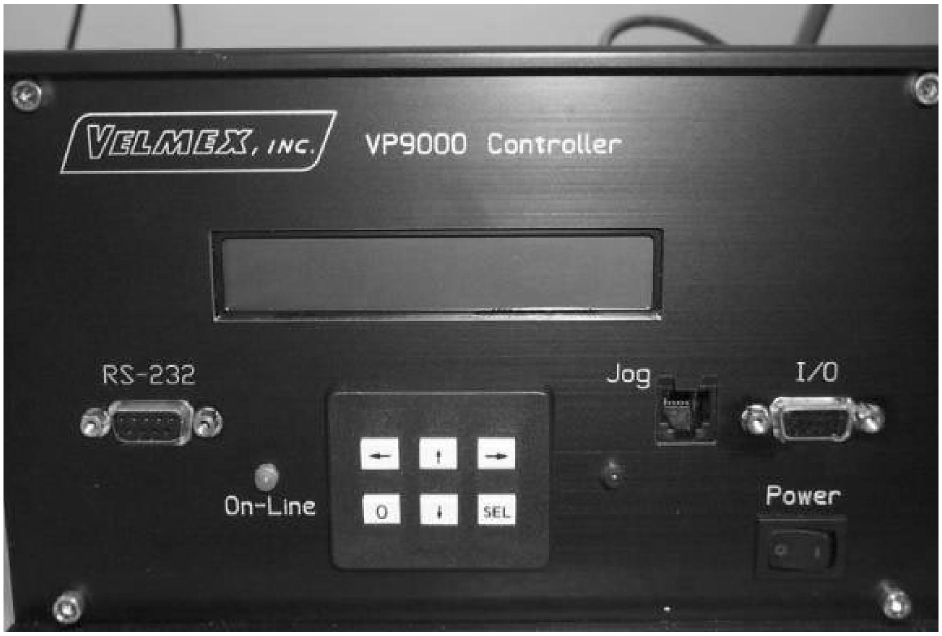
The following facility/components will be used in this laboratory:

1. Water towing tank facility.
2. VP9000 motion control system/LabView software package
3. Nd-Yag pulsed Laser system/optics.
4. Kodak ES 1.0 CCD camera/WinVu software package.

1. Towing tank facility:

The experiment will be performed in a water towing tank facility, which is 180 cm long and 43 by 55 cm in cross-sectional area. A stepping motor, interfaced to a computer-controlled stepping motor controller VP9000, drives the towing carriage with a towing speed varying from 0.2 to 20 cm/s. The cylinder model with a diameter of 2.54 cm is placed inside the tank. The camera is mounted on an extended platform to move with the cylinder (see above figure).

2. VP9000 motion control system

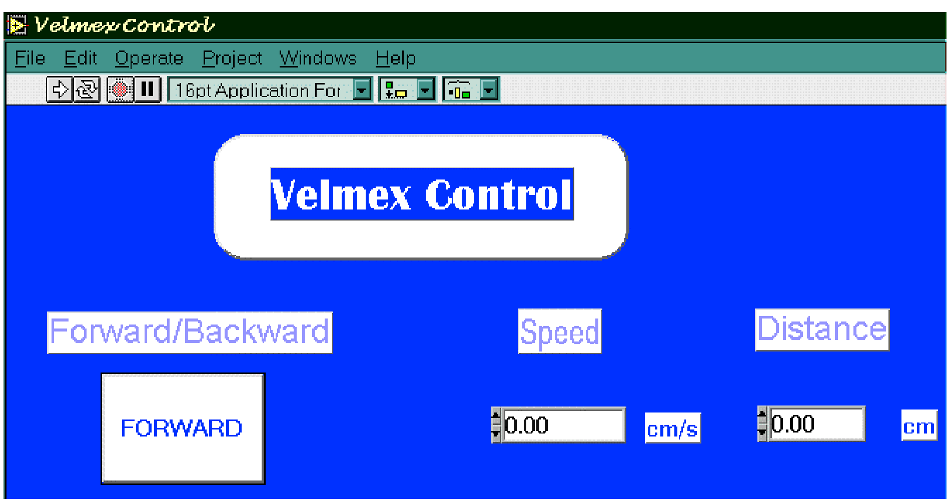


The motion of the towing carriage is controlled using a computer-interfaced stepping motor system. The Velmex VP9000 motor controller (see picture above) is used to to set specific speed and travel distance of the carriage via a computer by a LabView program. The system can also be controlled manually using the front panel control key pad as shown. The towing tank operating procedures are described in the following:

- Make sure the motor is connected to the motor controller and turn on the power of the motor controller.

- Put the motor controller online, so that it can receive the command from the computer. Confirm that the on-line status LED is on.

- Run the Labview program and open the towing.vi (as shown below) from the eml4304 folder.



- Input desired motor speed in cm/s and travel distance in cm.

- Assign moving direction (Forward to move away from the motor; Backward to move towards the motor).

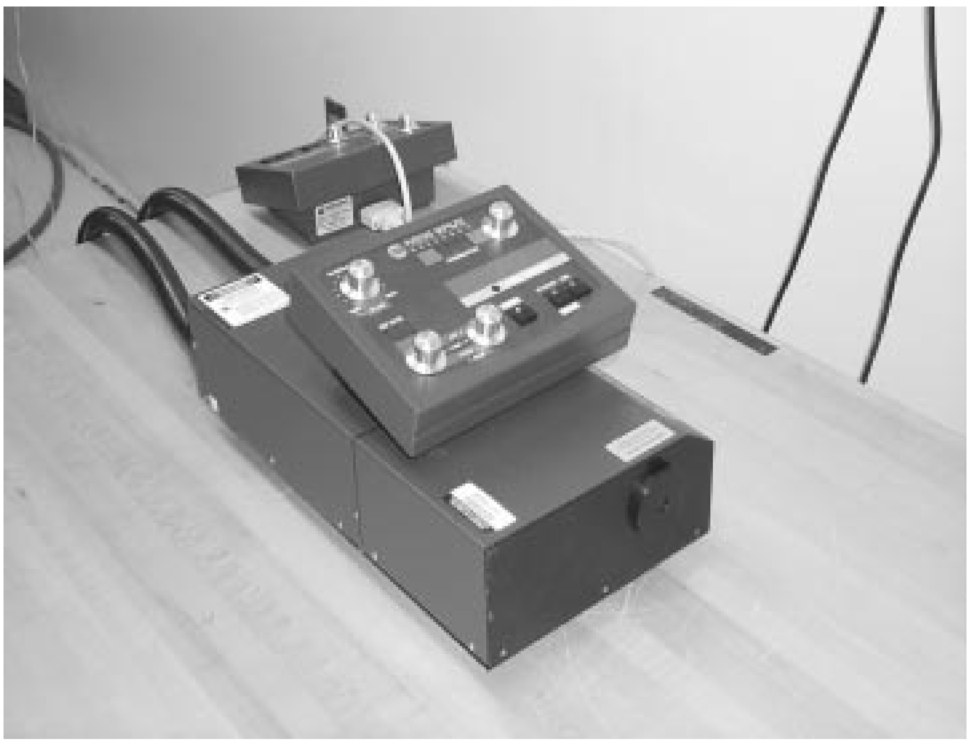
- Execute the towing.vi program by selecting the RUN command from the program to activate the motor.

3. Nd-Yag laser system/Optics:

A dual-pulsed laser system, consisting of two New Wave Nd-Yag pulsed laser systems, is used to provide the double illumination. A system of prisms and polarization cube combiner makes the two laser colinear. A cylindrical lens is used to expand the laser bean into a laser sheet illuminating the midspan section of the cylinder. The laser firing is synchronized with the digital video camera (Kodak Es1.0) to record the doubly-exposed particle image fields. The PIV image data will be stored and later processed by a image-processing package (WinVu program).

Laser operating procedures:

- Turn on the laser power. Make sure both of the cooling water pumps are functional.



- Make sure to turn the flashlamp power and the attenuator setting to the minimum before activating the laser.
- Press the standby button to activate the flash lamp;
- Press the fire button to activate the Q-switch and fire the laser. Make sure there is no stray reflection from the laser.
- Gradually increase the flashlamp power until laser sheet is visible.
- Adjust the light intensity to desired level.

4. Digital camera/Image Processing program

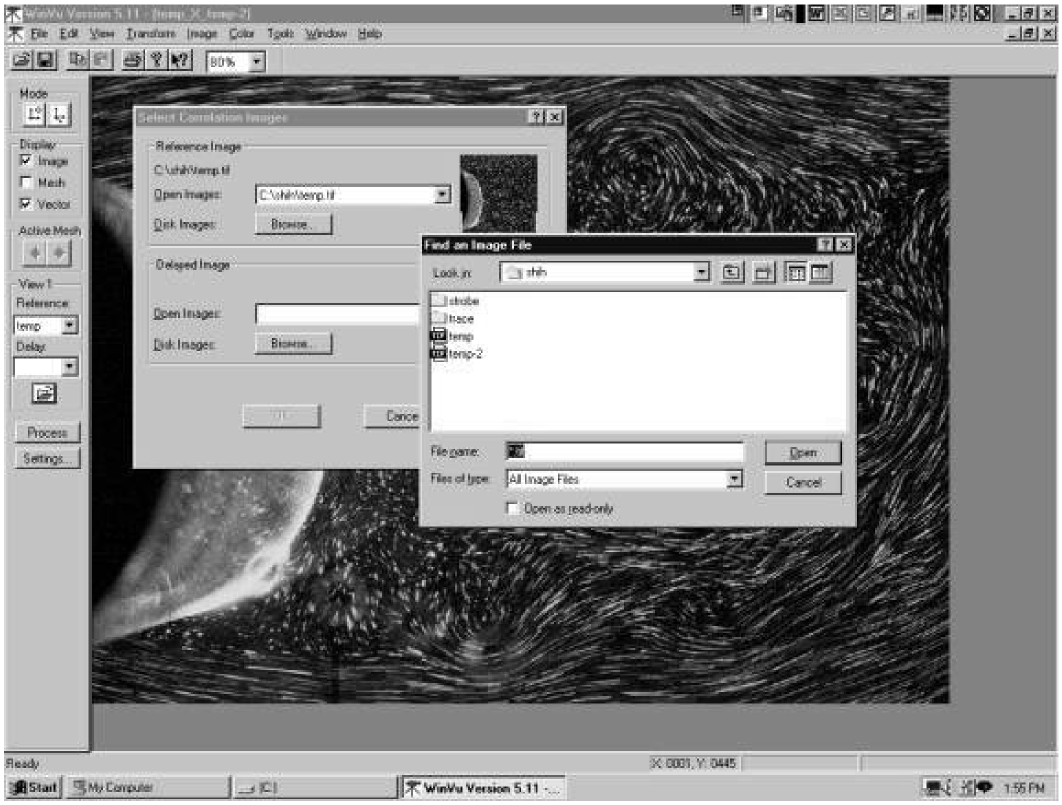
A Kodak Es 1.0 double-exposure digital camera will be used to record the particle imaging fields. An imaging acquisition package, WinVu, is used to digitize each particle field into a 1k by 1k pixels image data file. Two consecutive PIV images, separating by a given short period Δt, will be cross-correlating to determine the local particle image displacement Δx. The local velocity V can then be obtained by the following relationship, $V=\Delta x/\Delta t$. The following is a description of the imaging processing procedures:

- Open WinVu program by clicking on the WinVu icon on the desktop.
- Open the video window inside the view tool menu box.
- Input the data file names and select the trigger mode for the camera. Choose to acquire 10 image-pair for the run. (Note: a sequential number will be assigned to the end of each file for identification purpose).
- Activate the program. (Note: run the PIV program after the towing tank move for a short distance so that the wake flow stabilizes).
- At the end of the acquisition, save all data files to your group's folder under eml4304 folder for later post-processing.

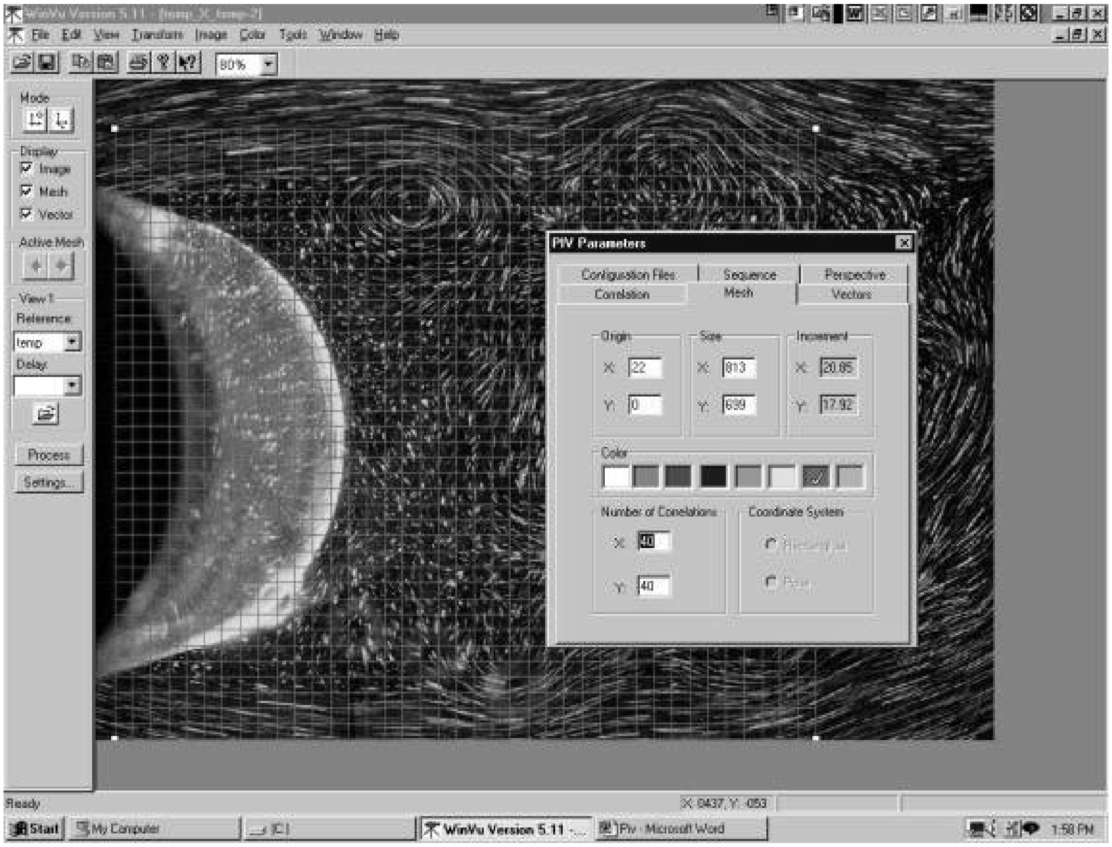
5. Post-processing

The recorded image files will be grouped into pairs of two consecutive imaging fields separated by a short time period Δt. The cross-correlation between these two imaging field will produce the displacement information for the entire flow field $\Delta X(x,y)$ and $\Delta Y(x,y)$, where ΔX and ΔY are the local displacement as a function of the coordinate x and y. The local velocity U(x,y) and V(x,y) can be determined by differentiating these displacements with time: $U=\Delta X/\Delta t$ and $V=\Delta Y/\Delta t$. The cross-correlation calculation for the entire flow field will be performed using the WinVu program based on the following procedures:

- Open the WinVu program and choose to open another PIV window (inside the tool menu box).
- Open the file sequence you want to perform cross-correlation (for example: c:\shih\temp and c:\shih\temp1 as shown below).



- Choose the flow direction (left to right for the present experiment) and the interrogation size (use 24 pixels in this case).
- Drag an interrogation box on the image (represented by the purple grid lines as shown below.), and choose a 40 by 40 grid size.



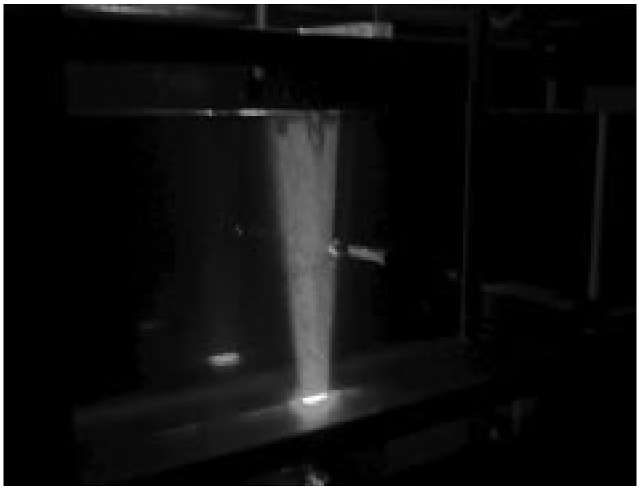
- Click the process icon to calculate the vector field.
- The corresponding whole-field velocity data file will be stored in the file temp.vec.
- Save all vector files to your subdirectory for future uses.

Experimental Procedures

Wake Measurement:

For the wake measurement, run the towing tank at 20 cm/s and for a distance of 100 cm. Start image acquisition after the cylinder has traveled for 30 cm, that is after it has traveled for more than 12 diameters.

- Seed the water towing tank with tiny particles (glass beads coated with silver, 10 µm in diameter). Stir up the water to mix the particles thoroughly. Wait at least 10 minutes to let the sloshing water to calm down before the experiment.



- Turn on the laser and adjust the laser sheet to cover the wake region (see photos above).
- Adjust the camera to make sure that it can capture the entire wake flow field, including the boundary layer flow on the cylinder's surface, separating shear flow and the wake region.
- Activate the VP9000 motion control program. Move towing carriage forward.
- Start the imaging acquisition WinVu program after the cylinder has traveled beyond 30 cm. Take a sequence of 30 doubly-exposed image pairs.
- Post-processing these imaging fields into vector files and save them to your folder.
- Move towing carriage backward to its original position.

Appendix B

An optical bench is made with the support provided by the NSF. A heavy duty aluminum plate, equipped with optical mounting threaded holes, is used as the base plate of the system. A strobescope unit is used to provide the illumination. It can be replaced with a laser source if the laser speckle displacement technique is to be used. The optical system is made of a set of two parabolic mirrors and two front surface mirrors as shown. Traditional Schlieren and shadowgraph techniques can be easily arranged on the bench.

Compressed Helium and compressed air tank are equipped for both buoyancy-driven convective flow and high-speed supersonic flow visualizations.

Quantitative Flow Visualization

Objectives

The main objective of this project is to use the quantitative flow visualization technique into the teaching of thermal and fluids related dsisciplines by Providing visualization-enhanced courseware.   This web-based package allows the students to learn many important subjects interactively by using bookmarks and hyperlinkages.   The integration of the visualization results and quantitative data provides an optimal way of learning the thermal and fluids subjects because it not only motivates students' interests and enhance their comprehension of the subjects but also stimulates indepth understanding by allowing full interaction.

Introduction

"Thermal Sciences" are among the most difficult curricula for undergraduate engineering students. A primary reason is the difficulty in conceptually visualizing thermal/fluid phenomena because most fluids are transparent and their motions and the associated thermal processes are invisible to human eyes. Consequently, visualization methods play an important role in grasping such concepts. In thermal fluid sciences, the understanding of the physical thermal/flow processes can be greatly improved if the flow pattern of interest can be visually observed. In light of this, we produce the following visualization-based courseware as supplement to the teaching of the fluid mechanics. Different from the traditional flow visualization technique, we use a quantitative visualization technique that can not only provide qualitative visualization but also can produce useful velocity and vorticity data. With the aid of this technique, students will be able to directly compared the data to theoretical or numerical results.

