September 2018

AER210 – Flow Visualization and Measurement Laboratory

The objective of this laboratory is to generate, observe and measure a variety of flow situations; from straight-forward steady flows to complex unsteady flows. The apparatus is designed to provide multiple methods of visualization and measurement. You will be required to do a number of basic calculations using two simple equations governing fluid flows:

Bernoulli Equation: $p + \frac{1}{2}\rho v^2 + \rho g h = constant$

Incompressible Continuity Equation: $Q = Av = constant$

where:
A = cross-sectional area
g = acceleration due to gravity
h = relative height
p = static pressure
Q = volumetric flow rate
v = velocity
ρ = density

The following conversion factors from Imperial units to SI units may be useful in this lab:

1 Gallons Per Minute (GPM [US]) -> **6.309×10^{-5}** cubic metres per second (m^3/s)
1 Pounds per Square Inch (PSI) -> **6894.76** Pascals (Pa)

All work is to be completed within the 3-hour lab period.
Completed assignment sheets are to be handed in to the TA before leaving the lab.

CAUTION: Students that are sensitive to stroboscopic (flashing) light effects should inform the TA as these will be encountered during the lab.

Description of Apparatus

Flowcoach Experimental Apparatus

This lab makes use of the Flowcoach apparatus developed by Interactive Flow Studies Corporation (http://www.interactiveflows.com). The equipment consists of a desktop recirculating water tunnel that can be used to investigate the flow around various model inserts. The flow around these inserts can then be examined using qualitative visualisation techniques, through the introduction of air or foam bubbles, or using quantitative measurements of pressure or Particle Image Velocimetry (PIV). The accompanying FlowEx™ software also contains a Computation Fluid Dynamics solver that can be used provide a numerical comparison for the experimental flow. The Flowcoach apparatus is shown in Figure 1.

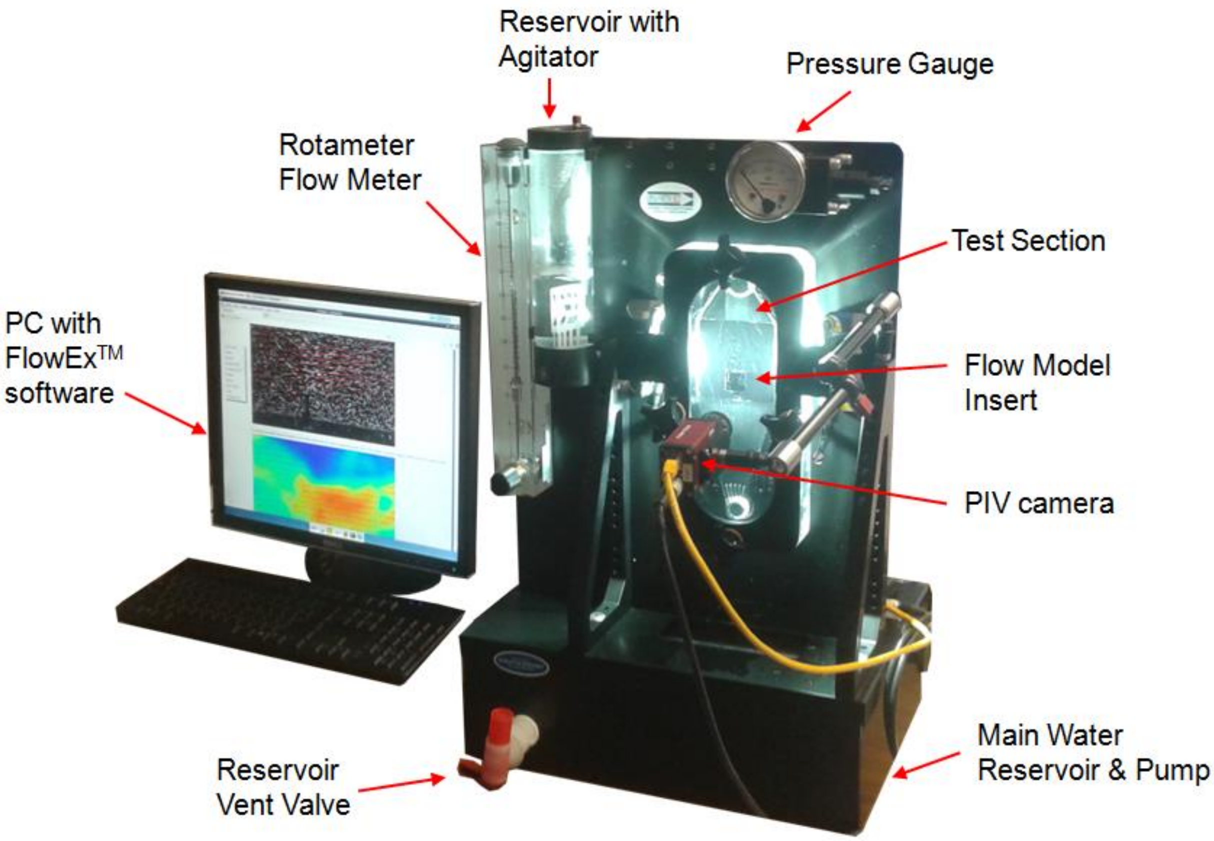


Figure 1: Flowcoach experimental apparatus.

