ME 347

FLUID MECHANICS
LAB EXPERIMENTS

7TH Edition

Upgrades to the Fluid Mechanics Lab Made Possible By:

Donald E. Bently Center for Engineering Innovation
Mechanical Engineering Student Fee Allocation Committee

Mechanical Engineering Department
California Polytechnic State University
San Luis Obispo, CA

Copyright © 2009 by Cal Poly Mechanical Engineering Department
All rights reserved. Reproduction or translation of any part of this work beyond that permitted by sections 107 and 108 of the 1976 United States Copyright Act without permission of the copyright owner is unlawful.
# CONTENTS

Introduction to Lab ........................................................................................................ iii

1. Wind Tunnel Testing and Pitot-Static Tubes .............................................................. 1-1

2. Thrust from a Water Rocket ....................................................................................... 2-1

3. Flow Through a Rectangular Duct with a 90° Bend ............................................... 3-1

4. Pipe Flow Analysis .................................................................................................... 4-1

5. Industrial Ventilation System Analysis ..................................................................... 5-1

6. Flat Plate Boundary Layer Flow ................................................................................ 6-1

7. Cylinder in a Uniform Flow Stream .......................................................................... 7-1

8. Flow Through a Converging-Diverging Nozzle ....................................................... 8-1

9. Water Turbine Performance Test .............................................................................. 9-1

10. Axial Fan Performance Test ..................................................................................... 10-1

Appendix A. Textbook Reading .................................................................................... A-1

Appendix B. Measurement Error Review ..................................................................... B-1
INTRODUCTION TO LAB

Welcome to the fluid mechanics lab located in Engineering IV (Building 192) Room 102. All of the experiments are designed to illustrate many of the concepts that were presented to you in either Fluid Mechanics I or II. The first objective is to give you hands-on experience with fluid mechanics experiments. This includes working with standard facilities and instrumentation found in fluid mechanics laboratories. The second objective is to learn how to use experimentally collected data to either validate an analytical model for a range of flow conditions or to characterize the performance of a flow passage or turbomachine. This includes learning about experimental design and instrument selection concepts needed to achieve the goals of each lab. The third objective is to continue to develop your ability to effectively document your results.

There are a total of ten experiments in this lab manual presented in Sections 1 through 10. Your lab instructor will select from these experiments for your lab class. The lab manual is organized to prepare you for the experiment prior to arriving in lab; therefore, it is critical that you read each manual section prior to lab. In addition, you should review the corresponding sections from your fluid mechanics textbook to refresh your memory on the concepts (see list in Appendix A).

Experiments in this lab manual are organized as follows: (1) Objectives, (2) Introduction, (3) Experiment, (4) Report, and (5) References (except for the Industrial Ventilation System Analysis experiment). The objective describes the goals of the experiment. The introduction gives background information for the lab, describes general lab procedure, outlines each measurement along with required data reduction steps, and develops models for data comparison. For the data reduction and modeling, we will begin by applying the basic governing conservation equations for fluid mechanics. Derivations for these equations can be found in your textbook. The experiment section gives detailed step-by-step instructions needed to run the experiment and collect data. The report section gives detailed instructions for reducing the experimental data, producing tables and/or figures, and performing additional calculations that will be used when completing the report. Be advised that the experimental procedure and report requirements listed for the experiment can be changed at the discretion of your instructor. If in doubt ask your instructor for guidance.

There are three types of documentation that this class will typically require: (1) lab report, (2) technical memo, and (3) assignment. A lab report is a formal presentation of your results and includes a complete description of your experiment, results, and conclusions. A technical memo is a less formal and more concise presentation of your results. For both the lab report and technical memo you should assume your audience is your boss and coworkers that are NOT familiar with the details of each experiment. Thus, you must give sufficient information so they can understand your document and could reproduce your results. An assignment is an informal presentation of your results where you submit answers to specific questions, similar to a homework assignment. Your lab instructor will select the type of documentation required for each lab and if they should be completed by a group or individually.
Safety is a top concern in executing these lab exercises. The fluid mechanics lab contains equipment that uses electricity, pressure systems, and compressed air. Also, there are numerous experiments with rotating machinery. All equipment must be used in a safe manner. If you have any questions, please ask the instructor prior to using the equipment. You must read the *General Lab Practice Procedures* included in this Introduction before performing these labs.
California Polytechnic State University
Mechanical Engineering Department

**General Lab Safe Practice Procedures**

1. Read the following course Safe Practice Procedures information.
   
   a. **Wind Tunnel**: Hearing protection is required when the wind tunnel is in operation. The wind tunnel accelerates air to approximately 100 mph in the test section; therefore, objects must not be placed near the entrance of the wind tunnel. These could be sucked into the tunnel and cause damage. The motor and fan are connected by a belt. Keep hands away from the belt as injury could result.

   b. **Airflow Bench**: This experiment blows high velocity air at the outlet. Do not put your face in the path of the outlet air. Follow the procedures carefully to avoid blowing manometer fluid from the tubes and do not operate the damper with the lock on.

   c. **Water Rocket Experiment**: High pressure air is used in this experiment. When pressurizing the bottle, make sure the door to the bottle chamber is shut and you are wearing safety glasses. Do not open the door to the bottle chamber when it is pressurized.

   d. **Pipe Flow Experiment**: The bucket used to weigh the water may be heavy. Be sure to lift it properly to avoid injury.

   e. **Compressible Flow Apparatus**: The compressor and motor rotate at high speed. Keep hands and hair away from the moving belt. Do not place small objects near the entrance to the duct. These could be sucked into the compressor and cause damage. Do not turn on the apparatus until the speed control knob is verified fully counterclockwise. Finally, in case of emergency, push the emergency stop switch located on top of the display panel.

   f. **Axial Fan Experiment**: The fan and motor are rotating at high speed. Hands and objects must be kept clear of the fan inlet. While the fan is not loud, the experiment is in close proximity to the wind tunnel that is loud; therefore, hearing protection is required. Use of a ladder may be required. When using a ladder, do not use either of the top two steps.

   g. **Water Turbine Experiment**: This experiment contains rotating machinery. Also the turbine rotates at high speed and is noisy. Observe all safety limits on the equipment to avoid injury and damage.

2. Use proper Personal Protective Equipment (safety glasses or goggles, ear plugs, gloves etc.) when operating lab equipment or experiments.

3. Wear appropriate attire when operating equipment. Secure long hair around rotating equipment or open heat source, proper shoes where drop hazards exist, etc.
4. No equipment shall be operated without the instructor’s permission.

5. No unsupervised use of laboratories without prior written authorization by the instructor.

6. No working alone in the laboratories.

7. Any accident or illness must be immediately reported to the instructor and/or the Mechanical Engineering Department office.

8. Any unsafe or hazardous condition in lab (liquid spills, electrical hazards etc.) must be immediately reported to the instructor.

9. In case of an emergency dial 911 and tell the dispatcher:
   • The nature of the emergency
   • Your name
   • The location of the emergency

10. To evacuate the building in an emergency exit either by the door in front of lab (by the roll-up door) or by the door next to the inlet of the wind tunnel. All students are to congregate in the loading dock of Engineering III should the building be evacuated.
1. **Wind Tunnel Testing and Pitot-Static Tubes**

![Cal Poly Mechanical Engineering Department wind tunnel.](image)

**Figure 1-1** Cal Poly Mechanical Engineering Department wind tunnel.

I. **Objectives**

The objectives of this experiment are: (1) to introduce the basic operation of the wind tunnel shown in Figure 1-1, (2) to measure test section velocity using a Pitot-static tube, and (3) to calibrate contraction differential pressure measurements to also provide test section velocity.

II. **Introduction**

Wind tunnels are tools used to evaluate aerodynamic properties, such as drag and lift forces, for a wide range of objects (Ref. 1). Scaled models are often used for large complex objects such as airplanes, cars, trucks, wind turbines, and cities. The use of models allows for lower cost testing and ease of development compared to the construction and testing of actual objects. The wind tunnel also allows, with careful oversight, the opportunity for very high accuracy measurements and repeatability as the conditions of the test can be carefully controlled. The testing of scaled models requires an understanding of how forces and flow kinematics are scaled so that we can predict full-scale performance and characteristics from sub-scale testing. We will not go into the details of scaling parameters for this lab. We will consider how the air velocity in the wind tunnel can be accurately measured, critical for wind tunnel testing, and some of the methods used to ensure accuracy.
Pitot-Static Tube

The Pitot-static tube, shown in Figure 1-2, is a common device used to measure the local dynamic pressure which can be used to calculate the local velocity. The relationship for this calculation is developed by applying the Bernoulli equation

\[
\frac{p}{\rho} + gz + \frac{V^2}{2} = \text{constant}
\]  

where \( p \) is pressure, \( \rho \) is density of the fluid, \( g \) is gravitational acceleration, \( z \) is elevation, and \( V \) is velocity. The Bernoulli equation can be derived by (1) applying Newton's Second Law to a fluid element moving along a streamline under ideal conditions and (2) integrating this equation along the streamline (over distance). Thus, each term represents mechanical work (force integrated over distance) and the Bernoulli equation is a statement of the conservation of mechanical work for ideal flow. The ideal conditions required for the Bernoulli equation to apply include that the flow must be steady, incompressible, inviscid, and along a streamline.