Whole-field flow measurement and visualization:

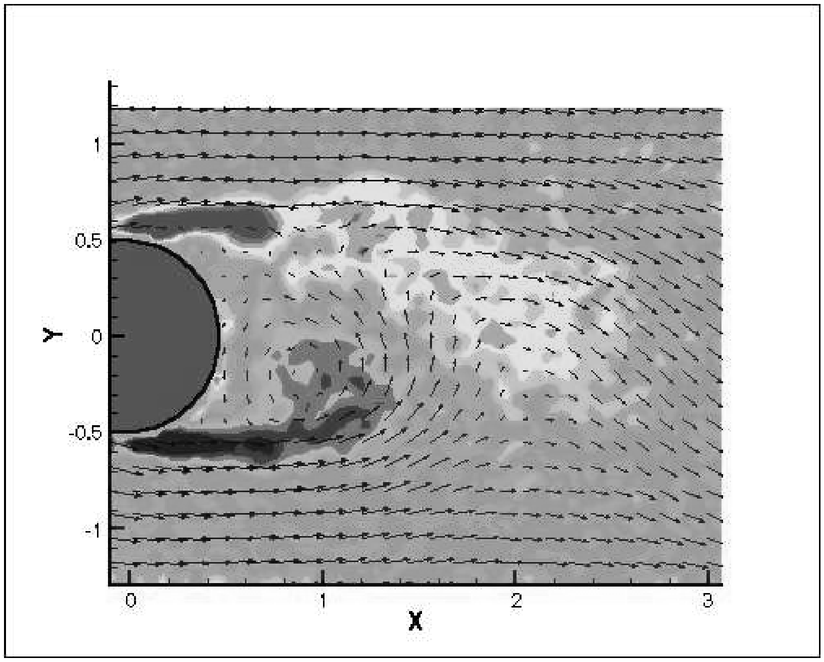
The Particle Image Velocimetry (PIV) is used to enhance undergraduate laboratory teaching. PIV can provide measurement/visualization of global velocity field information of complex flow fields. Sample projects presented include:

- **Flow separation and turbulent wake**
- **Droplet injection process of a Hewlett-Packard inkjet printhead**

- **Jet**
- **Wall jet**

Both the visualization photography and the corresponding whole-field PIV data will be integrated into one database. Useful information of the flow, such as velocity profiles and color-coded vorticity contours, are available on selected regions on the photography using hyperlinks.

Flow Separation and Wake

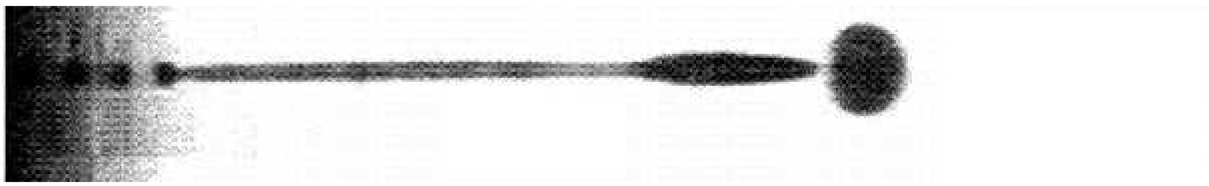


When the fluid is moving adjacent to a solid surface, its motion depends on the balance of several forces acting upon the fluid particles: frictional force at the surface, viscous force by the neighboring flow stream, and pressure force depending on the external flow condition. The fluid can continue its forward motion if the pressure gradient is favorable. However, if the fluid experiences an adverse pressure gradient, that is when the pressure force is against the motion of the fluid, the fluid particles can be stopped and sometimes be forced to move backward. The retardation and backward-moving motion always occur at the surface since the friction force exerted by the wall is greatest and the fluid momentum is much lower there. When the fluid moves backward at the surface, the following particles have to move away from the surface and the flow is said to be "separated" from the surface. This phenomenon is called "flow separation".

When the flow is passing by a bluff body, a strong adverse pressure gradient always presides over the rear half of the body and the flow separates. The flow separation prevents the pressure being recovered in the rear half of the object, therefore, causing a difference in pressure between the front and the back sides of the object. Consequently, a net drag force is acting on the body as is usually called the form drag or pressure drag. In additional to this drag force, the wake flow field is highly unsteady and can induce significant fluctuating lateral loading on the object. The unsteady nature of the wake flow is discussed in more details in the cylinder wake flow web page.

- **Wake flow behind a circular cylinder**

Inkjet Droplet Injection Process



Thermal bubble jet technology has been used widely in commercial inkjet printer. It is one of the most cost-effective ways to produce high quality color printing. Tiny ink droplets are ejected out of the inkjet cartridge to form the print pattern when they impinging on the paper. Inside the inkjet chamber, a thin strip heater vaporizes the liquid ink when an electric current passes through it. The increasing pressure inside the chamber forces the remaining liquid ink to eject out of a nozzle as a liquid jet column. The bubble collapses as the electric current is turned off and the chamber is replenished by liquid ink outside the chamber. This on and off cycle repeats at a very high frequency (~kHz range) and streams of inkjet droplets form the print pattern. This process is illustrated in the following schematic.

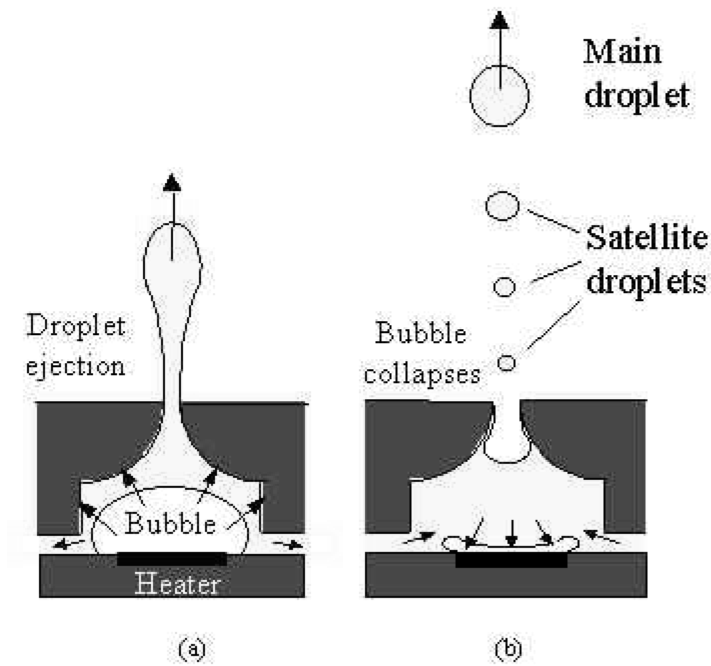


Illustration of a typical inkjet ejection process of a thermal bubble jet
(a) growth of bubble and ejection of liquid column
(b) collapse of bubble and droplet formation
Schematic courtesy by Dr. Fang-gang Tseng

Due to the capillary instability, the ejected jet column breaks up into a primary droplet and a stream of samll droplets (satellite droplets) as shown. This ejection process and other interesting subjects related to the inkjet droplet formation can be found in the following web page.

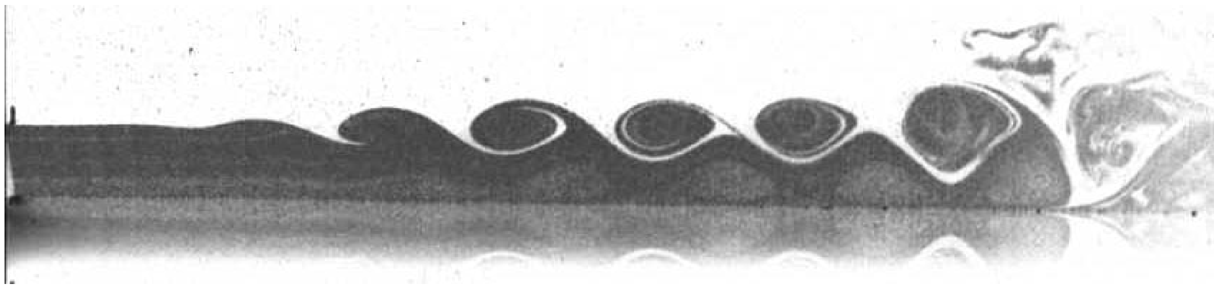
- **Inkjet Droplet Injection Process**

Jet

A jet is formed by flow issuing from a nozzle into ambient fluid which can be at rest (a free jet) or moving (a coflowing or counterflowing jet) or tangential to a solid surface (a wall jet). A jet is one of the basic flow configurations which has been widely used in many practical applications such as the fuel injection, combustor, jet engine propulsion exhaust and others. One of the most important considerations for jet flow is the entrainment and mixing process of a jet with respect to its surroundings. For example, in a fuel injedctor, more uniform mixing and fuel distribution means higher combustion efficiency and better engine performance. In order to characterize the jet mixing process, a jet flow field measured using the Particle Image Velocimetry (PIV) is presented in the following page.

- **PIV Jet Flow Field**

Wall Jet



A wall jet is a stream of fluid blown tangentially along a wall and it has a wide range of applications, such as boundary-layer separation control over a wing, film cooling on turbine blades, etc. The photograph shown above is a wall jet flow exiting from a nozzle and the flow is from left to right. Smoke is used to seed the flow in order to visualize the jet and take PIV images. The surface can be seen clearly as the interface line between the jet image and its reflected image from the surface. There are two distinct regions in the wall jet configuration: first, an outer free shear layer that is subjected to the inviscid Kelvin-Helmholtz instability and the formation of large scale vortices; and an inner layer that behaves like a viscous boundary layer. The interaction between large scale structures from the outer layer and inner layer eventually leads to the laminar-to-turbulence transition. This process can be understood better by browsing through the following pages:

- **Comparison between PIV and Numerical Simulation Results**
- **PIV Velocity and Vorticity Data**

[Introduction](#)

[Background](#)

[Flow separation](#)

[Wake](#)

[Vortex shedding](#)

[Vortex-induced vibrations](#)

[Aerodynamic loading](#)

[Momentum balance](#)

[PIV technique](#)

[CCD technology](#)

[Animated vortex shedding sequence](#)

[Particle pathline visualization](#)

[Other links](#)