The test section of Flowcoach is accessed by removing the acrylic face plate. A schematic of the test section is shown in Figure 2. The flow enters through the hole at the bottom and passes up through the guide vanes towards the model. Three models are available for this setup; a Venturi model, a square bluff body model and an orifice comparison model. Each model is designed to demonstrate different flow phenomena.

In the test section, two pressure ports are located to coincide with the entrance and throat of the Venturi model. These two ports are connected to either side of the differential pressure transducer. Once the flow passes the model, it is exhausted from the test section through the outlet at the top. An O-ring is seated around the test section to create a seal with the face plate, thus preventing water leaking from the system when it is operational.

The height of the test section is 5 mm for all inserts.

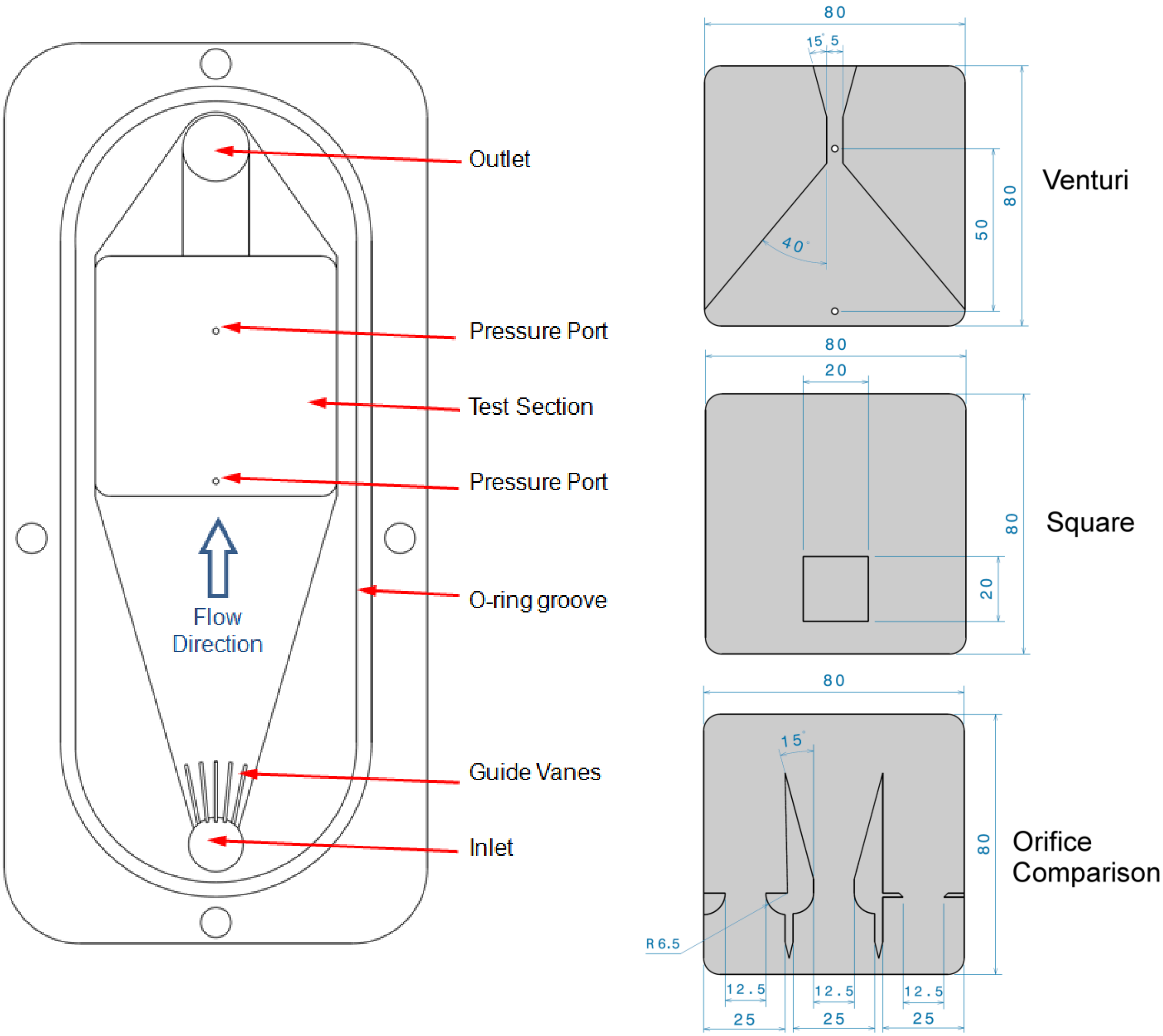


Figure 2: Flowcoach test section and model inserts

Operation of Flowcoach

Prior to operating Flowcoach, read the instructions fully and take a look at the equipment to identify the major controls and how they operate. If you have any questions or doubts about how to use the equipment, ask your TA.