![Figure 1-2](image)

**Figure 1-2** Pitot-static tube and differential manometer cross-section schematic.

The pressure term, \( p \), in the Bernoulli equation is the thermodynamic or static pressure. This is the pressure that would be measured by a sensor moving with the flow. In practice this is extremely difficult to measure, but in steady incompressible flow we can show that there is no change in pressure normal to the streamlines if the streamlines are straight (Ref. 2). Thus, for steady, isentropic and straight flow, like in a wind tunnel, a simple pressure tap on the wall will measure the static pressure at that longitudinal cross section. For a Pitot-static probe the static ports are aft of the leading edge of the probe and generally consist of multiple, ganged ports about the circumference of the probe as shown in Figure 1-2.
When a flowing fluid of velocity $V$ and static pressure $p$ is slowed isentropically to rest at constant elevation the resulting pressure is called the total or stagnation pressure, $p_T$. We can apply the Bernoulli equation to a constant elevation streamline to relate the free stream conditions (at $p$ and $V$) to the Pitot tube conditions (at $p_T$ and zero velocity) to get

$$\frac{p + \frac{V^2}{2}}{\rho} = \frac{p_T}{\rho}$$

(1-2)

and then rearrange to get

$$p_T = p + \frac{1}{2} \rho V^2 .$$

(1-3)

This is the definition of stagnation pressure and introduces the dynamic pressure term

$$p_T - p = \frac{1}{2} \rho V^2 .$$

(1-4)

Rearranging we can obtain a relationship for velocity based upon the dynamic pressure

$$V = \sqrt{\frac{2}{\rho} (p_T - p)} .$$

(1-5)

The stagnation and static pressure ports from the Pitot-static tube are typically connected to opposite sides of a differential pressure measurement device, thus reading the dynamic pressure directly. A common differential pressure measurement device (that we will use extensively in this lab) is a U-tube manometer as shown in Figure 1-2. It displays the pressure reading as a height, $h$, of a column of liquid (in this case water). This corresponds to the pressure a column of liquid of height $h$ would produce. Thus, the unit of pressure for this manometer reading is given as inches of water or “in H$_2$O.” To convert this to standard units of pressure we must use the hydrostatic equation (force balance for an incompressible liquid at rest)

$$p_T - p = \rho_{H_2O} g h$$

(1-6)

where $\rho_{H_2O}$ is the density of water. Note that this is different than the air density used above in Equation 1-5. For our lab we use a differential pressure transducer calibrated to indicate pressure as in H$_2$O. With these relationships we can simply use a differential pressure transducer to measure the dynamic pressure and, for known fluid density, the velocity can be calculated from Equation 1-5. Alternatively, a total pressure probe, also called a Pitot tube or total head probe, can be used in concert with a remote static pressure tap as long as the steady, uniform, incompressible and isentropic flow assumptions are met.
Wind Tunnel Contraction Pressure Differential Velocity Measurement

A schematic diagram of the Cal Poly Mechanical Engineering department’s wind tunnel is shown in Figure 1-3 where the flow of air through the wind tunnel is from left to right. The main sections of the wind tunnel are in order: (1) rounded inlet, (2) settling chamber, (3) honeycomb flow straightener, (4) contraction, (5) test section where models are placed, (6) primary diffuser/transformation, (7) flexible coupling, and (8) fan used to induce the flow. Most wind tunnel testing is performed at constant velocity or dynamic pressure across most of the test section. The air flow in the wind tunnel test section can be treated as inviscid flow because the boundary layers are very small, i.e. still in the entrance region of this very large pipe. When testing a model in a wind tunnel it is desirable to use the largest model practical in the test section while minimizing the interaction from the physical boundaries of the tunnel. It is also desirable to minimize intrusions into the test section that may influence the measurements. However, the ability to successfully evaluate the results from a wind tunnel experiment is dependent on knowing the wind velocity very accurately in the test section. To minimize disturbances and maximize the available volume for larger models it would be preferable to not have a Pitot-static tube in the test section of the tunnel when possible. Additionally, the presence of the model itself will influence the pressure field in the test section and can lead to erroneous readings from a Pitot-static probe.

Figure 1-3 Cal Poly Mechanical Engineering Department wind tunnel schematic.

An alternative method for determining the velocity in the test section is to measure the static pressure drop across the contraction and use a calibration constant to calculate the test section velocity. The relationship used for this calculation is developed by applying the energy equation for incompressible, steady flow to a control volume with a single inlet and exit

\[
\frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 - \frac{p_2}{\rho g} - \frac{V_2^2}{2g} - z_2 = h_c
\]  

(1-7)
where $\alpha$ is the kinetic energy coefficient, $h_L$ is the head loss term, and the subscripts 1 and 2 indicate the inlet and exit, respectively. For a control volume that includes all the air flowing from the contraction inlet to the contraction exit and assuming uniform flow across control surfaces and constant elevation we can reduce this to

$$\left( \frac{p_L}{\rho g} + \frac{V_L^2}{2g} \right) - \left( \frac{p_C}{\rho g} + \frac{V_C^2}{2g} \right) = K_1 \left( \frac{V_C^2}{2g} \right)$$

(1-8)

where $K_1$ is the minor head loss coefficient for the contraction and the subscripts $L$ and $C$ indicate the contraction inlet and exit, respectively. Solving for the pressure drop across the contraction and factoring out the dynamic pressure at the contraction exit gives

$$p_L - p_C = \left[ 1 + K_1 - \left( \frac{V_L}{V_C} \right)^2 \right] \frac{1}{2} \rho V_c^2.$$  

(1-9)

We can eliminate $V_L$ by applying conservation of mass for incompressible flow to the same control volume to get

$$A_L V_L = A_C V_C.$$  

(1-10)

To simplify our equation, we define

$$K_2 = \left( \frac{V_L}{V_C} \right)^2 = \left( \frac{A_C}{A_L} \right)^2$$  

(1-11)

and substitute this into Equation 1-8 to yield

$$p_L - p_C = \left( 1 + K_1 - K_2 \right) \frac{1}{2} \rho V_C^2.$$  

(1-12)

If the test section does not have the same cross sectional area as the contraction, then we can again use conservation of mass for incompressible flow applied to a control volume between the end of the contraction and the test section to obtain

$$A_C V_C = A_S V_S$$  

(1-13)

where the subscript $S$ denotes the test section, define

$$K_3 = \left( \frac{V_C}{V_S} \right)^2 = \left( \frac{A_S}{A_C} \right)^2,$$  

(1-14)

and finally substitute this into Equation 1-12 to yield
Thus, the static pressure differential across the contraction can be directly related to test section dynamic pressure (or velocity) through a simple calibration by comparing a known velocity in the test section to the measured contraction pressure differential.

\[
p_L - p_C = K \frac{1}{2} \rho V_s^2 \quad \text{or} \quad p_L - p_C = K \left( p_T - p \right)
\]

\[
K = \left( 1 + K_1 - K_2 \right) K_3
\]
III. Experiment

Equipment

Cal Poly ME Wind Tunnel Facility
Pitot-static probe
x-y traverse
MKS Baratron® Series 200 pressure transducer

Preparations for Test (Completed by Instructor)

1. Energize the 230V power supply for wind tunnel. Turn on wind tunnel instrumentation system (Figure 1-4) and allow to warm up for at least 15 minutes.

2. Ensure that the zero and span for the pressure transducer located on the instrumentation system front panel (Figure 1-4) have been properly calibrated to display differential pressure from 0.00 to 10.0 in H₂O on the pressure transducer display.

3. Ensure that the Pitot-static probe is mounted correctly in the traverse and that the traverse is properly connected to the back panel of the instrumentation system (Figure 1-5).

4. Ensure that the zero and span for the traverse are set correctly. The x-direction (streamwise position) should be zero at the entrance of the test section. The y-direction (vertical position) should be zero at the bottom of the test section. The span for the x-axis and y-axis should be set to display the position in inches. Verify by checking that the length of the test section reads 48.0 in and the height of test section reads 24.0 in.

5. Locate the Pitot-static probe at the midpoint of the test section.
6. Ensure that the total and static pressure tubes from the Pitot-static probe are connected to the total and static pressure barbed fittings on the back panel of the instrumentation system (Figure 1-5). Port 0 on the pressure selector (Figure 1-4) corresponds to the Pitot-static probe reading.

7. Ensure that the pressure tube located before the contraction is connected to the white tube (corresponds to Port 1 on the pressure selector in Figure 1-4) on the pressure bulkhead connector on the back panel of the instrumentation system (Figure 1-5).

