Document in Progress

AAE 33301 Lab Manual
Fluid Mechanics Lab
School of Aeronautics and Astronautics
Purdue University

Draft: Prepared in Fall 2023 - Summer 2024

Last Updated: August 13, 2025

Author: Dr. Sally Bane

Compiled by: Adarsh Agrawal

Contents

Chapter 1

Flow Visualization of Boundary Layers and Wakes

1.1 Introduction

In the realm of fluid dynamics, understanding boundary layers and wakes is critical for elucidating the intricate behaviors of fluids flowing past solid surfaces. The formation of boundary layers on solid surfaces and subsequent boundary layer separation and formation of wakes is a direct result of fluid **viscosity**.

Viscosity is crucial in fluid flow because it determines how resistant a fluid is to deformation or flow. In practical terms, viscosity influences the ease with which a fluid moves through pipes, channels, or over surfaces. For example, high-viscosity fluids like honey flow sluggishly, while low-viscosity fluids like water flow more freely.

Viscosity plays a pivotal role in fluid flow near solid surfaces due to the phenomenon known as **boundary layer** formation. As a fluid flows adjacent to a solid boundary, the viscosity determines the velocity gradient within this boundary layer, which significantly influences heat transfer, drag forces, and mass transport near surfaces, making viscosity a critical factor in engineering applications.

If the boundary layer separates from the surface of a solid body in the flow, a region of recirculating flow forms immediately behind the body, which is referred to as a **wake**. The wake is one of the most fundamental flow configurations in fluid mechanics and usually involves significant momentum and energy transfer. Its importance is due to a variety of phenomena that have scientific as well as practical interests. Frequently observed examples include flow behind flying aircraft, ground vehicles such as trains and cars, and boats or submarines.

Flow visualization refers to the technologies that make the invisible flow visible. Many flow visualization technologies rely on the addition of tracers, either dye or particles, whereas others rely on existing properties of the fluid, e.g., density gradients. Flow visualization provides a quick, qualitative assessment of the flow field, guiding initial concept development, and the design of a more elaborated investigation. Advanced visualization techniques also provide detailed quantitative information. Flow visualization also has the advantage of describing the entire flow field, unlike many velocity probes that provide information at only one point or along one line. For unsteady flows, in particular, visualization can render details of the flow field far more quickly than a compilation of point-by-point measurements. Often the most effective scheme is to use flow visualization to describe the general characteristics of the field and then to use select point measurements to provide detail.

1.2 Lab Overview

This experiment is designed to demonstrate two fundamental flow fields that occur as a consequence of the fluid viscosity: 1. laminar wakes behind cylinders and 2. flat plate boundary layers. The concepts of boundary layers, flow separation, and the formation of Kármán vortices will be discussed in an introductory manner. These flow fields will be visualized in a water tunnel using small hydrogen bubbles or the large oxygen bubbles that follow the flow. The bubbles are generated through electrolysis by applying a voltage across a very thin wire immersed in the water. At the beginning of the lab, you will program a power supply and function generator to produce pulsed streams of hydrogen bubbles and track these streams to calibrate the water tunnel (motor frequency versus flow velocity). Then, various cylinders will be placed in the water tunnel and the hydrogen bubbles will be used to visualize the wakes behind the cylinders so the shedding frequency of the Kármán vortices can be measured. Bubble visualization will be used to track the boundary-layer growth on the exterior and interior of the plates forming the channel.

1.3 Objectives

Upon the completion of this experiment you will be able to:

1. Understand the operation of a water table/tunnel.
2. Use simple techniques to visualize fluid flow around a body.
3. Understand basic concepts of viscosity, boundary layers, and boundary layer separation.
4. Calculate the Strouhal number from a measured frequency.
5. Measure the boundary layer growth on a flat plate.

1.4 Background

1.4.1 Boundary Layers

The boundary-layer concept was introduced at the turn of the century by L. Prandtl to treat the effect of viscosity in a flow along the surface of a body. Viscosity is the effect of friction between fluid elements, or fluid elements and surfaces.

Prandtl considered the flow to be composed of two regions:

1. A very thin region in the neighborhood of the body where the viscous (friction) effects cannot be neglected. This region is called the boundary layer.
2. The region outside the boundary layer where the viscous forces can be neglected. Usually the inviscid region is irrotational.

One of the first cases in which the boundary layer theory was tested is that of the flow along the horizontal flat plate. Figure 1.1 shows the viscous boundary layer region and the inviscid region.

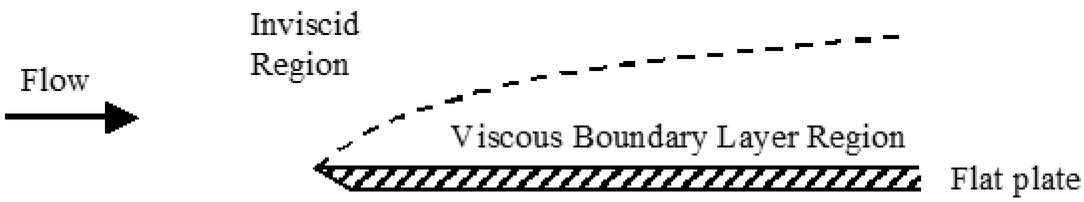


Figure 1.1: Schematic of Boundary Layer on a Flat Plate

1.4.1.1 Boundary Layer Region

This region is a very thin layer along the surface of the body in which the fluid speed changes very rapidly, from zero at the surface, to the freestream speed, U_∞, at the edge of the boundary layer. This is shown in Figure 1.2. The condition of zero relative velocity at a surface is called the no-slip condition, and the "edge" of the boundary layer is commonly defined as the location in the fluid where the fluid speed parallel to the body is equal to 0.99 U_∞. The boundary layer thickness, $\delta(x)$, is the distance normal from the body to this edge. The boundary layer thickness increases along the plate due to the diffusion of vorticity as the fluid travels along the body. Friction between the layers of fluid gradually slows down fluid that is farther and farther from the wall.

The velocity gradient is greatest near the wall. It decreases as the edge of the boundary layer is approached and the fluid speed approaches that of the free stream. A shear stress is present as a result of viscosity and the velocity gradient. Newton's law of friction, generally valid for flows of gases and water, gives equation 1.1.

$$\tau = \mu \frac{du}{dy} \tag{1.1}$$

where τ is the shear stress, μ is viscosity, u is the velocity in the direction parallel to the flow, and y is the direction normal to the flow.

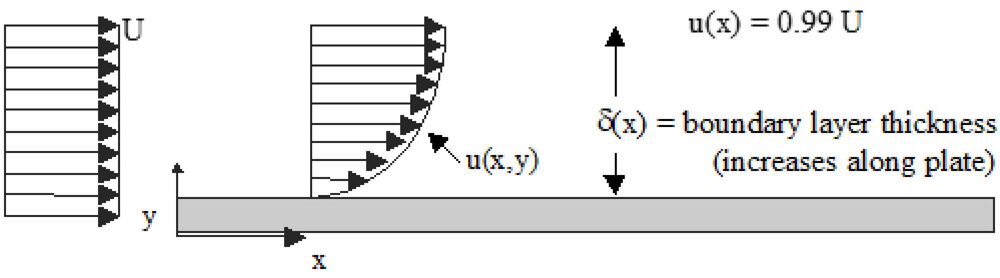


Figure 1.2: Sketch of Boundary-Layer Growth

Analytical results (derived from Anderson's, "Fundamentals of Aeronautics") based on Blasius theory give the boundary layer thickness as represented in equation 1.2

$$\frac{\delta}{x} = \frac{5}{\sqrt{\mathrm{Re}_x}} \tag{1.2}$$

where δ is the thickness of the boundary layer, x is the distance from the leading edge, and Re in the Reynolds number based on the distance from the leading edge.

1.4.1.2 Inviscid Region

This region is the region outside the boundary layer in which viscosity may be neglected. Because of this, no shear stresses are present in the fluid, and the net amount of vorticity is not changed. If this region is considered irrotational, then potential flow solutions can be applied to predict the flow field in this region. The differences between a Viscous and Inviscid Fluid is outlined in 1.1

Flow in the boundary layer is always rotational. This is due to the velocity gradient, as illustrated in Figure 1.3. As a fluid element in a boundary layer moves downstream, leg AA remains nearly horizontal. But the top of leg BB is moving faster than the bottom; thus the fluid element is rotating.

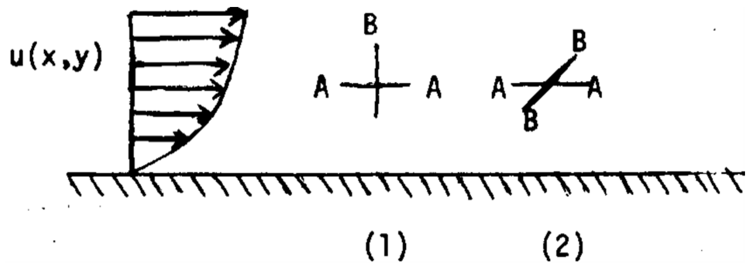


Figure 1.3: Sketch of Rotationality in a Boundary Layer

Viscous ("Real") Fluid	Inviscid Fluid
There is no slip at the wall; the fluid velocity at the wall is zero.	The no-slip condition cannot be imposed at the wall; velocity exists at the wall.
Shear Force (stress) is nonzero at wall and within the fluid.	Shear Force (stress) must be assumed zero at the wall and within the fluid
A boundary layer is present at the wall.	No boundary layer exists.

Table 1.1: Difference between a viscous and an inviscid fluid.

1.4.2 Streamlines and Bernoulli's Equation

Fluid flow is composed of individual fluid particles which move from one point to another. There are two ways to describe the flow:

- **Langrangian Description** is one in which individual fluid particles are tracked. So to say, the observer moves with an individual particle.
- **Eulerian Description** is one in which a control volume is defined, within which fluid flow properties of interest are expressed as fields. So to say, the observer is fixed and is observing particles as they (particles) cross their field of view.

1.4.2.1 Streamlines

While performing theoretical analyses with Langrangian description, we tend to focus on a single particle in the flow. **Streamlines** is a path traced out by a massless particle moving with the flow. The velocity of the particle is tangential to the streamline. It is a powerful tool to understand fluid flow. For an inviscid flow around a cylinder (shown in red), the streamlines can be represented by blue lines in the Figure 1.4

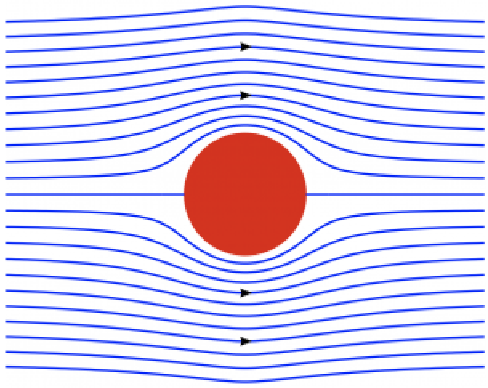


Figure 1.4: Streamlines around cylinder for inviscid fluid flow

1.4.2.2 Bernoulli's Principle/Equation

One of the simplest ways to characterize fluid flow mathematically is by the use of Bernoulli's principle. It states that if the velocity of the fluid flow increases, the static pressure or the fluid's potential energy must decrease. The Bernoulli's equation is derived from a steady-state application of momentum conservation along a streamline. It can be written as in equation 1.3.

$$p + \rho gh + \frac{1}{2}\rho V^2 = constant \tag{1.3}$$

where p is the static pressure, ρ is the density (assumed constant), g is the gravitational acceleration, h is the height of fluid flow above the reference line and V is the velocity. Note that the Bernoulli's equation makes several assumptions to make tha analysis easier. It assumes that the flow is

- inviscid (no viscosity in the fluid)
- steady (flow parameters like velocity do not change with respect to time)
- incompressible (density remains constant along the streamline); possible when $M << 1$

1.4.3 Flow Separation and Kármán Vortices

Figure 1.5 represents the flow about a circular cylinder. Point A is known as the stagnation point. This is the point where a streamline approaches normal to the surface of the cylinder; the velocity here is zero. A streamline is a line drawn in the fluid at a given instant of time so that there is no flow across the line. The velocity of every particle in the streamline is tangent to the streamline. All streamlines above Point A are turned above the cylinder and all streamlines below Point A are turned below the cylinder. Between Points A and B a boundary layer grows.

At point B in Figure 1.5, the velocity at the boundary-layer edge reaches a maximum, and the static pressure reaches a corresponding minimum. The two are related by Bernoulli's equation, as shown in equation 1.3

Point C in Figure 1.5 represents the approximate location where separation occurs. Separation, the breakaway of surface streamlines from the boundary, is caused by a flow reversal in the boundary layer due to an adverse pressure gradient. **Boundary-layer separation is a complex but critical viscous effect (for example,**

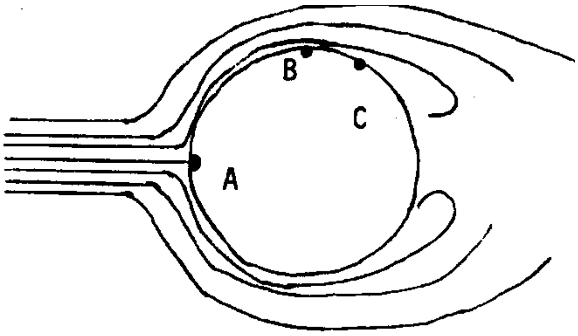


Figure 1.5: Boundary-Layer Separation in Flow Past a Circular Cylinder

it causes stall on airfoils and wings). Separation can cause a flow to be dramatically different from one computed using simple inviscid methods.

A simple mathematical definition of an adverse pressure gradient is given by equation 1.4

$$\frac{\partial p}{\partial x} > 0 \tag{1.4}$$

This means that the pressure increases as the flow moves in the positive x direction, positive being in the direction of the general flow. Figure 1.6 represents a closeup of Points B and C of Figure 1.5 and illustrates the concepts of separation and adverse pressure gradient.

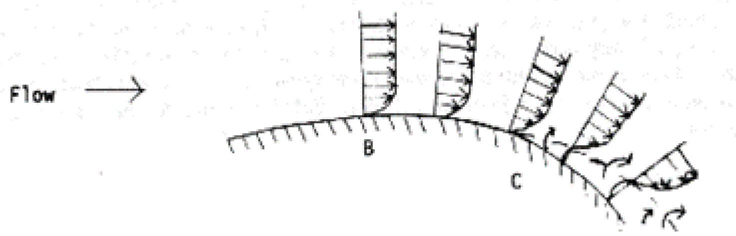


Figure 1.6: Detailed Sketch of Boundary-layer Separation

At Point B the flow velocity has attained its maximum value. As the flow moves toward Point C the velocity near the wall decreases due to viscosity, and the pressure begins to increase because of the shape of the body. This adverse gradient slows the flow down until at Point C the velocity is zero not only at the surface (no-slip condition) but also a small distance above it. This is the initial point of separation. As the flow moves past Point C, the gradient becomes sufficiently strong to turn the flow in the opposite direction.

1.4.3.1 Flow Around a Circular Cylinder

The character of the flow behind the cylinder depends on the Reynolds number. The flow patterns that develop in the wake at different Reynolds numbers will now be discussed: [1].

1. At very low Reynolds numbers ($Re < 4$) the streamlines close behind the cylinder (no separation occurs). The flow is symmetrical fore and aft and therefore no wake is formed (Figure 1.7A).
2. At higher Reynolds numbers ($Re > 4$) the streamlines no longer close and a wake is formed behind the moving body. As will be discussed later in more detail, the velocity in the wake is smaller than the uniform free stream velocity. This velocity deficit is associated with the drag acting on the body. By measuring the velocity distribution in the wake we can estimate the drag on the moving body.

   In the range of Reynolds numbers between 4 and 6 ($4 < Re < 6$), a pair of stationary vortices, called Foppl vortices, are formed behind the cylinder (Figure 1.7B).
3. At still higher Reynolds numbers, from about 6 to 35 ($6 - 10 < Re < 30 - 35$), the vorticity behind the cylinder increases as the pair of vortices moves downstream and elongates (Figure 1.7C).

[1]These ranges of Reynolds number have been adopted from R. L. Panton, "Incompressible Flow", John Wiley & Sons, New York, 2013 and Kundu, Pijush K., Ira M. Cohen, and David R. Dowling. Fluid mechanics. Academic press, 2015

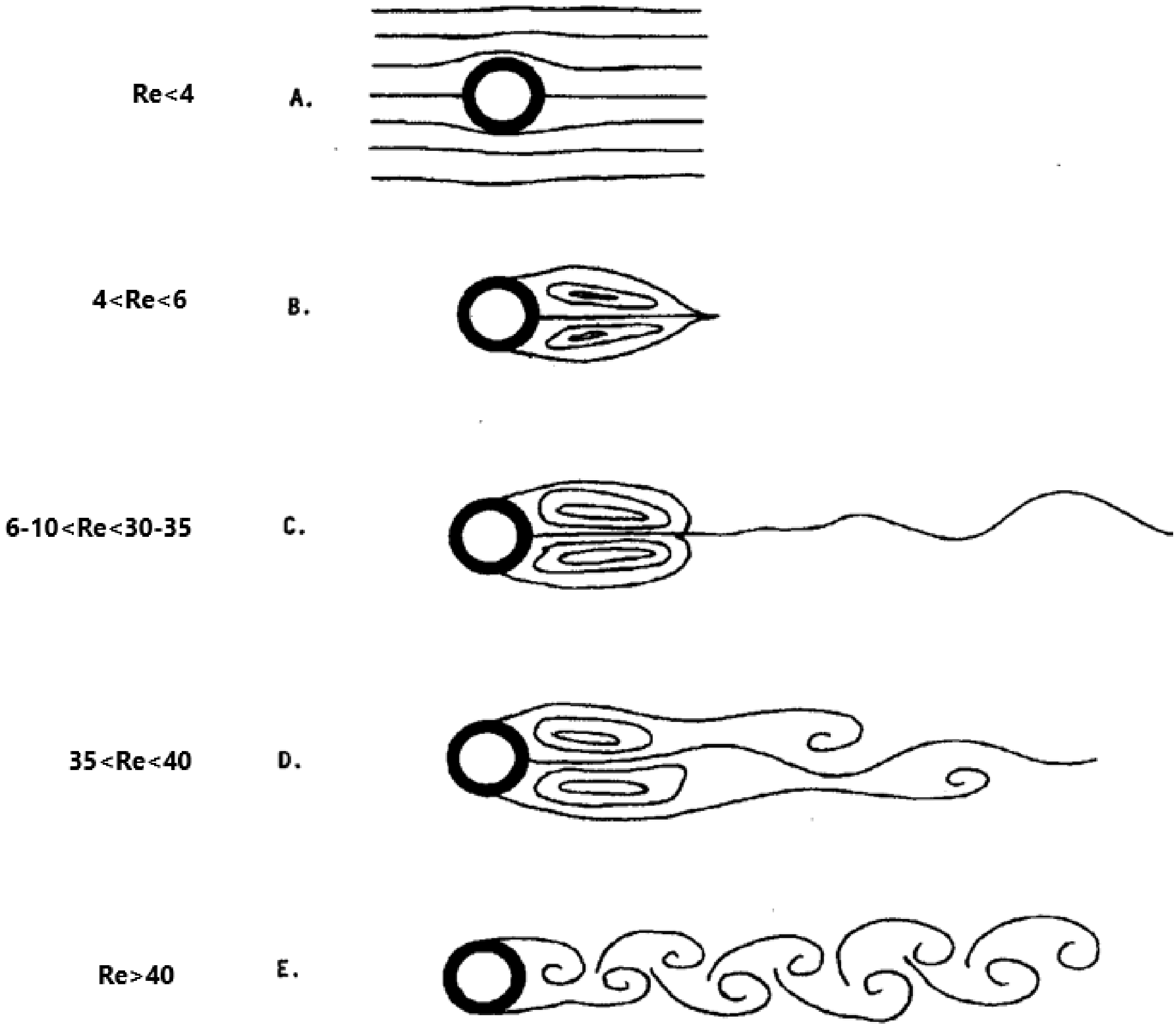


Figure 1.7: Flow about a circular cylinder at different Reynolds numbers

4. At Reynolds numbers of about 35 to 40 ($35 < Re < 40$) the vortices are periodically shed behind the cylinder, thus forming the well known von Kármán vortex street consisting of two rows of vortices equally spaced and arranged with each vortex located at the center of the two vortices of the opposite row (Figure 1.7D. The shedding of the vortices is due to an instability that develops in the flow.
5. At much higher Reynolds number ($Re > 40$), the Vortex shedding increases and the flow becomes even more turbulent behind the cylinder as shown in Figure 1.7E.

The flow reversals caused by the adverse pressure gradient results in a flow phenomenon known as Kármán vortices (refer to 1.8.1). Figure 1.8 shows the Kármán vortices shed by a cylinder. The vortices are alternately shed periodically from two sides of the cylinder. The flow is dramatically different from the simple steady attached flow predicted by inviscid theory.

The frequency with which vortices are shed behind a cylinder has been extensively studied, and it has been found that a non-dimensional frequency S, called the Strouhal number, depends uniquely upon the Reynolds number. The Strouhal number is defined in equation 1.5.

$$S = \frac{nD}{U} \tag{1.5}$$

where n is the shedding frequency (from **one side** of the cylinder only), D is the diameter of the cylinder, and U is the freestream velocity. While performing the experiment, you will be required to count the number of vortices shedding from the flow around the given specimen for a fix amount of time. Vortices can flow past pretty fast, especially when the water speed is high. A simple method to calculate the number of vortices is to

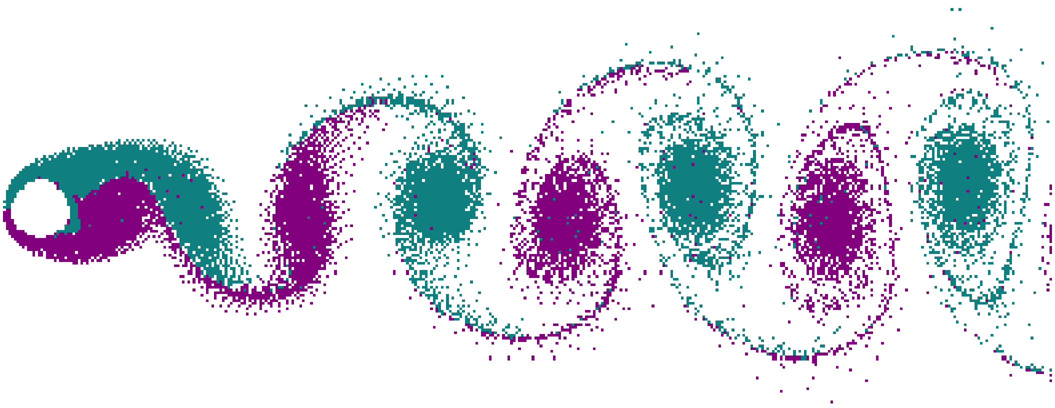


Figure 1.8: Vortex Shedding Animation

count the "mushroom" like humps created as a result of the alternate shedding of the vortex from upper and lower surface. Figure 1.9 shows the shape being referred to.

The cylinder Reynolds number is defined using the diameter, as in equation 1.6.

$$\boxed{Re = \frac{\rho U D}{\mu}} \tag{1.6}$$

where ρ is the density of fluid, μ is the dynamic viscosity of the fluid.

The compilation of data for the $S - Re$ curve is shown in Figure 1.10. For Reynolds numbers below about 40, the circular cylinder wake is steady. For Reynolds numbers between about 40 to 180, the wake is unsteady but still laminar. The wake transitions to turbulent and three-dimensional by about Re = 260, with the range $180 < \text{Re} < 260$ encompassing the wake transition regime. Williamson and Brown [2] fit the best available data for $40 < \text{Re} < 1200$ and found the following empirical correlations between Strouhal number (S) and Reynolds number (R):

$$S = 0.2665 - \frac{1.018}{\sqrt{\text{Re}}} \quad \text{for} \quad 40 < \text{Re} < 180 \tag{1.7}$$

$$S = 0.2234 - \frac{0.3490}{\sqrt{\text{Re}}} \quad \text{for} \quad 260 < \text{Re} < 1200 \tag{1.8}$$

For Reynolds numbers above about 300 and up to 10,000 Roshko[3] proposed the empirical curve-fit:.

$$S = 0.212(1 - \frac{12.7}{R}) \quad \text{for} \quad 300 < \text{Re} < 10,000 \tag{1.9}$$

1.5 Recommended Reading

It is highly recommended that students read the following sections in Fundamentals of Aerodynamics by John D. Anderson:

- 3.18 Applied Aerodynamics: The Flow Over a Circular Cylinder - The Real Case
- 15.2 Qualitative Aspects of Viscous Flow
- 17.2 Boundary-Layer Properties

[2] Williamson and Brown, *Journal of Fluids and Structures*, 1998
[3] NACA Report 1191, 1955

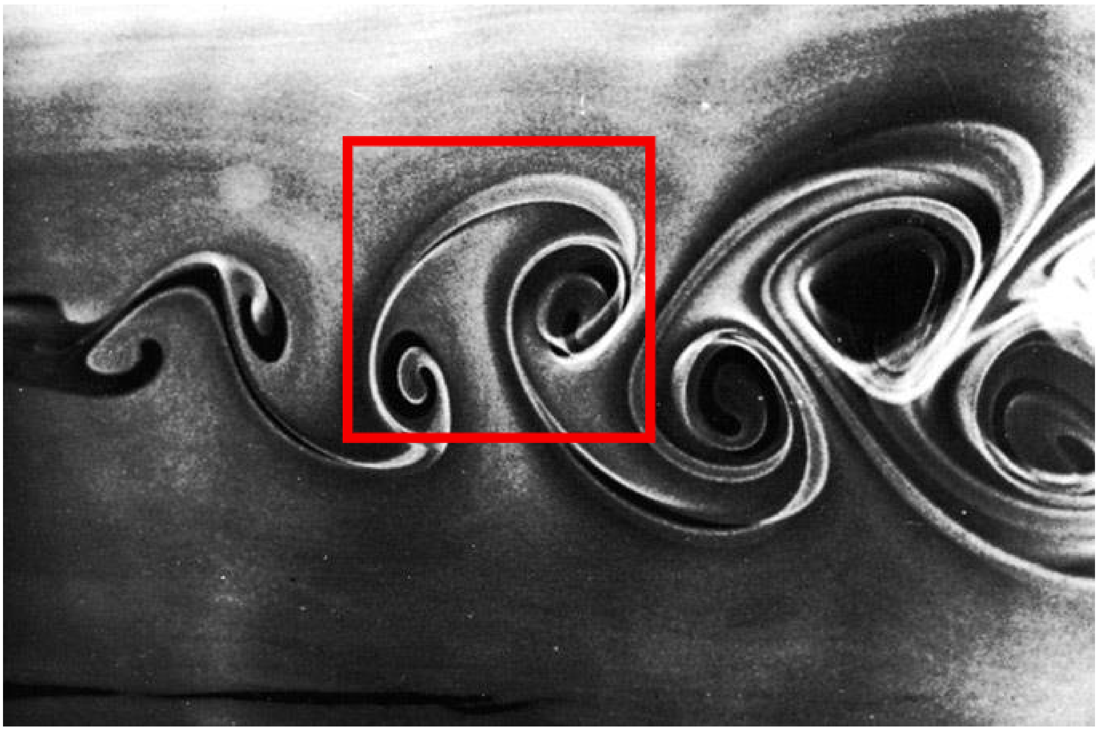


Figure 1.9: Vortex shedding observed during flow around a cylinder. Red box shows the shape formed by two alternating vortices. *Adapted from Large-scale particle image velocimetry for resolving unsteady flow features at cylinders, Basnet, 2010*

1.6 Data to be Acquired

During this lab you will acquire the following data:

1. Time for pulsed hydrogen bubbles in the flow to travel a specified distance and frequency of the Water Table motor (to estimate flow velocity vs. speed controller setting)
2. Temperature of the water (for calculating viscosity)
3. Dimensions of the cylinder models
4. Number of vortices shed in the cylinder wake for a given time (to calculate the shedding frequency)
5. Capture images of the flow when necessary

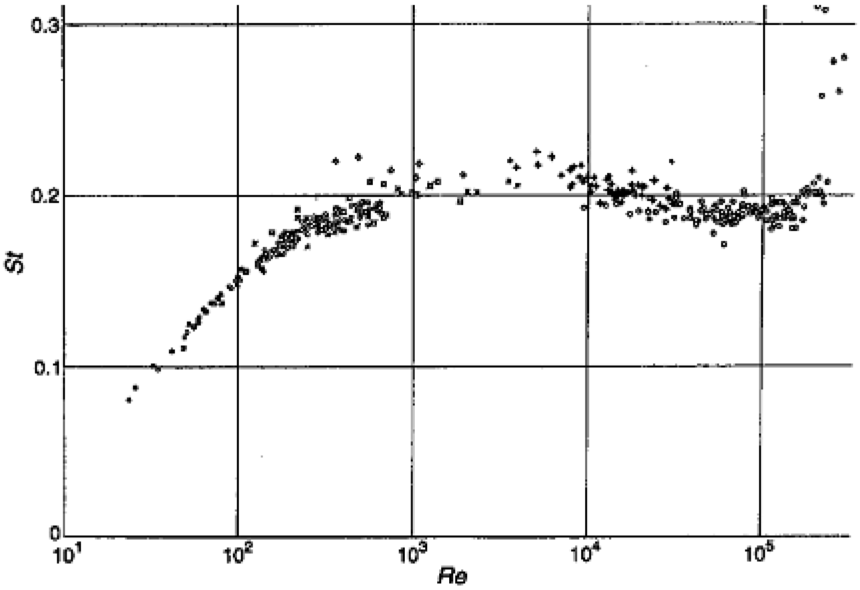


Figure 1.10: Reynolds Number versus Strouhal Number (from Zdravkovich)

1.7 Lab Setup and Procedure

Throughout the lab, periodically, you will be asked to check in with your TA to ensure that you are achieving reasonable results. **Yellow bold highlighted text** indicates that student should show the data/plot obtained to the TA present at the lab.