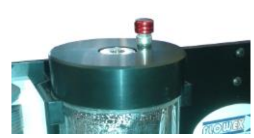
- Fill the main reservoir with water - The water level should be around 10cm from the bottom when the system is running (just covering the pump). Once the system is operational, the water level can be filled up if too low.
- Insert model into test section
  - With the pump off, remove flow model cover plate using the four thumb screws. If a model is in the test section already, be careful to ensure the model does not fall out when the cover plate is removed.
  - Insert new model and appropriate back plate into test section. For the Venturi model, the back plate contains two drilled holes for access to the pressure ports, whereas for the other models the plate has no holes. To prevent a dry model from falling out when the cover is removed, wet the back of the model inserts prior to placing them in the test section.
  - Carefully clean the inside of the model cover plate using the Armorall wipes. Gently dry using the microfibre cloth.

  - Ensuring the O-ring seal is correctly located, replace the model cover plate and tighten with the four thumb screws

- Ensure the Reservoir vent valve is closed - You should see the black side of the handle facing you. For extra precaution, position Flowcoach such that the vent valve is off the edge of the table or over a bucket.

- Ensure the cap on the upper reservoir (containing the agitator) is securely closed.
- Switch on the pump and LED lights at the power bar

  *** The LED lights are water cooled and therefore to prevent overheating and damage, the pump must be running if the LED lights are turned on. ***
- Increase water level in upper reservoir
  - When the pump is running and water is flowing, press the red nipple on the cap of the upper reservoir to increase the water level in the upper reservoir.



- To prevent air bubbles from being drawn into the system when not wanted, a water level that fully covers the agitator is desired.

- Control the flow rate using the rotameter control knob
  - Turning the knob clockwise reduced the flow rate and therefore reduces the flow speed. Turning the knob anticlockwise increases the flow rate and flow speed.
  - The maximum flow rate attainable is approximately 2 Gallons Per Minute (US Gallons).
  - The equipment can be operated with the pump running and the flow meter fully closed, however prolonged operation in this state is undesirable and may cause problems with the pump.

Stopping the Flow

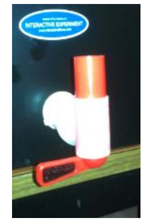
- Ensure the red vent valve is fully closed (the black side is facing away from the system)
- To stop the flow, turn off the pump using the switch on the power bar. Turn off the LED lights in a similar manner to prevent them from overheating.
- Emptying the water from the test section
  - When the power is switched off, water may remain in the system.
  - Press and hold the red nipple on the upper reservoir to drain as much water as possible.
  - When no more water drains, slowly open the thumb screws on the test section cover plate. Water will then drain from the bottom of this plate back into the main reservoir through the opening in the reservoir housing. Be careful to perform this action slowly to prevent spilling water onto other components.

Flow Visualisation

Two methods of flow visualisation are available for the Flowcoach equipment. These are the introduction of air bubbles or the introduction of foam bubbles. Both methods operate using the same procedure, however for the foam bubbles to be created, a foaming agent must be introduced into the water.

*** To prevent water spillage from the vent valve, ensure the red vent valve is closed before stopping the water flow or turning off the pump. ***

Bubble Generation



- When water is flowing, turn the red reservoir vent valve to open it and allow air to be drawn into the pump (red side facing away from system).
- The generation of bubbles is only effective at flow rates above ~1.3 GPM.
- To stop bubble generation, turn the reservoir vent valve until the black side faces outwards, indicating that the valve is closed.

Foam Generation

- Ensure the red vent valve is closed prior to introducing foaming agent. To add foaming agent,
  - fully close the rotameter valve to stop the flow, or turn main pump off
  - unscrew and remove the cap on the upper reservoir
  - add a SMALL drop of foam solution into reservoir
  - close upper reservoir cap
  - open rotameter valve to maximum flow rate to mix the foaming solution, or turn main pump back on if switched off
  - increase water level to desired height in upper reservoir using red nipple on reservoir cap
  - turn red vent valve to open position in order to introduce air into the pump and start the generation of foam bubbles.
  - The amount of bubble can be controlled by adjusting the flow rate or by varying the vent valve position to restrict the air being drawn into the pump.
- Pay attention to the buildup of foam in the main water reservoir to ensure it doesn't overflow.
- The generation of foam bubbles is only effective at flow rates above ~1.3 GPM
- To stop the generation of foam bubbles, turn the vent valve so that the black side of the handle is facing away from the reservoir.