**Procedure**

1. As with any piece of equipment, the condition of the equipment and safety of operation must be checked before operating. First verify, along with the instructor, that the inlet area of the wind tunnel is free from obstructions and foreign objects and the honeycomb flow straightener is clean and undamaged. Inform the instructor of any issues and document your observations. Next, ensure that there are no foreign objects in the wind tunnel. This completes the most basic of checks before operation of a wind tunnel and must be performed every time the wind tunnel is operated.

2. Record the atmospheric pressure, $p_{\text{amb}}$, and temperature, $T_{\text{amb}}$. Record the initial value of the pressure gauge, $p_{\text{zero}}$, which is the offset for both of your differential pressure readings for this lab. Use the data sheet at the end of this section.

3. Select Basic Parameters Mode on the motor speed controller panel by pressing the dial, <enter>, on the keypad shown in Figure 1-6.
4. If necessary, set the adjustment increment to 1Hz increments by pressing the F3 function key. Finer increments of 0.1Hz are possible by selecting the F4 key.

5. Turn the dial clockwise to increase the inverter frequency setting to 10Hz.

6. Press the dial, <enter>, to return to the Top View Mode.

7. Press the **RUN** button to start the tunnel. (This step is not required when the tunnel is already running. The inverter controller will automatically ramp up or down the motor speed as the frequency setting is changed.)

8. Allow the tunnel to settle, i.e. reach steady state which typically takes approximately 1 minute, and record 5 readings each for the differential pressure output of the Pitot-static probe (Port 0) and the contraction gage pressure (Port 1). To change ports use the pressure selector on the front panel of the instrumentation system (Figure 1-4).

9. Repeat Steps 2 and 7 for 20Hz, 30Hz, 40Hz, 50Hz and 60Hz speed settings.

10. When data collection is complete press STOP to shut down the wind tunnel motor and fan.

**IV. Report**

1. Tabulate the raw data.

2. Calculate and tabulate the average differential pressure measurement from the Pitot-static probe (or test section dynamic pressure) in lb/ft², the average pressure drop across the contraction in lb/ft², and the average test section velocity in ft/s for each fan speed setting from the pitot-static probe and the contraction data. Use the atmospheric pressure and temperature to calculate the density of the air using the ideal gas law. Include sample calculations with your table.

3. The manufacturer specifies that the top speed for the wind tunnel is 110 mph (at 60 Hz). Can you verify that this specification has been met for this wind tunnel?

*Figure 1-6 Motor speed controller keypad shown in Top View Mode.*
4. Plot the calculated average test section velocity in ft/s versus fan speed in Hz. In your caption describe the trends seen and if they make sense.

5. Plot the average contraction pressure drop versus the average Pitot-static probe differential pressure measurement (or test section dynamic pressure). Find the calibration constant, \( K \), for Equation 1-15 using a linear curve fit to the data. Compare your measured value for \( K \) to a value calculated from area ratios (see Figure 1-3) and a contraction loss coefficient estimated from the tables in your textbook.

6. Calculate and tabulate the uncertainty with 95% confidence for (a) the differential pressure measurement from the Pitot-static probe in lb/ft\(^2\), (b) the pressure drop across the contraction in lb/ft\(^2\), and (c) the test section velocity in ft/s for each fan speed setting. See Appendix B. Measurement Error Review to aid you in these calculations. For parts (a) and (b) include the minimum error and statistical uncertainty in your total uncertainty calculation. For part (c) use the Uncertainty in Calculated Quantities method. Include sample calculations with your table.

7. Give two reasons why a differential pressure measurement is advantageous for the Pitot-static tube measurement compared to measuring individual pressures. For your answer consider how you can minimize error by what we choose to measure.

8. Did you take enough measurements for reasonable accuracy for the velocity measurement? In general, how many measurements do you think are sufficient?

9. Describe three design features, based on the assumptions of the Bernoulli equation and other concerns, that may influence the accuracy of the Pitot-static probe (such as port placement, fabrication issues, alignment) and why?

V. References


# Data Sheet
## Wind Tunnel Testing

<table>
<thead>
<tr>
<th>Fan Speed (Hz)</th>
<th>No.</th>
<th>Pitot-static Pressure (in H₂O)</th>
<th>Contraction Pressure (in H₂O)</th>
<th>COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 Hz</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20 Hz</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30 Hz</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>40 Hz</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50 Hz</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>60 Hz</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
2. THRUST FROM A WATER ROCKET

I. Objectives

The objectives of this experiment are to measure the thrust produced by the water rocket shown in Figure 2-1 and to compare it to the thrust predicted by a simple model developed using the integral form of the momentum equation.

II. Introduction

Water rockets typically consist of a plastic cylinder that is partially filled with water, sealed with a stopper at the neck, and then pressurized with air. Upon release of the stopper, the compressed air rapidly forces water out of the nozzle propelling the rocket upwards.

Figure 2-1 Water rocket experimental apparatus.
Experiment

In this experiment a 2-liter soda bottle is used as the cylinder. At the beginning of each experiment, the height of water in the bottle (used to calculate the volume) and the initial pressure of the air above the water will be measured. Instead of allowing the rocket to move upwards, the top of the bottle is attached to a load cell as shown in Figure 2-2. The thrust produced by the rocket (minus the weight of the water and bottle) is measured by the load cell as a function of time. The weight of the full water bottle will be zeroed out from the load cell measurement so that the thrust can be recorded directly (neglecting the change in the weight of the water as the bottle empties). The pressure of the air inside the bottle will also be measured during the experiment and used to interpret the type of process for the air expansion.

Modeling

The measured thrust will be compared to the theoretical thrust predicted by a simple model developed from the momentum equation in integral form

\[ \frac{\partial}{\partial t} \int_{CV} \rho \ V \ dV + \int_{CS} \rho \ V \cdot \hat{n} \ dA = \sum F \] (2-1)

where \( t \) is time, \( V \) is velocity, \( \rho \) is density, \( \vec{V} \) is volume, \( A \) is area, \( \hat{n} \) is the outward unit normal for area, and \( F \) are forces. Equation 2-1 represents Newton’s Second Law applied to a finite control volume, thus the rate of change of momentum inside the control volume, \( CV \), and the net momentum flux through the control surfaces, \( CS \), is equal to the sum of the forces acting on the control volume. For our model, we define the control volume as a rectangular box around the bottle and apply Equation 2-1 in the vertical \( z \)-direction.

Figure 2-2 Schematic of water rocket.
To reduce Equation 2-1 for this control volume and process, we will consider each term separately. For the first term on the left side, the momentum of the air and water inside the bottle does change with time as the water is ejected. However, in our analysis this transient process will be divided into many small time steps. For each time step a steady state analysis will be performed as an approximation. This is called a quasi-steady approximation and it will greatly simplify the analysis (Ref. 1). Thus, the rate of change in momentum inside the control volume, the transient term in Equation 2-1, is assumed to be small compared to the other terms for each time step and neglected. For the second term on the left side, the only momentum flux across the control surface corresponds to the water jet issuing from the bottom of the rocket. For the right side, the only force acting on the control volume is the reaction force at the load cell. Again, this reaction force will include the upwards thrust and the weight of the bottle and water. Using all of these Equation 2-1 now reduces to

\[ \int_{A_{\text{jet}}} V_{\text{jet},i} \rho_w V_{\text{jet},i} dA = F_{\text{LC},i} \]  \hspace{1cm} (2-2)

where \( V_{\text{jet},i} \) is nozzle jet velocity and \( F_{\text{LC},i} \) is force on the load cell, both at state \( i \) (during a small time step). Also, \( \rho_w \) is density of the water. Assuming uniform flow across the nozzle we get

\[ \dot{m}_i V_{\text{jet},i} = F_{\text{LC},i} \]  \hspace{1cm} (2-3)

where mass flowrate of the water at the nozzle at step \( i \) is

\[ \dot{m}_i = \rho_i A_{\text{jet}} V_{\text{jet},i} \]  \hspace{1cm} (2-4)

where \( A_{\text{jet}} \) is nozzle exit area.