1.7.1 Experiment Setup and Preparation

Lab Location: ARMS B098, Water Table

Relevant Safety Hazard: Significant voltage and current are used to drive electrolysis in the water.

- Before starting to work with the Water Table, the voltage and current knobs on the bubble wire system power supply should be turned fully counter-clockwise (zero position).
- DO NOT touch any metal parts due to risk of electric shock.
- While performing the experiment do not touch the metal electrodes, the water, or the hydrogen bubble wire fixture.
- Always turn the power supply off before replacing or moving the models. You may designate an individual to replace models, and another person to control the analog electronics.

1. **Preparation**

   1.1 ☐ One member should log into the lab computer using their career account and navigate to the Lab 1 folder at the destination: `C:\ Temp \ AAE33301 \ Lab 1 - Flow Visualization of Boundary Layers and Wakes`.

   1.2 ☐ Open the Camera app, using the shortcut in the Lab 1 folder. You should now be able to see the water table setup from the top.

   - In order to take photos, toggle to "Camera" icon. Click the icon again to take a photo and click a sample photo.
   - In order to take a video, toggle to "Video" icon. Click the icon again to take a video and take a sample video.

   1.3 ☐ Open the folder `C:\ Temp \ AAE33301 \ data_33301_lab1`. You should see your photo and videos here. All videos and photos have a data stamp in their name - use that to locate your sample photos and videos.

   1.4 ☐ Also download the Lab 1 data spreadsheet from Brightspace. The spreadsheet will help you record data and perform calculations.

2. **Bubble Wire System Setup**

   2.1 ☐ Take a few minutes to review the basics of the bubble wire system in Section 7.2.2.2.

   2.2 ☐ Verify that the voltage and current knobs on the power supply are turned fully counter-clockwise, all the way to zero position.

   2.3 ☐ Visually confirm that the bubble wire is installed in the tunnel and that the wire is not broken. Please do not touch the wire (it is very delicate), let the TA know if it needs to be replaced.

   2.4 ☐ Verify that the red output cable from the control box is connected to the bubble wire fixture and the black output cable is connected to the copper wire.

3. ☐ Check that the water table is filled to the "Fill Line."

4. ☐ Verify that the speed controller has power, is set to 60 Hz frequency, and there is a red light next to "FWD" mode. Take a few minutes to review the water tunnel operation in Section 7.2.1.

5. ☐ Locate the models that will be used in the experiment: three cylinders (labelled A, B and C) and one duplicate of cylinder "C". Record the following values in Lab 1 Data spreadsheet (download from course Brightspace)

   5.1 Measure the diameters of the cylinders.

   5.2 Measure the temperature of the water in the water table using the thermometer and record it in the data spreadsheet.

   5.3 Find the viscosity and density of water based on water temperature (look up information online) and record in the data spreadsheet.

6. **Bubble Wire System Operation**

   NOTE: You already confirmed that the power supply was zeroed in Step 1.2. This should always be the first step before operating the bubble wire.

   To keep the bubble wire clean during operation, you should switch polarity every 30-60 seconds. Switching polarity will remove debris that has accumulated on the wire through electrostatic repulsion. It is recommended that one member of your team assumes the responsibility of changing the polarity regularly when producing bubbles.

   6.1 ☐ Set the Water Table speed controller to 60 Hz and turn on the motor using the "GREEN" button.

   6.2 ☐ Turn on the bubble wire system power supply by pressing the large green power button on the bottom left. The amperage and voltage output should both read zero.

   6.3 ☐ Turn on the waveform generator by pressing the power button on the bottom left.

   6.4 ☐ Turn on the control box by flipping the leftmost toggle switch to "ON." Flip the rightmost toggle switch to "Cont." The red lights above both switches should be on.

   6.5 ☐ Turn on the lights using the light switch on the leg of the table, behind the computer. The light will help with visualizing the bubbles.

   6.6 **Produce Continuous Bubbles**

      6.6.1 ☐ Turn the COARSE current knob clockwise until maxed out. You DO NOT need to adjust either current knob during the experiment.

      6.6.2 ☐ Turn the COARSE voltage knob clockwise slowly to increase the voltage applied to the wire to 25 V. You should see bubbles start to appear $\sim$ 10 V and the bubble density increase with voltage.

   6.7 **Produce Pulsed Bubbles**

      6.7.1 ☐ Leaving the voltage at 30 V, switch to "Pulsed" on the control box. The red light should turn off (indicating no incoming signal from the waveform generator) and the wire will stop generating bubbles.

      6.7.2 ☐ Press the "Pulse" button from the six available waveform buttons near the screen of the generator, at which point it should light up.

6.7.3 ☐ Press the Output button above the "CH1" output on the waveform generator. The Output button will light up, indicating that the waveform generator is sending pulses to the control box. The light above the "Pulse/Cont." toggle switch on the control box should also light up. **You should not see any bubble generation by the wire. Why do you think this is the case?**

6.7.4 ☐ Press the "Output" button to turn off the pulses. The light on the Output button should turn off.

6.7.5 ☐ Change the frequency to 1 Hz by highlighting "Freq" in blue on the right side of the screen. Type "1" and select "Hz" from the "Unit" options on the right side of the screen.

6.7.6 ☐ Change the pulse width to 500 ms by pressing the button next to "PulWdith" to highlight it in blue. Type "500" and select "ms" from the "Unit" options on the right side of the screen.

6.7.7 ☐ Press the "Output" button and observe the pulsed bubbles. The light above "Pulse/Cont" on the control box will flicker at the same frequency and width as input into the waveform generator.

6.7.8 ☐ Turn off the bubbles by pressing the "Output" button and ***turn the voltage down to zero.***

1.7.2 Experiment Procedure

1. **Calibrate the Water Table Velocity vs. Motor Frequency**

1.1 ☐ Using the same pulse settings (1 Hz, 500 ms), capture video of the pulsed hydrogen bubbles over speed controller settings 60 to 30 Hz by 10 Hz decrements using your smartphone camera. Keep the ruler on the side of the water tunnel in the field of view of the camera and track the pulsed bubble sheets as they move with the flow. *Due to parallax, you might want to limit your distance observation to 10 inches.* It is recommended that you use the larger oxygen bubbles (negative polarity) for this part of the lab since they are more visible. You can switch off the lab's ambient lights for a better visualization. ***Remember to switch polarity every 30-60 seconds to keep bubble wire clean.***

1.2 ☐ Calculate the velocity by measuring the time for the front of the pulsed bubble sheet to travel a chosen distance along the ruler. Calculate the velocity for three different bubble sheets and take the mean for a given speed controller setting). Use the provided data spreadsheet to record your measurements and perform your calculations.

1.3 ☐ When you are done recording videos at all four speeds, turn off the bubbles by pressing the "Output" button on the function generator and turn the voltage supply down to zero. Ensure that the voltage output reads zero before proceeding.

1.4 ☐ **Plot the velocity versus the speed controller setting and show it to the TA.**

2. **Measure the Vortex Shedding Frequency of Circular Cylinders for a Range of Reynolds Numbers.**

2.1 ☐ Keeping the speed controller setting at 30 Hz, mount cylinder A in the center hole directly in front of the bubble wire. Be careful not to hit the bubble wire with the model.

2.2 ☐ Switch the bubbles to Continuous mode and increase the voltage to 30 V. At this low of a flow speed, buoyancy will be an issue for the larger oxygen bubbles. Therefore, visualize and count the vortices using the smaller hydrogen bubbles (positive polarity on the control box). ***Remember to switch the polarity every 30-60 seconds to keep bubble wire clean.***

2.3 ☐ Start a timer for 15 seconds and count the number of vortices shed from one side of the cylinder (top or bottom). Record your count in the data spreadsheet. Note that it may be easier to count the total number of vortices and divide by 2. You can record a video and play it at reduced speed if the vortex shedding is too fast to count the vortices.

2.4 ☐ Repeat Step 2.3 two additional times and calculate the average number of vortices from the three measurements. Record in the data spreadsheet and find the average number of vortices from the three measurements.

2.5 ☐ Calculate the average shedding frequency (vortices per second) and the Strouhal number using the formula from Section 1.4. Record all values in the data spreadsheet.

2.6 ☐ Repeat Steps 2.3 to 2.5 for cylinder A for speed controller frequencies of 40, 50, and 60 Hz.

2.7 ☐ Repeat Steps 2.3 to 2.6 for cylinders B and C. The cylinders mount in the same hole as cylindar A, directly in front of the bubble wire.

2.8 ☐ Mount both cylinder C and the duplicate cylinder in the two outer holes. Observe the flow around and between the two cylinders. **Is the flow what you expected? Discuss with your TA.**

3. ☐ **Return the experiment and the lab to its state when you first arrived.** Return all tools and equipment back to their designated location, turn off any equipment that was started for the purpose of your lab, and clean up your workspace. **Obtain approval from the TA before leaving the lab.** Thank you!

You will use the information recorded in the Lab 1 data spreadsheet to complete the post-lab assignment. Remember that the post-lab should be completed **individually** and uploaded in Gradescope within 1 week of your lab session.

1.8 Additional Information

1.8.1 Kármán Vortex Street

Wikipedia

A Kármán vortex street is a term used in fluid dynamics for a repeating pattern of swirling vortices caused by the unsteady separation of flow of a fluid over bluff bodies. It is named after the engineer and fluid dynamicist, Theodore von Kármán and is responsible for such phenomena as the "singing" of suspended telephone or power lines, the vibration of a car antenna at certain speeds, and the fluttering of Venetian blinds as the wind passes through them". It is often observed in nature as shown in Figure 1.11.

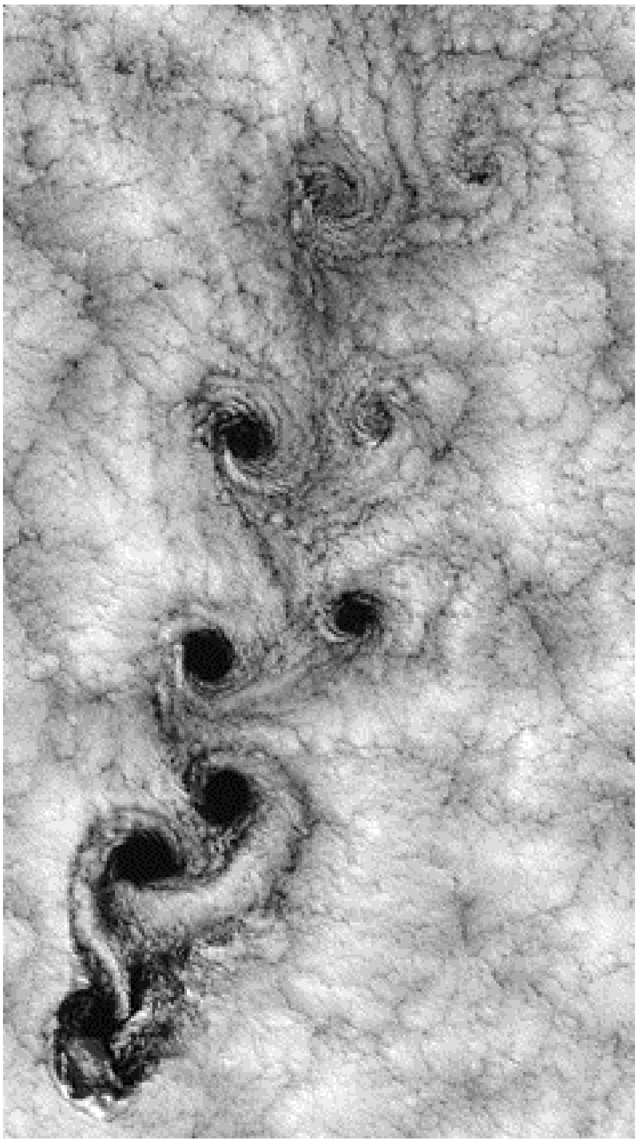


Figure 1.11: Von Kármán vortex street off the Chilean coast near the Juan Fernandez Islands

Chapter 2

Reynolds Pipe Flow

2.1 Introduction

In both nature and laboratory experiments, fluid flow can occur in two distinct regimes: laminar and turbulent. In laminar flows, fluid particles move in parallel layers, sliding past each other with minimal mixing between the layers. Conversely, turbulent flow is characterized by chaotic, seemingly random fluctuations that result in extensive mixing between fluid layers.

We can predict whether the flow in a given system should be laminar or turbulent, thanks to Osborne Reynolds, a prominent 19th-century British engineer and physicist. He conducted an experiment where he varied the flow rate through a glass pipe and visualized the flow by injecting flow into the water. He observed that at low velocities, the layer of dye kept its shape, but at higher velocities, the dye layer would break apart and diffuse through the pipe cross-section. This was the transition between what is now known as laminar and turbulent flow, and Reynolds found that the condition of the flow (laminar vs. turbulent) depended on a single dimensionless number that took into account the fluid's static properties (viscosity and density), the flow velocity, and the diameter of the pipe. In 1908, Arnold Sommerfeld named this dimensionless number after Reynolds in a paper on hydrodynamic stability he presented at the 4th International Congress of Mathematicians in Rome. The Reynolds number is given by:

$$Re = \frac{\rho V L}{\mu} \tag{2.1}$$

where ρ and μ are the density and viscosity of the fluid, respectively, V is the characteristic velocity of the flow, and L is the characteristic length scale of the flow (the inner diameter in the case of a pipe). Physically, the Reynolds number represents the ratio of intertial forces to viscous forces in the flow.

The Reynolds number plays a critical role in predicting and analyzing flow behavior in many different scientific and engineering contexts. It is also important for dimensional analysis - for example, if you want to study drag in the laboratory to inform the design of a ship or aircraft, the Reynolds number of the flow used with a model in the lab should match the real-life value in the field.

The specific case of viscous flow through a pipe is relevant to flows in many practical systems. Applications in aerospace and mechanical engineering include liquid propellant feed systems in satellites and boosters, turbine engine oil systems for lubrication and cooling of critical engine seals, hydraulic systems in aircraft controls, life-support systems in spacecraft, and duct flows in aerodynamics.

2.2 Lab Overview

Understanding the dynamics of fluid flow in pipes is essential for numerous engineering applications. In this laboratory, we delve into the fundamental regimes of fluid flow: laminar and turbulent. By examining these distinct flow types, we aim to elucidate the conditions under which each occurs, their respective characteristics and the relationship between flow regime and Reynolds number. Additionally, this lab will explore the concept of the friction factor, a crucial parameter that quantifies the resistance exerted by the pipe walls on the flowing fluid. The lab setup is inspired by Reynolds classic pipe flow experiments, where water flow through a clear pipe will be visualized using dyed water injection. The height of the reservoir will be varied to change the average flow velocity through the pipe, and two different pipe diameters will be used. The Reynolds number for each case will be calculated and the flow regime will be observed. In addition, the pressure head loss in the pipe will be measured and used to estimate the friction factor.

2.3 Objectives

At the completion of this experiment you will be able to:

1. Distinguish between laminar, transitional, and turbulent pipe flow.
2. Calculate pressure loss in a pipe using friction factors.
3. Explore the effect of streamwise pressure gradients on transition.

2.4 Background

2.4.1 Reynolds Pipe Flow

In pipe flow, viscous effects become important, and the behavior of the boundary layer must be taken into account. At the entrance of a pipe the boundary layer is generally so thin that the flow in this region can be considered non-viscous, except near the pipe walls. However, as the flow progresses down the pipe, there is a thickening of the boundary layer until it encompasses the whole pipe cross-section, and can no longer be considered a boundary layer. Fully developed flow is a flow that no longer changes in the streamwise direction (see Figure 2.1).

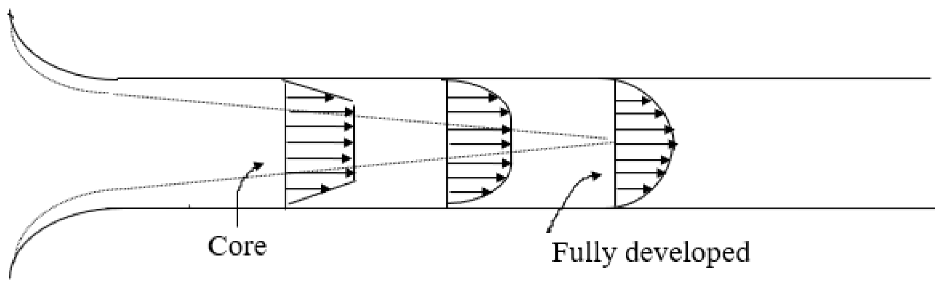


Figure 2.1: Boundary Layer Development

In laminar flow, fluid layers slide smoothly over one another in a well-ordered pattern. In the pipe, after the flow becomes fully developed, laminar flow is purely axial, and the velocity profile is independent of the coordinate along the direction of flow. Laminar instability can readily be seen in the classic Reynolds experiment on viscous flow, in which dye is injected into the flow of water through a glass pipe. The thread-like form of the moving dye indicates the laminar behavior. When the velocity of the water is increased a fluctuating motion appears in the dye, indicating a transition to an unsteady flow. At higher velocities the thread of dye becomes mixed with the fluid; irregular radial velocity fluctuations are superimposed upon the axial motion, and the flow is said to be turbulent (see Figure 2.2).

Another method of determining the flow regime involves observation of the fluid as it exits the pipe. For laminar flow, the stream of fluid leaving the pipe appears smooth and glassy. The stream will also have a steady well-defined shape. A turbulent stream, on the other hand, appears to be rough and dull in luster. A turbulent stream as it exits the pipe is also characterized by fluctuations and small disturbances. For any given pressure difference, a turbulent stream will have less momentum (and thus speed) as it exits the pipe, compared to a laminar stream.

2.4.2 Calculation of Darcy Friction Factor

Oswald Reynolds determined that the transition from laminar to turbulent flow depends on the Reynolds number (symbolized by Re or R), defined as in equation 2.2

$$Re = \frac{\rho V D}{\mu} = \frac{V D}{\nu}, V = Q/A \tag{2.2}$$

where ρ is the density, D is the pipe diameter, μ is the dynamic viscosity, ν is kinematic viscosity, defined as $\nu = \frac{\mu}{\rho}$, V is the mean velocity, Q is the volumetric flow rate of the fluid, and A is the pipe cross-sectional area.

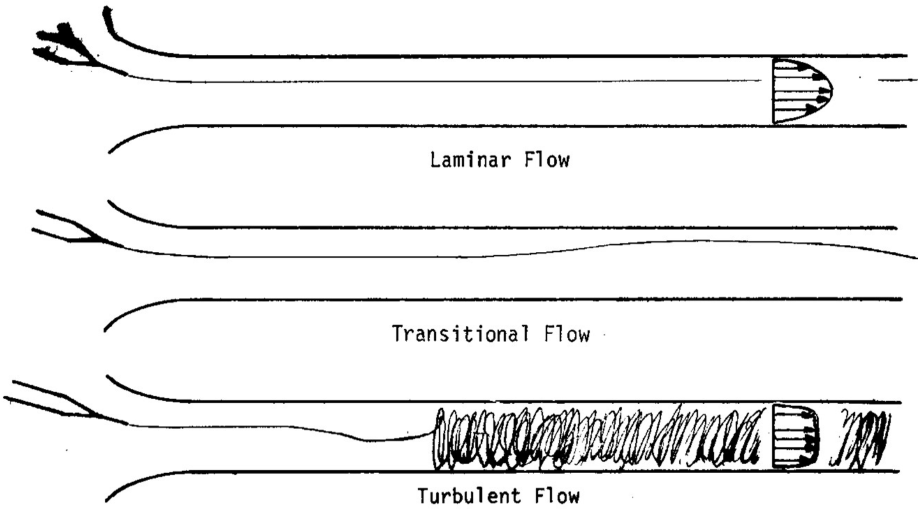


Figure 2.2: Dye Streaks showing flow stability. These are typical profiles for laminar and turbulent flow (time averaged). Note that your hardware could either be oriented for right to left or left to right flow but we generally illustrate flows as being from left to right.

The critical Reynolds number is the Reynolds number at which transition from laminar to turbulent flow occurs. This critical Reynolds number (R_{crit}) depends upon the initial disturbances in the fluid entering the pipe. A critical Reynolds number of 40,000 or greater can be reached if great care is taken to reduce disturbances. However, the critical Reynolds number below which even strong disturbances are damped by viscosity and do not cause turbulence is approximately $R_{crit} = 2300$.

Consider incompressible, steady flow between sections ① and ② of the inclined constant-area pipe shown in Figure 2.3. The energy equation for the steady, incompressible flow between sections ① and ② can then be written as in equation 2.3.

$$\frac{p_1}{\rho} + \frac{{V_1}^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{{V_2}^2}{2} + gz_2 + gh_l \tag{2.3}$$

where each symbol has its usual meaning. This looks like Bernoulli's equation except for the additional head loss term, h_l, which accounts for any energy loss associated with the flow. In other words, it represents the loss of mechanical energy per unit of mass flowing through the pipe. In this case, the energy loss is the direct consequence of viscous dissipation that occurs in the pipe. For the ideal (inviscid) case, $h_l = 0$ and henceforth we obtain Bernoulli's equation.

From the one-dimensional continuity equation (conservation of mass), given that density and area are constant, we know that (equation 2.4)

$$Q_1 = Q_2 = constant; V_1 = V_2 = V \tag{2.4}$$

so equation 2.3 reduces to equation 2.5

$$h_l = \Delta z + \frac{\Delta p}{\rho g} \tag{2.5}$$

where $\Delta z = z_1 - z_2$, $\Delta p = p_1 - p_2$. Therefore, the pipe head loss equals the sum of the elevation change (gravity head) and the change in pressure. For a horizontal pipe like the one used in the lab experiments, $\Delta z = 0$ so we obtain equation 2.6.

$$h_l = \frac{\Delta p}{\rho g} \tag{2.6}$$

For details, see for example *Sabersky et al., Fluid Flow, MacMillan, 1971, sec. 5.4.* Therefore we can consider the head loss as the loss in pressure head, $\Delta p/\rho g$, due to friction. This head loss is dependent upon the velocity profile, the type of fluid, and sometimes the roughness of the pipe.

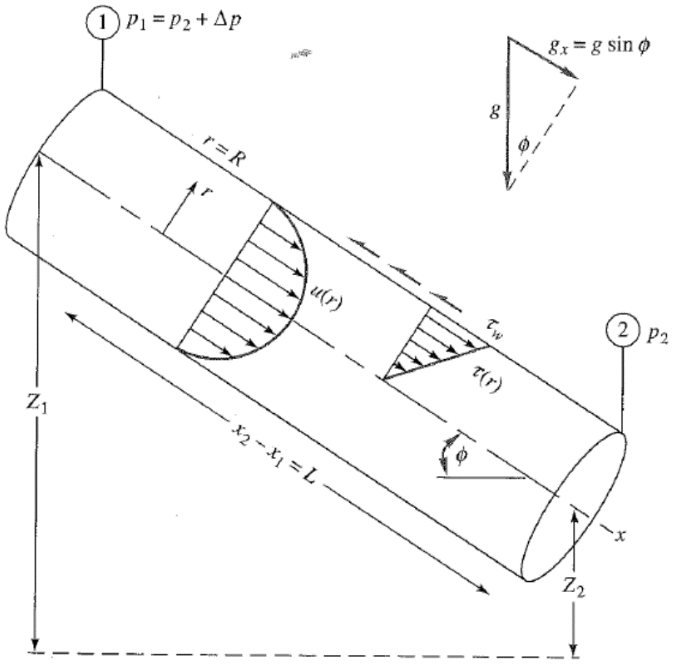


Figure 2.3: Control volume of steady, fully developed flow between two sections in an inclined pipe (from *Fluid Mechanics by F. M. White, pg. 351*)

We have used conservation of mass and energy to derive Equation 2.5. If we apply conservation of momentum, we can relate the head loss h_l to the wall shear stress τ_w as in equation 2.7.

$$h_l = \frac{4\tau_w}{\rho g}\frac{L}{D} \tag{2.7}$$

where L and D are the length and diameter of the pipe, respectively.

An equation for the head loss was proposed in 1850 by Julius Weisbach, a German professor who published the first modern textbook on hydrodynamics. Using dimensional analysis, common-sense reasoning (e.g. the head loss should be proportional to the length of pipe), and experimental data, he proposed the following correlation (equation 2.8)

$$h_l = f\frac{L}{D}\frac{V^2}{2g} \tag{2.8}$$

where f is called the Darcy friction factor and is a function of the Reynolds number, pipe roughness (ϵ/d), and the pipe cross-section shape. Equating Eqs. 2.7 and 2.8 gives another expression for the friction factor (equation 2.9)

$$f = \frac{8\tau_w}{\rho V^2} \tag{2.9}$$

For laminar, fully developed flow, we can derive an expression for the velocity profile in the pipe and then find the wall shear stress. This results in an exact expression for the friction factor for laminar flow (equation 2.10)

$$\boxed{f = \frac{64}{\mathrm{Re}}} \tag{2.10}$$

There is no exact theoretical solution for turbulent flow, but for turbulent flow in SMOOTH pipes, empirical data is correlated as in equation 2.11

$$\boxed{f = \frac{0.316}{(\mathrm{Re})^{0.25}}} \tag{2.11}$$

These two equations (2.10 and 2.11) are plotted in a graph in section 2.8.1 called the **Moody Diagram**. The line for laminar flow corresponds to Equation 2.10, and the curve for smooth pipes corresponds to Equation 2.11. This diagram is helpful in evaluating the friction factor for a pipe. It can be used for many types of pipes and flow ranges. Experiments determined the friction factor for a range of Reynolds numbers for turbulent flow (see 2.8.1). Values of f can be obtained for the case of the plexiglass pipe by following the smooth pipe curve. Note from the graph that in the laminar region, f is independent of the roughness, as is expected; roughness has no effect on the head loss in laminar flow.

For turbulent flow in rough pipes, the friction factor can be obtained from the Moody diagram. The Moody diagram was constructed using a huge number of experiments for a range of Reynolds numbers.

Intuition suggests that the pressure change is proportional to the length of the pipe, L. The non-dimensional pressure change can be written as equal to the non-dimensional pipe length times a function of Reynolds number and non-dimensional pipe roughness given by equation 2.12.

$$\frac{\Delta p}{\rho V^2} = \frac{L}{D} f_0 \left(\frac{\rho V D}{\mu}, \frac{\varepsilon}{D} \right) \tag{2.12}$$

where ϵ is the average roughness height and f_0 is a function of the Reynolds number and non-dimensional pipe roughness (ϵ /D). Letting $2f0 = f$ and assuming horizontal streamlines and fully developed flow, this becomes equation 2.13.

We can also write an expression for friction factor as a function of the pressure change in the pipe by combining Eqs. 2.7 and 2.8 as in equation 2.13.

$$h_l = \frac{V^2}{2g}\frac{L}{D} f\left(\mathrm{Re}, \frac{\varepsilon}{D}\right) \implies h_l = \frac{V^2}{2g}\frac{L}{D} f = \frac{\Delta p}{\rho g} \implies f = \frac{\Delta p}{\left(\frac{\rho V^2}{2}\right)\left(\frac{L}{D}\right)} \tag{2.13}$$

This relation is known as the **Darcy-Weisbach formula**. Therefore, if we can measure the pressure change and flow velocity in the pipe, we can calculate the friction factor. For details, see *Schlichting, Boundary-Layer Theory, McGraw-Hill, seventh ed., sections V.a. and XX.a.)*

Equation 2.13 can be rewritten based by modifying Δp to accommodate the height of a reservoir filled with a fluid. Consider the apparatus shown in figure 2.4.

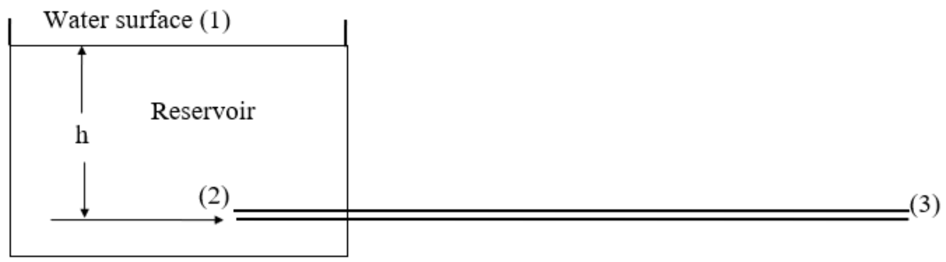


Figure 2.4: Application of the Bernoulli equation to Reynolds pipe flow

Since the reservoir and pipe exit are open to the atmosphere (vs. a pump, turbine, etc.), we know that the pressure at locations ① and ③ is 1 atm (equal to atmospheric pressure). The pressure at location ②, the entrance to the pipe, is simply the hydrostatic pressure given in 2.14

$$p_2 = \rho g h + 1atm \tag{2.14}$$

Therefore, the change in pressure or the pressure drop in the pipe given by

$$\Delta p = p_2 - p_3 = \rho g h + 1atm - 1atm = \rho g h \tag{2.15}$$

Therefore, one can calculate the friction factor directly using equations 2.14 and 2.13.

$$f = \frac{\rho g h}{\left(\frac{\rho V^2}{2}\right)\left(\frac{L}{D}\right)} \tag{2.16}$$

This can be simplified further, to produce equation 2.17

$$\boxed{f = \frac{2hDg}{LV^2}} \tag{2.17}$$

2.4.3 Calculation of Drag in pipes

For fully developed laminar flow, the drag on a pipe is given by the Hagen-Poiseuille equation 2.18.

$$D_P = \frac{8\mu LQ}{R^2} \tag{2.18}$$

This is developed from the Navier-Stokes equations and Newton's law of friction, which also show that the velocity profile in laminar flow is parabolic (see Figure 2.2, and Schlichting, 1979). It can be shown for laminar flow that the drag is proportional to the first power of the mean velocity by equation 2.19.

$$D_P = 8\pi\mu LV \tag{2.19}$$

As noted, this theory is based upon the assumption of fully developed pipe flow. Boussinesq was the first to make theoretical investigations in laminar pipe flow development and found the length of transition from entrance conditions to fully developed flow to be represented as in equation 2.20.

$$z \cong (0.02 - 0.08\mathrm{Re})\, D \tag{2.20}$$

In the turbulent region, the pressure drop and therefore the drag becomes approximately proportional to V^2. Turbulent mixing dissipates a large amount of energy, causing a substantial increase in resistance to flow. In a time-averaged sense, the speed profile is nearly constant across the entire diameter of the pipe, and falls off very rapidly to the no-slip condition near the wall (see Figure 2.2). There is a vigorous exchange of momentum between the main flow and the slow fluid near the wall, and for this reason the wall is often considered a "momentum sink". Turbulent flow is unsteady - at a given position the velocity is not constant in time but exhibits irregular fluctuations, which are often averaged over a time period to determine a mean flow.