Removing Excess Bubbles

When using the flow visualisation techniques, excess bubbles can form on the model and in the test section. The following procedure can be used to try and remove some of these bubbles prior to having to disassemble and clean the model insert.

- Fully close the rotameter knob to stop the flow of water through the test section.
- Press and hold the red nipple on the upper reservoir cap to drain the water from the upper reservoir.
- This will generate large air bubbles in the test section that will clean some of the bubbles from the surface.

- Open the rotameter knob to restart the water flow.

If the model is still covered in bubbles and these need to be removed, the procedure detailed previously to stop the flow and replace the model insert should be followed. Once you have removed the model insert, use the Armorall wipes to clean the appropriate areas.

FlowEx^{TM} Software

The FlowEx^{TM} software accompanies the Flowcoach experimental apparatus and provides additional experimental functionality through both Particle Image Velocimetry, as well as a Computational Fluid Dynamics Solver. The FlowEx^{TM} software is a purpose built Linux distribution. The lab computers will automatically boot into the FlowEx^{TM} desktop when turned on. To run the Flowex service, click on the "Flowex Web Interface" icon on the desktop. This will open the welcome page in the internet browser.

Instructions to operate the different features of the software can be found on this welcome page. The main navigation is achieved using the sidebar, shown in Figure 3.

After booting into Flowex^{TM}, the system needs to be setup. The setup page is shown below in Figure 3. The first step is to update the server IP address. The computer is connected to a router, which in turn is connected to the PIV camera. To ensure proper operation, the IP should be updated using the update button. The next step is to load the default settings for the Flowcoach system by clicking on the image of the Flowcoach apparatus, as highlighted. For now the only parameter that must be changed is the camera type in the "General" settings section. This should be changed to "PIV". The other settings for the PIV and CFD sections will be discussed later and can be left as the default values. Click the "Save" button.