**Nozzle Jet Velocity**

To calculate the nozzle jet velocity, we will use the Bernoulli equation for steady, incompressible, inviscid flow along a streamline

\[ \frac{p}{\rho} + g z + \frac{V^2}{2} = \text{constant} \]  \hspace{1cm} (2-5)

where \( p \) is pressure, \( g \) is gravitational acceleration, \( z \) is elevation, and \( V \) is velocity. Again, since the analysis consists of looking at many small time steps, each time step can be assumed to be approximately steady at step \( i \). Bernoulli’s equation will be applied to a streamline attaching states 1 and 2 (in the water only) in Figure 2-2. The pressure at state 1 is the same as the air pressure in the bottle, \( p_{a,i} \), and the pressure at state 2 is the same as the atmospheric pressure, \( p_{\text{atm}} \). The change in potential energy is small compared to the change in pressure energy, so it will be assumed negligible. From conservation of mass and due to the significant decrease in area, the velocity at 1 will be much less than the velocity at 2, thus it will be assumed negligible. The velocity at state 2 is the nozzle jet velocity, \( V_{\text{jet},i} \). Based on these assumptions and variable substitutions we get
\[
p_{a,i} = \frac{p_{am} + \frac{V_{jet,i}^2}{2}}{\rho_w}.
\]

Solving for the jet velocity at step \(i\) this reduces to

\[
V_{jet,i} = \sqrt{\frac{2(p_{a,i} - p_{am})}{\rho_w}}.
\]

Air Pressure in Bottle

To calculate \(V_{jet,i}\) we need to calculate the air pressure in the bottle at each subsequent time step. During the experiment, you will measure the initial air pressure, \(p_{a,0}\). From this initial measurement, you will calculate the air pressure at subsequent time steps, \(p_{a,i}\), by assuming a polytropic process for an ideal gas

\[
\frac{p_{a,i+1}}{p_{a,i}} = \left(\frac{\frac{V_{a,i}}{V_{a,i+1}}}{\frac{V_{a,i}}{V_{a,i+1}}}\right)^n
\]

(2-8)

where \(V_{a,i}\) is the volume of air at step \(i\) and \(n\) is the polytropic exponent (Ref. 2). Therefore, the pressure at the next time step \(p_{a,i+1}\) is a function of the previous pressure and the volume at steps \(i\) and \(i+1\). The polytropic exponent can vary from \(0 \leq n \leq \infty\). For \(n = 0\), the process is constant pressure. For \(n = 1\), the process is constant temperature. For \(n = k\), the process is adiabatic and reversible where \(k\) is the specific heats ratio. Finally for \(n = \infty\), the process is constant volume. For this experiment, the polytropic exponent will be determined experimentally and you will need to evaluate the type of process that has occurred.

To calculate \(p_{a,i}\) we also need to calculate the air volume at each subsequent time step. You will measure the initial volume of water, \(V_{w,0}\), in the bottle. The initial air volume, \(V_{a,0}\), is calculated by assuming the total volume of the bottle, \(V_{bottle}\), is constant

\[
V_{bottle} = V_{a,i} + V_{w,i}.
\]

(2-9)

Since this transient process is non-linear with time, each step will be based on a fixed change in air volume fraction, \(\Delta y_a\). The air volume fraction at each step, \(y_{a,i}\), is defined as

\[
y_{a,i} = \frac{V_{a,i}}{V_{bottle}}.
\]

(2-10)
The initial air volume fraction, $y_{a,0}$, is calculated from the initial measurements. Next, the air volume fraction is incremented using

$$y_{a,i+1} = y_{a,i} - \Delta y_a .$$  

(2-11)

Then, Equation 2-10 is used to calculate $\frac{\partial V}{\partial x}$ and Equation 2-9 $\frac{\partial V}{\partial y}$ for subsequent steps.

**Time Step**

To calculate the time step that corresponds to the increment in air volume fraction, conservation of mass in integral form is applied to our original control volume

$$\frac{\partial}{\partial t} \int_{CV} \rho dV + \int_{CS} \rho \mathbf{V} \cdot \hat{n} dA = 0 .$$  

(2-12)

The only mass leaving the control volume is the water exiting at the nozzle exit reducing this to

$$\left. \frac{\partial m_w}{\partial t} \right|_{t_i} + \dot{m}_i = 0$$  

(2-13)

where $m_w$ is the mass of water in the control volume. We will use finite differences to approximate the differential in this equation as

$$\left. \frac{\partial m_w}{\partial t} \right|_{t_{i+1}} \approx \frac{m_{w,i+1} - m_{w,i}}{\Delta t_{i+1}}$$  

(2-14)

where $\Delta t_{i+1}$ is a small time step. Substituting this into Equation 2-13 and solving for $\Delta t_{i+1}$ we get

$$\Delta t_{i+1} = \frac{m_{w,i+1} - m_{w,i}}{\frac{1}{2}(\dot{m}_i + \dot{m}_{i+1})}$$  

(2-15)

where we have used the average mass flowrate between steps $i$ and $i+1$ for $\dot{m}_i$.

**Sample Calculation**

With the equations above and the geometric measurements for the 2-liter bottle given in Table 2-1, the solution technique can now be presented. The following analysis is a sample calculation demonstrating how the theoretical force as a function of time is calculated. For this sample calculation, use an initial water volume of $V_{w,0} = 0.0426 \text{ ft}^3$ and an initial air pressure of $p_{a,0} = 32.0 \text{ psig}$ (both of which will be measured for your experiment).
Table 2-1 2-liter bottle geometric measurements.

| Bottle Volume ($V_{bottle}$): | 0.0727 ft³ |
| Outlet Jet Diameter ($D_{jet}$): | 0.0700 ft |
| Jet Area ($A_{jet}$): | 0.00385 ft² |

The first row in Table 2-2 corresponds to the initial condition or step 0. Enter $V_{w,0}$ and $p_{a,0}$ into columns 4 and 5, respectively. The initial air volume is calculated using Equation 2-9 and entered in column 3. The initial air volume fraction is calculated using Equation 2-10 and entered into column 2. Calculate the jet velocity using Equation 2-7 and enter the result in column 6. Note that the pressure difference in Equation 2-7 is the measured gage pressure in column 5, but that you will need to convert from psig to pressure in lb/ft². The mass flowrate of water leaving the bottle is calculated using Equation 2-4. Enter this result in column 7. The initial mass of water in the bottle in slugs in column 8 is obtained by multiplying the water volume by its density. Finally, column 11 is the load cell force (or water jet thrust) calculated from Equation 2-3.

The next row, corresponding to step 1, is obtained by incrementing the air volume fraction by a small amount $\Delta V_a = 0.001$. The new air volume fraction is calculated using Equation 2-11 and entered in column 2. The air volume is calculated using Equation 2-10 and entered in column 3. The water volume is calculated using Equation 2-9 and entered in column 4. As you can tell from Table 2-2, the air volume increased and the water volume decreased as water is expelled from the rocket, as expected. The new air pressure in column 5 is calculated based on the polytropic relation, Equation 2-8, where we will assume $n = 1.4$ (adiabatic and reversible) as an initial guess. Be careful to use absolute pressures in this equation by adding atmospheric pressure (14.7 psi was used in this sample) to the original gage pressure, then subtract atmospheric pressure from the result to get the new gage pressure. The jet velocity, jet mass flowrate, mass of water in the bottle, and water jet thrust are calculated the same as before. The time interval for this step is calculated using Equation 2-15 and entered in column 9. The time interval is then added to the total elapsed time and entered in column 10.

Table 2-2 Sample thrust calculation.

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Step</td>
<td>Air</td>
<td>Air</td>
<td>Water</td>
<td>Air</td>
<td>Jet</td>
<td>Mass</td>
<td>Water</td>
<td>Time</td>
<td>Total</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fraction</td>
<td>Volume</td>
<td>Volume</td>
<td>Pressure</td>
<td>Velocity</td>
<td>Flowrate</td>
<td>Mass</td>
<td>Interval</td>
<td>Time</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(ft³)</td>
<td>(ft³)</td>
<td>(psig)</td>
<td>(ft/s)</td>
<td>(slug/s)</td>
<td>(slug)</td>
<td>(slug)</td>
<td>(s)</td>
<td>(s)</td>
</tr>
<tr>
<td>0</td>
<td>0.4143</td>
<td>0.03012</td>
<td>0.0426</td>
<td>32.0</td>
<td>68.9</td>
<td>0.515</td>
<td>0.0826</td>
<td>0.00000</td>
<td>0.00000</td>
<td>35.5</td>
</tr>
<tr>
<td>1</td>
<td>0.4153</td>
<td>0.03019</td>
<td>0.0425</td>
<td>31.8</td>
<td>68.7</td>
<td>0.513</td>
<td>0.0825</td>
<td>0.00028</td>
<td>0.00028</td>
<td>35.3</td>
</tr>
<tr>
<td>2</td>
<td>0.4163</td>
<td>0.03027</td>
<td>0.0424</td>
<td>31.7</td>
<td>68.6</td>
<td>0.512</td>
<td>0.0823</td>
<td>0.00027</td>
<td>0.00055</td>
<td>35.1</td>
</tr>
</tbody>
</table>
Additional steps are entered using the same calculations performed for step 1. These calculations are continued until either the water empties from the bottle (column 4 is zero) or the pressure reaches atmospheric pressure (column 5 is zero). The force versus time is then plotted and compared to the experimental force versus time measurement from the load cell as shown in Figure 2-3. The fluctuations in the measured data are due to test stand oscillations.