Using a control volume and applying Newton's second law of motion results in equation 2.21

$$D_P = \Delta p \pi R^2 \tag{2.21}$$

where D_p is the drag on the walls of pipe, Δp is the pressure change over which the drag acts, and R is the pipe radius.

Recall the derivation for h_l. Assuming horizontal fully developed flow results in equation 2.22.

$$D_P = \frac{\rho V^2}{2}\frac{L}{D}\pi R^2 f = \frac{\rho V^2}{8} L\pi D f \tag{2.22}$$

where D_p is the drag on the walls of a pipe, V is the mean velocity, ρ is the fluid density, L is the length of pipe over which drag acts, D is the pipe diameter, R is the pipe radius, and f is the friction factor.

2.5 Recommended Reading

It is highly recommended that students read the following sections in Fundamentals of Aerodynamics by John D. Anderson:

- 15.2 Qualitative Aspects of Viscous Flow

2.6 Data to be Acquired

Following is the data that you need to acquire over the course of this lab:

1. Length (L) and Diameter (D) of the pipe being used.
2. Temperature of water (a thermometer is attached to the reservoir).
3. Height of water column (this will change based on the dam being used)
4. Height of the centerline (this will stay fixed for all cases)
5. Amount of water filled in a given time (take three trials for each setup).

You will be able to measure the flow velocity (from measuring the volumetric flow rate), the pipe length, and the pipe diameter. You will not be measuring the pressure change, however, in this particular setup we can calculate it. The setup is shown in Figure 2.4.

2.7 Lab Setup and Procedure

2.7.1 Experiment Setup

Lab Location: ARMS B098, Reynolds Pipe Flow Apparatus
We have two identical setups of the Reynolds flow experiment, you may use either one to get started. Two pipes with different inner diameters (3/8 in. and 1/4 in.) will be used in the experiment. The pipes can also be fitted with 3D printed smooth inlets. The flow velocity through the pipes will be varied by changing the height of the water in the tank using three different dams. Note that the tank will fill to the level of the highest dam. The volumetric flow rate will be measured using a graduated cylinder to collect the water flowing from the end of the pipe over a known period of time. Dye will be injected into the pipe flow to observe the nature of the flow (laminar, transitional, or turbulent). The flow regime will also be observed by studying the condition of the flow as it exits the pipe. A schematic of the pipe flow experiment is given in Figure 2.5 and Table 2.1 describes various components of the apparatus.

Important: While running the various cases, allow a few minutes for disturbances to die out in the tank. The lowest Reynolds number case should give fully laminar flow where the dye is ribbon is laminar all the way to the end of the pipe. If you do not observe laminar flow, ask your TA to help you adjust the apparatus (e.g., improve alignment of the dye injector tube, reduce disturbances in the tank, etc.). Higher Reynolds number cases should exhibit transitional or fully turbulent flow. Make sure to take pictures and notes on the flow characteristics for each case.

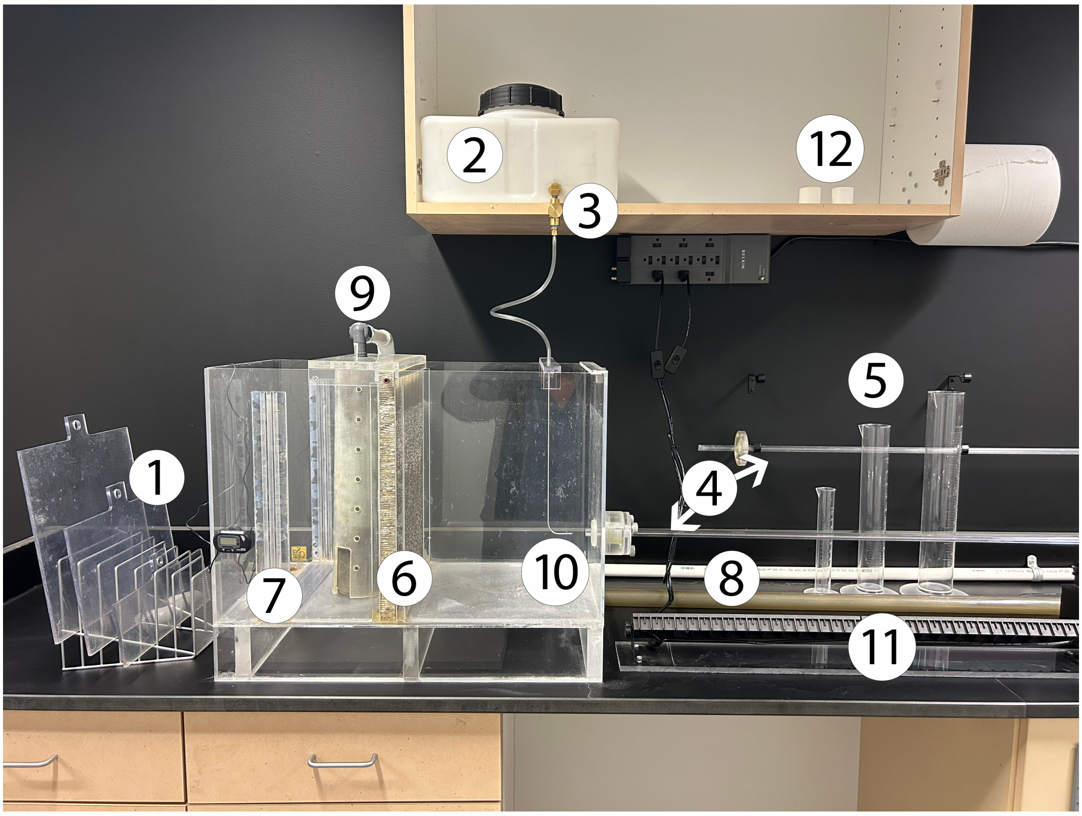


Figure 2.5: Schematic of pipe-flow apparatus

Before you begin: Check to see that the **1/4-inch diameter pipe** is installed on the tank and make sure the pipe is level. Also, make sure the inlet attachment is NOT on the inlet of the pipe. Then, position the LED lamp underneath and the black background behind the pipe.

Item Number	**Item Description**
1	Dams (to Control the Height of the Water)
2	Dye Container (contains special dye)
3	Stopcock Valve to Control the Flow of Dye
4	Varying Diameter Pipes
5	Graduated Cylinder (to Measure Volumetric Flow)
6	Flow Straightener
7	Slots to hold various Dams
8	Overflow Drain
9	Water Supply line
10	Dye Injector
11	UV Light
12	Nozzle Attachments

Table 2.1: Description of various items in the experiment apparatus

Figure 2.6: Reynolds pipe flow experiment

2.7.2 Experiment Procedure

Yellow bold highlighted text indicates that student should show the data/plot obtained to the TA present at the lab.

1. ☐ Insert dam A (the small dam) into slot A in the tank (the slot nearest to the overflow outlet).

2. ☐ Turn the cold water (green knob) on slowly and set the rate so that a small amount is always spilling over the dam. Thus, a constant reservoir height can be maintained. A large amount of fluid spilling over the dam will disturb the water in the reservoir (*what will this do to the transition Reynolds number?*).

3. ☐ Record the temperature of the water in the reservoir in the Lab 2 Data Spreadsheet. Find the viscosity and density of water based on the measured temperature (you can look up the information online) and record in the data spreadsheet.

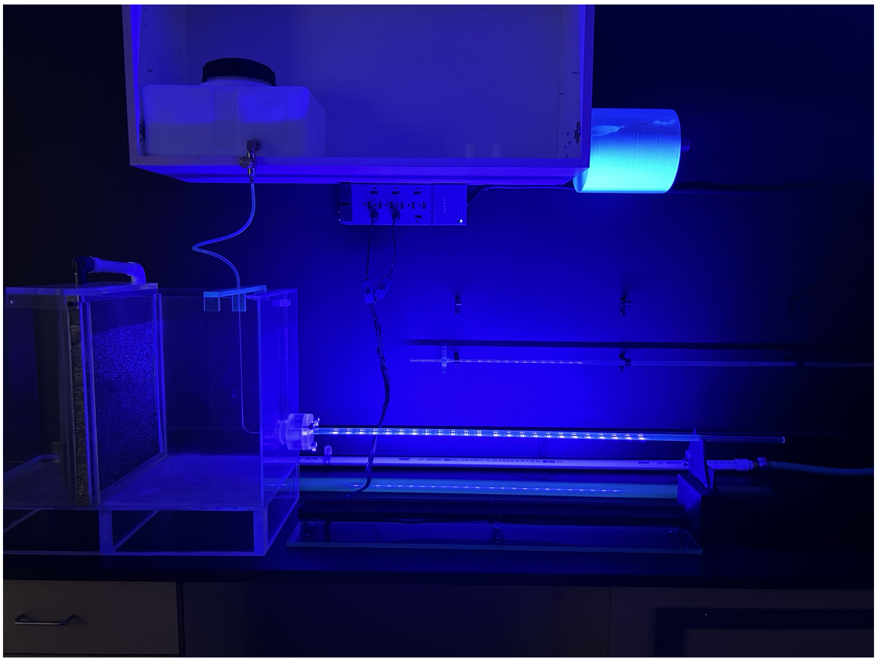


Figure 2.7: Pipe flow experiment with lighting for dye visualization

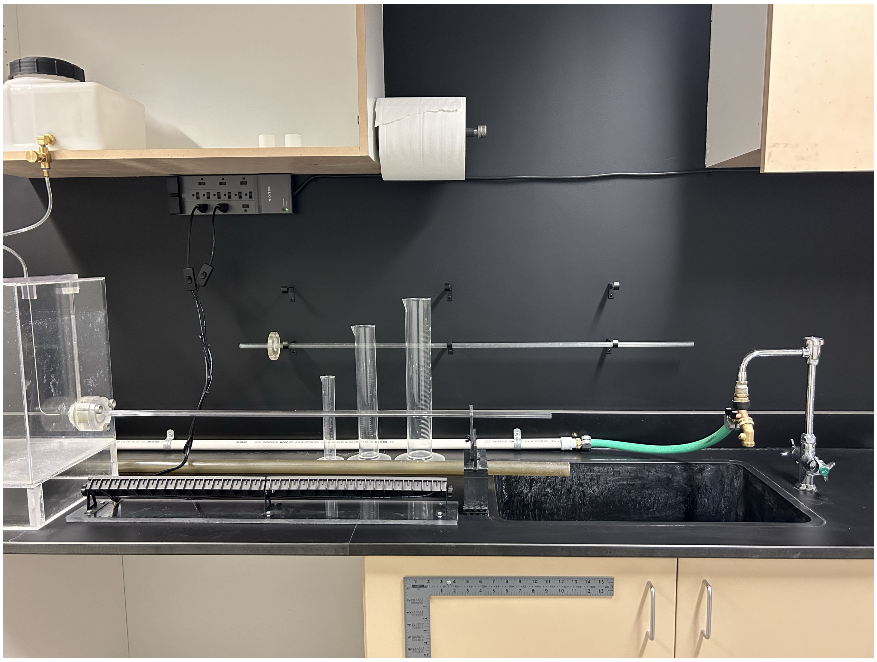


Figure 2.8: Another Side view of the apparatus showing the collection tank, pipes, light, measuring flasks and water supply valve

4. Measure the height of the reservoir and height of the pipe centerline with respect to a reference (a good reference is the table top) using a framing square. Record the measurements in the data spreadsheet.

5. Measure the volumetric flow rate using the 100 mL graduated cylinder and the timer on your phone. If the cylinder fills too quickly, opt for a larger graduated cylinder. Fill the cylinder from the pipe outlet for 10 seconds and record the volume in the data spreadsheet. Repeat two additional times and calculate the average volumetric flow rate from the three trials. **Show your results to your TA and discuss.**

6. Compute the flow velocity, Reynolds number, theoretical and experimental Darcy friction factor using the formulas derived in Section 2.4.2. Compare your experimental result to the theoretical friction factor. **Show your results to your TA and discuss.**

7. **Dye Visualization**

7.1 ☐ Turn on the violet LED lights for easy visualization of the dye and turn of the ambient light using the flip switch. Refer figure 2.7.

7.2 ☐ Position the dye tube outlet approximately 1 cm away from the mouth of the pipe. Make sure that the dye tube is approximately aligned with the center of the pipe. Open the dye valve and make any additional adjustments so that the dye flows smoothly into the tube.

7.3 ☐ Open the needle valve a couple of turns to get the dye flowing. You should see layers of green dye flowing through the 1/4 inch pipe.

7.4 ☐ Once the dye is flowing, close the needle valve all the way and then open it slightly to inject a very small flow rate of dye. If the water in the pipe is green throughout, reduce the dye flow.

7.5 ☐ Observe the condition of the flow and note the flow regime in the Lab 2 data spreadsheet. **Show the flow to your TA and discuss your observations.**

7.6 ☐ Repeat the Dye Visualization steps after attaching the smooth inlet to the pipe. *You do not need to measure the flow rate or do any calculations for this case.*

8. ☐ Insert dam B (the medium sized dam) into slot B in the tank, leaving dam A in place. Again, adjust the water flow rate so that a small amount of water spills over the dam. Repeat the steps 4 through 7 and record your data in the Lab 2 Data spreadsheet.

9. ☐ Insert dam C (the large dam) into slot C in the tank, leaving dam A and B in place. Again, adjust the water flow rate so that a small amount of water spills over the dam. Repeat Steps 4 through 7 and record your data in the Lab 2 Data spreadsheet.

10. ☐ After taking measurements for all three reservoir levels, turn off the dye and the water and remove the dams.

11. ☐ Remove the three screws holding the pipe to the tank and install **the 3/8" pipe**. Make sure the pipe is level. Insert dam A into slot A and repeat Steps 5 through 10 for the 3/8" pipe. **Make sure that the thumb screws are tightly fastened.**

12. ☐ When you are finished with this part of the experiment, connect the 1/4" pipe to the reservoir and make sure it is level.

13. ☐ **Streamwise Pressure Gradient Effects**
With the guidance of your TA, gently raise or lower the outlet of one of the flexible tubes. What happens to the point of transition? Relate your observations to the streamwise pressure gradient you have created.

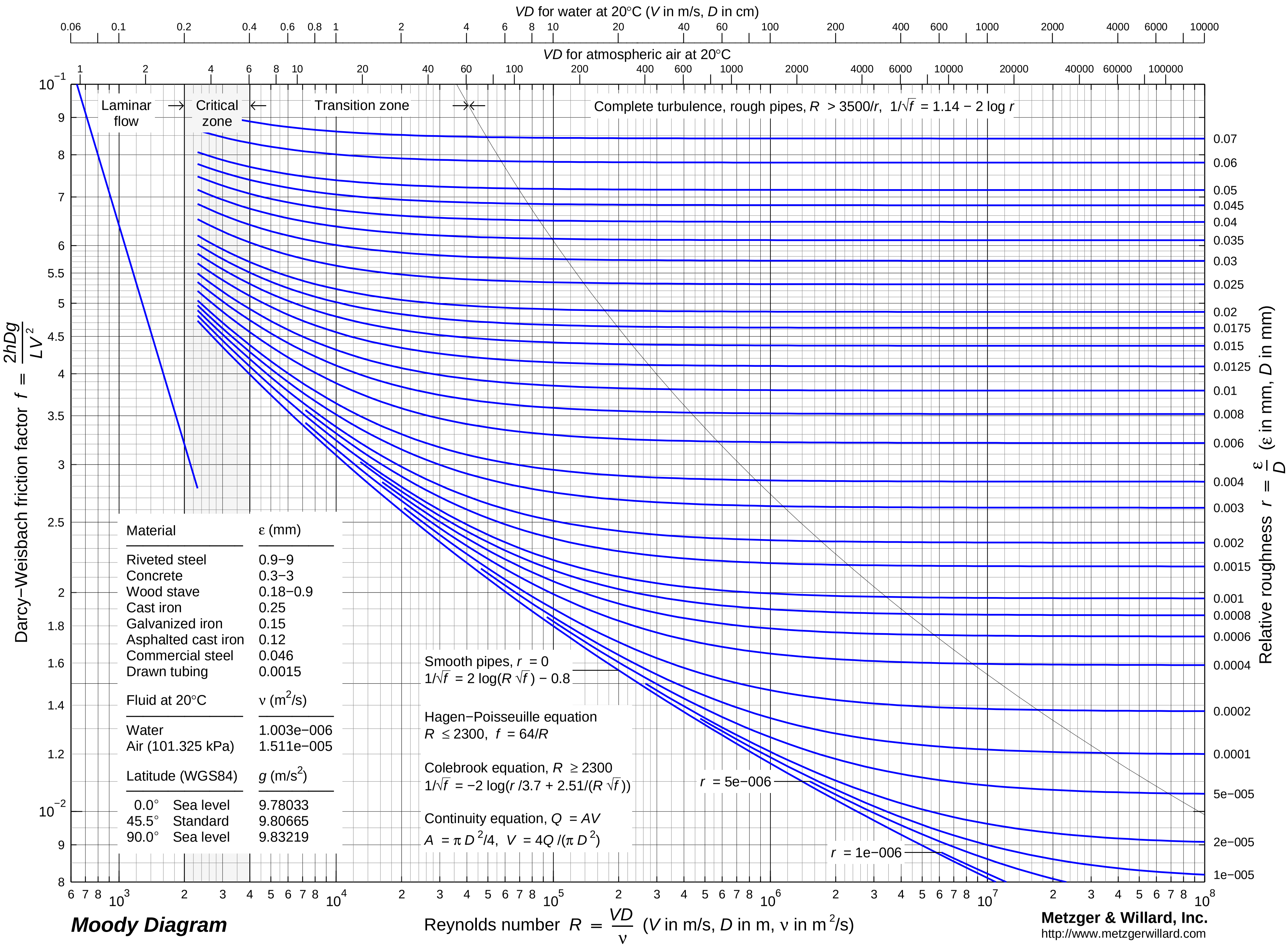
14. ☐ **Return the experiment and the lab to its state when you first arrived.** Return all tools and equipment back to their designated location, turn off any equipment that was started for the purpose of your lab, and clean up your workspace. **Obtain approval from the TA before leaving the lab.** Thank you!

You will use the information recorded in the Lab 2 data spreadsheet to complete the post-lab assignment. Remember that the post-lab should be completed **individually** and uploaded in Gradescope within 1 week of your lab session.

2.8 Additional Information

2.8.1 Moody Diagram

The Moody chart or Moody Diagram is a graph in non-dimensional form that relates the Darcy–Weisbach friction factor f_D, Reynolds number Re, and surface roughness for fully developed flow in a circular pipe. Please refer to the Moody diagram on the next page for all lab related calculations.


VD for water at 20°C (V in m/s, D in cm)
VD for atmospheric air at 20°C
Darcy−Weisbach friction factor f = 2hDg / LV^2
Relative roughness r = ε / D (ε in mm, D in mm)
Laminar flow
Critical zone
Transition zone
Complete turbulence, rough pipes, R > 3500/r, 1/√f = 1.14 − 2 log r
Material | ε (mm)
Riveted steel | 0.9−9
Concrete | 0.3−3
Wood stave | 0.18−0.9
Cast iron | 0.25
Galvanized iron | 0.15
Asphalted cast iron | 0.12
Commercial steel | 0.046
Drawn tubing | 0.0015
Fluid at 20°C | ν (m^2/s)
Water | 1.003e−006
Air (101.325 kPa) | 1.511e−005
Latitude (WGS84) | g (m/s^2)
0.0° Sea level | 9.78033
45.5° Standard | 9.80665
90.0° Sea level | 9.83219
Smooth pipes, r = 0
1/√f = 2 log(R √f) − 0.8
Hagen−Poiseuille equation
R ≤ 2300, f = 64/R
Colebrook equation, R ≥ 2300
1/√f = −2 log(r/3.7 + 2.51/(R √f))
Continuity equation, Q = AV
A = π D^2/4, V = 4Q/(π D^2)
r = 5e−006
r = 1e−006
0.07
0.06
0.05
0.045
0.04
0.035
0.03
0.025
0.02
0.0175
0.015
0.0125
0.01
0.008
0.006
0.004
0.003
0.002
0.0015
0.001
0.0008
0.0006
0.0004
0.0002
0.0001
5e−005
2e−005
1e−005
Reynolds number R = VD / ν (V in m/s, D in m, ν in m^2/s)
Moody Diagram
Metzger & Willard, Inc.
http://www.metzgerwillard.com

Chapter 3

Pressure in Aerodynamics

3.1 Introduction

The concept of pressure is fundamental in fluid dynamics and aerodynamics because it directly influences how fluids (liquids and gases) behave and interact with surfaces. Pressure is a measure of the force exerted by a fluid per unit area, and it determines the movement and distribution of fluid particles. In fluid dynamics, pressure gradients drive fluid flow, enabling the analysis and prediction of fluid behavior in various systems, such as pipelines, rivers, and weather patterns. In aerodynamics, pressure differences across surfaces, such as airfoils or wings, are crucial for generating lift and understanding drag forces, which are essential for the design and performance optimization of aircraft and other vehicles. Additionally, pressure variations affect buoyancy, stability, and the efficiency of propulsion systems, making it a critical parameter for engineering and scientific applications involving fluid motion.

Pressure measurements are essential not only for understanding basic flow physics but also for performing and interpreting many types of experiments involving aircraft, propulsion systems, hydraulic systems, and any other systems involving fluid flow. For example, in this laboratory class and AAE 33401, you will study and measure pressure distributions on wings, cylinders, supersonic nozzles, and wind tunnels. Other times you will measure the integrated effects of pressure, such as the lift force on a wing.

Presently, *Pressure Sensitive Paint* (PSP) is a vibrant research area. These are fluorescent paints where the spectrum of the fluorescence changes with pressure. Other similar paints are made to be temperature sensitive. Surface pressure measurements using PSP have been conducted in wind tunnels from low speed to hypersonic, on the fan of an operating turbofan engine, and in-flight on a transonic aircraft wing.

3.2 Lab Overview

In this lab, multiple experimental setups are used to measure the pressure in different flow conditions and relate the pressure measurements to the underlying fluid dynamics. Experiments will be conducted to explore the concept of hydrostatic pressure in static fluids, dynamic pressure in flowing fluids, and Bernoulli's principle relating pressure and flow velocity. In addition, various instrumentation for measuring pressure will be employed. The dynamic pressure of a flow will be measured using a pitot-static probe connected to an inclined manometer that operates according to the principles of hydrostatics. The static pressure distribution on the surface of a cylinder in a uniform flow will be measured using piezoelectric pressure transducers, and Bernoulli's principle will be demonstrated using measurements of static pressure of flow through a tube with changing area.

3.3 Objectives

The objectives of this lab are to study and experience a variety of topics in aerodynamic pressure. Specifically, you will:

1. Observe and experiment with stagnation, static, and dynamic pressure of air.
2. Learn about absolute, gauge, and differential pressures.
3. Learn about and use an inclined manometer to determine pressure on the surface of a body.
4. Use a pitot-static tube and manometer to determine the velocity of an incompressible stream of air.

5. Measure the pressure distribution on a circular cylinder in an air flow.
6. Learn to convert between various units of pressure.

3.4 Background

3.4.1 Pressure

What is pressure? The word pressure is used commonly in everyday conversations. We speak of air pressure in a car or bicycle tire, atmospheric pressure in weather reports, water pressure, the "pressure" of the exam that is coming up, peer pressure, blitzing in football to pressure the quarterback, and so on. But what precisely is the physical definition of pressure we use in science and engineering? A useful macroscopic technical definition of pressure is: a distributed force applied perpendicular to a surface and measured as force per unit area. In other words, pressure is the point wise normal force per area acting on a surface. Pressure is perpendicular force per area, or symbolically, $P = F/A$ where P is pressure, F is normal force, and A is area.

Pressures occur in solids, liquids, and gases. Molecules rebounding or recoiling from collisions with other molecules cause pressure. Except at absolute zero temperature, all molecules are vibrating. The molecules travel in straight lines but collide with other molecules. Each time a molecular collision occurs, a force accompanies the direction change of the molecules (force vector = change in momentum vector as in billiard balls colliding). When collisions occur between a gas molecule and the wall of the container, the force of the collision is part of the pressure of the contained gas or liquid. The collisions happen billions of times per second per unit area in materials under ordinary conditions, but perhaps much less in a partial vacuum. The sum total of all the very small forces of the collisions is the pressure.

Clearly the units of pressure must agree with the definition of pressure, force per area. In the metric system, pressures are usually given as newtons per square meter (N/m^2), which are known as pascals (Pa). In the English system, pressures are usually given as pounds force per square inch (lb/in^2 or psi) or per square foot (lb/ft^2 or psf). Note too that a force per area is also energy per volume, so it is often useful to think of pressure as a spring-like energy per unit volume.

Energy is the ability to do work, so a pressurized container of air is able to do work. It is for this reason that gas cylinders, which are tanks of pressured air, nitrogen, or other gases, are kept chained to the wall in our labs. If one tank were to fall over and snap the valve off the top, much of the energy of the pressurized gas could be transformed into kinetic energy of the valve, hurling it dangerously through the room. But note that in hydraulic systems, such as power brakes on your car, you do not wish to store energy in the fluid but use it to transmit force. Thus, highly incompressible liquids are chosen for these applications so that very little energy is stored in them. That is, recall from sophomore thermodynamics that *differential work done is pressure times differential volume change*, $dW = PdV$. A hydraulic fluid has very small dW even for sizeable pressures, so there is not much work done on the fluid. On the other hand, gases will change volume rather easily with pressure and thus will store significant energy.

Similarly, you can think of the dynamic pressure at a point in a flow as the kinetic energy per mass of the continuum air (not to be confused with the kinetic energy of the random molecular motion which exists for air that is not in motion). Thus for an incompressible (constant density) flow, Bernoulli's law states that total pressure is the sum of the static and dynamics pressures as shown in equation 3.1.

$$p_0 = p + \frac{1}{2}\rho V^2 = p + q \tag{3.1}$$

Thus, in an incompressible flow with constant total pressure (frictionless and irrotational), energy is exchanged between static pressure and dynamic pressure. This is analogous to spring energy and kinetic energy, respectively, in the simple mass and spring oscillator in freshman physics. In both cases, the total energy is constant.

Figure 3.1 presents a chart of definitions for use in measuring and expressing pressures. Your career will (*does?*) depend on effective technical communication and proper use of pressure definitions is one part of this. Let us discuss these pressures:

1. The first is ***absolute pressure***. Remember that the absolute pressure of a perfect vacuum is zero. When you use the ideal gas law, you must use absolute pressure (and don't forget to use absolute temperature, such as degrees Kelvin or Rankine). Absolute pressure and one other property can define the thermodynamic state of the fluid. When we speak of atmospheric pressure in a weather forecast, it is absolute pressure.

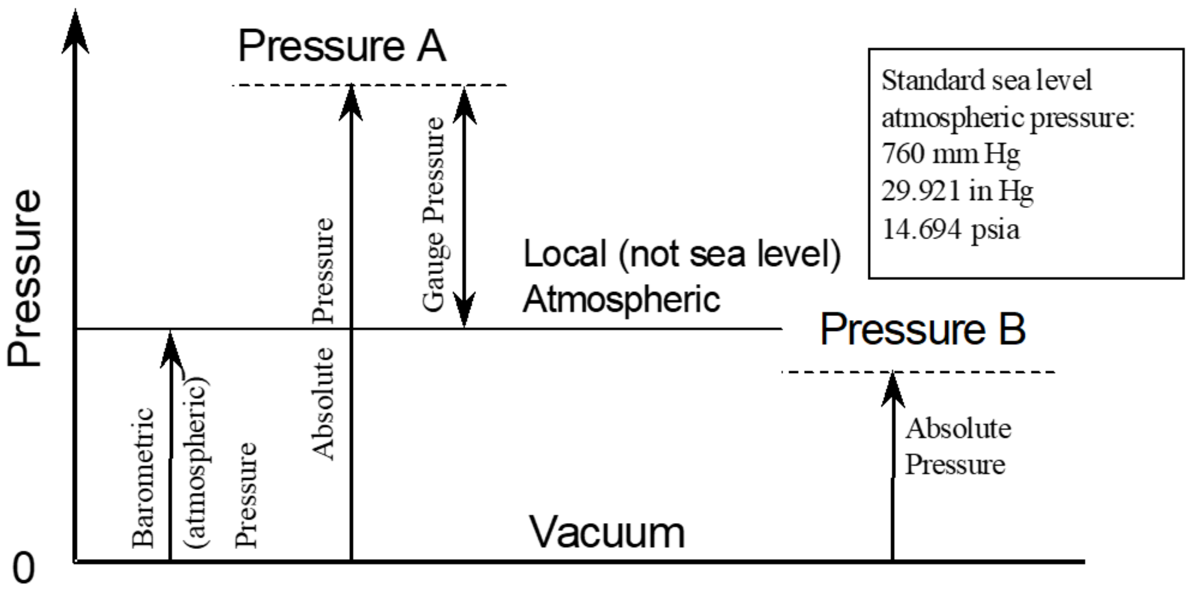


Figure 3.1: Definitions for use in measuring and expressing pressures

2. The second way of presenting a pressure is ***gauge pressure***. Gauge pressure is measured with respect to the current local atmospheric pressure. Therefore, gauge pressure is equal to the absolute pressure minus the current atmospheric pressure. Thus, the current atmospheric pressure is always zero gauge pressure, or 0 $psig$.
3. The third pressure is ***differential pressure***. Differential pressure is the difference between two pressures either in a measurement or in analysis you may be performing. Gauge pressure (see #2 immediately above) is a special case of differential pressure that is used so much that we give it it's own name.