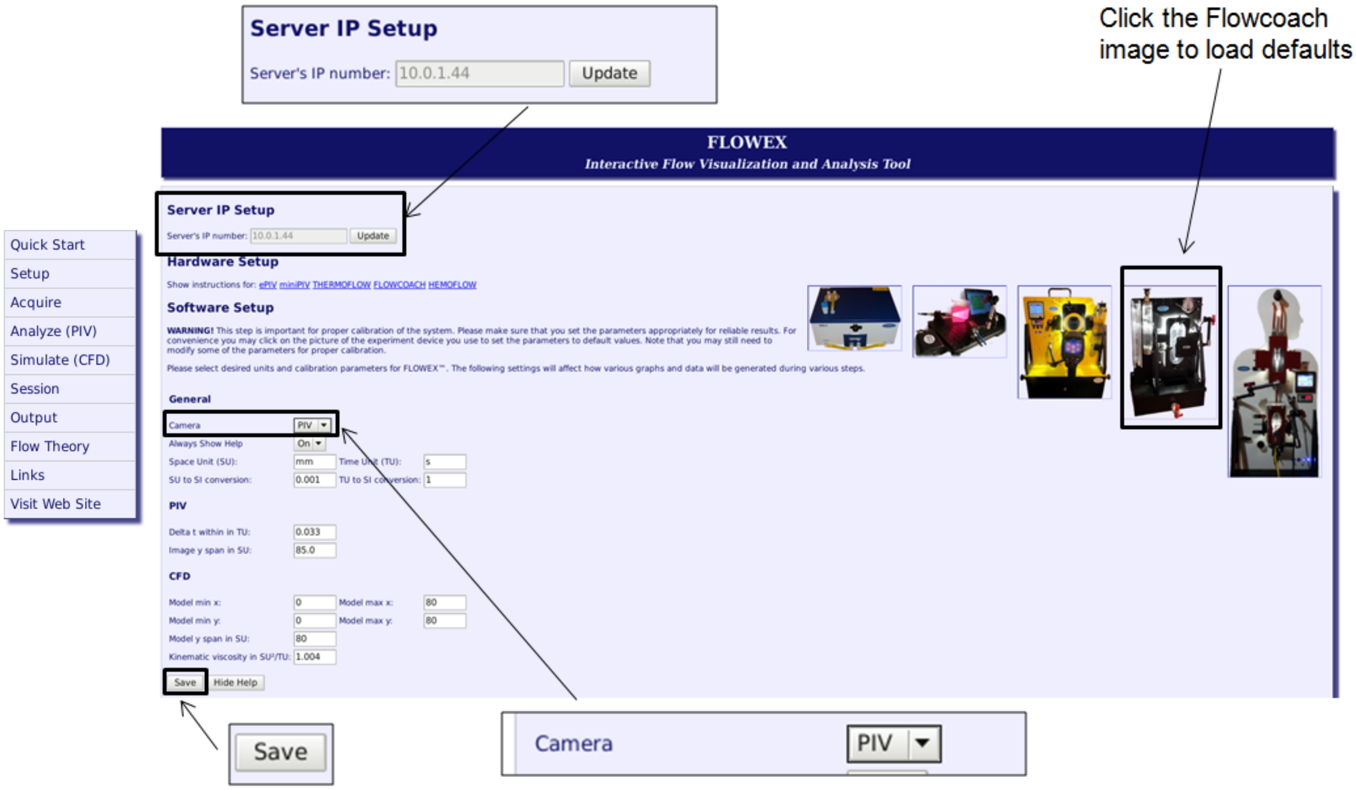


Figure 3: Setup page within FlowEx^{TM} software

With the initial setup complete, take time to explore the other pages shown on the sidebar. At the bottom of each page, there is help information relating to the options and settings on that page. If this is not visible, click on the "Show Help" button and it will appear.

Under the "Session" page, the data generated from operating FlowEx^{TM} can be saved to a USB stick. If you have a session already saved, it can also be loaded here. For the current labs, the sessions should not have to be exported as all work will be performed during the lab session.

To view the output data from either the PIV or the CFD computations, clicking on the "Output" sidebar option will open the output folder in the web browser. Alternatively, the generated files can be viewed in the Output folder directly, which is located on the desktop.

Particle Image Velocimetry

Particle Image Velocimetry (PIV) is a technique used in the experimental study of fluid flows that enables the measurement of velocity vectors for whole flow fields in a non intrusive manner. The basic principle behind PIV measurements is the determination of the time dependant displacement of tracer particles that have been introduced into the flow being studied. This is generally achieved using a high powered illumination source, such as a laser or LED, which is flashed twice with a set, known time interval between pulses. A digital camera captures two frames showing the illuminated tracer particles at the time of each illumination pulse. Each image in this image pair is broken down into a number of discrete interrogation windows. For each interrogation window, the two images in the pair are compared using a cross correlation algorithm. For any given window, the subsection of the first image represented by the window is displaced in two dimensional space and compared to the second image, the displacement of the window that provides the highest cross correlation peak is the average displacement for that window. Once the 2D displacement is known for each window in the whole image and if the relationship between camera pixel size to physical dimensions are known, the velocity can be calculated based on the time interval between the two illumination flashes. This therefore provides a two dimensional velocity vector field for the flow at one incidence in time. Repeating this process multiple times in quick succession permits the capture of the time variant flow velocities. Alternatively, multiple snapshots can be averaged to determine the mean flow properties.

When performing PIV measurements, a number of parameters must be carefully considered to ensure the validity of the results being obtained:

Tracer particles - the fluid velocity is indirectly inferred from the movement of the tracer particle inside the flow. This means that the tracer particles must be small enough and (for liquid applications) neutrally buoyant in order to faithfully follow the flow. The tracer particles that are employed for the current experiment are Polyamide particles, with a mean diameter of 50 μm and a specific gravity of 1.03.

Density and distribution of tracer particles - to obtain a good quality velocity vector field, the tracer particles should be evenly distributed throughout the whole field of view. As the PIV processing uses the cross correlation peak between two images to determine the window displacement, the greater the number of particles contained in each window results in a better correlation. It is therefore desirable to have a homogenously dense seeding of the interrogation area.

Illumination - the tracer particle illumination is important to ensure that the camera is able to adequately capture the particles with high enough light intensity so that they stand out from the background. Achieving suitable tracer illumination is a combined balance of the particle size, its light scattering abilities, the intensity of the illumination source, the length of the illumination exposure and the sensitivity of the camera. In the current experiment, the tracer particles are provided, meaning their size and reflectivity are fixed. The illumination is provided by two 16W LED units mounted beside the test section. These can be adjusted to alter the light intensity within the test chamber by loosening the two thumb screws that fasten the LEDs to the back plate, adjusting the position of the LEDs with respect to the test chamber and then retightening the screws.

The length of the illumination exposure is indirectly controlled within FlowEx™. It is a function of the camera frame rate and the time between images. The shorter time between image pairs results in a shorter exposure time, thus less light illuminating the particles. If the illumination time it too short, the particles will not be bright in the image. However, if the illumination time is too long, the particles will appear as streaks on the image rather than discrete particles. The sensitivity of the camera can be controlled using the aperture settings on the camera lens.

Time delay between illumination pulses - the time delay between illumination pulses must be set so that it is able to capture the displacement of all the particles in the flow with sufficient resolution. For example, in a flow with a large dynamic velocity range, the tracer particles in the area of high velocity will show a large displacement for a given illumination delay, whereas the tracers in the low velocity area will move very little. If the delay between images is too small, any slow speed flow that is below a certain threshold value (based on the delay) will appear to the PIV camera to be stationary. Once processed, this will result in large areas of zero flow speed when in fact the flow is actually moving, albeit slowly. Conversely, if the delay between images is too large, the higher speed particles will be displaced a large amount. Because the PIV algorithm limits how much each interrogation area can be displaced, if many of the particles move further than these limits, they will not contribute to the cross correlation and therefore the mean flow velocity for that window will not be a true representation of the flow speed. Furthermore, for three dimensional flows where the illumination source is a light sheet and the tracer particles can be displaced in three dimensions, if the delay between image pairs is too long the tracer particles may have moved out of the illumination plane at the time of the second pulse. Therefore they would not contribute towards the cross correlation.