![Figure 2-3 Thrust curve for water rocket.](image)
III.  Experiment

Equipment

Water Rocket Apparatus
Pressurized Air
Thermometer
Computer with data acquisition system

Procedure

1. Start the data acquisition system by clicking on the RocketLab icon on the desktop. When the window similar to Figure 2-4 opens, change the filename for your first test run. Make sure that the file is saved in the usertemp directory. Click on the Run button  . The load cell, pressure transducer, and thermocouple are now activated.

![Figure 2-4 Program window.](image)

2. Verify that the pressurization valve is fully shut (Figure 2-5) and then attach the air hose to the air supply. Adjust the regulator to 45 psig by turning the knob on the regulator clockwise. Should you raise the pressure greater than 55 psig, the relief valve will lift.

3. Record the temperature of the water in the bucket and atmospheric pressure.
4. Install the plug in the bottom of the bottle, by unscrewing the adaptor on the bottle neck (Figure 2-6). Insert the plug with o-ring until it is firmly seated in the bottle. Verify that the o-ring is seated all the way around the bottle neck or a leak will occur. Reattach the adaptor and insert the pin through the shroud. The bottle is now ready for filling.

5. With the vent valve open on the top of the bottle, start the pump. Open the water fill valve (Figure 2-7) and fill the bottle to approximately 8-in using the scale located on the side of the bottle (Figure 2-8). As the bottle is filling check for leaks from the plug. If leaking stop filling and fix the plug. When the water is at the desired level, stop the pump and shut the fill valve. Shut the vent valve.

6. Shut the door to the cabinet then open the pressurization valve slowly and pressurize the cylinder to 40 psig. Shut the pressurization valve.
7. Zero the weight of the water from the load cell by adjusting the zero knob on the signal conditioner (Figure 2-9) until the load cell reading is approximately zero. Be advised that you will not be able to get this value to be exactly zero due to signal noise.

8. Record the initial water level and pressure. Start collecting data by pressing the Begin Recording button on the display (Figure 2-4). Data will only be collected for 5 seconds, so quickly pull the pin from the bottle neck. This step will require coordination between two people. The plug will shoot out into the bucket below along with the water jet and the bottle will empty in less than 1 sec.

9. Review the data collected. The pressure will record in volts and the load cell will record in volts. Record the load cell and pressure transducer calibration curve constants posted on the experiment that are required to convert the measured voltages to force and pressure, respectively.

10. Repeat steps 1-9 for the same pressure of 40 psig and the same water level of 8-in. Make sure to use a different file name to not overwrite your first data set.
11. Repeat steps 1-9 two more times at 25 psig air pressure and a water level of 8-in.

12. Insert the plug and pin back into the bottle per step 4.

**Figure 2-9** Signal conditioner.

**IV. Report**

1. Tabulate the measured force and pressure data converted to lbf and psig, respectively, for each time step. When submitting tables for your report/assignment, use only the first 10 rows of data.

2. Calculate the initial volume of water, $V_{w,0}$. For heights greater than 4-inches, it can be calculated from the initial height of water in the bottle, $h$, using

   $$V_{w,0} [\text{ft}^3] = (0.008096 \text{ ft}^3/\text{in}) h [\text{in}] - 0.018096 \text{ ft}^3$$

3. Construct four tables similar to Table 2-2 for each of the experimental test conditions. Use an assumed polytropic exponent of $n = 1.4$. This will be changed later. Include sample calculations for one of your tables. When submitting tables for your report/assignment, use only the first 10 rows of data.

4. Plot air pressure versus time using your experimental measurements (symbols with no lines) and from your model predictions (lines with no symbols). You will have a total of four different graphs corresponding to the different test conditions. Only plot the pressure during the draining event. Modify the polytropic exponent in the model until its pressure curve matches the experimental pressure curve. Comment on the agreement or lack of agreement between the predicted air pressure and the actual air pressure. What kind of process did the air undergo?
5. Plot thrust versus time using your experimental measurements (symbols with no lines) and from your model predictions (lines with no symbols). You will have a total of four different graphs corresponding to the different test conditions. These graphs are similar to Figure 2-3. Only plot the thrust during the draining event. Explain which test conditions generated the largest and smallest forces. Comment on the agreement between the model and the experimental data from the thrust curve. Explain any differences.

6. Use data from your model to justify the following two assumptions made while reducing the Bernoulli equation to calculate the nozzle jet velocity: (1) negligible velocity at state 1 compared to state 2 for the chosen streamline and (2) negligible change in potential energy compared to change in pressure energy for each step. Comment on when these assumptions might become invalid.

7. Based on the experimental polytropic exponent, calculate the boundary work and heat transfer that the air in the bottle undergoes. Use an undergraduate thermodynamics textbook to review how to calculate boundary work for an ideal gas undergoing a polytropic process and how to calculate heat transfer from the 1\textsuperscript{st} Law of Thermodynamics for a closed system (for example Section 3.8 from Ref. 2). Comment on the magnitude of the work and heat transfer that the air undergoes. Are they significant (have the same order of magnitude) compared to each other? Is the heat transfer significant?

V. References


3. Flow Through a Rectangular Duct with a 90° Bend

![Figure 3-1 Rectangular bend experimental apparatus.](image)

I. Objectives

This experiment compares the measured pressure distribution for airflow through a rectangular duct with a 90° bend to that predicted by a model that assumes inviscid incompressible flow. Figure 3-1 shows the experimental apparatus that will be used to investigate the airflow. The objectives are to determine the validity of this model for this case and to show the relationship between velocity and pressure in a flow with curvature.

II. Introduction

An important type of flow passage component found in many flow systems is the 90° bend or elbow. The flow through these bends has been studied extensively to understand how the radius of curvature affects the flow, pressure distribution, and losses through the bend. This can be used to predict flow system performance or to calculate the flowrate from the measured pressure distribution and a calibration curve. More generally, studying this flow will aid in our understanding of the variation in velocity and pressure throughout flows with curvature.
Experiment

Figure 3-2 shows a schematic of the rectangular bend used for this experiment and the location of static pressure taps. There are ten pressure taps on the inside of the bend and ten on the outside. These pressure taps will allow the measurement of the static pressure variation both along the direction of flow (θ-direction in the bend) as well as between the inner and outer surface. Finally, there are nine pressure taps located in the radial or r-direction in the middle of the bend (θ = 45°) to measure the radial pressure distribution.

![Figure 3-2 Rectangular bend schematic.](image)

The average velocity, \( \bar{V} \), in the rectangular duct is determined by measuring the static pressure drop across the inlet contraction shown in Figure 3-3 and by using a simple model. The relationship used for this calculation is developed by applying the energy equation for incompressible, steady flow to a control volume with a single inlet and exit

\[
\left( \frac{p_1}{\rho g} + \alpha_1 \frac{\bar{V}_1^2}{2g} + z_1 \right) - \left( \frac{p_2}{\rho g} + \alpha_2 \frac{\bar{V}_2^2}{2g} + z_2 \right) = h_L
\]

(3-1)

where \( p \) is pressure, \( \rho \) is density, \( g \) is acceleration due to gravity, \( \alpha \) is the kinetic energy coefficient, \( z \) is elevation, \( h_L \) is head loss term, and subscripts 1 and 2 indicate the inlet and exit, respectively. Our control volume is defined as the air moving through the airflow box and contraction. Assuming the velocity in the airflow box is negligible compared to the velocity at the contraction exit (
\( \bar{V}_1 \ll \bar{V}_2 \), uniform flow at the contraction exit \((\alpha_2 = 1)\), constant elevation \((z_1 = z_2)\), and negligible head losses \((h_L = 0)\) this reduces to

\[
\frac{p_{box}}{g \rho_{air}} - \left( \frac{p_{inlet}}{g \rho_{air}} + \frac{\bar{V}^2}{2g} \right) = 0
\]

which can be solved for the average velocity as

\[
\bar{V} = \sqrt{\frac{2(p_{box} - p_{inlet})}{\rho_{air}}}.
\]

**Figure 3-3** Pressure tap locations to measure average velocity.

**Modeling**

Modeling the flow through a 90° bend can be very difficult because the actual viscous flow can include complicated multi-dimensional effects such as secondary flow and flow separation. However, for our simple geometry (rectangular cross-section) and conditions, the flow can be approximately modeled as inviscid and incompressible. These are reasonable assumptions for high Reynolds number and Mach number below 0.3 (Ref. 1). Finally, we assume the flow is two-dimensional. Thus, velocity and pressure will only vary in the plane shown in Figure 3-2 and there is no velocity component or change in pressure across the duct.