The nomenclature shown in table 3.1 is used when expressing pressure values (as an example lbf/in^2 and N/m^2 are used, it would be similar for any other set of units). Note that "a", "g" and "d" stand for Absolute, gauge and differential pressures respectively.

Pressure	Units	SI Units
ABSOLUTE	psia	N/m^2 (a)
GAUGE	psig	N/m^2 (g)
DIFFERENTIAL	psid	N/m^2 (d)

Table 3.1: Nomenclature for expressing Pressure

3.4.2 Measuring Differential Pressure

Most types pressure measurement instruments measure a differential pressure, and thus the choice of the reference pressure is critical. The following sections describe some of the most common instruments used for measuring differential pressure.

3.4.2.1 Manometers

U-Tube Manometer The simplest type of manometer is the U-tube manometer. The U-tube manometer consists of a U-shaped glass tube containing a liquid (water, oil, mercury, etc.). When the gas pressure above the two legs differs, the higher gas pressure will push liquid up the low-pressure leg as shown in Figure 3.2.

If ρ is the density of the liquid and A is the cross sectional area of the tube, then summing the vertical forces on the liquid gives

$$-p_A A = -p_B A - \rho g h A \tag{3.2}$$

or

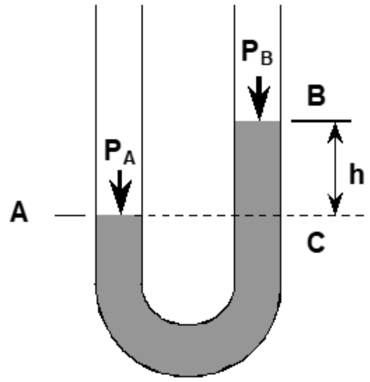


Figure 3.2: The U-tube Manometer. The shaded region is the liquid in the tube

$$p_A - p_B = \rho g h = \Delta p \tag{3.3}$$

Equation 3.3 above is the basic equation used in all manometers. Many special types of manometers use special equations but these equations are all derived in the manner. You see from the left side of the equation that this manometer (and all others too) measure the difference between two pressures, $(p_A - p_B)$.

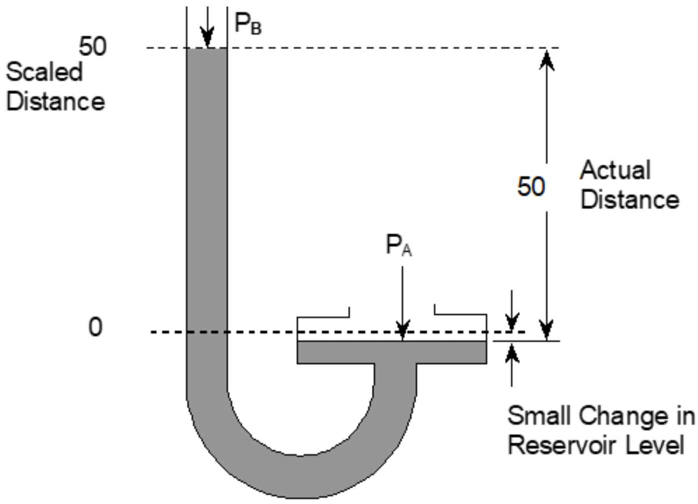


Figure 3.3: The reservoir manometer

Example: Suppose you are measuring a pressure with a manometer with mercury as the liquid. You find Δp from the manometer to be 21.5 in Hg (d). "d" refers to the differential pressure. What is Δp in psi? Using the conversion factor

1 psi = 2.036 in Hg,

we find that

$$\Delta p(\text{psi}) = 21.5\text{inHg(d)} \times \frac{1\text{psi}}{2.036\text{inHg}} = 10.56\text{psid} \tag{3.4}$$

Alternatively, using the hydrostatic equation and noting that mercury (Hg) is 13.5 times denser than water (H_2O),

$$\Delta p = \rho g \Delta h \tag{3.5}$$

$$= 21.5 \times 1.94\frac{\text{slug}}{\text{ft}^3} \times 32.174\frac{\text{ft}}{\text{s}^2} \times 21.5\text{in} \times \frac{1\text{lb}_\text{f}}{1\frac{\text{slug}\times\text{ft}}{\text{s}^2}} \times \frac{1\text{ft}^3}{(12\text{in})^3} = 10.48psid \tag{3.6}$$

Note: Δ p is a pressure difference between p_A and p_B, hence the label *psid.*

Some U-tube manometers have one end connected to a large reservoir. When the mercury (or other liquid) rises in the tube the level of the reservoir will decrease only slightly because of its larger surface area. Usually a scaled distance is then read to give h (see Figure 3.3).

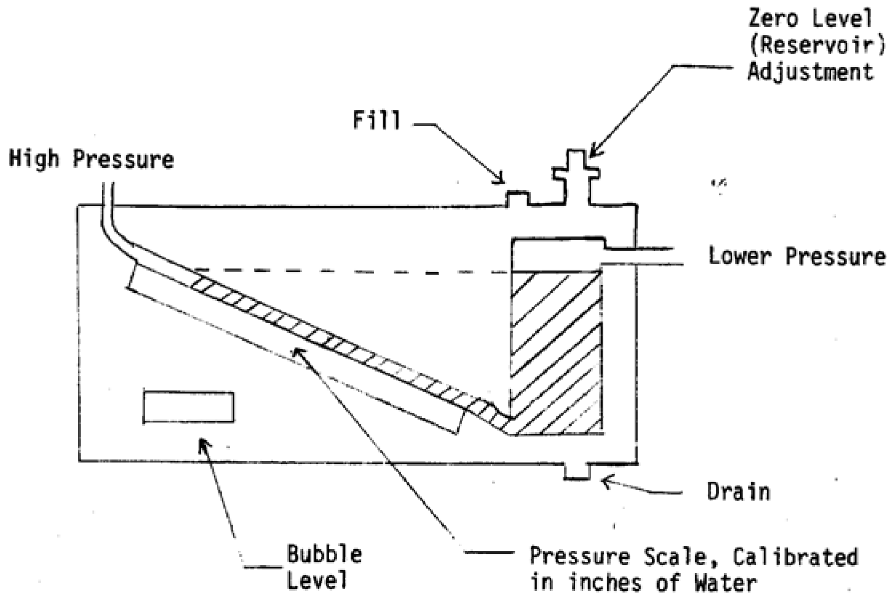


Figure 3.4: Meriam Inclined Differential Manometer

Inclined Manometer The capabilities of the U-Tube manometer have been extended to various other types of manometers. One such manometer is the Inclined Differential Manometer (Figure 3.4).

The inclination increases the length over which the liquid moves for a given pressure change, improving the measurement sensitivity.. This type of manometer can measure pressure differences in the range from zero to two or three inches of water, in increments of 0.01 inches of water. The higher pressure is connected to the end of an inclined tube, the lower pressure to a reservoir. When using oil with a specific gravity of 0.827, pressure differences can be read directly from a scale attached to the inclined tube. Initially, the reservoir is adjusted up or down to give a scale reading of zero. When recording pressure differences, the value should be read at the bottom of the meniscus. Note that the pressure scale has been calibrated in inches of water so no conversion is necessary.

Although the manometer must be read by eye, it has the advantage of being a (nearly) primary standard. The pressures computed depend only on accurate measurement of Δh and accurate knowledge of ρ and g. No calibration is generally required.

This lab makes use of a Series 420 Durablock inclined Manometer by Dwyer Instruments Inc. The instrument uses red gage fluid pressure level. Red gage fluid is specifically created Dwyer instruments, as a means to measure pressure in fluid columns and has a specific gravity of 0.826. Note that the instrument shows the pressure in mm of water and not the red gage fluid; the fluid's properties and the instrument design makes sure that the levels correspond to inches of water. One does not need to make any conversions based on the density of red gage fluid.

To read the pressure from the instrument, connect the positive pressure port (port with higher pressure) on the left side and the negative pressure port (port with lower pressure) on the right side.

If the instrument is calibrated such that the inclined scale reads 0 mm when both pressure ports are open to atmosphere, then the reading when pressure ports are connected can be accepted readily, without any manipulation.

The red gage manometer is shown in the figure 3.5.

Figure 3.5: Red gage incline manometer from Dwyer

3.4.2.2 Electric Pressure Transducers

An electrical pressure transducer consists of an outer shell usually made of metal or plastic (Figure 3.6). Inside this outer shell is a cavity with a flat piece of metal called a diaphragm, AB, dividing the cavity into two chambers. These two chambers can be pressurized to different pressures p1 and p2. A strain gauge is mounted on the diaphragm.

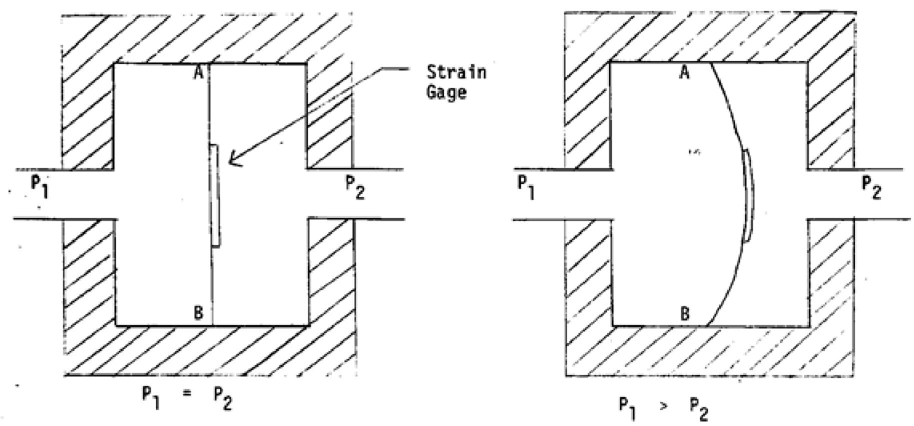


Figure 3.6: Cross-section of an electrical pressure transducer.

When p_1 and p_2 are different, the diaphragm will bow toward the low-pressure side. This causes the strain gauge to deform and therefore change its electrical resistance. This change in resistance can be detected and converted to a voltage. Thus the pressure difference (deformation of the strain gauge) can be measured as a voltage. Therefore equation 3.7 can be represented as a voltage.

$$p_1 - p_2 = \Delta p \tag{3.7}$$

EXAMPLE: If for $p_1 = 20$ psia, and $p_2 = 7$ psia, the voltage output is 12 volts, then $p_1 - p_2 = 13$ psid corresponds to 12 volts. If at some later time we have $p_2 = 12$ psia and an output voltage of 12 volts, then $p_1 - 12$ psia $= 13$ psid, so $p_1 = 25$ psia.

It is important to note that the voltage output from the strain gauge can only represent a pressure difference (similar to a manometer). It should also be noted that such a transducer must be calibrated in terms of a standard such as a manometer or dead weight tester. The calibration process involves measuring the transducer output voltage for a range of known pressures. Assuming that the transducer properties do not change, the pressures can then be obtained from the voltage. If the diaphragm is stretched or yields due to an overpressure, what will happen to the calibration?

3.4.2.3 Mechanical Pressure Gauges

Pressure gauges are generally constructed from Bourdon tubes, which are metal tubes bent into a helical shape with 1 or 2 coils. When a pressure is applied to the inside of the tube, the coil bends; if the pressure is higher than ambient, the coil straightens, if the pressure is lower, the coil bends more. The device inherently measures "gauge" pressure - that is, pressure with respect to ambient. A mechanical mechanism translates movement of the tube into movement of a point adjacent to a calibrated dial.

Pressure gauges must be calibrated by the manufacturer, and come in various grades and ranges. As long as all deformations are elastic, and the mechanism is in good condition, they should repeat within specification. If the mechanism is damaged, or inelastic deformations or creep occurs in the tube, all readings will be in error, and there is no inherent warning of the problem. There is thus a much greater risk of systematic error with these devices, as compared to the simple manometer. They are easier to use, however.

3.4.3 Velocity Measurement Using Differential Pressure

In steady, inviscid, incompressible flow, Bernoulli's equation provides a relationship between the fluid velocity and static pressure. This relationship, combined with the principle of continuity, can be used to determine flow velocity or flow rate from a differential pressure measurement. Two devices that use Bernoulli's equation for flow velocity measurement, the pitot-static tube and the venturi meter, are discussed in the following sections.

3.4.3.1 The Pitot-Static Tube

Pitot static probes are commonly used to measure velocities in aerospace applications. Average velocity measurements can be made with a pitot-static tube (Figure 3.7). The operation of a pitot-static tube is based upon the Bernoulli equation, which for a steady incompressible flow takes the form as in equation 3.8.

$$p_s + \frac{1}{2}\rho V^2 = p_t \tag{3.8}$$

where p_s is the static pressure, V is the average flow velocity, ρ is the fluid density, and p_t is the total or stagnation pressure.

The pitot-static tube is a combination of a pitot or total head tube for measuring total pressure and a static tube for measuring static pressure in the flow, thereby allowing the velocity to be determined at the point of measurement. The velocity at the open end of the tube is zero and therefore provides the stagnation pressure for the flow. Holes for the in the wall of the tube provide for static pressure measurement. To obtain an accurate measurement, the static-pressure holes must be positioned carefully, and the pitot tube must be aligned with the flow to within a few degrees (for details, see R.G. Folsom, "Review of the Pitot Tube", Transactions of the ASME, v. 79, pp. 1447-1460, 1956).

The p_t port is in the center of the front blunt end (Figure 3.8 a), which faces the oncoming flow. The flow stagnates (V drops to zero) at the port location, so that locally $p = p_t$ (at stagnation port, where $V = 0$) which is then transmitted by the port tube to the pressure-sensing device. The static port for p_s is on the side of the stream-aligned tube (Figure 3.8 b), so there is no local change in V, and so the local p_s is transmitted into the port tube. Solving Bernoulli's equation for the velocity gives the equation 3.9.

$$V = \left[\frac{2\left(p_t - p_s\right)}{\rho}\right]^{1/2} \tag{3.9}$$

EXAMPLE: Find velocity from dynamic pressure of 1.0 in H_2O.
Given: Ambient temperature 73°F, and Barometric pressure 754.3 mm Hg. Use the equation of state to correct air density for temperature and pressure, given as in 3.10.

$$\rho = \rho_0 \left(\frac{p}{p_0}\right)\left(\frac{T_0}{T}\right) \tag{3.10}$$

where $p = 754.3\mathrm{mmHg} \times \frac{1.0\mathrm{inHg}}{25.4\mathrm{mmHg}} \times \frac{0.49116\mathrm{lb_f/in^2}}{1.0\mathrm{inHg}} = 14.586\mathrm{lb_f/in^2(a)}$

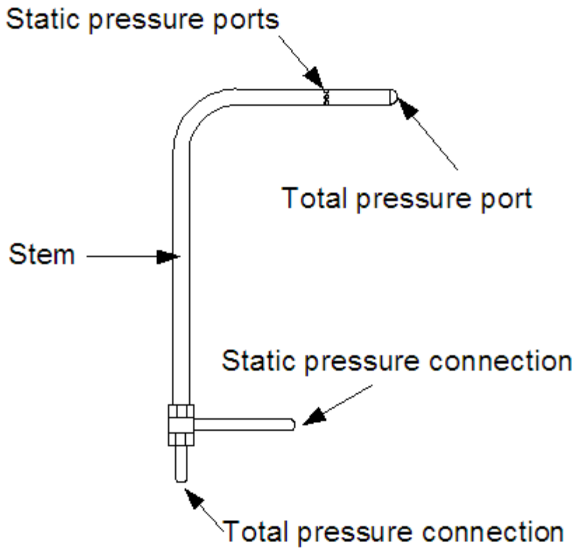


Figure 3.7: Schematic of a pitot-static tube

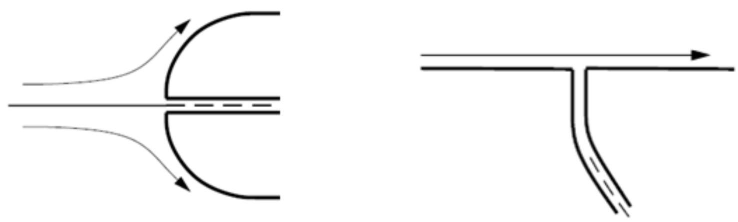


Figure 3.8: Pitot-static tube pressure ports. (a) Total Pressure (p_t) port, *left*; (b) Static Pressure (p_s) port, *right*

Also, T = 73 °F = 533 °R

$\rho_{air} = 0.002378 \mathrm{slug/ft^3} \times \frac{14.586 \mathrm{lb_f/in^2}}{14.696 \mathrm{lb_f/in^2}} \times \frac{519°\mathrm{R}}{533°\mathrm{R}} = 0.002298 \mathrm{slug/ft^3}$

Now find the velocity:

$V = \left[\frac{2(p_t - p_s)}{\rho}\right]^{1/2} = \left[2 \times \frac{1 \mathrm{inH_2O}}{0.002298 \mathrm{slug/ft^3}} \times \frac{0.03613 \mathrm{lb_f/in^2}}{1 \mathrm{inH_2O}} \times \frac{1 \mathrm{slug \cdot ft}}{1 \mathrm{lb_f \cdot s^2}} \times \frac{144 \mathrm{in^2}}{1 \mathrm{ft^2}}\right]^{1/2}$

V = 67.3 ft/s

Thus, for approximate calculation only,

$V = 67.3\sqrt{h}$

where V is in ft/s, and h is in inches of water.

3.4.3.2 The Venturi Tube

Venturi Tube Basics

A venturi tube is a tube purposefully built to be narrow in the middle, as shown in Figure 3.9. Such a tube has a converging section (where the area decreases along the fluid flow direction), a throat (where the area of the tube is a minimum) and a diverging section (where the area increases along the fluid flow direction). Venturi tubes are used in many engineering applications for measuring flow rate.

For the purposes of this lab, the fluid flow is assumed to be incompressible. For an incompressible flow, $\rho = constant$, which gives us the quasi-one-dimensional continuity equation:

$$\boxed{A_1 V_1 = A_2 V_2} \tag{3.11}$$

which states that the volume flow rate through a duct with changing area is constant. If such a fluid moves through a venturi tube, the continuity principle tells us the fluid velocity must increase through the converging section, thus increasing the kinetic energy. Conversely, the flow velocity decreases in the diverging section of the tube, decreasing the kinetic energy.

Conservation of energy can also be applied between two locations along the venturi tube. Neglecting friction (we will return to assess this assumption later) and assuming steady flow through the tube, the energy equation is simply Bernoulli's equation:

$$p_1 + \rho g z_1 + \frac{1}{2}\rho V_1^2 = p_2 + \rho g z_2 + \frac{1}{2}\rho V_2^2 \tag{3.12}$$

If the tube is level, there will be negligible difference in elevation ($z_1 = z_2$) between different points of the tube's centerline, which means the elevation head remains constant.

Recall that the continuity equation (Equation 3.11) tells us that velocity increases and decreases in the converging and diverging sections of the tube, respectively. From Bernoulli's equation, we can then deduce that the **pressure will decrease along the converging section, reach the minimum pressure at the throat, and then gradually increase along the diverging section**. Thus, energy conservation tells us that the pressure head energy decreases to account for the increase in kinetic energy in the converging section, and vice versa in the diverging section. The changes in velocity and pressure along the venturi tube are illustrated in Figure 3.9. As shown, the pressure is highest at the inlet, lowest at the throat, but less than what it was upstream as it exits the venturi tube. This is due to **pressure loss** in the divergent section.

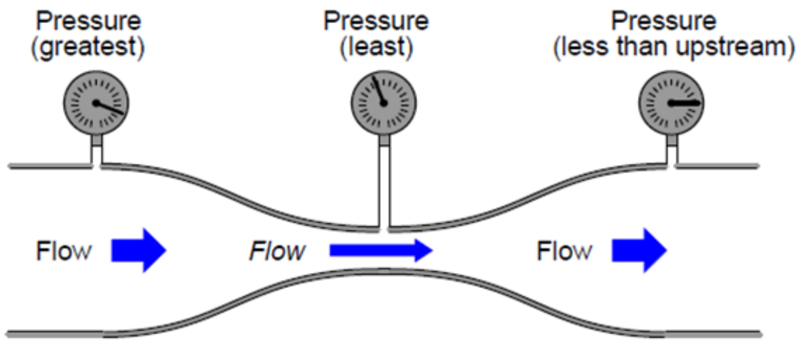


Figure 3.9: Pressure Variation in a Venturi Tube

Pressure Head Loss

Ideally, the pressure at a location downstream of the throat should be the same as the pressure at a location upstream if the cross-sectional area of the tube is the same. However, in practice the downstream pressure gauge will show slightly less pressure than the upstream gauge due to some inevitable energy loss as the fluid passes through the venturi. There is energy loss due to fluid friction against the walls throughout the entire tube, which converts kinetic energy of the flow to heat. Also, in the diverging section, the flow experiences an adverse pressure gradient (pressure increasing) which promotes boundary layer separation and turbulence generation. The energy lost to heat and turbulence introduces irreversible pressure losses in the system and so the pressure will not increase back to the inlet value, as illustrated in Figure 3.10. The difference between the upstream and downstream pressure is called **permanent pressure loss**, while the difference in pressure between the throat and the tube exit is called the **pressure recovery**.

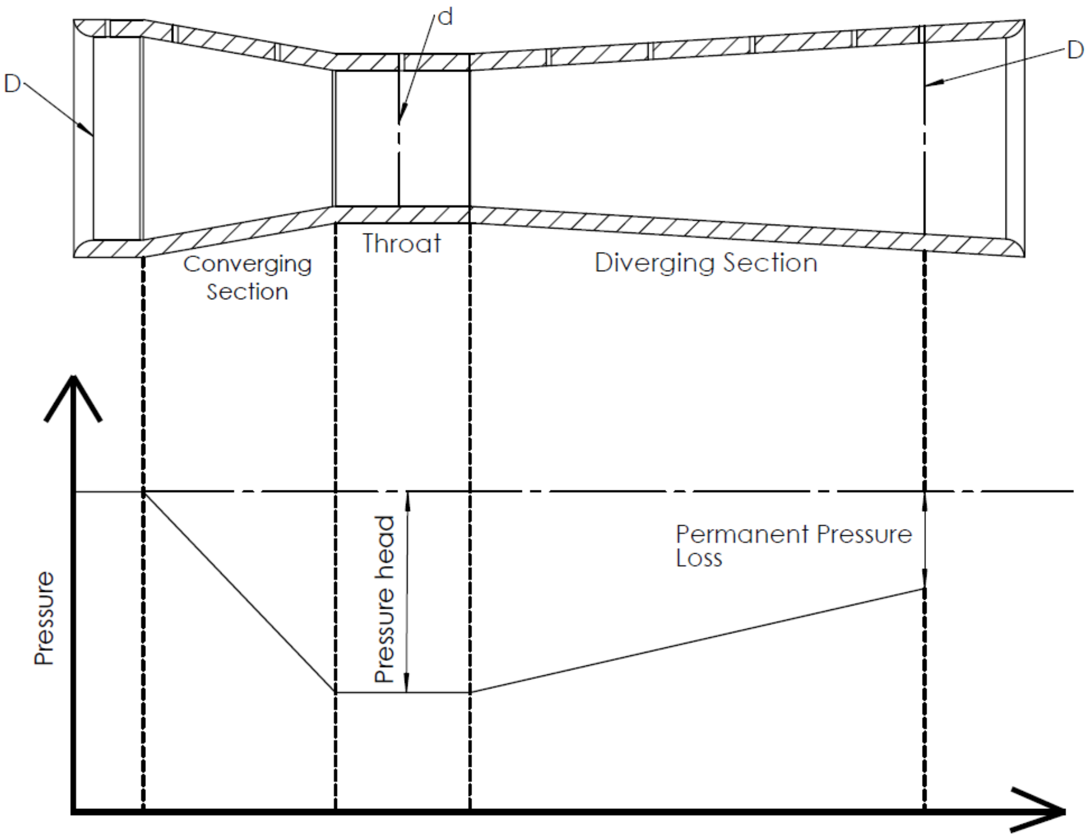


Figure 3.10: Pressure Variation along the Venturi Tube

Velocity and Flow Rate Calculation Using a Venturi Tube
A venturi tube can be used to measure velocity and flow rate if the pressure difference between two points can be measured. Bernoulli's equation can be applied between any two locations along the venturi tube where we can measure the static pressure:

$$p_i + \frac{1}{2}\rho V_i^2 = p_j + \frac{1}{2}\rho V_j^2 \tag{3.13}$$

where $i, j \in \{1, 2, 3, .., n\}$. where n is the total number of pressure taps in the venturi tube. Thus, the velocity difference between tap i and tap j is related to the static pressure difference by:

$$p_i - p_j = \frac{1}{2}\rho(V_j^2 - V_i^2) \tag{3.14}$$

The continuity equation (Equation 3.11 relates the velocity at tap i to the velocity at tap j:

$$V_j = \frac{A_i}{A_j} V_i \tag{3.15}$$

Substituting Equation 3.15 into Equation 3.14 and solving for V_i, we obtain:

$$\boxed{V_i = \sqrt{\frac{2(p_i - p_j)}{\rho\left[\left(\frac{A_i}{A_j}\right)^2 - 1\right]}}} \tag{3.16}$$

We can similarly solve for the velocity at tap j:

$$\boxed{V_j = \sqrt{\frac{2(p_i - p_j)}{\rho\left[1 - \left(\frac{A_j}{A_i}\right)^2\right]}}} \tag{3.17}$$

Thus, to calculate the flow velocity at two points along the venturi tube, one can needs to know the pressure difference between the two taps, the ratio of the tube cross-sectional areas at the two taps, and the density of the fluid. In practice, venturi tubes are typically used to measure the flow rate $Q = (VA)$ using pressure taps at the tube inlet and the throat. If we measure the pressure difference, $p_{throat} - p_{inlet}$, the flow rate is then:

$$Q = \text{constant} = Q_{throat} = A_{throat}V_{throat} = A_{throat}\sqrt{\frac{2\left(p_{throat} - p_{inlet}\right)}{\rho\left[1 - \left(\frac{A_{inlet}}{A_{throat}}\right)^2\right]}} \tag{3.18}$$

3.4.4 Flow Around a Cylinder and the Pressure Coefficient

From friction less potential theory (Figure 3.11), the fluid velocity about a circular cylinder at the surface with no circulation is given by equation 3.19

$$U_\theta = -2V_\infty \sin\theta \tag{3.19}$$

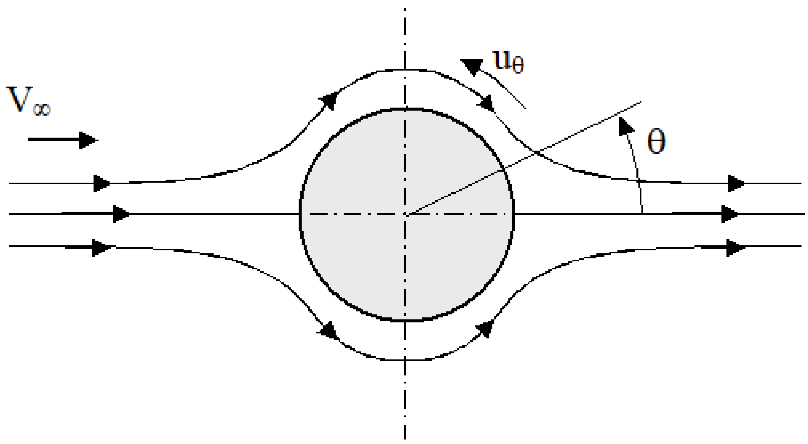


Figure 3.11: Frictionless potential flow around a non-rotating circular cylinder.

The pressure at the surface of the cylinder is, from Bernoulli's equation 3.20.

$$p_s + \frac{1}{2}\rho U_\theta^2 = p_\infty + \frac{1}{2}\rho V_\infty^2 \tag{3.20}$$

Here p_s is the static pressure at a point on the surface of the cylinder, p_∞ is the freestream static pressure, U_θ is the tangential velocity at a point on the surface of the cylinder, and V_∞ is the freestream velocity.

The pressure coefficient is defined as in equation 3.21.

$$C_P = \frac{p_s - p_\infty}{\frac{1}{2}\rho V_\infty^2} \tag{3.21}$$

So, for surface of the cylinder, the pressure coefficient is given by equation 3.22 for ideal frictionless potential flow.

$$C_P = \frac{\left(\frac{1}{2}\rho V_\infty^2\right) - \left(\frac{1}{2}\rho U_\theta^2\right)}{\frac{1}{2}\rho V_\infty^2} = 1 - \left(\frac{U_\theta}{V_\infty}\right)^2 = 1 - 4\sin^2(\theta) \tag{3.22}$$

3.5 Recommended Reading

It is highly recommended that students read the following sections in Fundamentals of Aerodynamics by John D. Anderson:

- 1.9 Fluid Statics: Buoyancy Force
- Chapter 3: Inviscid Incompressible Flow, specifically:
  - 3.2 Bernoulli's Equation
  - 3.3 Incompressible Flow in a Duct: The Venturi and Low-Speed Wind Tunnel
  - Pitot Tube: Measurement of Airspeed
  - 3.5 Pressure Coefficient
  - 3.18 Applied Aerodynamics: The Flow Over a Circular Cylinder - The Real Case

3.6 Data to be Acquired

Following is the data that you need to acquire over the course of this lab:

1. Dynamic pressure from
   1.1 a digital readout and
   1.2 an inclined manometer vs. wind tunnel frequency
2. Surface pressure measurements on a cylinder from $\theta = 0$ to $\theta = 180$ degrees.
3. Static pressure measurements from several taps along a venturi tube.

3.7 Lab Setup and Procedure

Lab Location: ARMS B098, Low-Speed Wind Tunnel

There are three parts to this lab:

- **I. Wind Tunnel Speed Calibration Using a Pitot-Static Tube**
- **II. Pressure Distribution Around a Cylinder**
- **III. Flow with Area Change**.