Turbulent Wake Flow
Behind a Circular Cylinder

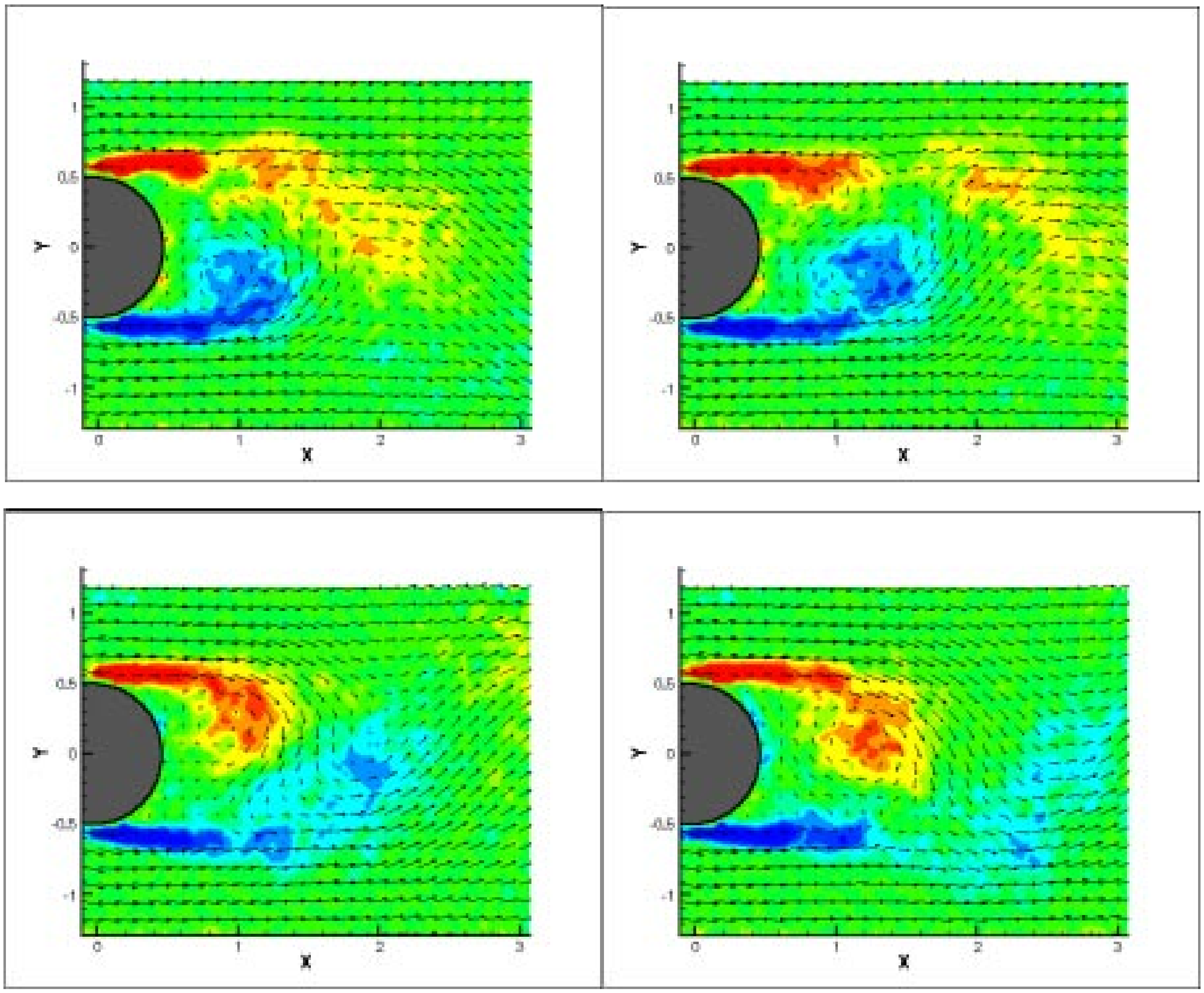


Figure 1. A sequence of four PIV fields depiciting different developing stages within a vortex shedding cycle. Vortices are shed alternatively from the upper and lower surfaces of the cylinder, creating this periodic flow pattern. Color-code is used to indicate the strength of the flow vorticity. PIV velocity field is represented by the velocity vector data.

- **Animated vortex shedding sequence**
- **Particle pathline visualization.**

Introduction

External flows past objects have been studied extensively because of their many practical applications. For example, airfoils are made into streamline shapes in order to increase the lifts, and at the same time, reducing the aerodynamic drags exerted on the wings. On the other hand, flow past a blunt body, such as a circular cylinder, usually experiences boundary layer separation and very strong flow oscillations in the wake region behind the body. In certain Reynolds number range, a periodic flow motion will develop in the wake as a result of boundary layer vortice being shed alternatively from either side of the cylinder. This regular pattern of vortices in the wake is called a Karman vortex street. It creates an oscillating flow at a discrete frequency that is correlated to the Reynolds number of the flow. The periodic nature of the vortex shedding phenomenon can sometimes lead to unwanted structural vibrations, especially when the shedding frequency matches one of the resonant frequencies of the structure. One example is the famous Tacoma Narrow bridge incident and this topic has been discussed in great details in the Tacoma bridge link. In this presentation, we are going to investigate the flow past a circular cylinder and study the turbulent wake flow field using the Particle Image Velocimetry (PIV) technique. Based on the PIV velocity field measurements and other reference information, a comprehensive discussion about many important flow concepts such as: boundary layer flow separation, wake flow, vortex shedding, vortex-induced oscillations, aerodynamic loading, momentum balance, and the lift and drag forces on an immerse body, will be given in the following section.

Background

Flow Separation: The presence of the fluid viscosity slows down the fluid particles very close to the solid surface and forms a thin slow-moving fluid layer called a boundary layer. The flow velocity is zero at the surface to satisfy the no-slip boundary condition. Inside the boundary layer, flow momentum is quite low since it experiences a strong viscous flow resistance. Therefore, the boundary layer flow is sensitive to the external pressure gradient (as the form of a pressure force acting upon fluid particles). If the pressure decreses in the direction of the flow, the pressure gradient is said to be favorable. In this case, the pressure force can assist the fluid movement and there is no flow retardation. However, if the pressure is increasing in the direction of the flow, an adverse pressure gradient condition as so it is called exist. In addition to the presence of a strong viscous force, the fluid particles now have to move against the increasing pressure force. Therefore, the fluid particles could be stopped or reversed, causing the neighboring particles to move away from the surface. This phenomenon is called the boundary layer separation.

Wake: Consider a fluid particle flows within the boundary layer around the circular cylinder. From the pressure distribution measured in an earlier experiment, the pressure is a maximum at the stagnation point and gradually decreases along the front half of the cylinder. The flow stays attached in this favorable pressure region as expected. However, the pressure starts to increase in the rear half of the cylinder and the particle now experiences an adverse pressure gradient. Consequently, the flow separates from the surface and creating a highly turbulent region behind the cylinder called the wake. The pressure inside the wake region remains low as the flow separates and a net pressure force (pressure drag) is produced.

Vortex Shedding: The boundary layer separates from the surface forms a free shear layer and is highly unstable. This shear layer will eventually roll into a discrete vortex and detach from the surface (a phenomenon called vortex shedding). Another type of flow instability emerges as the shear layer vortices shed from both the top and bottom surfaces interact with one another. They shed alternatively from the cylinder and generates a regular vortex pattern (the Karaman vortex street) in the wake (figure 1). The vortex shedding occurs at a discrete frequency and is a function of the Reynolds number. The dimensionless frequency of the vortex shedding, the shedding Strouhal number, St = f D/V, is approximately equal to 0.21 when the Reynolds number is greater than 1,000.

Vortex-Induced Vibrations: When vortices shed from the cylinder, uneven pressure distribution develops between the upper and lower surfaces of the cylinder, generating an oscillatory aerodynamic loading (lift) on the cylinder. This unsteady force can induce significant vibrations on a structure, especially if the "resonance" condition is met. The most famous example is the collapse of Tacoma Narrow bridge in 1940 under the action of wind-induced vibrations. It is believed that natural vortex shedding frequency behind the bridge matches one of the resonant modes of the bridge and eventually lead to a catastrophic vibration that destroys the bridge. Please refer to the other links section for a more comprehensive account of the incident.

Aerodynamic Loading: According to the Newton's second law, time rate change of the linear momentum is equal to the sum of all external forces acting on a system. Therefore, an integration of the linear momentum inside a control voulme surrounding the circular cylinder can provide information of the aerodynamic forces (lift and drag) acting on the cylinder. As shown in figure 1, and animated PIV vortex shedding sequence, and particle pathline visualization sequence, there are alternative upward and downward flows in the wake as the result of vortex shedding. Consequently, there must be also an oscillatory up and down force acting periodically on the cylinder. This periodic forcing exerting on the cylinder body is responsible for the vortex-induced vibrations as described earlier.

Momentum Balance: As stated earlier, the external force acting on an object can be determined using the momentum balance concept. In general, there is a momentum deficit in the wake profile along the streamwise direction as relative to the incoming momentum upstream of the object. Therefore, a simple balance of the momentum flow in and out of the control volume surrounding the object suggests that there is net force acting on the object. (note: the pressure is considered to be relatively constant if the momentum flow is measured far away from the object.) This net force along the flow direction is called the drag. Averaged velocity profiles of the flow past a circular cylinder is provided as a general representation of the wake flow field. Selected profiles at several representative regions is also presented for your reference. Near the cylinder, flow separates from the surface. Immediately behind the cylinder, a recirculation region exists with a strong reversing flow. The region between the cylinder and the end of the recirculation region is called the vortex formation region. The centerline velocity becomes zero at the end of the vortex formation region. Further downstream, the two separating shear layers merge and the velocity profile presents a typical wake profile. It is clear that there is a deficit in the center of the wake. This deficit in the momentum flow is the direct result of drag force acting on the cylinder.

- **Averaged PIV velocity profiles**

PIV Technique

The PIV is a quantitative flow visualization technique, which can be used to determine the instantaneous whole-field fluid velocities by recording and processing the multiply-exposed particle image pattern of small traces suspended in the fluid. The PIV particle image is first obtained by illuminating the seeded flow field with a thin laser sheet. The light scattered by the seed particles generates a particle image pattern. This pattern is recorded using a digital CCD video technique (see figure 2). The local fluid velocity is then obtained using digital signal processing techniques.