The previous discussion provides only a very brief introduction to the topic of PIV. For more information see Raffel (2007).

PIV in Flowcoach and FlowEx™

To use the PIV feature of Flowcoach, the PIV camera must be connected to the synchronizer and a gigabit router. The synchronizer is then connected to the LEDs and the gigabit router is connected to the FlowEx™ computer using an Ethernet cable. A schematic of the system setup is shown below in Figure 4.

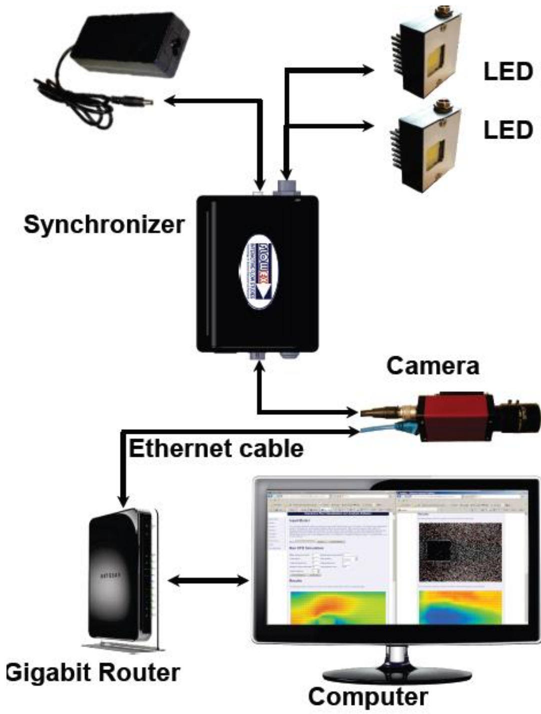


Figure 4: PIV hardware setup

- Adding tracer particles into the water
  - Fill a small jar or container with water to approximately 1/2 - 2/3 full.
  - Add a small scoop, approximately 1/4 teaspoon size of polyamide PIV particles to the water.
  - Close the container and shake to mix the particles with the water.
  - If the particles appear to be clumping and not mixing well, add a very small drop of dishwashing soap or foaming agent to the water. Close and shake to mix.
  - Once the particles are fully mixed, the liquid should look cloudy
  - Introduce the particle mixture into Flowcoach in the same manner as the foaming agent used for the flow visualisation
  - This step may have to be repeated to increase the seeding density once the mixture has dispersed.
  - An adequate seeding density will look like a dense star scene to the camera

The PIV camera is an Allied Vision Manta 46G. It has a maximum frame rate of ~67 frames per second and a sensor resolution of 780x580 pixels.

- Position the camera
  - The camera is mounted on an adjustable arm. Always support the camera with one hand and to loosen the arm, turn the handle located half way along the arm.

  - Move the camera into the desired position by roughly aligning the camera to your desired field of view.
  - Lock the arm in place by turning the arm handle clockwise.
    *** DO NOT OVERTIGHTEN ***
  - In the FlowEx™ software under the "Acquire" page, ensure that the camera type is set to PIV and then click the "Adjust Camera Settings" button. You will then be given the options to choose the camera frame rate (FPS) and time between image pairs (delta t). Select a suitable value for your given flow speed and model.