3.7.1 Experiment Setup and Preparation - I

1. ☐ Read more about the operation of the low speed wind tunnel in Section 7.1.
2. ☐ Familiarize yourself with the inclined manometer. The total pressure and static pressure ports from the Pitot tube are connected to the inclined and vertical arms of the manometer, respectively. Note that the manometer fluid meniscus is aligned with zero when there is no flow (total pressure = static pressure), and so when the tunnel is turned on, the higher total pressure will push the fluid down the inclined arm. The reading on the scale aligned with the bottom of the meniscus gives the differential pressure (total pressure - static pressure) in inches of water. Read more about the operation of the inclined manometer in Section 3.4.2.1.
3. ☐ Download and open the Lab 3 data spreadsheet from Brightspace. The spreadsheet will help you record data and perform calculations.

3.7.2 Experiment Procedure - Part I

3.7.2.1 Wind Tunnel Speed Calibration Using a Pitot-Static Tube and Manometer

Yellow bold highlighted text indicates that the students should show the data/plot obtained to the TA present at the lab.

1. ☐ Before turning on the wind tunnel, turn on the digital manometer connected to the pitot-static tube and press "ZERO" to tare the reading.
2. ☐ For a wind tunnel operating frequency of 10 Hz, read the dynamic pressure from the digital manometer. Record the value in the Lab 3 data spreadsheet.
3. ☐ For a wind tunnel operating frequency of 10 Hz, read the differential pressure from the inclined manometer. Record the value in the Lab 3 data spreadsheet.
4. ☐ Repeat the two pressure measurements for wind tunnel frequencies of 15, 20, 25, and 30 Hz. Record all measurements in the data spreadsheet.
5. ☐ Calculate the wind speed in the tunnel for each frequency using both the digital and inclined manometer data.
6. ☐ Plot the two wind speed measurements vs. frequency in the data spreadsheet. How different are the values obtained using the digital manometer vs. the inclined manometer? **Show your plot to the TA before proceeding.**

3.7.3 Experiment Setup and Preparation - Parts II and III

In these parts of the lab, pressure distributions on the surface of a cylinder and in a venturi tube will be measured. Two pressure scanner systems, labeled "A" and "B", are used to acquire the pressure data. Each scanner has 16 ports, labeled A1-A16 and B1-B16. Each port is connected to a piezoresistive pressure transducer that measures the differential pressure between the pressure at the port and a reference pressure. In Part II, the reference pressure port (labeled "RUN REF" on the scanner) is connected to the side of the wind tunnel so the reference pressure is the freestream static pressure p_∞. In Part III, the reference pressure port is connected to the inlet of the venturi tube. A pressure tap panel provides a convenient mechanism for connecting the pressure scanner ports to the pressure taps for a given test article. The pressure scanners are connected to the lab computer and a LabVIEW Virtual Instrument or VI is used to read the data from the 32 channels simultaneously. Click here to learn more about LabVIEW. **Please do not alter or modify any LabVIEW**

VIs - even very small changes can cause the program to stop working! The wind tunnel must be off when the program is started so the readings are properly "zeroed." Once the LabVIEW program is opened and running, the wind tunnel can be turned on and the experiment performed.

1. ☐ Verify that the pressure scanners are turned on (check that the "PWR and "LNK" LED lights are illuminated). They are located below the test section on the metal plates.

2. ☐ **Open the two VIs used for this lab.** The VIs used for these parts of the lab are `LowSpeedWT-Cylinder.vi` for Part II and `Venturi.vi` for Part III.

   Here WT stands for "Wind Tunnel". One member should log into the lab computer using their career account and navigate to the Lab 3 folder at the destination: `C:\temp\AAE33301\Lab 3 - Pressure in Aerodynamics`. Open the LabVIEW library file `Lab3_33301_PressureAero.llb` and then open the appropriate VI.

3. **Saving Experimental Data**

   3.1 ☐ Make a folder with your team name (e.g., Black16) in the `data_33301_lab3` folder. Make sure you save any and all data generated during the lab in this folder.

   3.2 ☐ Set the file path in the LabVIEW VI to your data folder. After the path, add a backslash and enter a name for the data file (in .txt format). For example, your final path might look like:
   `C:\temp\AAE33301\Lab 3 - Pressure in Aerodynamics\data_33301_lab3\Black16\Black16_pressure_data`.

   3.3 ☐ Click the Run arrow ⇨ at the top toolbar to begin running the program. The readouts from the pressure scanner channels should be fluctuating around zero. If the readouts are not fluctuating or there is an error message, get help from the TA.

   3.4 ☐ Verify that the program saves the data correctly as follows:

      3.4.1 ☐ Click the "Push to Write" button in the VI to write the pressure data to the file you specified in Step 3b. A push is confirmed when you see the green "Success" light and when the push counter increases.

      3.4.2 ☐ Press the STOP button on the VI [■ STOP] to stop the program (NOT the stop sign on the top toolbar but the word STOP on the screen under Main Program).

      3.4.3 ☐ Open the data file using Notepad, Wordpad or Excel. The first row of data is the pressure scanner channel number and the second row is the pressure readings from the channels at the time you clicked the "Push to Write" button. Note that these are *differential* pressures, i.e., the difference between the static pressure at the tap and a reference pressure. For the experiments in this lab, the reference pressure is the wind tunnel freestream static pressure, p_∞. (There is a port on the pressure scanner boxes labeled "RUN REF" that we have connected to a port in the sidewall of the wind tunnel.)

3.7.4 Experiment Procedure - Parts II and III

3.7.4.1 Part II: Pressure Distribution Around a Cylinder

The cylinder model has 11 pressure ports on its surface, spaced evenly from $\theta = 0°$ (front of the cylinder, facing the freestream flow) to $\theta = 180°$ (back of the cylinder). The pressure ports are connected to channels 1-11 of pressure scanner "A". The reference pressure port is connected to the side of the wind tunnel, so the pressure measurement displayed in LabVIEW is the static pressure at the port minus the freestream static pressure, $p_i - p_\infty$, in psi. The diameter of the cylinder is 2 inches.

1. ☐ If the VI is still running, press the STOP button [■ STOP]. It is important to restart the VI when the tunnel is off so the pressure readings zero out.

2. ☐ Click the Run arrow ⇨ at the top toolbar to begin running the program. The readouts from the pressure scanner channels should be fluctuating around zero.

3. ☐ Set the wind tunnel operating frequency to 15 Hz and turn it on.

4. ☐ Observe the surface pressure distribution on the VI. Does it look like you expect? Remember that the VI is plotting differential pressure (local surface pressure - p_∞) at each tap vs. angular location (0 to 180° in increments of 18°). **Discuss your observations with the TA.**

5. ☐ Click the "Push to Write" button in the VI to write the pressure data to the file you specified in Step 3.2.

6. ☐ Increase the wind tunnel frequency to 30 Hz and click "Push to Write" to record the pressure data.

7. ☐ Copy and paste your pressure data into the appropriate tab of the Lab 3 data spreadsheet. Convert the differential pressure measurements to pressure coefficients (be careful about units!).

8. ☐ On the same plot, show your experimental values for pressure coefficient vs. θ and the C_p distribution predicted by potential flow theory for inviscid flow around a cylinder. What differences in behavior do you see? Can you explain this from your knowledge of inviscid and viscid flow? **Discuss these questions with your TA.**

3.7.4.2 Part III: The Venturi Tube - Application of Bernoulli's Principle

A schematic of the Venturi Tube used in this experiment is shown in Figure 3.12. There are 9 pressure taps along the tube: 1 at the inlet, 2 in the converging section, 1 at the throat, and 5 in the diverging section. The locations of the pressure taps and the tube internal diameter at each tap are given in Table 3.2.

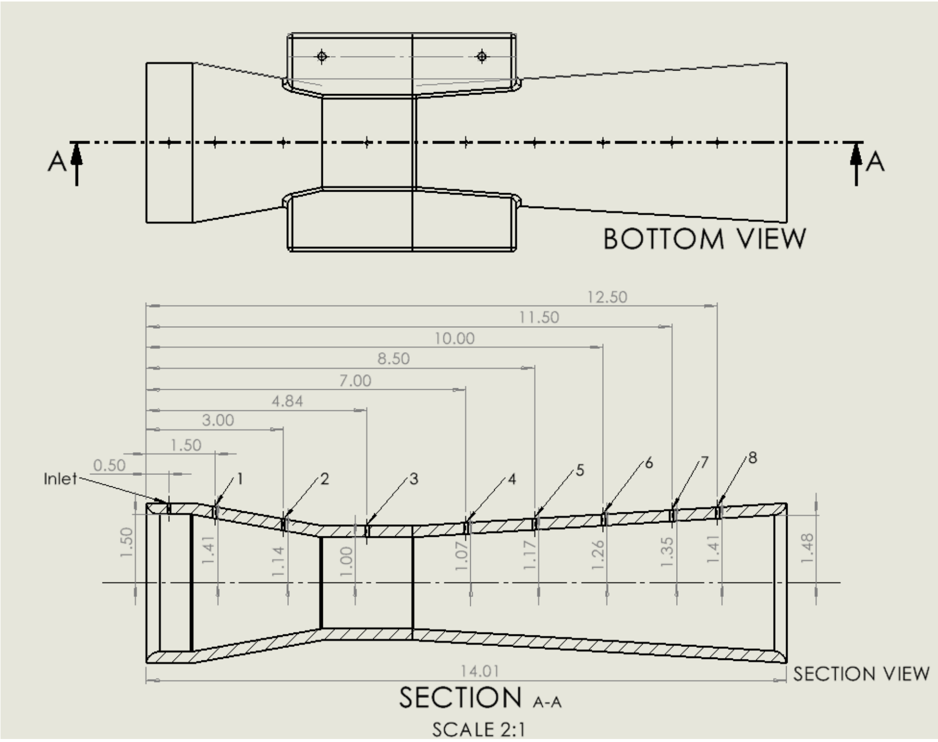


Figure 3.12: Bottom and section view of Venturi Tube

1. ☐ Verify that the tubing from the venturi tube is connected properly to the pressure tap panel. The first tap (at the tube inlet) should be connected to the Reference port on the panel, and the next 8 taps should be connected to ports B1 to B8 in the order shown in Table 3.2.

2. ☐ Run the LabVIEW program (`Venturi.vi`) and verify that the pressure values for all 8 ports are oscillating about zero.

3. ☐ Turn on the venturi tube inlet fan to the maximum speed and observe the pressure distribution. Does the pressure distribution look like you expected?

4. ☐ Click "Push to Write" to record the pressure data. Remember that we are measuring the *differential* pressure between each pressure tap and the inlet pressure tap in units of psi.

5. ☐ Turn off the fan and stop the VI. Transfer the pressure data to the Lab 3 Data spreadsheet.

Tap Number	**Nature**	**Tap Location (relative to inlet)**	**Tube Inner Diameter**
Inlet (Reference)	Inlet	0	3
1	Converging	1	2.82
2		2.5	2.28
3	Throat	4.34	2
4	Diverging	6.5	2.14
5		8	2.34
6		9.5	2.52
7		11	2.7
8		12	2.82

Table 3.2: Pressure tap locations (relative to the venturi tube inlet) and local tube inner diameter.

6. ☐ In the data spreadsheet, plot the differential pressure vs. tap location. **Show your plot to the TA and discuss.**

7. ☐ In the data spreadsheet, calculate the velocity (using Equation 3.16) and volumetric flow rate at the inlet and taps 1-8. For the best results, calculate the velocity in the converging section (taps 1-3) using the inlet as the reference, i.e., tap i in Equation 3.16 is tap 1, 2, or 3, and tap j is the inlet. In the diverging section (taps 4-8), use the throat (tap 3) as the reference (i.e., tap j in Equation 3.16). To calculate the velocity at the inlet of the venturi tube, it is recommended that you use tap 1 as the reference. Plot the calculated velocity at each tap on the same graph as the differential pressure. **Show your results to the TA and discuss.** Why do you think it's important to use different reference points for the converging vs. diverging sections of the tube? How well does Bernoulli's equation work in this experiment, i.e., how well is continuity satisfied?

8. ☐ **Return the experiment and the lab to its state when you first arrived.** Return all tools and equipment back to their designated location, turn off any equipment that was started for the purpose of your lab, and clean up your workspace. **Obtain approval from the TA before leaving the lab.** Thank you!

You will use the information recorded in the Lab 3 data spreadsheet to complete the post-lab assignment. Remember that the post-lab should be completed **individually** and uploaded in Gradescope within 1 week of your lab session.

Chapter 4

Wakes and Drag Measurement

A large part of this background document is adapted from Chapter 3 of Aerodynamics for Engineers by John J. Bertin (4th Edition).

4.1 Introduction

For low-speed flow around bodies, there are two types of drag: form/profile or pressure drag and skin friction drag. Form drag is a result of boundary growth and, in some cases, boundary layer separation on the pressure distribution on the body, whereas skin friction drag is a result of shear stress on the body surface. In this lab, a force balance will be used to measure the total drag (form + skin friction drag) on various body shapes over a range of Reynolds numbers.

4.2 Lab Overview

In this lab, you will measure the total drag on models of some basic, two-dimensional bodies, including flat plates, cylinders, and an airfoil. You will mount the models on the force balance in the low-speed wind tunnel and measure the drag for a range of velocities (and hence Reynolds numbers). You will then calculate the drag coefficient and compare with the approximate values shown in Figure 4.5.

4.3 Objectives

In this experiment you will:

1. Learn about form (pressure) and skin friction drag.
2. Study the effect of body shape and geometry on the drag force and drag coefficient.
3. Study the effect of Reynolds number on the drag force and drag coefficient.
4. Gain experience using a force balance and LabVIEW software to take drag measurements for various body shapes and sizes.

4.4 Background

4.4.1 Pressure on a Cylinder

In Lab 3 we studied the pressure on the surface of a cylinder. Using Bernoulli's equation, an expression for the local static pressure as a function of the angle can be obtained as in equation 4.1.

$$p = p_\infty + \frac{1}{2}\rho_\infty U_\infty^2 - 2\rho_\infty U_\infty^2 \sin^2\theta \tag{4.1}$$

The pressure coefficient is then given by 4.2.

$$C_p = 1 - 4\sin^2\theta \tag{4.2}$$

This theoretical curve for pressure coefficient versus angle on the cylinder is plotted in Figure 4.1, where in this case $\theta = 180°$ corresponds to the stagnation point on the front surface of the cylinder.

However, we know from previous labs that the actual flow field around a cylinder differs significantly from the potential flow solution due to the effects of viscosity. The flow field does not remain symmetric around the cylinder, but rather the boundary layer separations at some location and a low pressure wake forms behind the cylinder. Also shown in Figure 4.1 are curves corresponding to experimental data for the pressure coefficient on a cylinder. One data set is for experiments with a subcritical Reynolds number, and the other is for a supercritical Reynolds number. The critical Reynolds number, approximately 3×10^5, is the number at which the boundary layer becomes turbulent. Therefore, it is expected that in a flow with a supercritical Reynolds number, the boundary layer will stay attached to the surface longer and hence the wake will be smaller. In Figure 4.1, the data shows that for the supercritical Reynolds number, the flow separates at a later angle than for the subcritical Reynolds number. It should also be noted that the critical Reynolds number varies depending on the free stream turbulence and the surface roughness of the body.

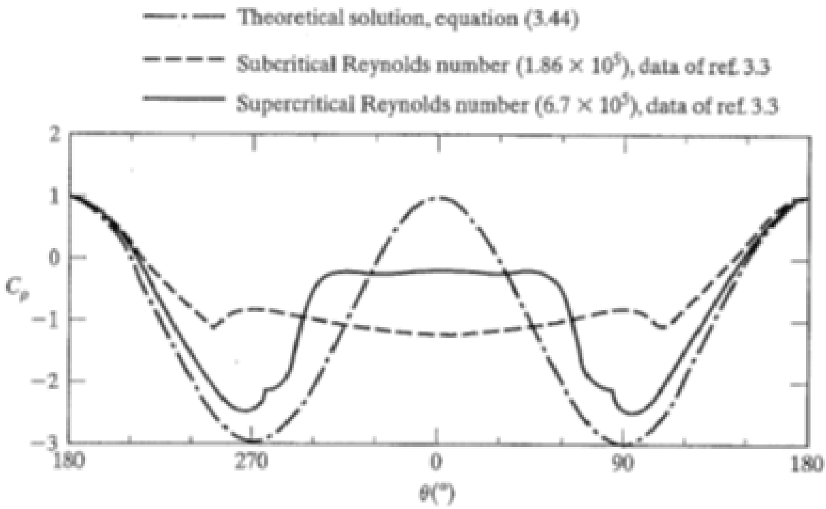


Figure 4.1: Theoretical and experimental values of the pressure coefficient versus angle on a cylinder (taken from Aerodynamics for Engineers, 4th Ed. By John J. Bertin, Chapter 3).

4.4.2 Lift and Drag

4.4.2.1 Lift and Drag on a Cylinder

As the air particles move around the cylinder, their motion produces a force on the surface. The force at a point on the surface has a normal component and a tangential component; the normal force is the pressure, and the tangential component is the shear force. Similarly, the overall resulting force on the cylinder can be resolved into two components, one perpendicular to direction of the free-stream velocity (lift) and the other parallel to the free-stream velocity (drag). These forces are illustrated in Figure 4.2.

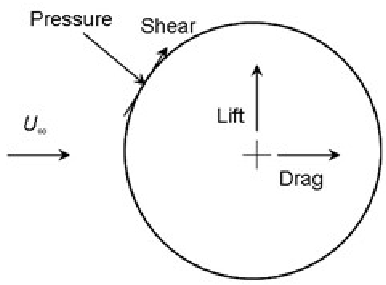


Figure 4.2: Forces acting on a cylinder

To calculate the contribution of the pressure to the lift, the equation for the pressure versus angular location for inviscid potential flow can be used. With careful consideration of the cylindrical geometry, the lift per unit span due to the pressure is

$$l = -\int_{0}^{2\pi} p \sin\theta \cdot R d\theta \tag{4.3}$$

$$= -\int_{0}^{2\pi} \left(p_\infty + \frac{1}{2}\rho_\infty U_\infty^2 - 2\rho_\infty U_\infty^2 \sin^2\theta \right) \cdot \sin\theta \cdot R d\theta \tag{4.4}$$

After evaluating the integral, one finds that $l = 0$, which makes good sense since the pressure distribution is symmetric about the horizontal plane. Also, the lift due to pressure can found using the pressure coefficient,

$$\frac{l}{q_\infty 2R} = -\frac{1}{2}\int_{0}^{2\pi} C_p \sin\theta d\theta \tag{4.5}$$

where q_∞ is the dynamic pressure, $q_\infty = \frac{1}{2}\rho_\infty U_\infty^2$, and therefore

$$C_l = \frac{l}{q_\infty 2R} \tag{4.6}$$

is the section lift coefficient for the cylinder. Similarly, the pressure contribution to the drag can be calculated:

$$d = -\int_{0}^{2\pi} p \cos\theta \cdot R d\theta \tag{4.7}$$

$$= -\int_{0}^{2\pi} \left(p_\infty + \frac{1}{2}\rho_\infty U_\infty^2 - 2\rho_\infty U_\infty^2 \sin^2\theta \right) \cdot \cos\theta \cdot R d\theta \tag{4.8}$$

As with the lift, when the integral is evaluated one finds that d = 0. However, we know from practical experience that there is in fact drag on a cylinder, so this result is known as d'Alembert's paradox. This calculation assumes that the flow is inviscid and so does not account for the wake behind the cylinder. The pressure in the wake is much lower than the theoretical pressure predicted by the equation, so the difference between the high pressure on the front of the cylinder and the low pressure behind results in a large drag component.

As stated before, the force produced by integrating the pressure force is called form or pressure drag. However, another component to the drag force is found by integrating the shear force over the body surface – this force is called the skin friction drag. For a blunt body with a large low-pressure wake, the majority of the drag is due to form drag, while the skin friction drag is small. The friction drag is found using the skin friction coefficient,

$$C_f = \frac{\tau_s}{q_\infty} \tag{4.9}$$

where the shear stress at the body surface, τ_s, is related to the viscosity, μ, as discussed in Lab 1:

$$\tau_s = \mu \left.\frac{\partial u}{\partial y}\right|_s \tag{4.10}$$

The skin friction drag coefficient is then defined as

$$C_{D,skinfriction} = kC_f \frac{S_{wetted}}{S} \tag{4.11}$$

where k is the form factor and is a weak function of the shape, S_{wetted} is the wetted surface area, and S is the reference area used to calculate the drag.

The drag coefficient per unit span for a cylinder is defined the same way as the section lift coefficient, and is given by equation 4.12.

$$C_d = \frac{d}{q_\infty 2R} \tag{4.12}$$

where $d = d_{form} + d_{friction}$. The form drag on a cylinder for subcritical Reynolds number changes very little with varying Reynolds number, as shown by the experimental data in Figure 4.3. As expected, for supercritical Reynolds numbers the drag coefficient is reduced significantly. While skin friction is small for blunt bodies like spheres and cylinders, for streamlined bodies (e.g. airfoils) the wake is very small and the skin friction drag dominates. Skin friction drag, unlike form drag, is always dependent on the Reynolds number as well as the surface roughness.

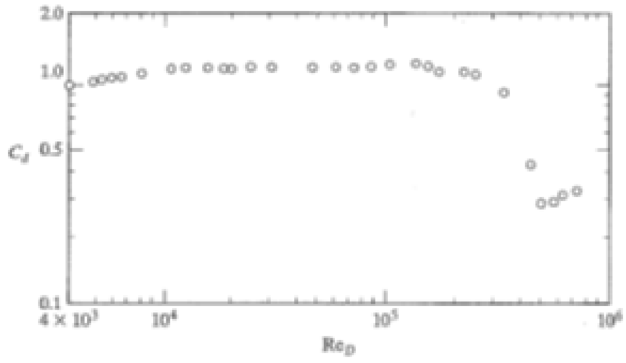


Figure 4.3: Drag coefficient vs. Reynolds number for a smooth cylinder (taken from Aerodynamics for Engineers, 4th Ed. By John J. Bertin, Chapter 3).

4.4.3 Drag of Various Bodies

As was discussed in the previous sections, the net drag is produced by both pressure and shear stress effects. In most instances, these two effects are considered together, and an overall drag coefficient, C_D is used:

$$C_D = \frac{D}{\frac{1}{2}\rho_\infty U_\infty^2 A} \tag{4.13}$$

where D is the total drag force and $A = bD$ is the appropriate reference area. For blunt bodies, since pressure drag dominates, the appropriate reference area is the ***frontal area*** where b is the span of the body perpendicular to the flow and D is the body thickness perpendicular to the flow. For thin streamlined bodies, where the friction drag dominates, it is customary to use the ***planform area*** as the reference area. In this case, b is still the span of the body perpendicular to the flow, but D is the length of the body *parallel* to the flow (i.e., in the flow direction). This makes sense because the shear stress acts on the platform area, rather than the much smaller (for thin bodies) frontal area.

There is an abundance of such drag coefficient data available in the literature. This information covers incompressible and compressible viscous flows past objects of almost any shape of interest—both man-made and natural objects. The drag coefficients C_D vs. Reynolds number for objects with various degrees of streamlining are shown in Figure 4.4. The effects of the body shape and subcritical vs. supercritical Reynolds number on the flow field and drag coefficient are illustrated in Figure 4.5. Some questions to consider while studying the figure 4.5:

1. Why is the drag force larger on some bodies than others?
2. In which cases is the profile drag dominant, in which cases is the skin friction drag dominant, and why?
3. What is the effect of cylinder size on the drag coefficient?
4. Why do some bodies have a larger drag force but lower drag coefficient?
5. What is the effect of increasing the Reynolds number?

4.5 Recommended Reading

It is highly recommended that students read the following sections in Fundamentals of Aerodynamics by John D. Anderson:

- 1.12 Applied Aerodynamics: The Aerodynamic Coefficients - Their Magnitudes and Variations
- 3.18 Applied Aerodynamics: The Flow Over a Circular Cylinder - The Real Case

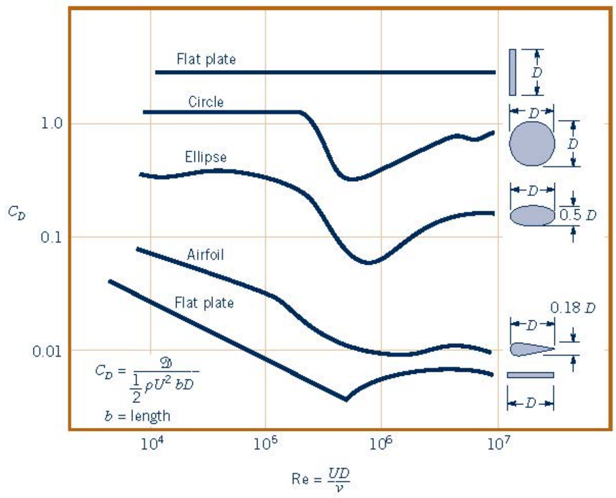


Figure 4.4: Character of the drag coefficient as a function of Reynolds number for objects with various degrees of streamlining, from a flat plate normal to the upstream flow to a flat plate parallel to the flow (taken from Fundamentals of Fluid Mechanics, 7th Ed. by Munson, Okiishi, Huebsch, and Rothmayer, Figure 9.22).

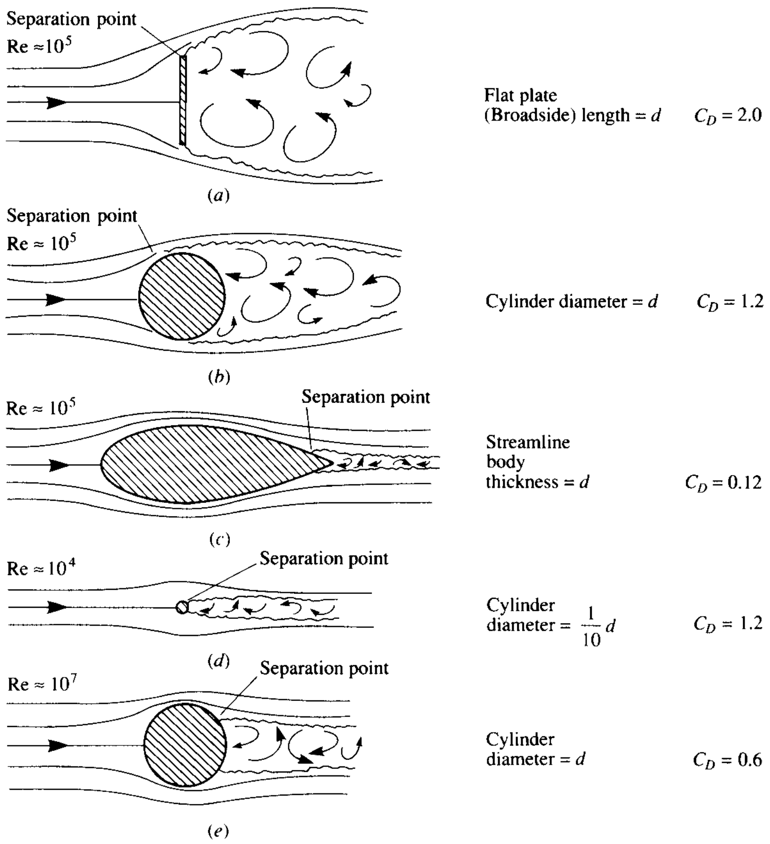


Figure 4.5: Comparison of section drag coefficients for the different bodies and Reynolds numbers (Source: Talay, T.A., Introduction to Aerodynamics of Flight, NASA SP-367, 1975).

- 4.12 Viscous Flow: Airfoil Drag

4.6 Data to be Acquired

1. Dimensions of all test models: Vertical and horizontal flat plates, small and large smooth cylinders, large rough cylinder, airfoil
2. Differential pressure from the pitot probe

3. Frequency of wind tunnel operation
4. Drag force using the wind tunnel force balance

4.7 Lab Setup and Procedure

4.7.1 Experiment Setup and Preparation

Lab Location: ARMS B098, Low Subsonic Wind Tunnel

In this lab, a force balance will be used to measure the drag force on various bodies as a function of Reynolds number. Bodies that will be tested include a **smooth horizontal flat plate, vertical flat plate, large and small smooth cylinders, large cylinder with rough surface, and a streamlined body (airfoil)**.

4.7.1.1 Preparation

1. ☐ One member should log into the lab computer using their career account and navigate to the Lab 5 folder at the destination: `C:\ Temp \ AAE33301 \ Lab 4 - Wakes and Drag Measurement`.
2. ☐ Download the Lab 4 data spreadsheet from Brightspace. The spreadsheet will help you record data and perform calculations.
3. ☐ Open the "**Lab4_33301_WindTunnelBalance**" VI. It measures the forces on the model being tested inside the wind tunnel's test section by the use of an electronic force balance system (read more about force balance in section 7.1.1).

4.7.2 Experiment Procedure

Yellow bold highlighted text indicates that student should show the data/plot obtained to the TA present at the lab.

1. **Calibration of Force Balance**
   Since we are only analyzing drag force, we will calibrate the balance for drag force (and not lift).

   1.1 ☐ Make sure the tunnel is not running, contains no test specimen in the test section, and that there are no weights placed on the force balance.

   1.2 ☐ Run the WindTunnelBalance VI. When you run the program, it zeroes out all the forces in the sensors.

   1.3 ☐ Record the initial values of drag in the data spreadsheet against zero actual weight (when no weights are placed on the balance) from the VI.

   1.4 ☐ Apply known weight(s) of 0, 0.5, 1, 2, 3, 4 and 5 lbs sequentially by **stacking** them to the pulley platform located just behind the F2 transducer and let the balance even out (also record the weight given by LabVIEW when no weights are placed). Place half pound and then the subsequent weights. Note that the curve is quite linear for most part, but remember that calibration is imperative to get accurate data.

   1.5 ☐ Record the values of drag from the program into the data spreadsheet.

   1.6 ☐ Plot "Actual Weight for Drag" vs "LabVIEW Weight for Drag".

   Compare your calibration values to the actual value of the weight you used and fit a calibration curve for your measurements. Show your calibration curve to the TA before proceeding.