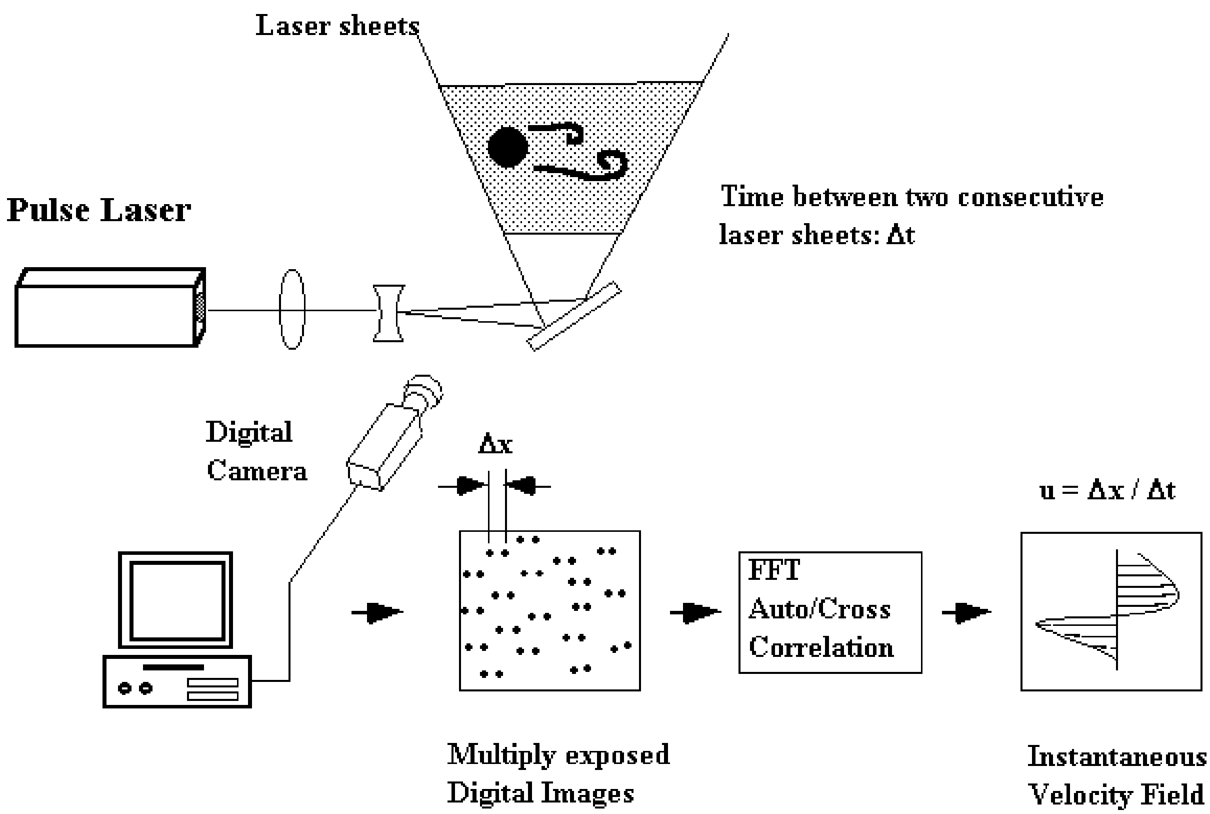


Figure 2. Schematic arrangement of the PIV system

Figure 3 is a sample PIV image pattern of the flow past a circular cylinder. The multiple illumination is provided using a rotating multi-faceted mirror. A total of four images are recorded for each particle in successive instants. Fast Fourier Transform (FFT) algorithm and autocorrelation scheme are used to determine the separation between particle pairs. The whole-field velocity vectors can be determined by time differentiation and the result is shown in figure 3.

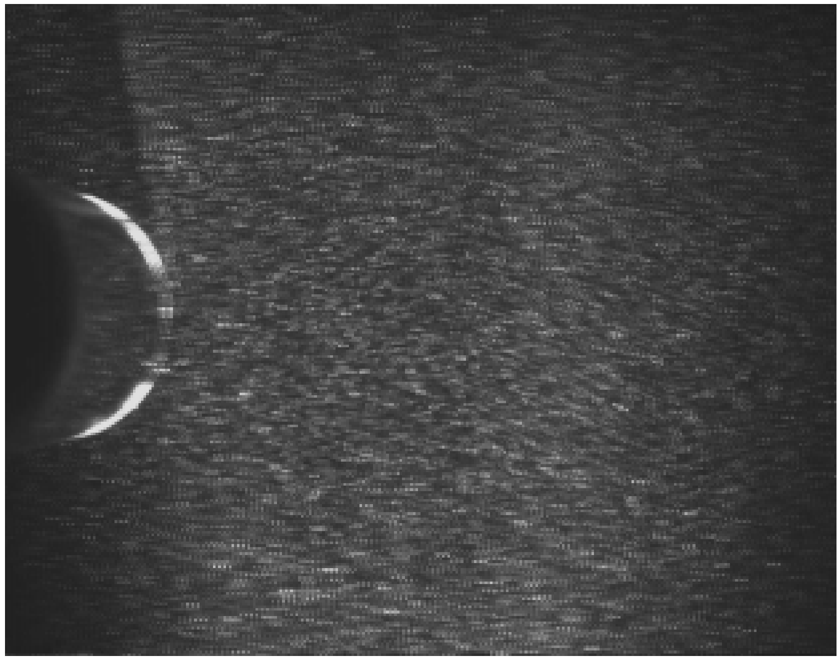


Figure 3 Multiply-exposed PIV particle image field of the flow past a circular cylinder

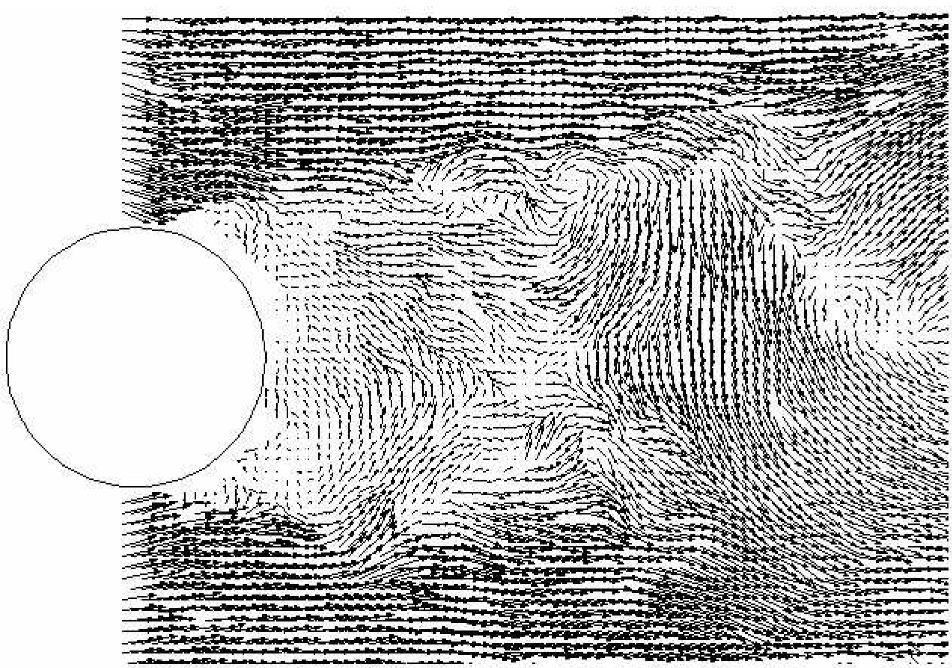


Figure 4 Corresponding PIV velocity vector field

CCD Technology

CCD (Charge-couple device) technology has been used commonly for digital video recording for many years. A CCD imager consists of an array of sensing elements, called pixels, connected to a set of Metal-Oxide Semiconductor (MOS) Capacitors. When a CCD sensor receives light, each pixel absorbed photons and generates electric charge to be stored into the MOS capacitor. The sensitivity of a CCD depends on the capacity (potential well) of the amount of charge it can hold. These charges will need to be transferred to attached electrodes and be amplified to convert into digital signal before the pixels can be exposed to another "picture". Full frame transfer is more sensitive since all sensing array is light sensitive. However, it is usually much slower and not suitable for PIV measurement. Interline transfer mode allows the charges to be moved from the MOS capacitors to neighboring masked area which serves as a buffer area between the readout elements and the photosensing elements. Continuous image acquisition at video rate (30 frame/sec) is therefore possible. One added advantage of this mode is that it enabes the double exposure mode, which allows two consecutive images to be recorded within a very short time period (in the order of μsec.) This feature is important for the PIV system as being described in the PIV section.

Other Useful Links

- **Viscous Vortex Method (Flow over a rotating cylinder and others)**
- **CFD Animations (Flow over a circular cylinder and others)**
- **Flow Over a Flexible Cable**
- **Tacoma Bridge Incident**

Inkjet Droplet Injection Process

[Injection process-animation](#)

[Injection process-still images](#)

[Droplet visualization set-up](#)

[Satellite droplets](#)

[Droplet trajectory](#)