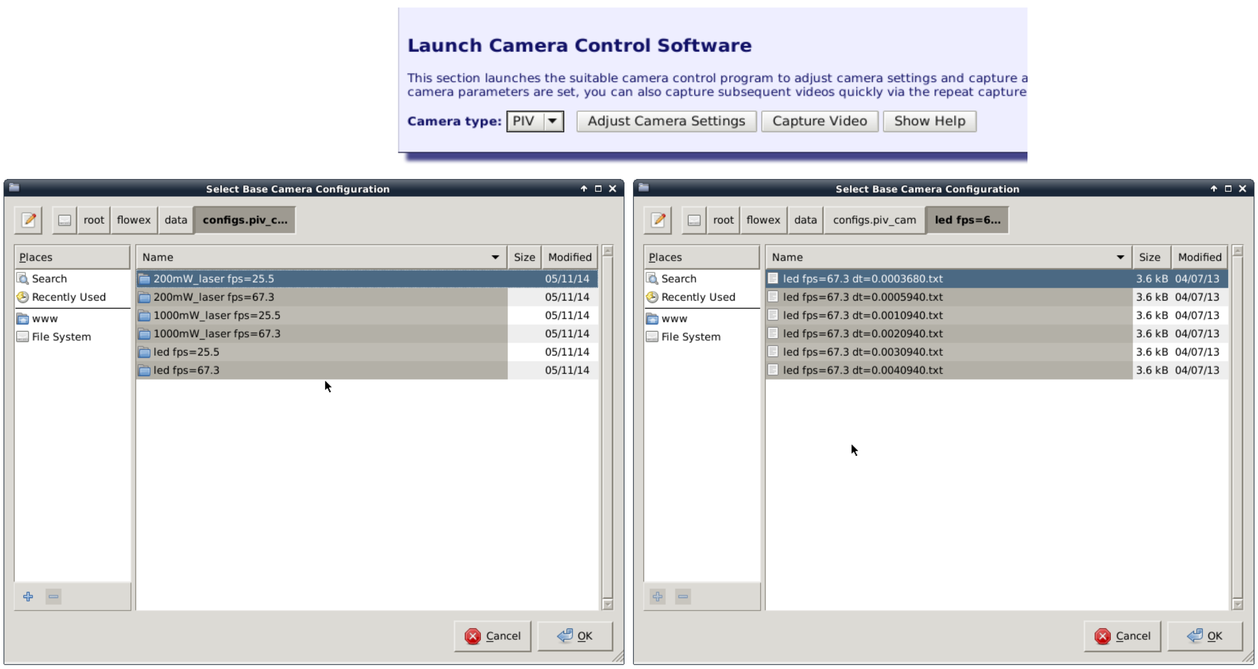
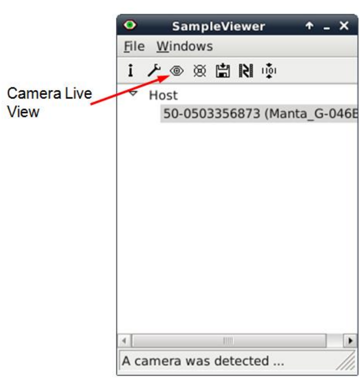


Figure 5 - PIV Camera setting option windows

  - In the camera window, click the eye icon to view the camera.



- Adjust the camera position to get the desired field of view in the image and then lock the support arm using the handle on the arm.
  *** NEVER FORCE THE CAMERA TO MOVE WHEN THE ARM IS LOCKED***
- Focus the camera using the focus ring on the lens and adjust the aperture if required using the aperture settings on the lens.
- Close the video window and the camera settings window.
- If you are not happy with the deltaT and camera FPS, experiment with different values. The smaller the deltaT the more suited to faster flows, although the illumination is also reduced.
- When you have a suitable FPS and deltaT, note it down for later.

• LED pulsing
  - To 'freeze' the particles in the image pairs, the LEDs must flash on for a short exposure time. The timing settings for the flashing is controlled in the previous step when adjusting the camera settings.
  - To switch the LEDs between constant light to flashing, flip the toggle switch on the synchronizer.

  - If the LED appear to be off when switched in flashing mode, go back into the camera settings and select a new FPS and deltaT setting.
  - To switch back from flashing mode to constant mode, flip the toggle switch on the synchronizer.

IF YOU ARE SENSITIVE TO FLASHING LIGHTS, YOU SHOULD NOT PERFORM PIV USING THIS SYSTEM

• Capturing Images
  - When you are happy with the camera positions and settings, capture images by clicking on the "Capture Video" button.
  - The software will capture a set of images and save them in the video folder, located on the desktop. It will also create a preview.gif image that will be displayed on the bottom of the page.
  - If you are happy with the image, you can proceed to processing.

- Processing PIV images.
  - Prior to processing the PIV images, return to the "Setup" page and write the deltaT value that you selected for the video capture into the appropriate box.
  - To convert the image pixel space into physical dimensions, a scale factor must be defined and applied to the image. In the PIV settings, the "Image y span in SU" is the scale factor. The physical scale of your camera field of view can be estimated using the ratio of the size of a known object (such as the flow model) in your camera image, compared to the overall image size.
  - In the sidebar, go to the "Analyze" page.
  - Here the PIV analysis can be performed and the results presented.
  - Descriptions of each of the parameters is provided in the help text at the bottom of the page. Read the details and discuss with your TA anything that you are unsure about.
  - When you have input the appropriate settings, click the "Run all steps" button and wait for the processing to be performed.
  - Depending upon the output settings, the PIV results will be presented at the bottom of the page once completed.
  - If you click at a location in the displayed PIV image, the velocity data at that point will be plot against the image pair snapshot number.
  - To view the analyzed PIV data, either click on the links at the bottom of the page or open the Output folder on the desktop.