For the first part of our model, we solve for the two-dimensional velocity field, \( \mathbf{V} = u \hat{i} + v \hat{j} \) for Cartesian coordinates or \( \mathbf{V} = V_r \hat{e}_r + V_\theta \hat{e}_\theta \) for cylindrical coordinates, using the stream function, \( \psi \), which satisfies conservation of mass for an incompressible fluid \( (\nabla \cdot \mathbf{V} = 0) \) and is defined as
for Cartesian coordinates where \( u \) and \( v \) are the \( x \) and \( y \) components of velocity, respectively, or

\[
V_r = \frac{1}{r} \frac{\partial \psi}{\partial \theta}, \quad V_\theta = -\frac{\partial \psi}{\partial r}
\]  

(3-5)

for cylindrical coordinates. If we assume uniform flow at the inlet, the flow is also irrotational (or without rotation). Surprisingly, the flow must remain irrotational through the bend for inviscid flow because there are no shear stresses which are required to get rotation started. Thus, the vorticity (or twice the rate of fluid rotation) is also zero. For two-dimensional flow there can only be rotation around the \( z \)-axis. For the \( z \)-component of vorticity we get

\[
\zeta_z = \frac{\partial u}{\partial y} - \frac{\partial v}{\partial x} = 0
\]

or

\[
\zeta_z = \frac{1}{r} \frac{\partial}{\partial r} \left( r V_\theta \right) - \frac{1}{r} \frac{\partial V_r}{\partial \theta} = 0
\]

(3-6)

For both coordinate systems, if we substitute our definition for stream function into these equations for vorticity we get the following

\[
\nabla^2 \psi = 0
\]

(3-7)

where \( \nabla^2 \) is the Laplace operator. This equation is a linear, second order, partial differential equation commonly referred to as Laplace's equation. We can solve this equation, along with the appropriate boundary conditions, for the stream function and then use it with Equation 3-4 or 3-5 to find the corresponding velocity distribution.

For the second part of our model, we solve for the pressure distribution by substituting \( \mathbf{V} \) into Euler's equation of motion which represents Newton's second law applied to a differential control volume for inviscid and incompressible flow

\[
\rho \mathbf{g} - \nabla p = \rho \left[ \frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} \right]
\]

(3-8)

where \( t \) is time. For steady flow through the bend and assuming negligible effects due to gravity this reduces to

\[-\nabla p = \rho (\mathbf{V} \cdot \nabla) \mathbf{V} \]

(3-9)

Thus, after substituting in our known \( \mathbf{V} \), we can integrate this equation to determine the pressure variations throughout the flow as a result of changes in fluid momentum.
Two-dimensional Model

We can solve Equations 3-7 and 3-9 for the domain shown in Figure 3-4 to determine the two-dimensional velocity and pressure distribution through the bend. For the boundary conditions at the inlet and outlet we assume uniform flow, thus the streamlines are normal to these boundaries ($\nabla \psi \cdot \hat{n} = 0$). For the boundary conditions at the inner and outer walls we use constant values of $\psi$. The value for the inner wall is arbitrarily set to zero, $\psi_1 = 0$. The value for the outer wall is calculated from the average velocity where $(\psi_2 - \psi_1) = \bar{V}(r_2 - r_1)$. These equations have been solved numerically for this domain using the PDE (Partial Differential Equation) Toolbox in MATLAB® (Ref 2). The streamlines through the bend and pressure contours are shown in Figure 3-5. In addition, results for the dimensionless pressure distribution through the bend are given in Table 3-1 where $p_1$ and $p_2$ are the pressure at the inner and outer wall, respectively, and $\xi$ is the distance along the inner wall (where $\xi = 0$ is at the beginning of the bend).

![Figure 3-4 Domain for two-dimensional flow through 90° bend.](image)

![Figure 3-5 Streamlines and normalized pressure distribution for inviscid incompressible flow in 90° bend.](image)
Table 3-1 Normalized pressure distribution for inviscid incompressible flow in 90° bend.

<table>
<thead>
<tr>
<th>TAP</th>
<th>$\xi/r_1$</th>
<th>$\frac{p_2 - p_1}{\sqrt{2 \rho V^2}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-1.00</td>
<td>0.001</td>
</tr>
<tr>
<td></td>
<td>-0.75</td>
<td>0.005</td>
</tr>
<tr>
<td>2</td>
<td>-0.50</td>
<td>0.024</td>
</tr>
<tr>
<td></td>
<td>-0.25</td>
<td>0.116</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.709</td>
</tr>
<tr>
<td></td>
<td>$\pi/16$</td>
<td>1.219</td>
</tr>
<tr>
<td>4</td>
<td>$\pi/8$</td>
<td>1.390</td>
</tr>
<tr>
<td></td>
<td>$3\pi/16$</td>
<td>1.456</td>
</tr>
<tr>
<td>5</td>
<td>$\pi/4$</td>
<td>1.472</td>
</tr>
<tr>
<td></td>
<td>$5\pi/16$</td>
<td>1.456</td>
</tr>
<tr>
<td>6</td>
<td>$3\pi/8$</td>
<td>1.390</td>
</tr>
<tr>
<td></td>
<td>$7\pi/16$</td>
<td>1.219</td>
</tr>
<tr>
<td>7</td>
<td>$\pi/2$</td>
<td>0.709</td>
</tr>
<tr>
<td></td>
<td>$\pi/2 + 0.25$</td>
<td>0.116</td>
</tr>
<tr>
<td>8</td>
<td>$\pi/2 + 0.50$</td>
<td>0.024</td>
</tr>
<tr>
<td></td>
<td>$\pi/2 + 0.75$</td>
<td>0.005</td>
</tr>
<tr>
<td>9</td>
<td>$\pi/2 + 1.00$</td>
<td>0.001</td>
</tr>
<tr>
<td>10</td>
<td>$\pi/2 + 1.50$</td>
<td>0.000</td>
</tr>
</tbody>
</table>

One-dimensional Model

Alternatively, from inspection of Figure 3-5, we can model the flow just through the bend as one quarter of a free vortex because the streamlines are concentric circles. This model will allow us to obtain an analytical solution for the pressure distribution in the radial direction half way through the bend (at $\theta = 45^\circ$) to compare to our experimental measurements. To find the velocity distribution we use Equation 3-6 for cylindrical coordinates and irrotational flow

$$\frac{1}{r} \frac{\partial}{\partial r} (r V_\theta) - \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} = 0$$  \hspace{1cm} (3-10)

For streamlines that are concentric circles there is no flow in the radial direction (or $V_r = 0$), thus Equation 3-10 reduces to

$$\frac{1}{r} \frac{d}{dr} (r V_\theta) = 0$$  \hspace{1cm} (3-11)

and integrate to obtain

$$V_\theta = \frac{K}{r}$$  \hspace{1cm} (3-12)
where $K$ is the integration constant called the vortex strength. This result shows that for this model the flow is now one-dimensional and the fluid velocity decreases as $r$ increases. This is opposite to rigid body rotation! This decreasing velocity outward from the center of curvature is similar to the fluid behavior outside the eye of a tornado or hurricane. Next, we will determine $K$ in terms of the average velocity defined as

$$
\bar{V} = \frac{1}{\rho A} \int \rho \mathbf{V} \cdot \hat{n} \, dA
$$

(3-13)

which for incompressible duct flow reduces to

$$
\bar{V} = \frac{1}{b(r_2 - r_1)} \int_0^b \int_{r_1}^{r_2} V_\theta \, dr \, dz
$$

(3-14)

where $b$ is the width of the duct. Substituting in Equation 3-12 we get

$$
\bar{V} = \frac{K \ln(r_2/r_1)}{r_2 - r_1}
$$

(3-15)

or solving for $K$ we get

$$
K = \frac{\bar{V} (r_2 - r_1)}{\ln(r_2/r_1)}.
$$

(3-16)

Substituting $K$ into Equation 3-12 and dividing by $\bar{V}$ we get the dimensionless velocity profile

$$
\frac{V_\theta}{\bar{V}} = \frac{r_2 - r_1}{\ln(r_2/r_1)} \frac{1}{r}.
$$

(3-17)

Thus, the normalized velocity is only a function of the geometry of the duct and the radial position.