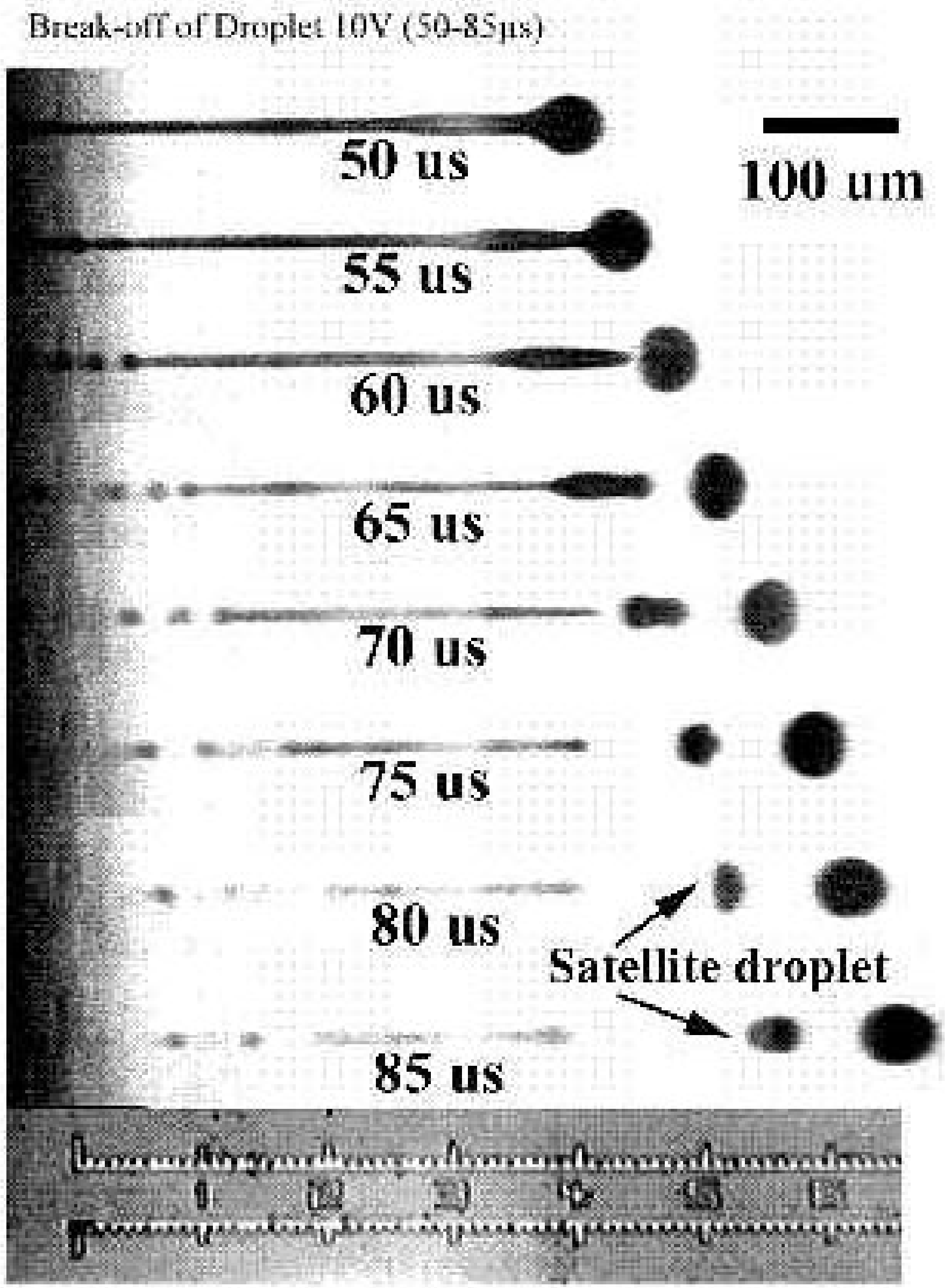
[Numerical simulation-1](#)

[Numerical simulation-2](#)

The following animated sequence shows the liquid ink column is ejected out of a HP printhead nozzle and begins to form a primary ink droplet. The existence of a very long tail and the emergence of the Raleigh jet instability force the narrow jet column to break up into several satellite droplets. The presence of satellite droplets can significantly degrade the printing quality of an inkjet printer.

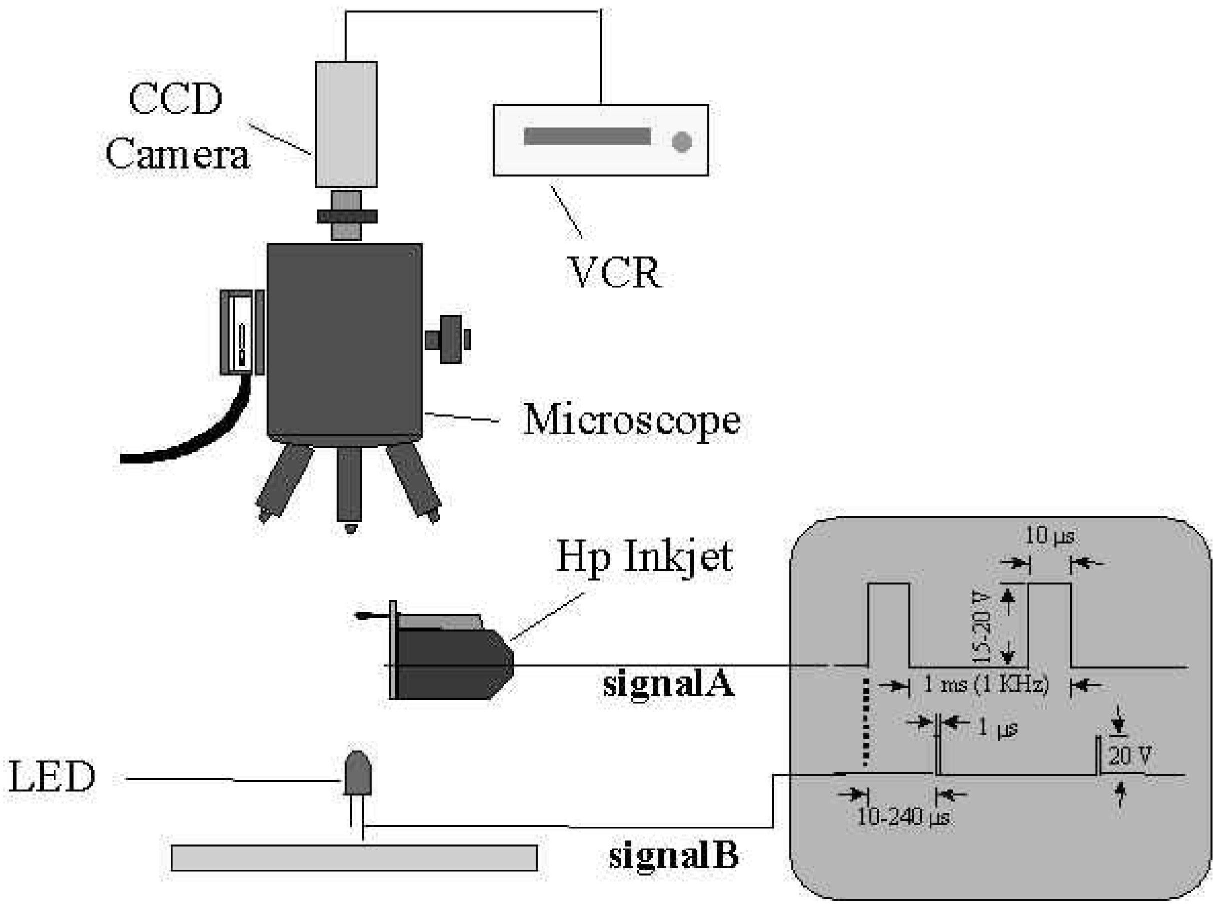
Injection Process, Animation

Injection Process, Still Images



The time associated with each droplet image indicates the dalay in μs between the time when the picture is taken and the time when the electric current pulse is activated. Photograph courtesy of Dr. Fan-Gang Tseng.

Droplet Visualization Set-Up

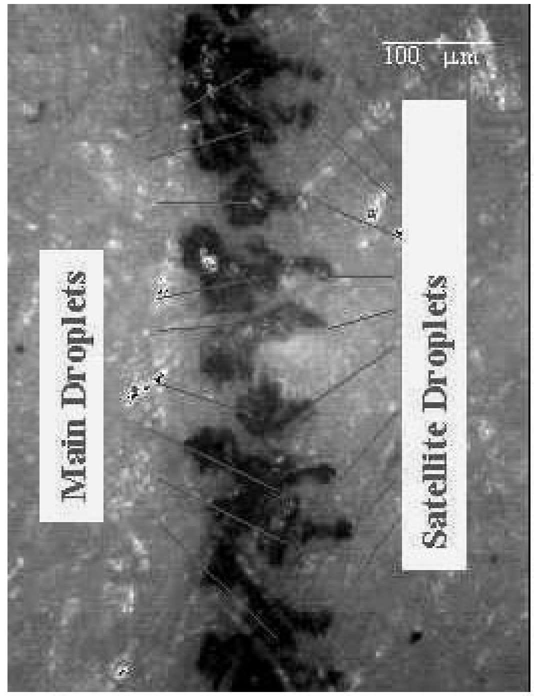


Schematic of the droplet visualization setup, courtesey of Dr. Fan-Gang Tseng.

The droplet injection process is visualized using a digital CCD camera via a microscope system as shown. A phase-triggered LED is used to backlite the process. A very short duration electric pulse is sent to the printhead to initiate the injection process at a frequency of 1 kHz.