2. **Calibration of Wind Tunnel Speed**
   - ☐ You do not have to calibrate the wind tunnel for flow speed vs. motor speed, as long as you still have the calibration data from Lab 3. You will use that same calibration for this lab. **Show your calibration curve to the TA before proceeding.**

3. **Drag Measurement of Various Bodies**

   Each model will be attached to the force balance using a stand, which will generate some additional drag. There is an extra stand for each model that you can mount by itself in the wind tunnel and measure its drag. Each model has been assigned a number and the corresponding stand is the number of the model, followed by the letter "S".

3.1 ☐ Locate the horizontal flat plate model (model 1) and the extra stand (model 1S). With the **wind tunnel off**, install the **stand** (model 1S) in the mounting block on the force balance.

3.2 ☐ Click the white "Run" arrow to start the LabVIEW program (WindTunnelBalance VI) **with the wind tunnel turned off** (this is crucial so the program resets the lift and drag to zero).

3.3 ☐ Turn on the wind tunnel to 10 Hz, and record the drag on the stand in the appropriate worksheet in the Lab 4 data spreadsheet. You will subtract this drag from the total drag you will record with the flat plate mounted on the stand.

3.4 ☐ Record the drag on the stand for wind tunnel speeds of 15, 20, 25, and 30 Hz.

3.5 ☐ Turn off the wind tunnel and install the **horizontal plate model** (model 1) on the force balance. Make sure the plate is secure, then close the top of the test section.

3.6 ☐ Start the LabVIEW program (with the tunnel off) and record drag readings for the horizontal plate at speeds of 10, 15, 20, 25, and 30 Hz. **Show your results to the TA**.

3.7 ☐ Repeat Steps 3.1 to 3.6 for the **vertical flat plate (models 2/2S), smooth small cylinder (models 3/3S), smooth large cylinder (models 4/4S), and rough large cylinder (models 5/5S)**. Remember to take drag measurements of the different mounting stands alone first, then with the body mounted on the stand. Also note that the mounting stands for smooth and rough large cylinder are same (4S and 5S are the same stand).

3.8 ☐ Repeat steps 3.1 to 3.6 using a streamlined body, the **airfoil (models 6,6'/6S)**. First, align the front of the airfoil with the airflow (model 6), then repeat with the airfoil facing the opposite direction (model 6'). The airfoil stand (6S) has a micrometer, which will not be used in this experiment. Also, check if the angle of attack of the airfoil is zero using a torpedo leveler.

3.9 ☐ Measure the dimensions of the bodies – length, width, and thickness for the flat plates, diameter and height for the cylinders, and chord and span for the airfoil.

3.10 ☐ Calculate Reynolds Number (Re) and Coefficient of Drag (C_d) for each of the cases in the spreadsheet.

3.11 ☐ **Return the experiment and the lab to its state when you first arrived.** Return all tools and equipment back to their designated location, turn off any equipment that was started for the purpose of your lab, and clean up your workspace. **Obtain approval from the TA before leaving the lab.** Thank you!

You will use the information recorded in the Lab 4 data spreadsheet to complete the post-lab assignment. Remember that the post-lab should be completed **individually** and uploaded in Gradescope within 1 week of your lab session.

***** ALWAYS TURN OFF THE TUNNEL BEFORE OPENING LID TO CHANGE OBJECT IN TUNNEL. CHECK THAT THERE ARE NO LOOSE OBJECTS IN THE TUNNEL.***
***** ALWAYS CLICK "RUN" TO START THE LABVIEW VI BEFORE TURNING ON THE TUNNEL TO RESET THE LIFT AND DRAG TO ZERO.***

Chapter 5

Wakes and Drag Measurement II

5.1 Introduction

A wake is a flow pattern that develops behind an object as the object moves relative to a real fluid. Wakes form behind moving bodies for all speeds (except at Re < 1) and flow regimes. The wake is one of the most fundamental flow configurations in fluid mechanics. Its importance is due to a variety of phenomena that have scientific as well as practical interest.

5.2 Lab Overview

In this lab, we will examine the specific case of the wake behind a cylinder. However, this does not limit the scope of our discussion on the fundamental processes in the wake. The cylinder is chosen because it has been investigated extensively by theory and experiment and is a good example of a bluff body with massive flow separation. The character of the flow behind the cylinder depends on the Reynolds number. Lab 1 you observed mostly laminar wakes behind circular cylinders characterized by periodic vortex shedding. The very low flow velocity in the water tunnel resulted in sufficiently low Reynolds numbers to observe laminar wakes. In this lab, much higher flow velocities are used in the low-speed wind tunnel, and thus the Reynolds number is high enough so the wake transitions to fully turbulent. The mean velocity in the wake is lower than the uniform free stream velocity due to loss of kinetic energy from the mean flow to the turbulent fluctuations. This *velocity deficit* is associated with the drag acting on the body, and so by measuring the velocity distribution in the wake we can estimate the drag.

5.3 Objectives

In this experiment you will:

1. Learn about the structure of a wake.
2. Learn to calculate the drag on a body using the velocity profile of the wake.
3. Compare the wake structure and drag of a smooth cylinder, rough cylinder, and airfoil.
4. Obtain experience in using a hot-film anemometer.

5.4 Background

5.4.1 The Turbulent Wake

In a laminar flow the fluid particles move downstream in smooth and regular trajectories, without appreciable mixing between different layers of fluid. In a turbulent flow, on the other hand, an irregular and seemingly random motion is superimposed on the average downstream motion of the fluid. This turbulence involves a good deal of mixing and interchange of mass between different average streamlines. This mixing exchanges lumps of fluid between various streamlines. It is important to note that there is not simply a mutual exchange of mass, but an exchange of momentum between different streamlines. In the mixing process, slow-moving fluid is invigorated and sped up by nearby fast-moving fluid, while the latter, in turn, is slowed down.

Turbulent flows are far more common than laminar flows, both in nature and in engineering devices. For example, the flow of water in rivers, the wakes extending from objects flying through the air and from ships on the sea, and the motion of the air in the atmosphere are practically always turbulent. A working knowledge of turbulent phenomena is absolutely essential in the practice of engineering. The energy of a body set in motion through an infinite volume of stationary fluid will eventually be dissipated as heat by viscous action within the fluid. For incompressible flow at high Reynolds number, the primary mechanism of dissipation is the turbulent wake. The energy of the body's motion is transferred to the kinetic energy of the turbulent fluctuations that define the wake. Viscous friction in turn causes these fluctuations to decay and their energy to be ultimately converted into heat.

Early investigators of this phenomenon found that energy "lost" from the free stream to the turbulent fluctuations was associated with a velocity deficit in the wake. From elementary consideration of the conservation of momentum it was clear that the magnitude of this deficit is a measure of the TOTAL drag acting on the body. The fluid mechanics of the wake are thus of considerable practical importance. The properties of technical interest are the velocity distribution, the rate at which the wake grows in the downstream direction, and the mechanism by which scalar quantities such as heat and matter are spread outward into the undisturbed fluid.

The wake is a region of entrained fluid moving at nearly the velocity of the body. After the body has passed a given point, the turbulent region spreads and the fluid slows until viscosity damps all motion. In a wind tunnel the body is stationary and the fluid moves past the body. From a wind tunnel frame of reference, the wake profile is shown in Figure 5.1.

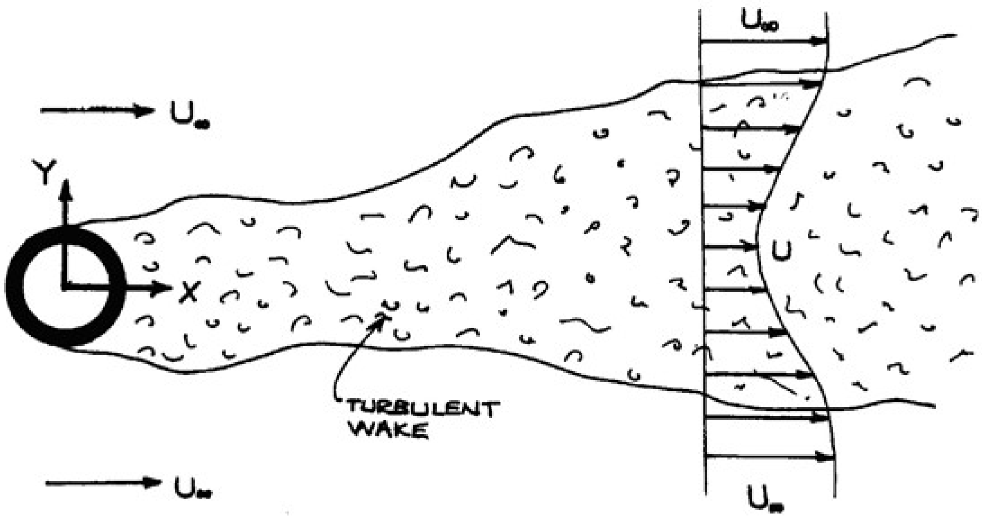


Figure 5.1: Sketch of a Turbulent Wake, in a Lab-Fixed Frame

The turbulent fluctuations that characterize the wake must be superimposed upon the mean profile shown. The boundary of the wake is determined by how far the velocity deficit and turbulent motion penetrate the surroundings. It is assumed the turbulent fluctuations decay as the mean velocity approaches the free stream velocity.

5.4.2 The Intermittent Wake Boundary

Observations have shown that turbulence does not taper off towards the boundaries of the wake. Rather, there is a sharp demarcation between turbulent and non-turbulent fluid. This surface shows up clearly in shadowgraphs of the turbulent wake, depicted in 5.2.

A stationary probe placed near the wake will find itself swept over by successive regions of turbulent and non-turbulent fluid, giving an intermittent quality to the output signal. The remarkable conclusion drawn from studies of this intermittent phenomenon is that a continuous range of states varying from fully turbulent to fully laminar does not ordinarily exist in nature. If both types of flow are present in a velocity field, they are distinct.

Actually, the phenomenon is a matter of common experience. If you have ever stood alongside a highway on which vehicles were moving at sufficient Reynolds number (say at 60 mph) and felt the wake pass over you, you have experienced the sharp boundary. The gust does not gradually rise to its full strength. Rather, at one

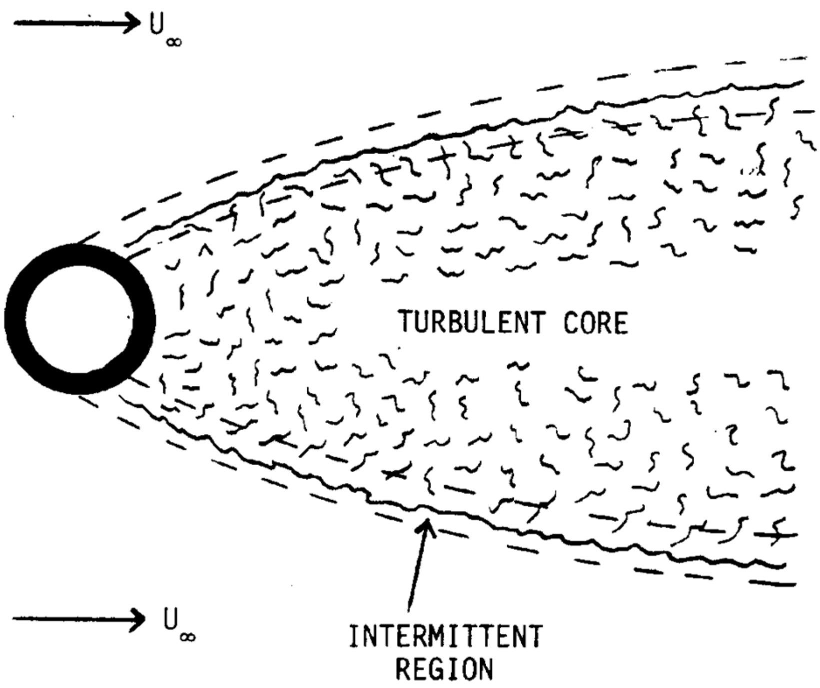


Figure 5.2: Schematic of a Wake Shadowgraph

instant you are standing in calm air and in the next the wind strikes you with its full force, surrounding you with the swirling gusts of the wake.

5.4.3 Relationship between Drag and the Wake

The velocity profile (a plot of speed vs. a distance normal to the flow and the cylinder) of the wake at different distances downstream of the cylinder sometimes shows similarity, i.e., a similar pattern of streamlines varying in a manner relative to a geometric dimension. Figure 5.4 presents the nomenclature of the wake. The velocity deficit in the wake is given by equation 5.1.

$$u_1 = U_\infty - u \tag{5.1}$$

where U_∞ is the free stream velocity, and u is the velocity at some point in the wake. Note that $u \to U_\infty$ outside the wake.

As was mentioned earlier the total drag acting on the body depends on the velocity deficit. Applying the momentum theorem to a control volume that encloses the cylinder gives us equations 5.2 and 5.3.

$$D = h\rho \int_{-\infty}^{+\infty} u(U_\infty - u)dy \tag{5.2}$$

$$D = h\rho \int_{-\infty}^{+\infty} u_1(U_\infty - u_1)dy \tag{5.3}$$

where ρ is the fluid density, h is the height of the body, D is the drag force, u is the velocity in the wake at the point y. The detailed derivation can be found in Schlichting's text. The drag on the cylinder can also be expressed as in equation 5.4.

$$\mathrm{D} = \frac{1}{2}\rho\, A\, U_\infty^2\, C_D \tag{5.4}$$

where D is the drag, C_D is the drag coefficient, ρ is the density, U_∞ is the freestream velocity, and A is the frontal area of the body. For a cylinder, $A = hd$, where d is the diameter. The drag coefficient is somewhere around 1 (see prelab question 4).

For a wing, A is usually the chord times the span, as in *Abbott and Von Doenhoff, Theory of Wing Sections, Dover, 1958, p. 3.* The same reference gives a typical drag coefficient of 0.01 at zero angle of attack. For a wing of 10% thickness, this makes a drag coefficient based on the thickness of about 0.1.

One classical analysis shows that for self-similar wake sections, one can use the equation 5.5

$$\frac{u_1}{u_{1\,\max}} = [1 - (y/b)^{3/2}]^2 \tag{5.5}$$

Figure 5.3 presents the curve of Equation 5.5. The similarity concept is displayed in this curve. In most cases, experiments show that the curve is independent of the distance downstream of the cylinder, provided that $x/(C_D d) > 50$.

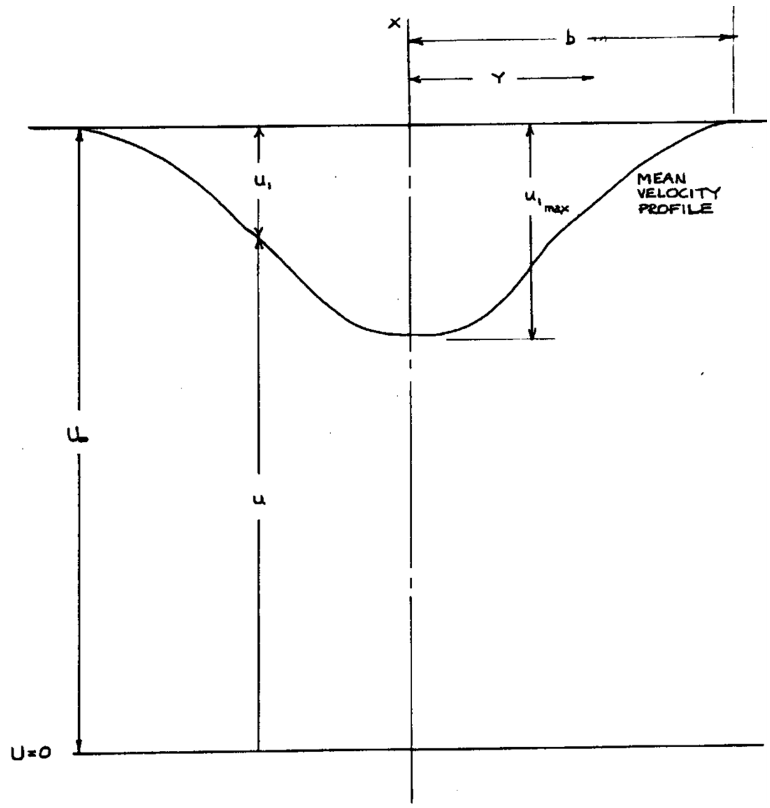


Figure 5.3: Velocity profile in a wake

5.4.4 Smooth and Rough Bodies

Figure 5.5 is a compilation of drag data. Note that the drag coefficient is based on frontal area. For a cylinder the area is the diameter times the length and for the airfoil it is the thickness times the span. Note the effect of roughness amplitude on the drag of cylinders.

5.4.5 Summary of the Wake Experiment

The cold resistance of an anemometer is found and the operating resistance is set. A calibration curve of the anemometer voltage vs. velocity is determined using a pitot-static tube. The velocity profile of the wake behind a body is then found using the anemometer.

The anemometer is driven across the wind tunnel behind the body by a stepper motor, controlled by a MATLAB program. By sampling the flow at several points in the wake, the velocity profile is generated. The data is for anemometer voltage vs. distance in the tunnel. This voltage can easily be converted to velocity by using the calibration curve obtained previously. The profile is then plotted. The wake profile and drag is found for smooth and rough cylinders and an airfoil, and the results are compared.

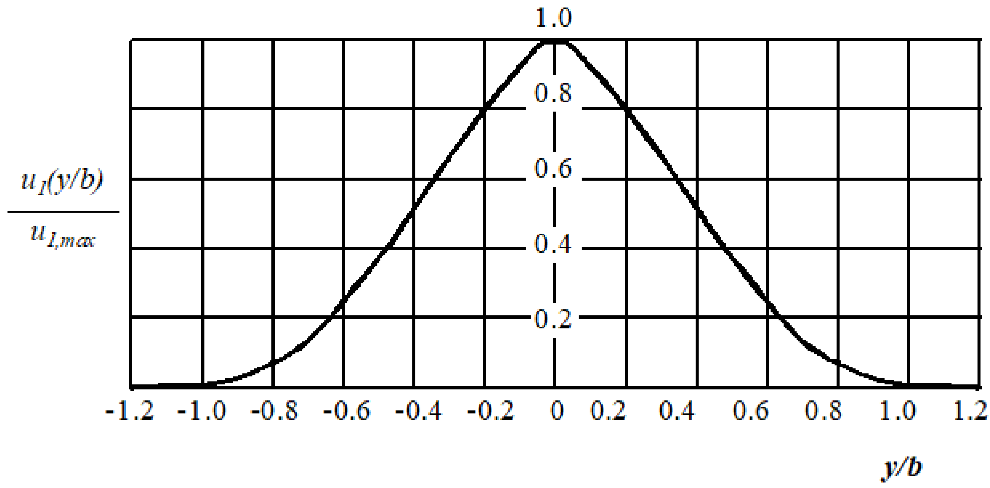


Figure 5.4: Velocity profile in a wake

5.5 Recommended Reading

It is highly recommended that students read the following sections in Fundamentals of Aerodynamics by John D. Anderson:

- 2.5 Momentum Equation
- 2.6 An Application of the Momentum Equation: Drag of a Two-Dimensional Body
- 3.18 Applied Aerodynamics: The Flow Over a Circular Cylinder - The Real Case
- 15.2 Qualitative Aspects of Viscous Flow

5.6 Data to be Acquired

1. Differential pressure from the pitot probe
2. Frequency of operation of wind tunnel
3. Voltage readings from the hot film probe

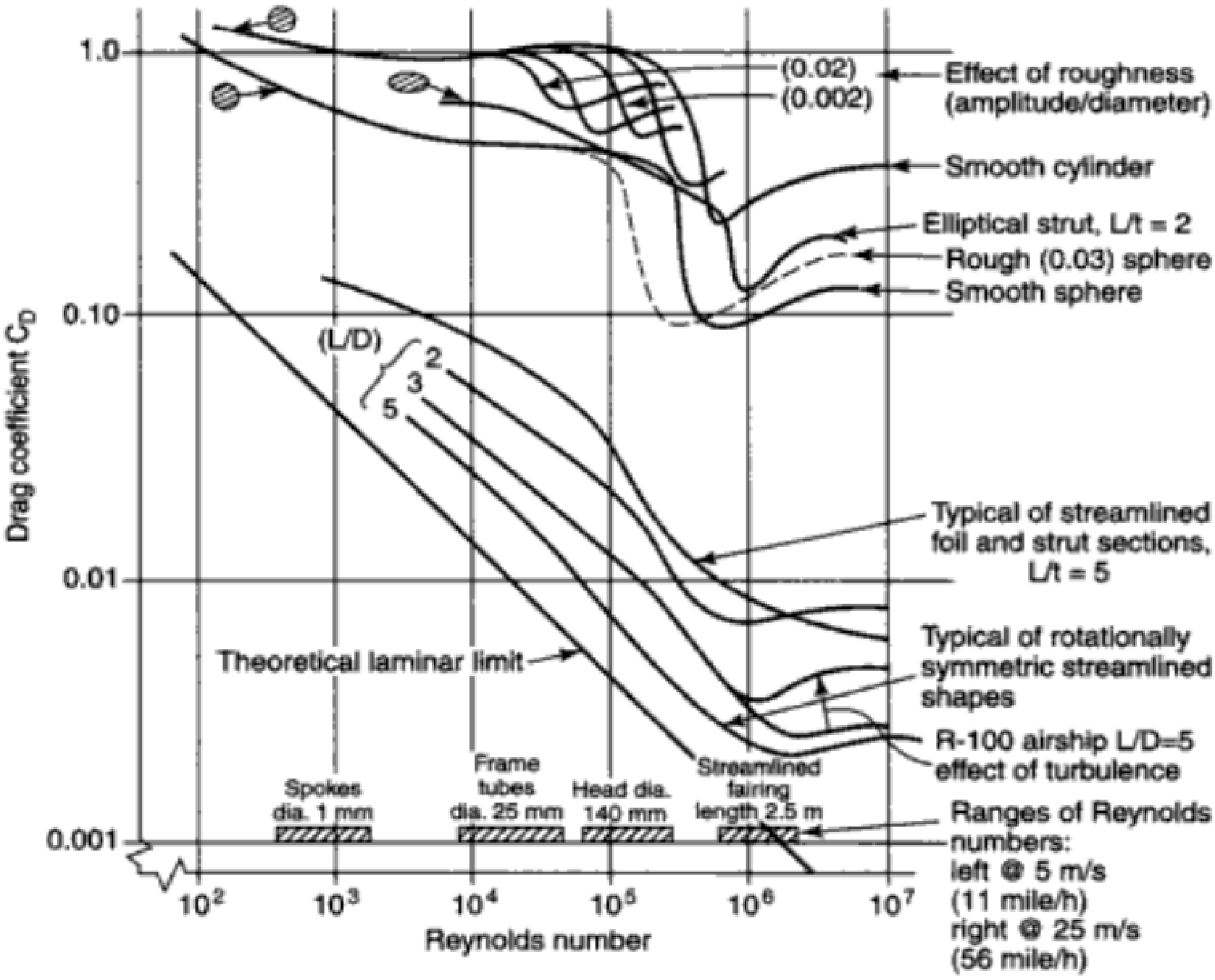


Figure 5.5: Drag of Various Bodies. *Source: "Bicycling Science" by David Wilson*

5.7 Lab Setup and Procedure

5.7.1 Experiment Setup and Preparation

Lab Location: ARMS B098, Low-Speed Wind Tunnel

A hot film anemometer will be used in this lab to measure the flow velocity across wakes behind cylinders and an airfoil. The hot film probe is mounted in the wind tunnel test section and can be moved from one wall to the other using a traverse driven by a stepper motor. The output of the hot film probe is processed by an analog to digital converter and then read using LabVIEW.

1. One member should log into the lab computer using their career account and navigate to the Lab 4 folder at the destination: `C:\ Temp \ AAE33301 \ Lab 5 - Wakes and Drag Measurement II`. Make sure you save any and all data generated during the lab in the `data_33301_lab4` folder. Make a folder with your team name (e.g. Black16) within that folder.

2. Download the Lab 5 data spreadsheet from Brightspace and open it on the lab computer. The spreadsheet will help you record data and perform calculations.

3. Open the "**Lab5_33301_Launchpad**" VI where you will see two SubVI buttons. These LabVIEW programs (called "virtual instruments" or "VIs") are:

   3.1 **MoveXDMM** - moves the probe traverse horizontally and displays the hot film probe voltage at the current location. The voltage is that needed for calibration ("DMM" stands for "digital multimeter")

   3.2 **Wake Scan** - moves the probe across the test section and writes data from the probe to a file

5.7.2 Experiment Procedure

Yellow bold highlighted text indicates that student should show the data/plot obtained to the TA present at the lab. Following is the nomenclature used for naming the cylinders and locations of placement.

Cylinder and Location Nomenclature	
Cylinder A	1.9 in Smooth Cylinder
Cylinder B	0.5 in Smooth Cylinder
Cylinder C	1.9 in Rough Cylinder
Location 1	8.5 in
Location 2	17 in

1. **Hot Film Probe Calibration**

   1.1 ☐ Confirm that the wind tunnel is empty, i.e., there are no test articles mounted. Check that the hot film probe is positioned correctly (straight vertically and not rotated relative to the flow direction). Turn on the wind tunnel lights.

   1.2 ☐ Click the Run button to start the launchpad, then click on "MoveXDMM" to open the Sub VI.

   1.3 ☐ Click the "Center" button to move the probe to the center of the tunnel.

   NOTE: Although it won't be needed in this lab, you can also move the probe to custom locations using this VI.

   1.4 ☐ Turn on the wind tunnel and set the motor frequency to 10 Hz.

   1.5 ☐ Record the differential pressure from the pitot-static probe (using the black readout on the side of the tunnel) and the hot film probe voltage (E), from the Simple DMM box on the VI in the Lab 4 data spreadsheet.

   1.6 ☐ Repeat the differential pressure and probe voltage measurements for motor frequencies of 15, 20, 25, and 30 Hz. Record all data in the spreadsheet.

   1.7 ☐ Calculate the freestream velocity U_∞ from each differential pressure measurement in the appropriate column in the data spreadsheet.

   1.8 ☐ Obtain a calibration curve for velocity vs. probe voltage by fitting a polynomial trendline in Excel, i.e.,

   $$u = C_0 + C_1 E + C_2 E^2 + C_3 E^3 + C_4 E^4 + ... \tag{5.6}$$

   Click here if you need a reminder about how to fit a trendline.

   NOTE: The curve-fit fails at low velocities where buoyancy effects become important. Thus, zero-velocity data should not be used in the calibration!

   1.9 ☐ **Show the calibration plot to the TA before proceeding to be sure that the probe is operating properly.** .

2. **Wake Surveys**

   2.1 ☐ Install Cylinder A in Location 1 in the test section. The diameter of Cylinder A is 1.9 inches and Location 1 is 8.5 inches upstream of the hot film probe.

   2.2 ☐ Click the "HOME" button in the MoveXDMM VI to move the hot film probe all the way to the left towards the wall. Verify the location visually: confirm that the probe aligns with the "HOME" marker in the tunnel as in Figure 5.6. *If not, inform your TA.*

   2.3 ☐ On the Simple DMM box, click the Stop button and close the MoveXDMM window.

   2.4 ☐ Open the Wake Scan VI from the Launchpad.

   2.5 ☐ Set the file path in the Wake Scan VI to your data folder. After the path, add a backslash and enter a name for the data file (in .txt format). **Make sure you change the file name each time you start a new experiment otherwise it will overwrite the data.**

   2.6 ☐ Turn on the tunnel and set the motor frequency to 10 Hz.

Figure 5.6: Desired hot film probe location before the wake survey (probe must be at "HOME" location).

2.7 ☐ Run the Wake Scan VI. The probe will move across the wind tunnel in 0.1 inch increments and take 1,000 voltage measurements over 1 second (i.e., data acquisition rate of 1 kHz) at each location. The VI will calculate the mean voltage and standard deviation at each location and write it to the data file.

2.8 ☐ After the wake scan is completed, turn off the wind tunnel and open the data file. There should be three columns of data – the first is the probe location (in inches), the second is the mean voltage at each location in the wake (in volts) and the third is the standard deviation of the voltage (in volts). Copy the mean voltage into the appropriate location in the lab data spreadsheet.

2.9 ☐ In the "Velocity" column in the data spreadsheet, calculate the velocity from the voltage measurements using your calibration equation.

2.10 ☐ In the data spreadsheet **calculate and plot the wake velocity vs. probe position. Show the plot to the TA before proceeding.**

NOTE: Due to wall effects, you will have regions of the wake near the walls where the voltage measurement will be higher than the voltage you measured for the freestream during calibration (i.e., mean velocity $> U_\infty$). When analyzing your data, **for any location where you calculate $u > U_\infty$, set that value equal to U_∞ (or the velocity deficit equal to zero),** otherwise your drag calculation will be negatively affected.

2.11 ☐ Repeat Steps 2.6 to 2.8 for motor frequencies of 20 and 30 Hz keeping the same test article (Cylinder A) in place. This will give you wake data for three different Reynolds numbers.

2.12 ☐ **Use the wake velocity measurements to estimate the drag coefficient for Cylinder A for at least one wind tunnel frequency.**

We have discrete data for velocity vs. location, so the integration in Equation 5.3 must be done numerically. Using the trapezoidal rule to perform the numerical integration, the equation for the drag can be written:

$$D = h\rho \left[\sum_{i=0}^{n} \frac{(u_{i+1}(U_\infty - u_{i+1})) + (u_i(U_\infty - u_i))}{2} \Delta y_i \right] \tag{5.7}$$

.

Here n is the number of measurements, u_i and $U_\infty - u_i$ are the wake velocity and velocity deficit at the i^{th} location, respectively, and Δy_i is the distance between measurement locations i and $i+1$. There are columns in the data spreadsheet to guide you through the steps to calculate the drag. **Show your calculated value of the drag coefficient to the TA before proceeding. Does your value of C_D seem reasonable?**

2.13 ☐ Move Cylinder A to Location 2 (17 inches upstream of the hot film probe). Use the stopper to close the hole at Location 1. **ALWAYS TURN OFF THE WIND TUNNEL BEFORE ATTEMPTING TO REMOVE THE MODEL.**

2.14 ☐ Repeat Steps 2.6 to 2.8 for a **motor frequency of 30 Hz only**.