Computational Fluid Dynamics Solver

Computation Fluid Dynamics is method used in the study of fluid flows that is based on numerically solving the governing fluid dynamic equations. In its most popular form, methods to numerically solve the Navier-Stokes equations are employed. However, the computational cost of solving the full equations is prohibitive, therefore simplified versions of the equations are more commonly chosen. These simpler equations neglect various terms of lesser importance for a given scenario. Through this simplification process, the computational cost of performing CFD studies on complex geometries can be reduced to an acceptable level, whilst still achieving satisfactory validity of the results. The validity of the results from CFD computations is not only dependent on the flow equations used in the code, it is also highly dependent on the spatial and temporal discretization used in the numerical scheme. For time varying flows, such as those which exhibit vortex shedding or high levels of turbulence generation, the simulated time step between iterations of the numerical solution must be small enough to allow accurate modeling of the flow behaviour. Similarly, the spatial discretization must be fine enough such that the small details in the flow can be properly resolved and not averaged out. Reducing the time step and increasing the number of grid cells in the spatial discretization both increase the computational cost of the code, so careful consideration must be paid to the balance between time and solution acceptability.

Within FlowEx^{TM}, two CFD implementation are available. The first is the Gerris CFD solver and the second is OpenFOAM. At present, only the first solver, Gerris, is fully operational and will therefore be used for this lab. Gerris is an open source CFD solver based on the time-dependent, incompressible Euler equations (Popinet, 2003). The Euler equations are a simplified version of the Navier-Stokes equations, where the viscous terms have been neglected. The Gerris solver uses a partially structured mesh for spatial discretization that is based on a quad/octree scheme. The meshing of the geometry is fully automatic with the mesh cell size reducing to a finer spacing close to the geometry body. The solver also has the ability to adaptively refine the mesh based on certain triggering criteria during the CFD computation. This ensures that the spatial discretization is adequate to capture the flow behaviour for features which may translate in the spatial domain.

- Running the CFD solver
    - Open the Setup page in FlowEx^{TM} by clicking the sidebar option in the web browser
    - The default settings in the CFD section of this page should be fine for the simulations performed on the models that are used in Flowcoach. However, if you would like to perform a simulation on your own model, these parameters may need to be changed. See the help to understand what each parameter means.
    - Open the CFD solver page in FlowEx^{TM} by clicking on the "Simulate (CFD)" option in the sidebar of the web browser
    - Select the model to be investigated. The models of the three inserts used in Flowcoach are available in the "Models/flowcoach" folder on the desktop. Note: The CFD solver only simulates the model insert, not the whole test chamber.
    - When the model is selected, click the "Upload Model" button.
    - Select the Gerris solver in the option window
    - The remaining options can be changed depending on the type of simulation that you would like to perform.
    - The most important parameter to control the simulation is the "Max flow speed in SI (m/s)". This defines the inlet velocity for the simulation. The inlet to the CFD simulation is equivalent to the location where the model insert starts (not the inlet to the test chamber with its diverging section). An estimate for the required inlet velocity can determined analytically, given that you know the volumetric flow rate through the system and the area of the channel where the model insert starts. Alternatively, the PIV

system can be used to get an estimate of the flow speed at the start of the model insert.
  - Read the help for the remaining settings and discuss with your TA any questions you may have.
  - Depending on the output options that you have selected, once the computation has completed, a summary of the data will be presented at the bottom of the screen. All the files are contained in the "Output/cfd" folder located on the desktop.

Cleaning Flowcoach after use

At the end of the day when Flowcoach will not be used for some time, the water should be emptied and the system flushed. This should be completed by the course TAs and not the students.

- Disconnect the pump power cable from the power bar.
- Ensure that the water has been drained from the model test section using the procedures defined previously for replacing the model inserts.
- Using the red vent valve on the main reservoir, empty as much water into a bucket as possible.
- When no more water will drain from the main reservoir, close the red vent valve.
- Unscrew the two thumb screws connecting the main water reservoir to the upper section of Flowcoach
- Disconnect the white, water inlet connector that is located at the back of the rotameter.
- Using the handle at the top of Flowcoach, carefully lift the upper portion away from the water reservoir.
- Take the water reservoir containing the main pump to a sink and drain the remaining water.
- Rinse the inside of the reservoir
- Fill the reservoir with fresh water, so that the level covers the pump.
- Take it back to the bench and replace the upper portion of Flowcoach
  - remember to connect the water inlet connector (white connector located behind the rotameter)
  - ensure the vent valve, the upper reservoir lid and the model insert face plate are all securely fastened.
- Run the clean water through the system for at least 1 minute
  - the pump may sound slightly strange when starting up from empty. This is normal and will stop as the pump fills with water.
- Repeat the process of emptying and running fresh water through the system once more.
- Drain the last water as described previously.
- The apparatus should now be clean.

References:

Raffell, M., (2007). *Particle Image Velocimetry - A practical guide.* 2nd ed. Springer-Verlag Berlin Heidelberg.

Popinet, S. (2003). *Gerris: a tree-based adaptive solver for the incompressible Euler equations in complex geometries.* Journal of Computational Physics 190(2): pp 572-600