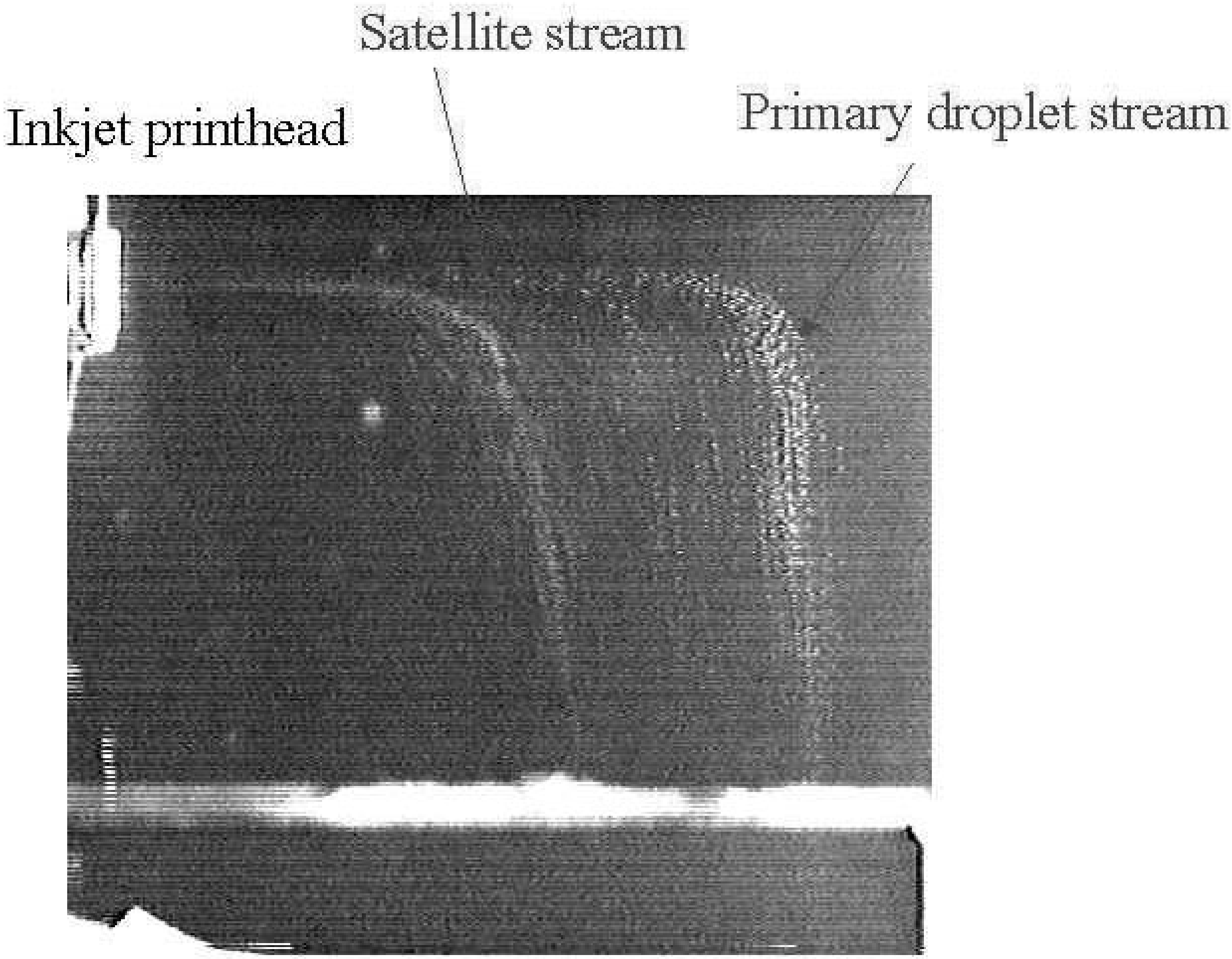
Satellite Droplets

It can be seen from the above animated sequence that the jet column experiences instability and breaks up into a stream of small satellite droplets following the primary droplets. The existence of these small droplets can significantly lower the inkjet printing quality since they have different trajectories from the primary one. As a result, they will fall on different locations from that of the primary droplet (as shown below).



Sample line inkjet printing from a HP printhead, showing the primary and the satellite droplets. Consequently, the printing resolution (as usually characterized by the dots per inch (DPI)) is degraded. The trajectories of these droplets can also be visualized using a CCD video camera. It is clear from the following inkjet injection photograph that the satellite droplet stream has a different trajectory from that of the primary droplet stream.

Droplet Trajectory



Inkjet injection process of a HP printhead.

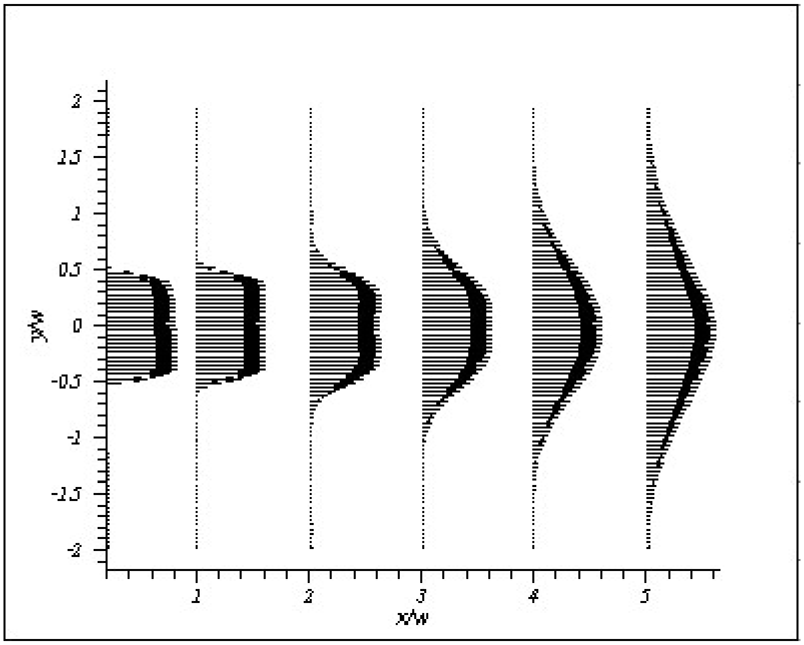
The above photograph shows the inkjet injection process as the droplet streams coming horizontally out of a HP inkjet print head. The ink droplets experience aerodynamic drag and gravitational pulling force when they are exiting from the nozzle. Since the droplets are relatively small (of an order of 50 µm in diameter), the Stokes' drag law is valid for the description of the drag on the droplets. Note: this is not true initially when the droplet velocity is high (of an order of 10 m/s), however, the velocity decreases very fast and the Stokes' law can be applied shortly after the droplet velocity decelerates. It is interesting to note that the droplet trajectory takes a very steep turn as the droplets lose their speed. Also, the satellite droplets follow totally different trajectories as they fall down earlier as compared to the primary. This is mainly due to the relatively lower speed of the satellite droplets.

Jet Flow Field

The following is a typical jet velocity field distribution measured using the PIV. A total of 190 instantaneous PIV velocity fields are ensemble-averaged in order to obtain this time-averaged flow field. The time-averaged jet velocity at the exit of the nozzle is a uniform, top-hat profile. Due to the large velocity difference between the jet and the ambient fluid, a thin shear layer is created and it is highly unstable. The shear layer is subjected to flow instabilities that eventually lead to the generation of strong turbulent fluctuations and the shear layer continuously grows downstream. This highly turbulent shear flow entrains ambient fluid into the jet and enhances the flow mixing. Consequently, the shear layer and the jet spreads laterally outward and the jet velocity decreases downstream. Near the nozzle exit and along the central portion of the jet, a region with an almost uniform mean velocity is called the potential core. Because of the spreading of the shear layer, the potential core eventually disappears when shear layers from all sides merge. The entrainment and mixing process continues beyond the potential core region such that the velocity distribution eventually relaxes to an bell-shaped profiles as shown.

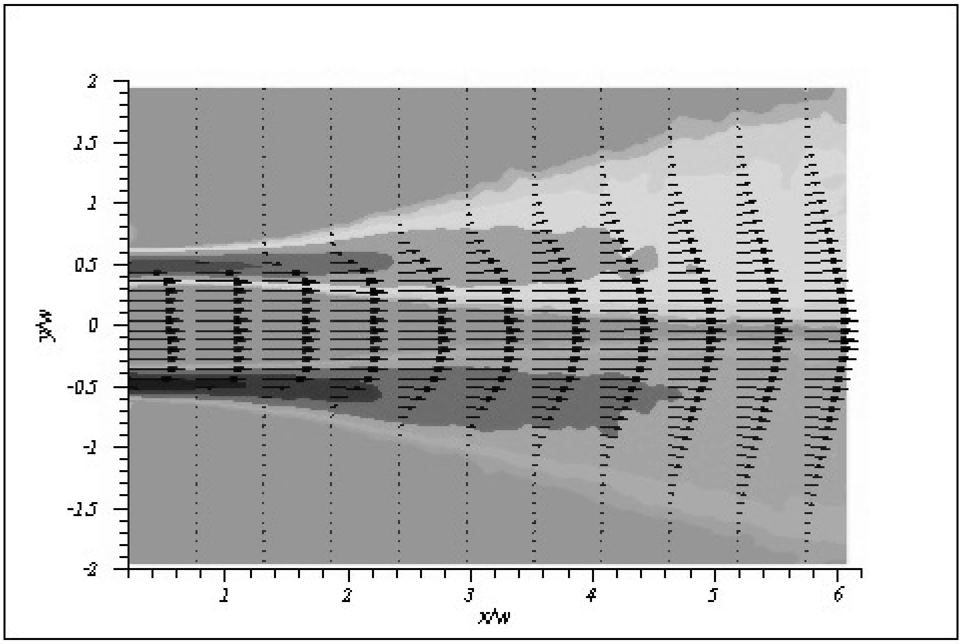
The following color-coded contours show the overall distribution of the velocity field. The jet boundary line can be interpreted as the outmost location where the velocity approaches to zero. It is clear that the jet spreads rapidly downstream.