2.15 ☐ Remove Cylinder A and install Cylinder B in Location 2. The diameter of Cylinder B is 0.5 inches.

2.16 ☐ Repeat Steps 2.6 to 2.8 for Cylinder B for a **motor frequency of 30 Hz only**.

2.17 ☐ Move Cylinder B to Location 1. Use the stopper to close the hole at Location 2. Repeat step 2.16.

2.18 ☐ Remove Cylinder B and install Cylinder C in Location 1. Cylinder C has the same diameter as Cylinder A (1.9 inches) but has roughness elements on the surface.

2.19 ☐ Repeat Steps 2.6 to 2.8 for motor frequencies of 10, 20 and 30 Hz.

2.20 ☐ Remove Cylinder C and install the airfoil model at Location 2. Insert the pin in the small hole in the airfoil to fix it at zero angle of attack.

2.21 ☐ Repeat Steps 2.6 to 2.8 to measure the airfoil wake for a **motor frequency of 30 Hz only**.

2.22 ☐ **Return the experiment and the lab to its state when you first arrived.** Return all tools and equipment back to their designated location, turn off any equipment that was started for the purpose of your lab, and clean up your workspace. **Obtain approval from the TA before leaving the lab.** Thank you!

You will use the information recorded in the Lab 5 data spreadsheet to complete the post-lab assignment. Remember that the post-lab should be completed **individually** and uploaded in Gradescope within 1 week of your lab session.

Chapter 6

Design your Own Experiment

6.1 Introduction and Lab Logistics

Lab 6 involves several steps outlined below (chronologically). Important dates are listed on the course Brightspace Calendar and in the syllabus. Highlighted parts of the text have relevant content and/or links in Brightspace, navigate to "Content > Lab 6 - Design Your Own Experiment".

1. ☐ **Proposal Development:** As a team, develop a proposal for your Lab 6 experiment using the template provided in Brightspace ("Lab 6 Proposal Template"). Proposals that do not follow the template will not be considered for evaluation. Based on the feedback you receive, you will be required to submit a revised proposal (please see the grading rubrics in Section 6.2). Student teams are required to submit a draft proposal and a revised proposal (after receiving feedback).

   Remember that a proposal is different than a standard lab assignment - the primary goal of a proposal is to convince the reader to accept and "buy in" to your ideas. Proposals are written frequently in industry and academia, for example, to help organizations obtain funds to operate. Stakes are often high in those instances, as an accepted proposal can bring millions of dollars into the organization. Always proofread and perform appropriate grammar checks before submitting a proposal.

2. ☐ **Proposal Submission:** Submit the proposal in pdf format as a group in Gradescope. Instructions for submitting a group assignment can be found here: Group Submission on Gradescope. You must also propose **3 potential times over the weeks before the final deadline** for completing your lab when submitting this proposal. Please base your proposed times on your team members' availability and a discussion with your TA. View the current schedule of various teams/experimental rigs on Brightspace ("Lab 6 Schedule"). Refer to "Schedule Testing time"(7) for more information.

3. ☐ **Preliminary Proposal Review:** Your TA will review your proposal and provide feedback as annotations in Gradescope within 2 to 3 days.

4. ☐ **Instructor Feedback:** The course instructor will briefly meet with each team to discuss their proposal and provide additional feedback.

   - Due to the large number of teams, meetings will be limited to 10 minutes.
   - At least 2 members of the team should be available to attend.
   - Schedule a meeting time (only one member needs to complete the online form, **sign up using the link in Brightspace**).
   - The goal is that by the end of this meeting, you will have a concrete plan for completing your project.

5. ☐ **Final Proposal submission** After the first round of review, your team will be needed to resubmit your proposal after incorporating the feedback from the instructor and your TA. Upload the revised proposal on gradescope.

6. ☐ **Fabrication/acquisition of test models:** Based on your proposal, you may need a physical model that can be tested in one of our facilities. The available test section dimensions for each facility are given in the table 6.1.

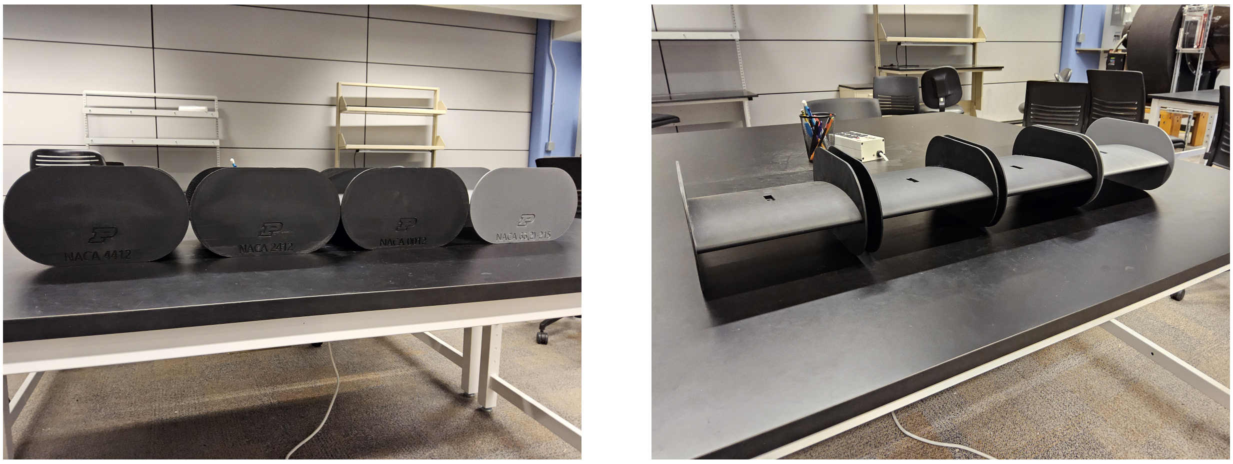
Facility	Location	Test Section Dimensions		
		Length (in)	Breadth (in)	Height (in)
Low-Speed Wind Tunnel	ARMS B098	14	14	14
Water Table	ARMS B098	8	10.5	6
Supersonic Tunnel	ARMS B098	1.625	0.875	1.25
Boeing Wind Tunnel	AERO	72	72	48

Table 6.1: Test Section Dimensions of various testing facilities

There are several options for acquiring or fabricating your test models:

6.1 *Ready-made models*: Use an off-the-shelf model for your test, e.g. a toy car, an action figure, sports balls, etc. We also have 3D printed airfoils compatible with the Armstrong subsonic wind tunnel, for the following NACA configurations, some of which have commonly been used in AAE 33300 lecture course: NACA0012, NACA2412, NACA2415, NACA4412 and NACA66(2)-215 (see figure 6.2. We also have a collection of models used by teams in the past. They are situated near the supersonic tunnel area that you are free to use (see figure 6.1). If you want to test a model that is not already available, we can purchase it for you (refer to section "Purchasing")

Figure 6.1: Models from previous semesters that are available to use



(a) Various NACA airfoils with endplate

(b) Isometric view of the airfoils with mounting holes

Figure 6.2: 3D printed NACA airfoil available for Lab 6 testing

6.2 *3D printing:* You may choose to 3D print a model for testing. Purdue has several options available for 3D printing:

- 3D print at Purdue AAE (preferred): Please read more about Purdue AAE 3D printing in section 6.3.1. Contact **Adarsh Agrawal** (agraw156@purdue.edu) for assistance in using the Purdue AAE 3D printers. Filament will be provided.
- 3D print at WALC (and others): For other printing options, see: `https://guides.lib.purdue.edu/3dprinting/Home`. Note that some of these options have a weight limit per print. We cannot

provide filament for any print done outside Purdue AAE.

- 3D print on your own: You can print your 3D model outside of Purdue. Again, we cannot provide filament for any prints done outside Purdue AAE.

6.3 *Foam cutting:* If you want to test airfoil/wing models, you can make them using the CNC foam cutter in the Design, Build, Test (DBT) lab (ARMS 2098). Please contact **Tom Bietsch** (tbietsch@purdue.edu) for scheduling a foam cutting session. Read the Foam Cutting Guidelines in section 6.3.2 before scheduling. Foam material will be provided.

6.4 *Machining:* In rare circumstances, you may need to have your model machined. Note that the lead time is high for machining so this option should be considered only when absolutely necessary. Bechtel Innovation Design center offers machining facilities, and AAE's AERO (Aerospace Sciences Lab) has machining services available. Please contact **Rob Hughes** (rrhughes@purdue.edu), AND Dr. **Sally Bane** (sbane@purdue.edu) if you think your experiment requires machining.

6.5 *Something else:* At Purdue AAE, the sky is not the limit. You may think outside the box and create something entirely different which may or may not include any of the above. Reach out to your TA if you want feedback on your ideas.

- **Purchasing**

We can purchase models or materials for your team, place request using the link on Brightspace "Lab 6 Purchase Request".

Purchase requests can include ready-made models, raw material (e.g., plywood, PVC pipe, etc.), specific fasteners/hardware, etc. There is no set budget for each team, but **please keep costs below $30 if possible.**

- **Mounting Considerations**

For almost anything that is being tested in the tunnels, mounting the models and understanding clever ways to do it need to be explored. While designing your model(s), you should also plan on how to mount the model in the experimental facility. Useful information is provided in the Mounting Guidelines section. If you have questions about mounting your model or need help with simple drilling, cutting, soldering, etc., contact the AAE Lab Technician **Rob Hughes** rrhughes@purdue.edu.

- **Emailing for help**

All correspondence sent out to any staff/faculty regarding Lab 6 must have the following subject line: "Lab 6 AAE33X01 Team YZ Color". X refers to the course code (3 for AAE33301, 4 for AAE33401), "YZ" is your team number, and "Color" is your team color (Black or Gold).

All requests for help with fabricating and/or mounting models, purchases, and other appointments with staff must be submitted by a specific date listed in the Brightspace calendar.

7. **Schedule testing time:** Contact your lab TA to schedule time to perform your experiments in the desired facility. Slots may become limited over time, so make sure you reserve your time as early as possible.

7.1 If necessary, your TA will schedule a virtual meeting with your team to discuss and finalize details for completing your project.

7.2 You will need to find times that work for your team, your TA, and during which the equipment you need is available.

7.3 Typically, teams like to perform their experiments during their usual lab time, but do not assume that this will be possible since other teams might reserve the equipment first.

7.4 The schedule for the subsonic wind tunnel in ARMS fills up quickly, so please schedule early!

7.5 You can access the "Read-only" lab schedule for various equipment for lab 6 on Brightspace. **Only your TA can schedule your lab 6 time for you.**

7.6 Fill out the Qualtrics survey "Lab 6 Slot Requester" with 3 potential times and the equipment you intend to use.

8. **Prepare and submit your Lab 6 report:** Prepare a complete technical report as a team and submit it in Gradescope within 1 week of completing your testing in the lab. Refer to the "Guidelines on Writing Lab Report" on Brightspace. Please confirm the due date with your TA.

9. **Complete the peer evaluation:** You must complete the online peer evaluation within 2 days of submitting the final report. You will receive an email from CATME regarding this. Please login to CATME.org tool, at `https://catme.org/login/index` to get started.

6.2 Rubrics

Lab 6 will be graded on the following components:

1. Draft Proposal: 5 points
2. Final Proposal: 20 points
3. Report: 100 points
4. Peer evaluation: 40 points

Total Lab 6 score: 165 points, 25% of your final grade

The point distribution for the proposal and report can be viewed in their respective templates that are uploaded on Brightspace.

6.3 Additional Services Information

6.3.1 3D printing at Purdue AAE

Point of Contact: Adarsh Agrawal (agraw156@purdue.edu)

3D printing has become an extremely common methodology for prototyping/manufacturing quickly, efficiently, and accurately. Although you can 3D print using a variety of materials, Polylactic acid (PLA) is widely used in plastic filament in fused filament fabrication (FFF) technology of 3D printing.

We have the following 3D printers available:

1. Bambu A1 Mini: Print Volume: 180 × 180 × 180 mm.
2. Bambu P1S: Print Volume: 256 × 256 × 256 mm.
3. Bambu X1 Carbon: Print Volume: is 256 × 256 × 256 mm.

Make sure that the maximum dimension of your model is within the volume limits listed above. We will use PLA for printing in these printers. There is no limitation on filament use on these printers and it will be provided by us. If you are building your own model, we would need an STL file which will then be programmed in the printers. If you are unsure about your design and would like assistance on making design modifications, please send the original part file to the point of contact. Read about the mounting guidelines in section 6.3.3, to incorporate the mounting holes within your model (drilling into 3D printed model is not advised).

Note that 3D printing is not straightforward. Unlike the popular notions like "anything can be 3D printed" or "you can probably print a 3D printer using another 3D printer," 3D printing requires some foresight. Since the FFF technology depends on depositing layers on top of each other to create a part, one needs to make sure that the design has fewer overhangs. If there are any, proper supports shall be provided to accommodate it.
Please email your design to the point of contact to get started.

Note: If you end up printing the models yourself, make sure you provide at least two to three walls near your hole to make sure that we can tap the hole

6.3.2 Foam Cutting Guidelines

Point of Contact: Tom Bietsch (tbietsch@purdue.edu)

The AAE Design-Build-Test Lab (ARMS 2098) has a CNC foam cutting machine that can be programmed to cut custom profiles. If you want to use the machine to make airfoil/wing models for testing, gather the following information before reaching out to the point of contact:

1. Airfoil profile you wish to test.
2. A .dat file containing the airfoil profile coordinates, which can be found online at www.airfoiltools.com. Save the .dat file to an external USB drive to transfer it to the foam cutter computer.
3. Have an idea about the chord length, span, and the maximum thickness of the airfoil you wish to create. Note that since the foam sheets are 2 inches thick, the maximum thickness must be 1.75 inches or less.
4. If you are planning to test your airfoil/wing in the ARMS Low-Speed Wind Tunnel, please note these additional design considerations:
   - The chord length of your model should not exceed 11.75 inches if you wish to test it in the Armstrong Wind Tunnel.
   - The span of the model can be no more than 12 inches.
   - For airfoil models, you should attach end plates to minimize the wing tip effects. You can cut these out of cardboard and glue them onto the foam.

6.3.3 Mounting Guidelines

Point of Contact: **Rob Hughes** (rrhughes@purdue.edu)

6.3.3.1 Low Speed Wind Tunnel at Armstrong

The "adjustable angle of attack" test stand has several options for mounting your model. The stand has two separate mount heads:

1. One with fixed studs (DO NOT take apart the mount head with fixed studs) which can facilitate material thickness up to 7/8" (22.3mm). This mount head is being referred to as Main head.
2. The other mount head has screw holes for material thicknesses from 1" (25.4mm) and above. This mount head is being referred to as Auxiliary head. Refer to the figure below.

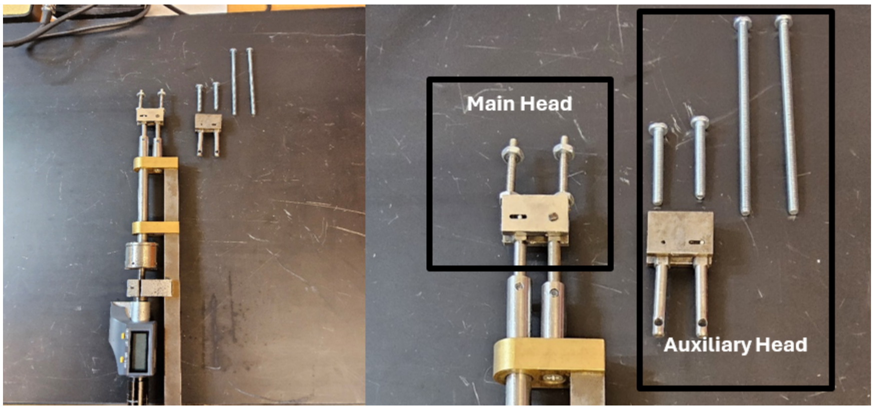


Figure 6.3: Main head and auxiliary mount head for mounting models which need change in Angle of Attack

The two heads are interchangeable on the test stand using two hex key screws located on the side of the linear shafts. If your model will not mount using the studs on the main head then remove the two hex key screws and change the head. Again, **do NOT try to remove the studs from the main head**. Refer to the figure below for more information.

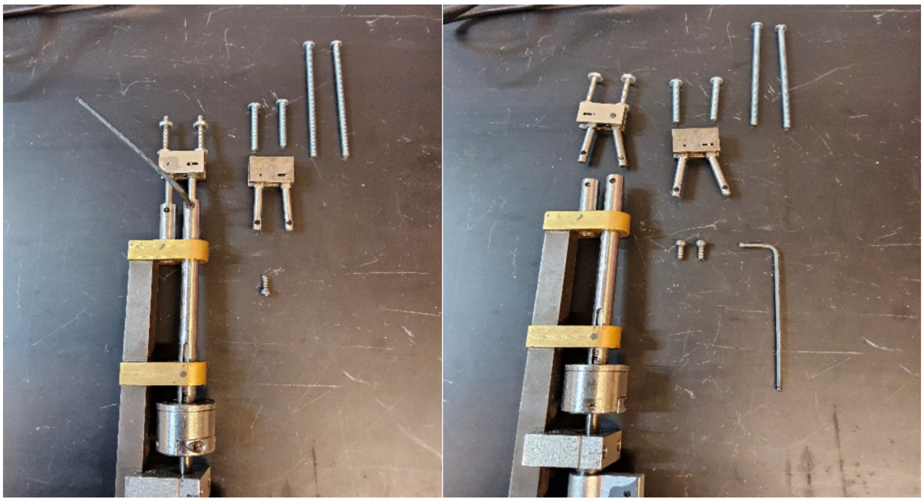


Figure 6.4: Detaching studs from airfoil stand

Both mount heads have a 0.82-inch (20.828mm) center to center distance between fasteners and are orientated parallel to the air flow of the tunnel. The Diameter of the studs on the main head is 0.142 inch (3.62mm) and a 9/64 drill bit works perfectly. The diameter of the screws for the auxiliary head is 0.171 (4.34mm) and a 11/64 drill bit works for this. Refer to the figure below.

Note for 3D printing: Please print the holes of diameter 0.145 in to take into account the contraction caused by 3D printing when using the fixed stud (0.142 in diameter stud), and print holes of diameter 0.174 in when using the detachable stud (0.171 in diameter stud).

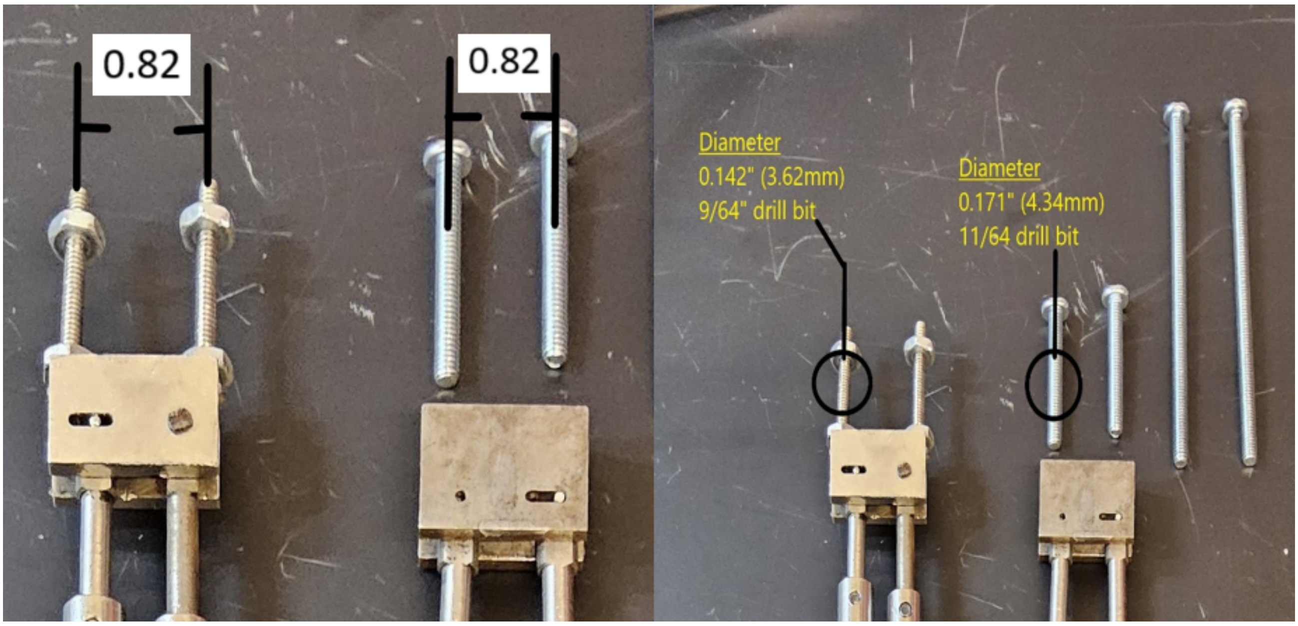


Figure 6.5: Dimensions related to mounting head

Important notes

1. Do not try to remove the studs from the main mount head.
2. Do not modify mount stands/heads in any way.
3. Do not disassemble any test/mount stands from other lab projects.

If your model requires any custom mounting considerations outside of these guidelines or if you would like any help in drilling/mounting your models contact Rob Hughes (refer to the email given at the beginning of this section for contacting him).

6.3.3.2 Supersonic Wind Tunnel

There are two model mount blocks available for use in the supersonic wind tunnel. Each has a small threaded stud to screw a model onto. One has an 8-32 threaded stud and the other has a 10-24 threaded stud. Refer to the figure below. If you want to make a model to mount on either of these mounts, include an untapped hole in your model. **For 8-32 stud, use diameter 0.137 in, for 10-24 stud use 0.15 in.**. The depth of the hole needs to be at least 0.5 in. We will use these mounts when the size of the model is around 1 in.

If the size of the model exceeds 1 in, we will have to mount it from the top.

In some cases, a mount block can be custom made to facilitate a special mounting circumstance; in this case, you must have the approval of your Professor and AAE Lab Tech. It is not advisable to put 3D printed models in the supersonic tunnels. We can help develop a machined model for testing purposes. If you want to propose a project in the supersonic tunnel, please contact Prof. Bane and Rob Hughes as early as possible to discuss feasibility.

Important notes

1. Special care should be taken not to overtighten models onto mount blocks.
2. Do not disassemble any models/blocks from other lab projects.

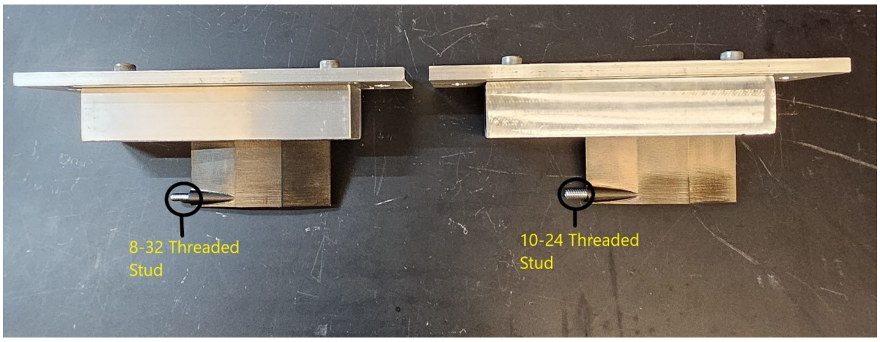


Figure 6.6: Mounting mechanisms for supersonic tunnel

3. Do not modify mount blocks or model mounts in any way.

6.3.3.3 Water Table and Boeing Wind Tunnel

Currently, no specific guidelines exist for model mounting in the water table or in the Boeing wind tunnel. All mountings for these locations are considered custom and will have to be approved by your Professor and the AAE Lab Tech.

Chapter 7

Appendix: Manuals

For the sake of keeping lab background and procedure documents succinct and precise, additional information pertaining to various parts of the lab has been added here. You will cross-reference to various sections of this chapter, based on what experiment you are conducting in the lab.

7.1 Armstrong Low-Speed Wind Tunnel

The Wind Tunnel in Neil Armstrong Hall of Engineering (ARMS) is located in Room B098 in the basement level of the building. The wind tunnel is designed and constructed by Dr. John P. Sullivan who is Emeritus Professor at the School of Aeronautics and Astronautics.

The wind tunnel has two test sections, with cross sectional dimension of 1 ft by 1.5 ft (12 in by 18 in or 3.66 m by 5.49 m). It can run at a max speed of around 40 m/s. The contracting intake portion and part of test section of the tunnel is shown in figure 7.1

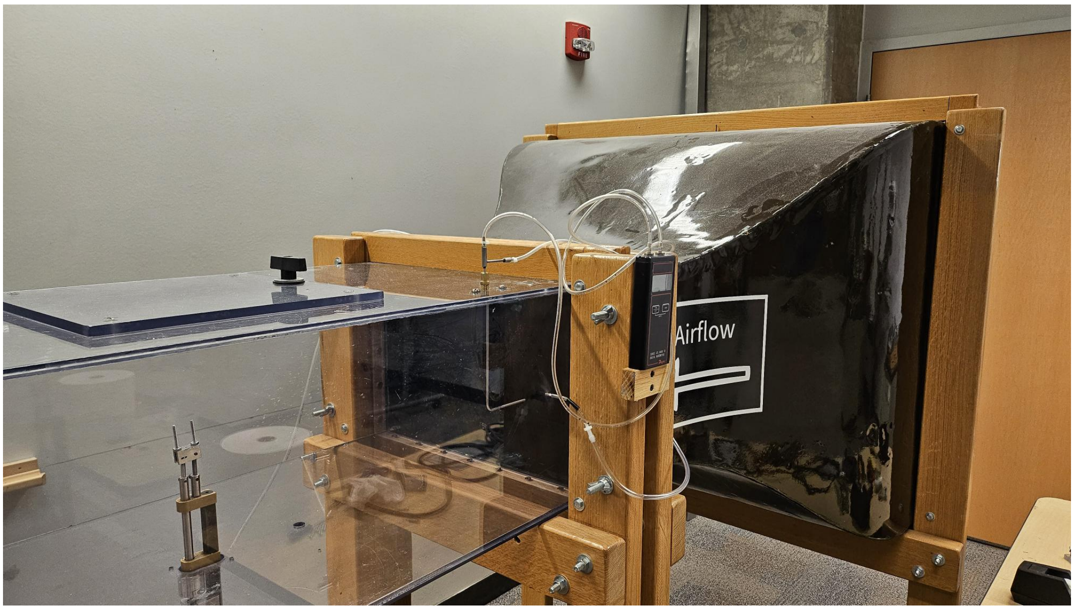


Figure 7.1: Armstrong Wind Tunnel at ARMS

Operation

1. Make sure Power Breaker (figure 7.2) is in "ON" position (red handle up).
2. Press the green “on” button (with vertical white line) on Allen Bradley controller (7.3)
3. Adjust speed (motor Hz) using up and down arrow buttons on the panel.
4. Press the red "off" (with white circle) button to turn off the wind tunnel.

Figure 7.2: Breaker for power control to wind tunnel

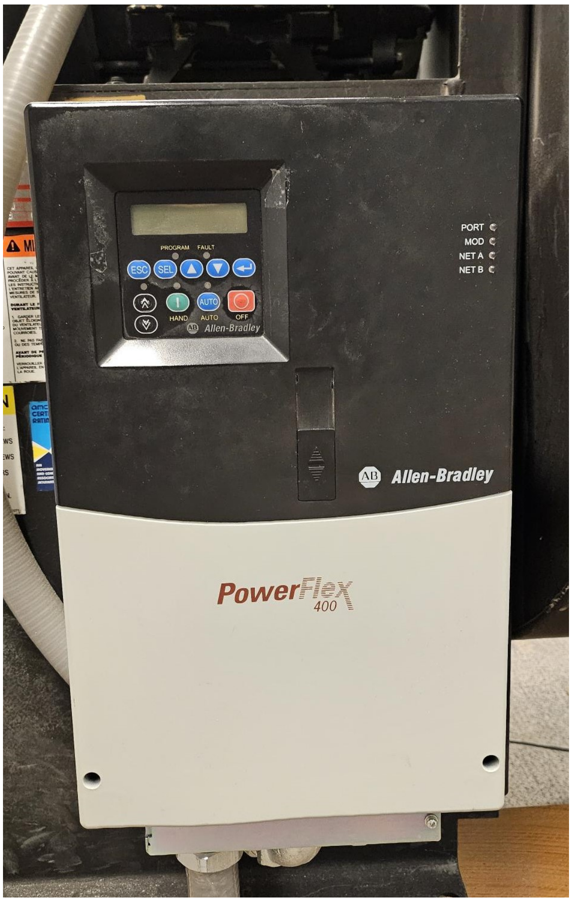


Figure 7.3: Allen Bradley Controller for controlling the variable frequency drive of the wind tunnel for speed control

7.1.1 Force Balance Calibration

7.1.1.1 Platform Balance

The platform balance is used to measure the drag, lift and pitching moment produced on an airfoil in the wind tunnel. It consists of four FUTEK Force Transducers. Three of the transducers are located under the top balance plate and measure the lift and pitching moment. These are labeled F1, F2, and F3. The fourth transducer is located inside the parallelogram balance measuring the drag and is labeled F0. The transducers are all connected to the National Instruments Channel Amplifiers. A schematic of the set up is shown below in

Figure 7.4.