Next, we use Euler's equation to find the pressure distribution. For cylindrical coordinates and two-dimensional flow, Equation 3-9 for the $r$ and $\theta$ directions becomes

$$
-\frac{\partial p}{\partial r} = \rho \left( V_r \frac{\partial V_r}{\partial r} + \frac{V_\theta}{r} \frac{\partial V_r}{\partial \theta} - \frac{V_\theta^2}{r} \right)
$$

(3-18)

$$
-\frac{\partial p}{\partial \theta} = \rho \left( V_r \frac{\partial V_\theta}{\partial r} + \frac{V_\theta}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{V_r V_\theta}{r} \right).
$$

(3-19)

Equation 3-19 reduces to $\partial p/\partial \theta = 0$, thus the pressure does not change along each streamline and is only a function of $r$. Using this result and Equation 3-12, Equation 3-18 reduces to

$$
3-7$$
For the free vortex, Euler's equation in the radial direction (normal to the streamlines) shows that there is a pressure variation due to the centripetal acceleration caused by the curvature of the streamlines. Separating variables we get

\[
\int_{p_1}^{p} dp = \rho K^2 \int_{r_1}^{r} \frac{dr}{r^3}
\]

where \( p_1 \), the pressure at \( r_1 \), is used in the definite integral. Integrating we get

\[
p - p_1 = -\frac{1}{2} \rho K^2 \left( \frac{1}{r^2} - \frac{1}{r_1^2} \right).
\]

Equation 3-22 indicates that the pressure increases as \( r \) increases. Thus, in the direction outward from the center of curvature of the streamlines the pressure increases as the velocity decreases. Substituting in Equation 3-16 to eliminate \( K \) we get

\[
p - p_1 = -\frac{1}{2} \rho \frac{V^2 (r_2 - r_1)^2}{\ln \left( \frac{r_2}{r_1} \right)} \left( \frac{1}{r^2} - \frac{1}{r_1^2} \right).
\]

Equation 3-23 is non-dimensionalized by dividing both sides by the dynamic pressure, \( 1/2 \rho V^2 \), to get

\[
\frac{p - p_1}{1/2 \rho V^2} = \frac{(r_2 - r_1)^2}{\ln \left( \frac{r_2}{r_1} \right)} \left( \frac{1}{r^2} - \frac{1}{r_1^2} \right).
\]

Thus, the normalized pressure difference across the bend is only a function of geometry.

Three-Dimensional Viscous Flow

The inviscid and two-dimensional flow model given above will provide reasonable agreement with the experimental pressure measurements for the flow conditions tested. However, the actual flow is both viscous and three-dimensional which contribute to differences between the data and the model predictions. First, the model predicts that there will be no pressure drop from the inlet (pressure at Tap 1) to the outlet (at atmospheric pressure). However, there will be a measurable loss in pressure through the flow passage due to wall shear stresses resulting from fluid viscosity and turbulent mixing. In addition, there may be significant pressure losses due to flow separation and secondary flow depending on the flow passage geometry and Reynolds number. Second, the model predicts that the flow velocity will be uniform at the outlet. Viscous effects, flow separation, and secondary flow can all contribute to a non-uniform velocity distribution after the bend.
Flow separation may occur for a turning internal flow if the boundary layer separates from the inner surface of the bend as shown in Figure 3.6. A boundary layer is a thin fluid region near a surface where the velocity goes from the free stream value to the surface velocity due to the no-slip condition. Along the inner surface of the bend the pressure is initially decreasing \((dp/d\theta < 0)\). As a result there is a net pressure force that pushes the boundary layer against the surface and keeps it attached, thus this is called a favorable pressure gradient. After the midway point through the bend (45° for this experiment) the pressure begins to increase \((dp/d\theta > 0)\). The net pressure force now pushes the boundary layer away from the surface. If the net pressure force exceeds the force due to the surface shear stresses the boundary layer will separate and a recirculation region will form just downstream of the separation point as shown. The flow is more likely to separate for an abrupt geometry such as a short radius of curvature.

![Flow separation schematic](image)

**Figure 3-6** Flow separation schematic for a rectangular duct with a 90° bend.

Secondary flow is defined as any fluid motion not in the main flow direction. Secondary flow for internal turning flows occurs when the high-pressure fluid on the outside of the bend moves towards the region of low-pressure fluid on the inside of the bend along the side walls resulting in the recirculation cells shown schematically in Figure 3.7. For typical engineering conditions, secondary flows begin to appear for a bend angle, \(\theta_b\), of approximately 20° (Ref. 3). They continue to grow stronger until about \(\theta_b = 80°\) and then begin to diminish in strength. They are very weak by \(\theta_b = 130°\) due to a reduction in centrifugal and pressure forces that eventually stops the secondary flow. Strong secondary flows also serve to reduce the magnitude of the adverse pressure gradient after the bend which can prevent flow separation.
The pressure drop through the bend resulting from viscous effects, flow separation, and/or secondary flow can be given in terms of a loss coefficient, $K$, defined as

$$K = \frac{P_{\text{inlet}} - P_{\text{out}}}{\frac{1}{2} \rho \bar{V}^2}$$  \hspace{1cm} (3-25)$$

Figures 3-8 through 3-10 show how $K$ varies with flow geometry and with Reynolds number, $Re$, for a rectangular duct with ideal inlet and exit conditions (Ref. 3). In these figures, the geometry is defined as follows: $b$ is the duct width, $W = r_2 - r_1$, and $R = (r_1 + r_2) / 2$. The Reynolds number is defined as $Re = \left( \rho_{\text{air}} \bar{V} D_h \right) / \mu_{\text{air}}$ where $D_h = 4 A_c / P$ is the hydraulic diameter, $A_c = b W$ is the cross-sectional area, and $P = 2 \left( b + W \right)$ is the perimeter. First, Figure 3-8 shows that $K$ increases with $\theta_h$. This increase is mainly due to an increase in the strength of the secondary flow initially, but then levels out after 130° due to its diminishing effects. Second, Figure 3-9 shows that $K$ initially decreases sharply with $R/W$ (or radius of curvature of the bend with fixed $W$) and then increases slightly. The sharp decrease is mainly due to the elimination of flow separation for a gentler bend. Minimal pressure losses are found for $R/W$ ranging from approximately 1.5 to 2. The slight increase in $K$ as $R/W$ continues to increase is due to the increased length of the passage resulting in greater viscous losses. Third, Figure 3-10 shows that $K$ initially decreases and then levels off with $Re$. This is similar to the trend found for the friction factor on the Moody diagram.
Figure 3-8 Loss coefficient versus bend angle for a rectangular duct (Ref. 3).

Figure 3-9 Loss coefficient versus dimensionless bend radius for a rectangular duct (Ref. 3).

Figure 3-10 Loss coefficient versus Reynolds number for a rectangular duct (Ref. 3).
III. Experiment

Equipment

TecQuipment AF10 Airflow Bench
AF15 Flow Around a Bend Apparatus, with:
   Inside radius $r_1 = 50\text{mm}$
   Outside radius $r_2 = 100\text{mm}$
   Width $b = 100\text{mm}$
AF10A Multi-tube Manometer
Pitot-static tube with inclined manometer

Procedure

1. Measure and record the temperature and pressure in the room.

2. Disconnect all tubes from the multi-tube manometer (Figure 3-11). Verify the angle of the tubes is at 60° from the vertical. Raise the reservoir on the multi-tube manometer until the fluid levels reach approximately halfway up the tubes.

![Figure 3-11 Multi-tube manometer.](image)

3. With all the manometer tubes open to the atmosphere, level the multi-tube manometer by adjusting the leveling feet at the bottom. The manometer is level if the water heights in the tubes are equal.

4. Connect pressure taps 1-10 from the OUTER radius of the bend to tubes 2-11 on the manometer (LEAVE THE FIRST AND LAST MANOMETER TUBES OPEN TO THE ATMOSPHERE).
5. UNLOCK and close the damper (fully counterclockwise) on the airflow bench and then turn the fan on (Figure 3-12). SLOWLY AND CAREFULLY open the damper, while watching to make sure the water levels in the manometer do not go over the top or below zero. Use the portable Pitot-static tube connected to the oil manometer (Figure 3-13), to set the free-stream velocity as determined by the instructor. When the flow is at the desired velocity, lock the damper open. DO NOT EVER ADJUST THE DAMPER WITH THE LOCK ON! You may have to adjust the reservoir height in order to control the liquid levels in the manometer.

![Image](image_url)

**Figure 3-12** Wind tunnel damper (shown fully open).

6. Measure the liquid levels in the manometers. Also record the height of the water in the open tubes; the relative heights of the water columns (i.e. gage pressures) will be used in this experiment. Disconnect the tubes when done.

7. Convert the measured vertical height difference, \( h \), between the tube and open tube to differential pressure, \( \Delta p \), using the hydrostatic equation, \( \Delta p = \rho_{\text{water}} \ g \ h \), where \( \rho_{\text{water}} \) is the density of the manometer fluid (in this case water). Since the manometer is inclined at 60°, the vertical height is one-half the measured difference, therefore, \( h = 0.5 \left( h_{\text{tube}} - h_{\text{open tube}} \right) \).
8. Repeat steps 4-7 for the inner surface taps. Be careful to record the open tube reading as well. This value changes every time you change the pressure taps.

9. Repeat steps 4-7 for the side surface taps. When connecting these tubes, reconnect the tube from the inside bend tap 5, and the outer bend tap 5. These represent the pressures at \( r = 50\text{mm} \) and \( r = 100\text{mm} \), respectively. Again, be sure to record the open tube reading as well.

10. Measure the pressures at the air box and the inlet, Figure 3-3. These two pressure taps will be used to determine the average velocity, \( \bar{V} \).

11. Examine the velocity profile of the exiting flow by inserting the Pitot-static probe (Figure 3-13) into the flow. Traverse the flow along the centerline of the flow; first side-to-side, then bottom-to-top. You need not record values of velocity, merely qualitatively observe how the velocity varies. However, you may want to record a reading of the Pitot tube near the centerline, to verify your calculated average velocity in the duct.