Averaged Jet Flow Field

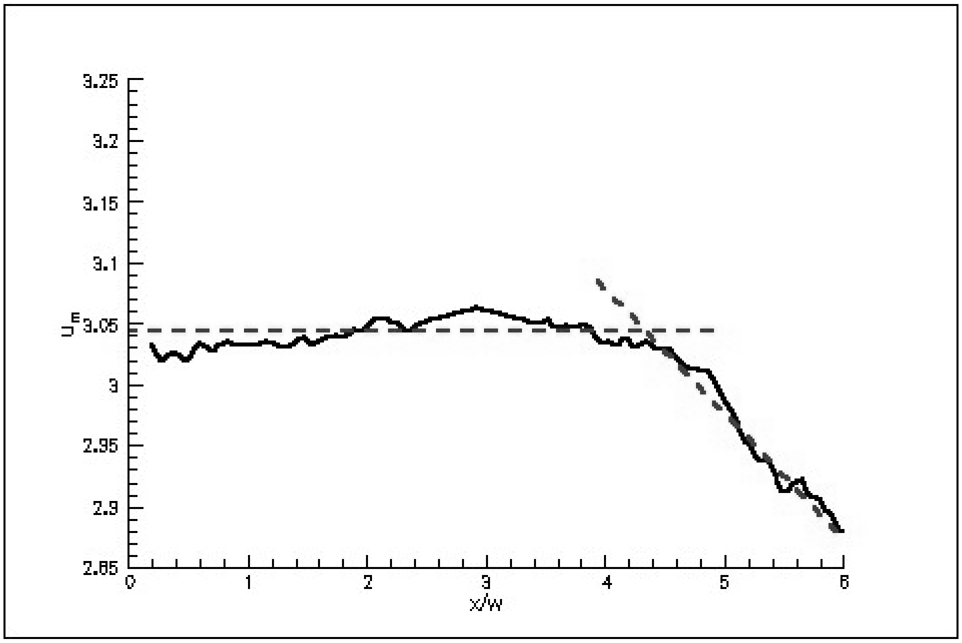


Courtesy PIV jet data from Mr. Bahadir Alkislar and Dr. Luiz Lourenco

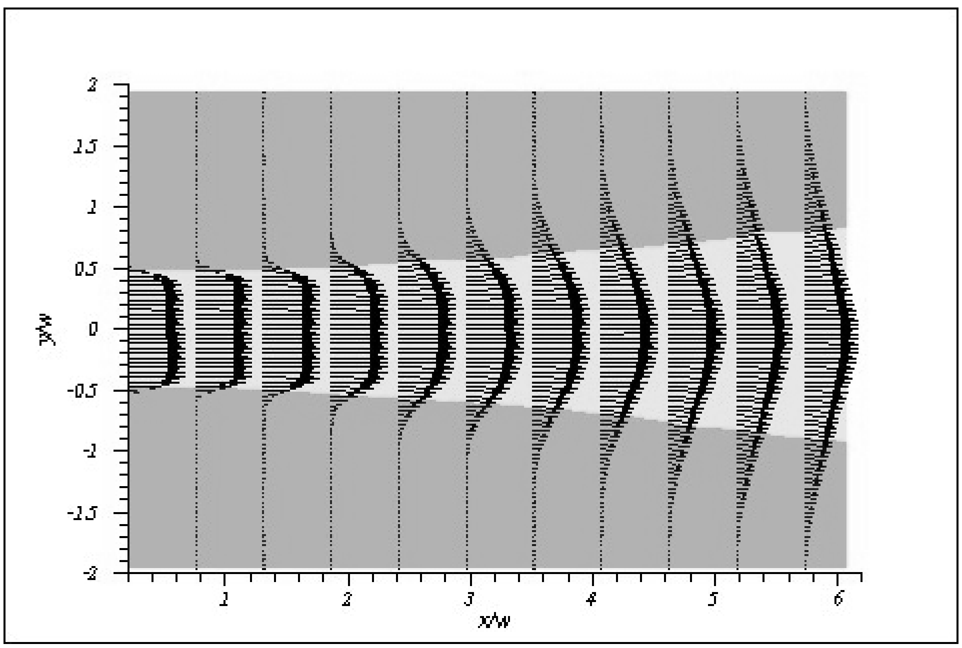
The most effective way to characterize the shear layer is to plot the constant vorticity contour. Inside the shear layer, large velocity difference exists and the vorticity level is high. Away from the shear layer, the velocity gradient is small and the vorticity level is low. Therefore, the contour distribution of the vorticity level represents a good measure of the growth of the jet shear layer.



As evident from the vorticity contour and mean velocity plots, the jet spreads outward and entrains ambient fluid into the jet. Another good measure of the spreading of the jet is the jet centerline velocity. The jet centerline velocity is relatively constant inside the potential core of the jet. As the shear layers merge toward the center, the centerline velocity decreases downstream. The decay rate of the centerline velocity can then be used to characterize the mixing level of the jet.



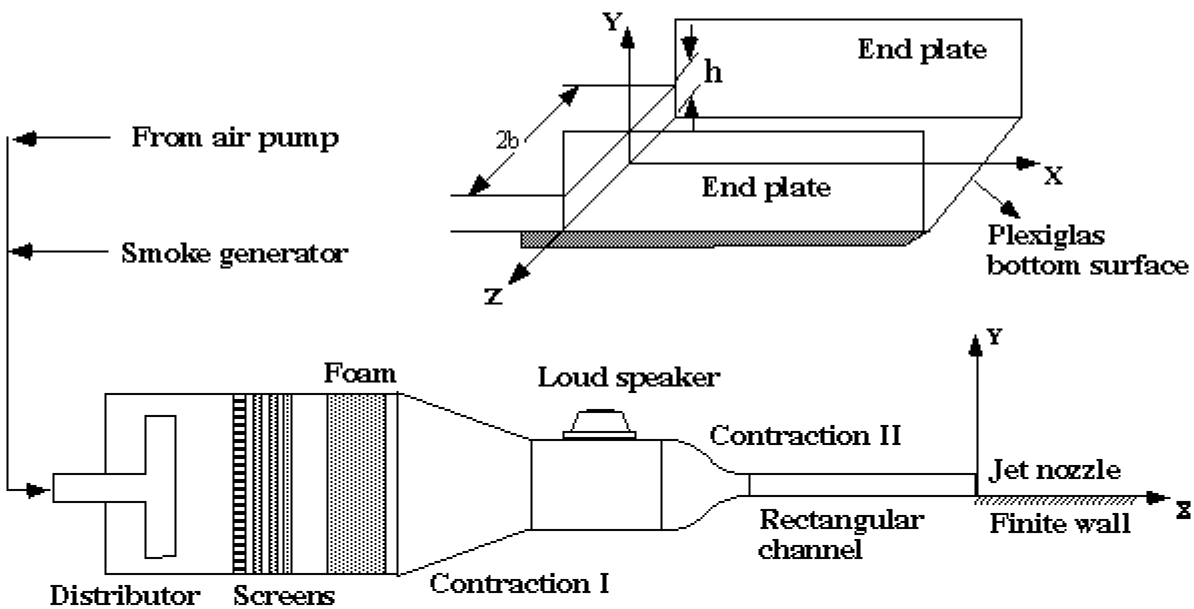
Another commonly used measure to characterize the growth of the jet is define the half jet width. It is defined as the distance measure from the centerline of the jet where the local mean velocity is equal to half of the local centerline mean velocity.



Wall Jet

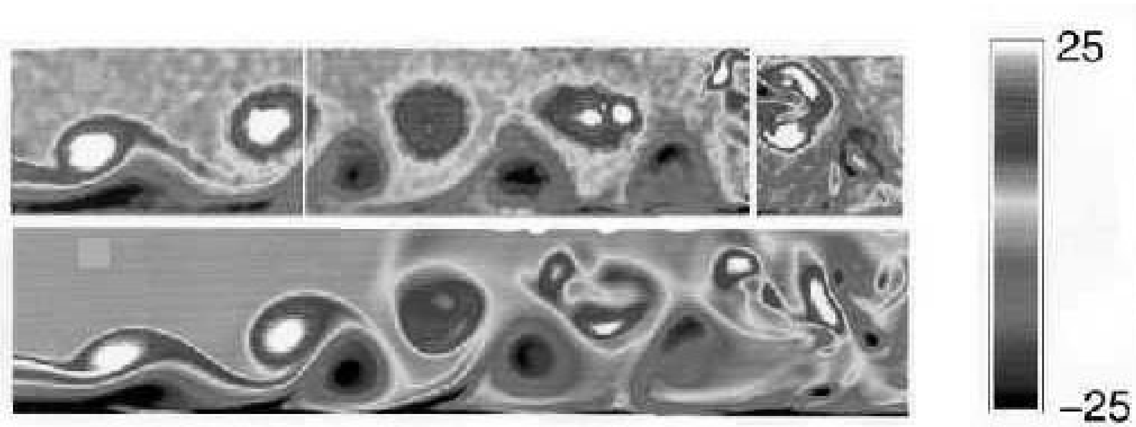
Introduction:

A wall jet is a stream of fluid blown tangentially along a wall and it has a wide range of applications, such as boundary-layer separation control over a wing, film cooling on turbine blades, etc. There are two distinct regions in the wall jet configuration: first, an outer free shear layer that is subjected to the inviscid Kelvin-Helmholtz instability and the formation of large scale vortices; and an inner layer that behaves like a viscous boundary layer. The interaction between large scale structures from the outer layer and inner layer leads to the eventual laminar-to-turbulence transition. In the following, a plane wall jet flow is measured using the Particle Image Velocimetry (PIV), and the experimental data are compared to a direct numerical simulation results calculated using the Navier-Stokes equations. The following schematic shows the experimental setup of the wall jet. Color-coded contour plots are used to characterize the vorticity distrinbution inside the wall jet flow. The red color spectrum represents vorticity rotating in the counterclockwise direction, while the blue spectrum shows the vorticity in the clockwise direction.



Schematic of wall jet facility

The following graph shows a comparison between the PIV data and the direct numerical simulation results. It can clearly be seen that the interaction between the surface vortices (coded blue) and the shear layer vortices (coded red) is strong and the wall jet transitions from laminar to turbulent very fast. Numerical simulation can predict very well the ejection of fluid from the wall because of this interaction.

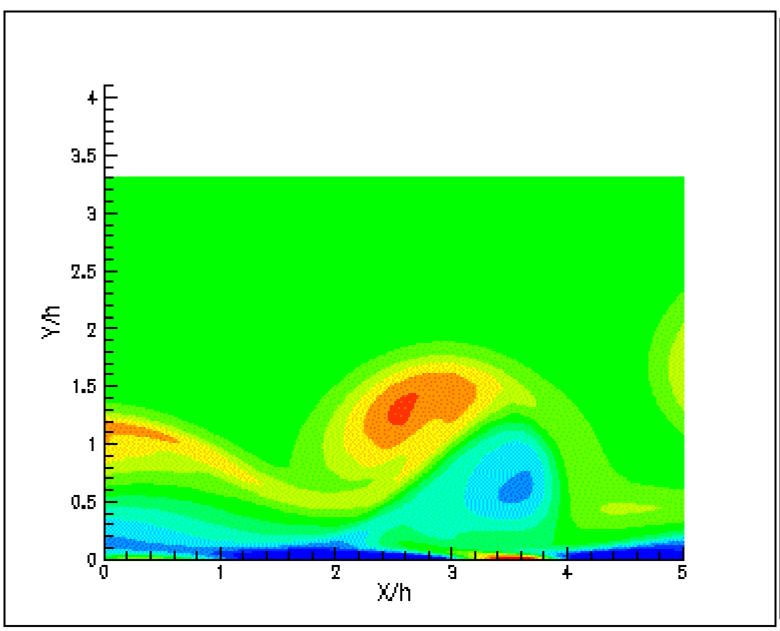


Comparison between the PIV data (top) and the numerical simulation reults (bottom)

Courtesy numerical simulation results from Dr. Migual Visbal

Instantaneous Wall Jet Vorticity Flow Field - A comparison

Direct Numerical Simulation Data



PIV Experimental Data