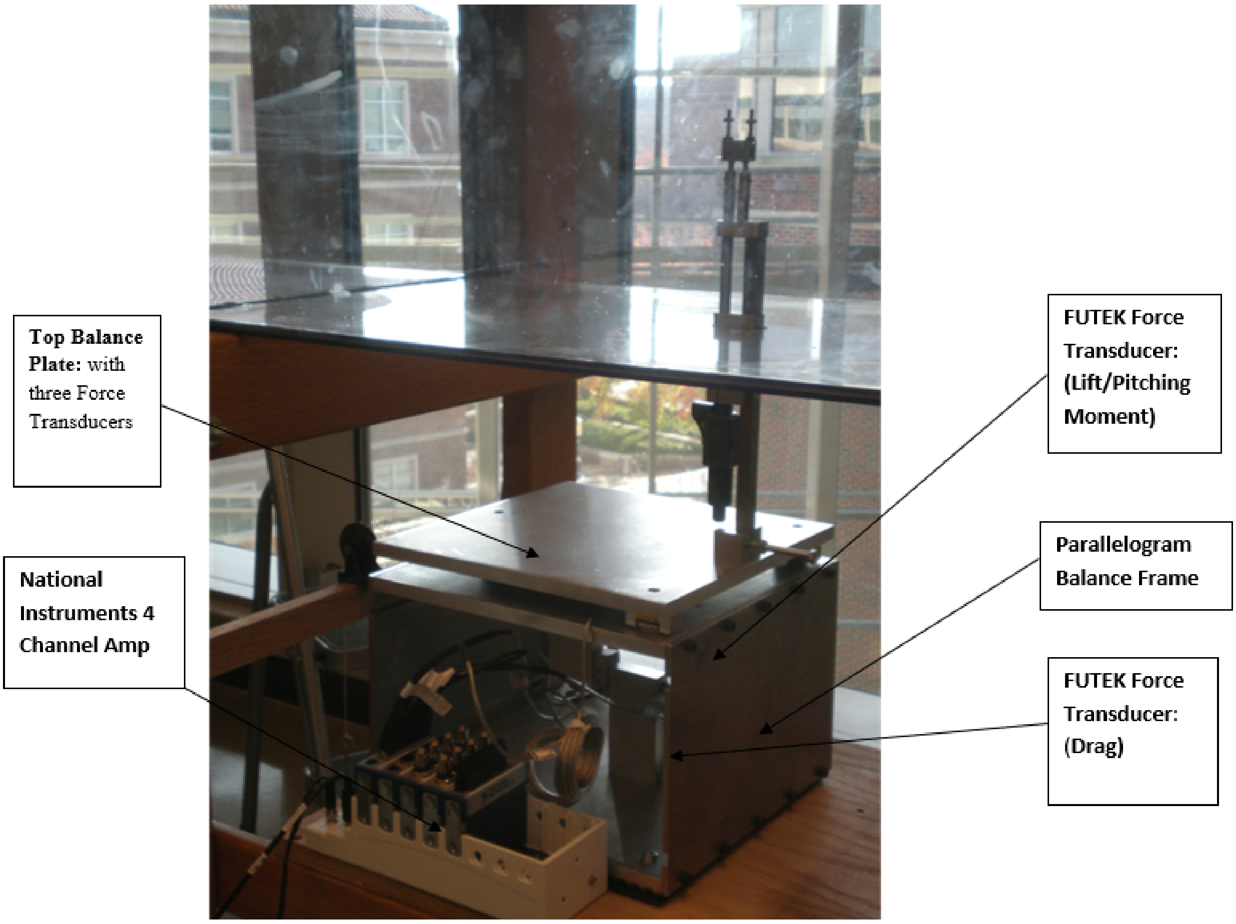


Figure 7.4: Force Balance Set-up

The balance works by taking the readings from the different transducers and manipulating them with the proper equations to output the lift, drag and pitching moment. To calibrate the four transducers on the platform balance, it is necessary to apply a known load at two different locations on the balance to measure the lift and drag.

- The Drag is measured by the F0 transducer reading inside the parallelogram. It measures the force induced between the top plate and the bottom plate. The drag force is given by equation 7.1.

$$D = F0; \tag{7.1}$$

- The Lift is measured by the F1, F2, and F3 transducers located beneath the top plate. When the airfoil is placed in the wind tunnel, the lift force pulls up on the balance. This Lift Force is the sum of the transducers. The force is given by the equation 7.2.

$$L = F1 + F2 + F3; \tag{7.2}$$

7.1.2 General process of calibration of force balance

This force balance needs to calibrated to get appropriate lift and drag readings. For calibrating lift, known weights should be placed on the **top balance plate** and for drag, put the weights **on the pulley platform** located just behind the F2 transducer. The procedure is as follows:

1. Make sure the tunnel is not running, contains no test specimen in the test section and that there are no weights placed on the force balance.
2. Run the WindTunnelBalance VI. When you run the program, it zeroes out all the forces in the sensors.
3. Record the initial values of lift/drag, against zero actual weight (when no weights are placed on the balance) from the VI.

4. Apply known weight(s) of 0, 0.5, 1, 2, 3, 4 and 5 lbs sequentially by stacking it based on which force (lift or drag) is being calibrated. Let the balance even out (also record the weight given by LabVIEW when no weights are placed). Place half pound and then the subsequent weights. Note that the curve is quite linear for most part, but remember that calibration is imperative to get accurate data.
5. Record the values of lift/drag from the program agasint the actual weight.
6. Plot "Actual Weight for Lift or Drag" vs "LabVIEW Weight for Lift or Drag".

7.1.3 Angle of Attack Adjustment

Although we do not change the angle of attack in any of our experiments in AAE 33301, you might make use of this apparatus as part of your lab 6 (design your own experiment). The angle of attack of the airfoil is changed using a micrometer which is mounted on a metal stand, which in turn is screwed to the force balance platform. Figure 7.5 shows the various parts of the test stand along with the micrometer.

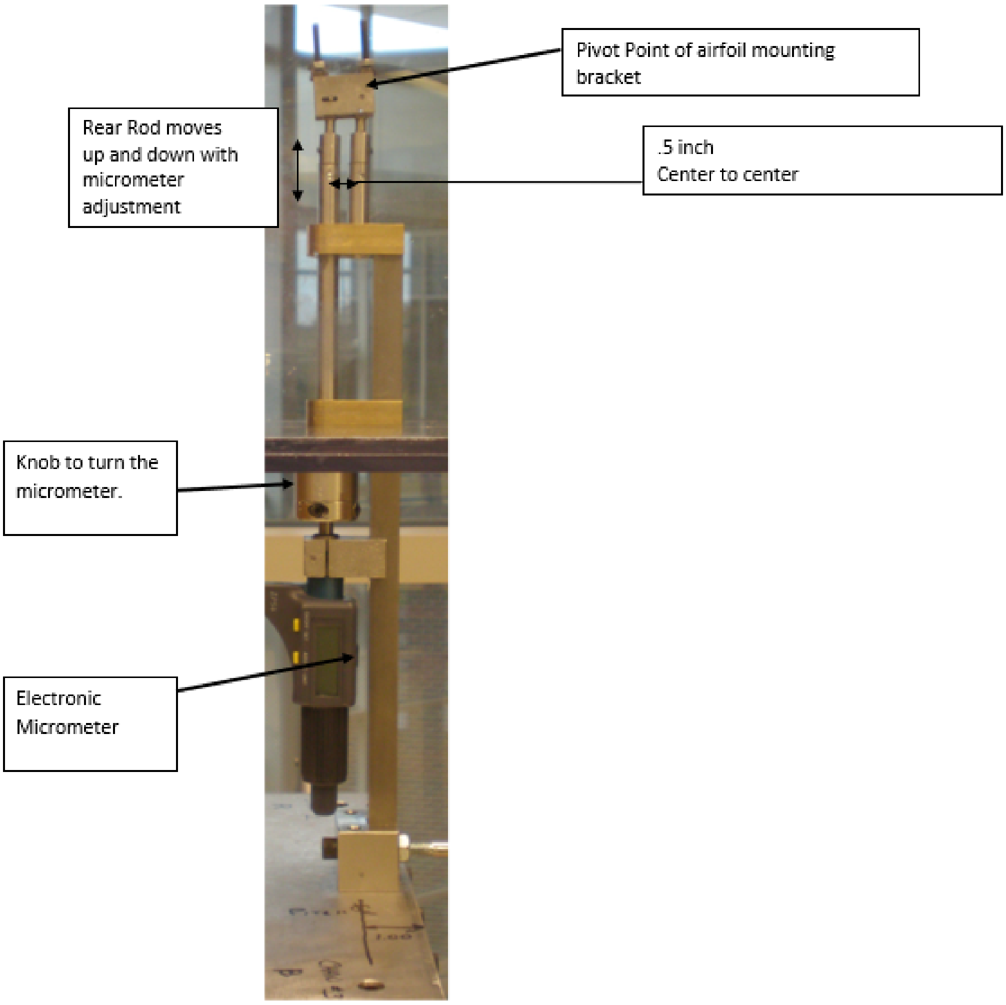


Figure 7.5: Anatomy of the airfoil test stand

Follow these steps after mounting your stand (with the micrometer) onto the force balance:

1. Mount the airfoil model on the stand using nuts and appropriately sized wrenches.
2. Use a level to zero out the angle of attack of the airfoil. The level must be set on top of the two mounting screws. Turn the knob to manipulate the angle of attack.
3. Once the angle of attack is zero (the level shows that it is perfectly horizontal), set micrometer value to zero by pressing ABS/INC button. This will "tare" the micrometer.
4. Use trigonometry to calculate the required micrometer value for a given angle of attack. Equation 7.3 can be used to find micrometer value in inches (h) corresponding to desired angle of attack (α).

$$\boxed{h = 0.5 \times tan\alpha} \tag{7.3}$$

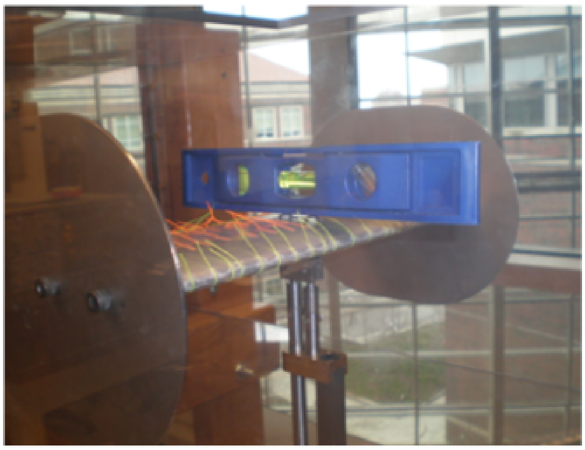


Figure 7.6: Level for Zeroing Angle of Attack

7.1.4 Micrometer Datasheet

The micrometer mounted on the airfoil stand is modified to affect the angle of attack as the knob is rotated. Refer to the datasheet (7.7) of the instrument.

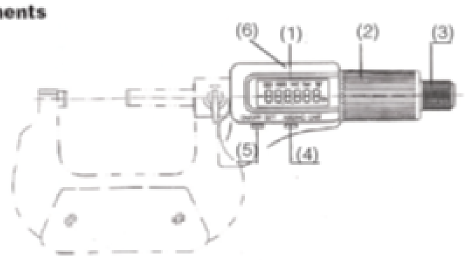
Fowler

Electronic Micrometer

Part Number: 54-850/860 series

Operation Manual

1. Functional elements

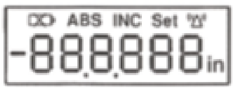


(1): LCD display
(2): Friction drive
(3): Quick drive
(4): ABS/INC•••UNIT key
(5): ON/OFF•••SET key
(6): Data output

2. Keys

ON/OFF•••SET key: Power switch. Datum set.
ABS/INC•••UNIT key: Absolute & relative measuring. Metric/Inch conversion.

3. LCD Display



"ABS" : Absolute measuring mode.
"INC" : Relative measuring mode.
"SET" : Datum set.
"in" : Unit: inch, otherwise mm.
: Battery voltage low.
: Data output is transmitting.

4. Operation

Two ways of pressing the keys are used in the following illustrations:
(1) Press and release; (2) Press and hold (2 sec. or more).

4.1 ON/OFF•••SET key:

Press and release: Power on/off.
Press and hold (2 sec. or more): Datum setting for absolute measurement; "Set" sign displayed on LCD.
Origin of metric is 0, 25, 50, 75 ••• 275mm. Origin of inch is 0, 1", 2", 3" ••• 11".
Sets datum automatically after battery reset.

4.2 ABS/INC•••UNIT key:

Press and release: Absolute and relative measuring mode conversion; "INC" sign displayed on LCD in relative measuring mode. "ABS" sign displayed on LCD in absolute measuring mode.
Press and hold (2 sec. or more): Metric/Inch conversion; "in" sign displayed on LCD for inch, otherwise mm.

5. Power

- A silver oxide cell SR44 is inserted into the back of the instrument with the positive pole outward . Replace the battery when the display data is dim or the " " sign is displayed in the upper left of the LCD.
- If not used for approximately five minutes, the power will auto-off. The micrometer will turn on by pressing "ON/OFF•••SET" key or turning the spindle. Power off the micrometer by pressing "ON/OFF•••SET" key to save the battery.

6. Specifications

Measuring force: 5 ~ 10N Power consumption: ≤ 20 μ A
Operating temperature: 0 ~ 40°C Storage temperature: -20 ~ 60°C

7. Data Output

1200 Baud, no parity, 7 data bits, 2 stop bits, no flow control.

8. Precautions

- Do not subject the instrument to blows or knocks. Do not drop it or apply excessive force.
- Do not disassemble the instrument.
- Do not press the keys with a pointed object.
- Do not use or store the instrument under direct sunlight, or in an excessively hot or cold area.
- Do not use the instrument near strong magnetic fields and high voltages.
- Use a soft cloth or a cotton swab that is dry to clean the instrument. Do not use organic solvent such as acetone or benzene. Alcohol may be used.
- Wipe the measuring faces of the instrument before using it.
- Remove the battery if the instrument is not used for a long period of time.

9. Troubleshooting

Failure	Causes	Repairing The Failure
Display "E 1" on LCD.	Data overflow.	Move spindle in reverse or press "ON/OFF•••SET" key.
Display "E 3" on LCD.	1. Sensor overflow. 2. Something wrong with sensor.	1. Reset battery (remove for 3 minutes). 2. Return the micrometer for repair.
Measuring data is not correct.	1. Dirty measuring surfaces. 2. Preset data is not correct.	1. Clean measuring surfaces. 2. Inspect preset data and reset it.
No display on LCD.	1. Battery voltage under 1.45v. 2. Battery is not properly set.	1. Replace battery. 2. Reset battery (remove for 3 minutes).
Display confused or remains blank. 1. Display blurred. 2. The output data is wrong.	Battery voltage under 1.45v.	Replace battery. Replace battery.

Fred V. Fowler Co., Inc. • 66 Rowe Street • Newton, MA 02466
617-332-7004 • 617-332-4137 (fax) • Internet: www.fvfowler.com

Figure 7.7: Datasheet for Electronic Micrometer

7.2 Water Table

7.2.1 Introduction and Operation

A water tunnel is an experimental facility used for analyzing the hydrodynamics behavior of submerged bodies in flowing water. Water tunnels are ideally suited to flow-visualisation studies, giving an insight into the physics of the flow.

Operation of Armstrong Water Tunnel

The flow in the water tunnel is controlled by a water pump, shown in Figure 7.8. The control module is depicted in Figure 7.9 Steps to start an experiment in the water tunnel:

1. Before starting the experiment, make sure the tunnel is filled to the "Fill line" on the side of the tunnel. If not, use the water hose, and fill the tunnel to the appropriate level.
2. You should be able to see floor of the water table. If the water is murky, drain the table and refill it.
3. Motor should be set to "FWD" so that the shaft rotates clockwise, the grey button to toggle. Start the motor to initiate the water flow across the tunnel. Push the GREEN button to turn the motor on. LED display will be illuminated when the power is supplied to the motor controller. The display shows the frequency at which the motor is operating.
4. Adjust the speed with the knob (do not adjust using the blue buttons). The knob will change the motor frequency and hence the speed of the water flow.
5. Push RED button to turn motor off.

Figure 7.8: Water pump used to create water flow in Armstrong Water Tunnel

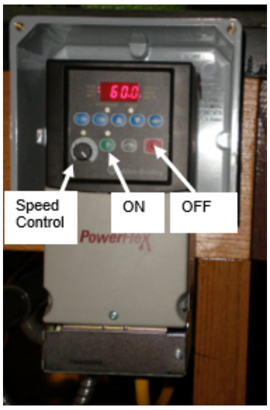


Figure 7.9: Control Module for Water Pump

7.2.2 Hydrogen Bubble Wire Setup and Operation

7.2.2.1 Introduction

In order to visualize the flow field around the specimen, we perform electrolysis in water to produce Hydrogen bubbles. The bubbles travel from the bubble wire to the electrode, via water. In between the bubble wire and the opposite electrode, the specimen is kept which allows the bubbles to flow around the specimen, thus showing the flow pattern. This process is illustrated in Figure 7.10. An example of how a pulsed line of hydrogen bubbles look like is depicted in Figure 7.11

Various metals can be used for making the bubble wire. The wire being used for our lab is made of platinum.

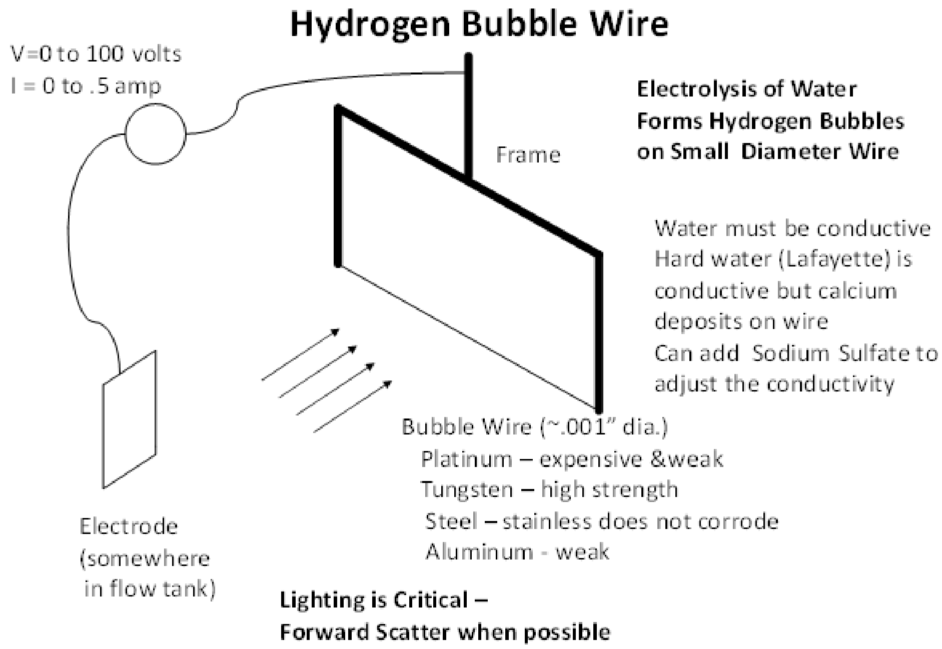


Figure 7.10: Hydrogen Bubble Wire Setup

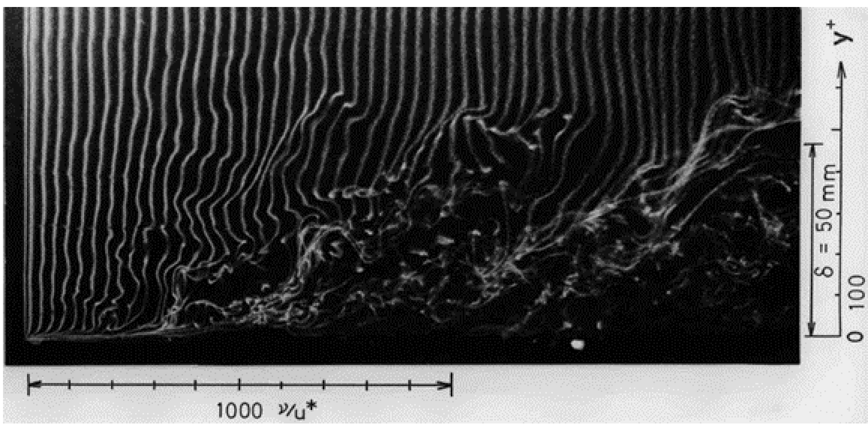


Figure 7.11: Hydrogen Bubble Wire Setup

7.2.2.2 Setup of Bubble Wire System

These instructions will walk you through the general setup of the hydrogen bubble wire system for use in the AAE 33301 Lab 1 involving the Armstrong water tunnel. The bubble wire system utilizes a GwINSTEK GPR-11H30D power supply, BK PRECISION 4054 waveform generator, and a custom control box developed by John Philips (retd). These instructions are meant as a base point for the initial generation of bubbles utilized in flow visualization, additional adjustments may be necessary based on flow speed and water conditions. The photograph pf the actual setup is attached for your reference (Figure 7.12).

NOTE: Please be sure before starting that the voltage and current knobs on the power supply are turned fully counter-clockwise, or OFF.

Instructions:

1. **Start the power supply:** After ensuring all knobs are zeroed on the power supply, begin by turning on the power supply by pressing the large GREEN power button on the bottom left. The amperage and voltage output should both read zero.
2. **Start the waveform generator:** Push the power button located at the lower left of the screen of the waveform generator.
3. **Start the control box:** Switch on the control box utilizing the left toggle, moving it to the "ON" position.
4. At this point the screen on the waveform generator should be on and at the sine-wave screen and the red lights on the control box should be lit above power and the "Pulse / Cont" switch assuming it is in the continuous position.
5. Begin setting the voltage in order to obtain the desired bubble size from the wire.
   - Selecting the "+" toggle from the control box will generate the smaller hydrogen bubbles desired, while "-" will generate the larger, more buoyant, oxygen bubbles.
   - To begin generating bubbles, first increase the "current" knob, clockwise turns, until the coarse knob is maxed out. Use the coarse and fine adjustment on the "voltage" knob to make the bubbles appear. **Only control the voltage for bubble generation, leave the current value setting maxed out.**

   NOTE: Switch polarity from positive to negative and back on the control box to help clear dirt and debris from the wire in the tunnel. A small brush may also be utilized to clean the wire of buildup, and is advisable in between model tests or when notable degradation of the bubble streamlines appears.
6. Once bubble size has been set, pulses can be added by the use of the waveform generator. Start by switching the upper control box to "Pulse", at which point the light above will turn off indicating no incoming signal from the waveform generator. In order to obtain this signal the generator must be programmed as follows:

   6.1 Start by selecting the "Pulse" signal from the six available waveform options near the screen of the generator, at which point it should light up.

   6.2 To activate the pulse, press the "Output" button above the yellow "CH1" output, which should also light up. At this point the light above the "Pulse/Cont" toggle on the control box should light up solid.

   6.3 Utilizing the five buttons directly right of the display screen, adjust period, pulse width, and duty cycle by first selecting the desired parameter and then using the large knob on the right of the panel to adjust the value. The light above "Pulse/Cont" on the control box will flicker at the same frequency and width as input into the waveform generator.

   6.4 A nearly infinite degree of variability exists here in order to allow for the best possible flow visualization. You may use a frequency of 1 Hz (time period: 1 s) with a pulse width of 500 ms (or 50% duty cycle) to generate waves. Then calculate velocity, and the Reynolds number of the flow as described in Section 1.7.
7. Once testing is complete, power down the system by first switching the control box to Cont. and then switching it OFF. Push the power button on the waveform generator to turn it OFF. And finally, ZERO the four voltage and current knobs on the power generator and then power it OFF as well.

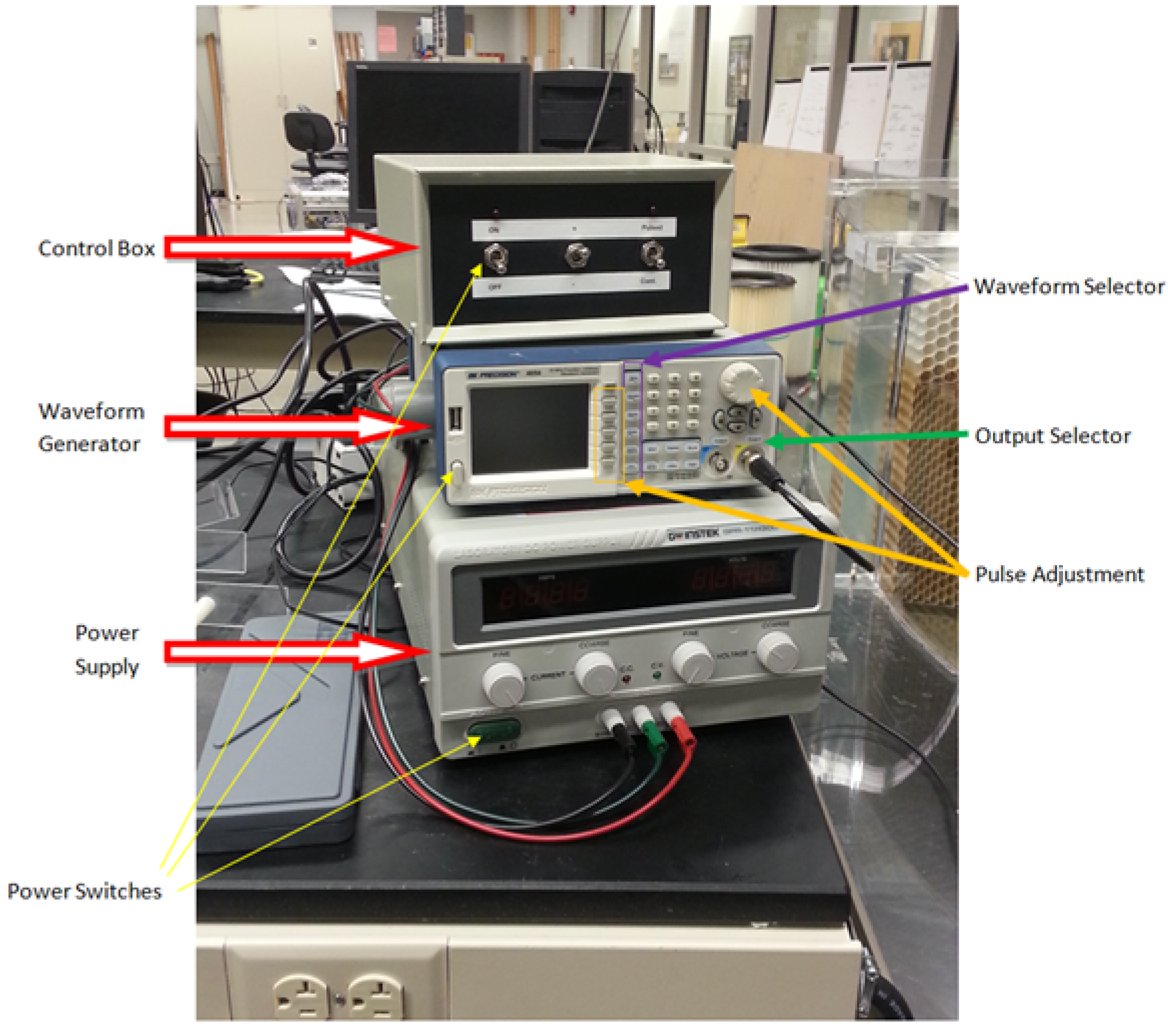


Figure 7.12: Electronic setup for operation of hydrogen bubble wire

7.3 Reynolds Pipe Flow Apparatus

The Reynolds pipe flow apparatus can be used to analyse the flow properties of water in various regimes.

7.4 Introduction to LabVIEW

LabVIEW (short for **Lab**oratory **V**irtual **I**nstrument **E**ngineering **W**orkbench) is a fully featured Integrated Development Environment (IDE) produced by National Instruments. It is a graphical programming environment designed for automated test and measurement. Engineers and scientists use LabVIEW in a wide array of applications and industries to integrate hardware and collect data in their test and measurement systems.

Here are some salient features of LabVIEW:

1. **Purpose-Built for Test and Measurement**
    - LabVIEW is specifically designed for creating test and measurement systems. It provides a unique approach to programming, connectivity to various instruments, and fully integrated user interfaces, via its graphical programming environment.
    - Engineers can use LabVIEW to build custom measurement and control applications, automate testing processes, and acquire data from sensors, instruments, and other hardware.
2. **Graphical Programming Environment**
    - LabVIEW uses a graphical programming language called G. Despite its name, many people refer to the language as "LabVIEW."
    - Instead of writing code in traditional text-based languages, LabVIEW users create programs by connecting graphical icons (nodes) that represent functions, data flow, and control structures.
    - This visual approach makes it easier to understand and develop complex systems.
3. **Hardware Integration and Data Analysis**
    - LabVIEW simplifies hardware integration for engineering applications. It provides a consistent way to acquire data from National Instruments (NI) hardware as well as third-party devices.
    - The software supports drag-and-drop user interface (UI) creation and integrated data viewers, allowing engineers to immediately visualize results.
    - Engineers can also develop algorithms for data analysis and advanced control using built-in math and signal processing functions or by reusing their own libraries.

7.5 Numerical Methods

7.5.1 Trapezoidal Rule

The trapezoidal rule is a technique for numerical integration, that can be used to perform integration of discrete values over a given interval. Essentially, it can be used for calculating a definite integral $\int_a^b f(x)dx$.

For a non-uniform grid spacing, one can use the following formula.

$$\int_a^b f(x)dx \approx \sum_{k=1}^{N} \frac{f(x_{k-1}) + f(x)}{2} \Delta x_k \tag{7.4}$$

where $\Delta x_k = x_k - x_{k-1}$

Glossary

irrotational a flow is called an irrotational flow when fluid particles does not rotate about their centre of mass in the flow field; no element of the moving fluid rotates in any direction from one instant to the next. 5

LabVIEW or **Lab**oratory **V**irtual **I**nstrument **E**ngineering **W**orkbench is a systems engineering software developed by National Instruments that is primarily used for data acquisition, instrument control and automation.. 59

manometer an instrument for measuring the pressure acting on a column of fluid, especially one with a U-shaped tube of liquid in which a difference in the pressures acting in the two arms of the tube causes the liquid to reach different heights in the two arms. 32

TA or Teaching Assistant is a graduate student who will supervise the lab operations; different group of students may have different TAs. Students are required to be in contact with TAs for their prelab and post lab assignments and can ask them questions about experiments.. 12

Venturi Tube is a tube which gradually tapers from full pipe diameter to a constricted diameter, which then tapers off again to a bigger diameter. This can be used to measure flow rate and to verify the Bernoulli's equation, if the flow is incompressible.. 44

VI or Virtual Instrument is the format/extension in which LabVIEW saves its files (.vi). People refer programs made in LabVIEW as "VI".. 42, 59