**IV. Report**

1. Use the air box and inlet pressures and room air conditions to calculate the average velocity, \( \bar{V} \), using Equation 3-3.
2. Tabulate the pressure tap number, raw manometer measurements (do not forget to correct pressure data for the inclined slope), and calculated gage pressures (in Pa) for each set of inside and outside taps.

3. Plot gage pressure (in Pa) versus tap number for each set of inside and outside taps. Briefly explain if the trends in the experimental data make sense. Why are the profiles asymmetric when comparing the inlet flow to the outlet flow?

4. Estimate the pressure drop from the bend inlet to outlet using $K$ from Figure 3-10 and Equation 3-25. Also, estimate the pressure drop for a straight duct of approximately the same length. Include sample calculations with your results. Compare these estimates to your experimental measurement of pressure drop. Does the magnitude of the pressure drop for the bend seem reasonable? What causes this pressure drop?

5. Tabulate the pressure tap number, normalized distance along the inner wall, $\xi/r_1$, and the experimental and two-dimensional model normalized pressure differences, $(p_2 - p_1)/\left(1/2 \rho V^2\right)$, for each set of inside and outside taps.

6. Plot normalized pressure difference versus normalized distance along the inner wall using the experimental data (points and no line) and two-dimensional model (line with no symbols). Briefly explain if the trends in the experimental data make sense. Comment on the agreement between the experimental data and the model. Why is there a pressure difference between the inside and outside surfaces of the bend?

7. Tabulate the pressure tap number, radial location, raw manometer measurements, and calculated gage pressures (in Pa) for each side tap (in the radial direction).

8. Tabulate the pressure tap number, normalized radial location, $(r-r_1)/(r_2-r_1)$, the experimental and one-dimensional model normalized pressure differences, $(p-p_1)/\left(1/2 \rho V^2\right)$, for each side tap. Include sample calculations with your table.

9. Calculate the flowrate using the measured pressure difference between the inner and outer pressure taps and the one-dimensional model. Compare this value with the flowrate calculated in step 1. Comment on using this technique to determine flowrate.

10. Plot normalized pressure difference versus normalized radial location using the experimental data (points and no line) and one-dimensional model (line with no symbols). Briefly explain if the trends in the experimental data make sense. Comment on the agreement between the experimental data and the model. What would you expect the plot for the velocity to look like for the bend?

11. Based on your observations from the Pitot-static tube measurements at the duct outlet, do you think there is flow separation and/or secondary flow? How would this affect your results for your plots above? What additional measurements could you make to verify if the flow has separated at if there is a secondary flow.
V. References


I. Objectives

The objectives of this experiment are to (1) measure flowrate through a tube with a well rounded entrance as a function of liquid height above the entrance, (2) compare the measured flowrate to that predicted by a simple model, and (3) investigate the transition from turbulent to laminar flow.

II. Introduction

The transport of fluids in pipes is a fundamental application for fluid mechanics. Examples surround us everywhere such as water supply systems, oil pipelines, and the flow of blood through our bodies. A course in fluid mechanics would be incomplete without their consideration. For this lab we will restrict ourselves to simple incompressible flow of a viscous fluid in a straight pipe. In addition, we will consider an important classification for these flows (laminar versus turbulent) that has a significant impact on the nature of these flows.
Experiment

In this experiment, you will examine the effect of tank height, $h$, on the flowrate, $Q$, through a tube with a rounded entrance as shown in Figure 4-1. To vary $Q$, $h$ will simply be maintained at different heights using overflow tubes and then measured. The flowrate will be measured experimentally using a bucket, digital scale, and stopwatch. Figure 4-2 shows a picture of a bucket being weighed on the digital scale. The bucket will be inserted into the flow stream discharging from the tube. The time will be recorded as the bucket is filling with water; therefore, an experimental measurement of the mass flowrate can be calculated as follows

$$\dot{m} = \frac{m}{t} = \frac{W}{g}$$  \hspace{1cm} (4-1)

where $m$ is mass of the water, $t$ is time, $W$ is weight of the water, and $g$ is the gravitational acceleration. You will want to calculate $\dot{m}$ in $\text{slug/s}$ so you will need to use the conversion factor $1 \text{ lb}_t = 1 \text{ slug} \cdot \text{ft/s}^2$. Dividing $\dot{m}$ by the density of water gives the volumetric flowrate

$$Q = \frac{\dot{m}}{\rho} = \frac{W}{\rho g t}.$$  \hspace{1cm} (4-2)

![Figure 4-2 Bucket on scale.](image-url)
Modeling

The relationship used to model this internal flow is developed by applying the energy equation for incompressible, steady flow to a control volume with a single inlet and exit

\[
\left( \frac{p_1}{\rho g} + \alpha_1 \frac{\overline{V}_1^2}{2g} + z_1 \right) - \left( \frac{p_2}{\rho g} + \alpha_2 \frac{\overline{V}_2^2}{2g} + z_2 \right) = h_L
\]

(4-3)

where \( p \) is pressure, \( \rho \) is density, \( g \) is acceleration due to gravity, \( \alpha \) is the kinetic energy coefficient, \( \overline{V} \) is average velocity, \( z \) is elevation, \( h_L \) is the head loss term, and subscripts 1 and 2 indicate pipe inlet and exit, respectively. For our model, the control volume is chosen to include all of the liquid in the tank and tube. Thus, the inlet control surface (section 1) is the top surface of the water and the outlet control surface (section 2) is at the exit of the pipe as shown in Figure 4-3. Initially in the entrance region of the tube, fluid boundary layers due to the no-slip condition are growing on the inside surface of the tube (Ref. 1). As the flow proceeds down the tube, these boundary layers grow and eventually merge until the velocity profile remains the same as you go further downstream. This is called fully developed flow. The tube length, \( \ell \), for this experiment is sufficiently large to guarantee fully developed flow over a majority of the pipe. As a result, we will assume fully developed flow for the entire pipe and that the error due to the entrance region is small.

![Figure 4-3 Schematic of tank draining experiment.](image)

Next, we consider how to simplify Equation 4-3 for our control volume. For the pressure terms, since both control surfaces are at atmospheric pressure (\( p_1 = p_2 = p_{atm} \)), the pressure terms in Equation 4-3 cancel. For the kinetic energy terms, the surface area of control surface 1 is much larger than the surface area of control surface 2, thus by conservation of mass \( \overline{V}_1 << \overline{V}_2 \) and \( \overline{V}_1 \) can be neglected. For the kinetic energy coefficient, recall that for fully developed pipe flow \( \alpha = 1 \) for turbulent flow and \( \alpha = 2 \) for laminar flow. For the elevation head terms, the \( z \)-axis is defined as
the vertical distance upwards from the midpoint of section 2, thus \( z_1 = h \) and \( z_2 = 0 \). Finally, the head loss term includes both major head loss, \( h_{L,\text{major}} \), and minor head losses, \( h_{L,\text{minor}} \).

Major head loss is due to friction as the fluid passes through the straight pipe. The no-slip condition requires zero fluid velocity at the pipe wall. As a result a velocity gradient is created where the fluid varies from zero to the maximum velocity in the middle of the pipe. This velocity gradient and friction results in shear stresses in the flow. Thus, \( h_{L,\text{major}} \) represents the energy expended by the fluid to overcome this shear stress. The major head loss is typically expressed with the Darcy friction factor, \( f \), as

\[
h_{L,\text{major}} = f \frac{\ell \bar{V}^2}{D 2g} .
\]

where \( \ell \) is the length of the pipe and \( D \) is the diameter of the pipe. The friction factor can be calculated using the Blasius correlation, Colebrook correlation, or read from the Moody Diagram (a graphical representation of these correlations found in Figure 8.20 in Ref. 1). In general, the friction factor is a function of the Reynolds number, for our pipe flow defined as

\[
Re_D = \frac{\rho \bar{V} D}{\mu} ,
\]

and pipe roughness, \( \varepsilon \), where for the drawn tubing used in our experiment \( \varepsilon = 5 \times 10^{-6} \) ft (Ref. 2). Note that because \( f \) depends on \( Q = \bar{V} A \) where \( A \) is the cross-sectional area of the pipe, to calculate flowrate you must guess a friction factor and then iterate to find a solution. For an initial guess for your model, \( f \) can be estimated using the experimental flowrate measurement. To iterate, calculate a new \( f \) from your calculated \( Q \) and continue until convergence (\( f \) and \( Q \) remain the same).

Minor head losses are those due to various pipe system components such as entrances, fittings, and valves. For our control volume the only minor loss that will be considered is the entrance loss between the tank and pipe. Figure 8.24 in Ref. 1 presents the entrance loss in terms of a loss coefficient, \( K_L \), where the minor head loss is given in terms of the kinetic energy in the tube as

\[
h_{L,\text{minor}} = K_L \frac{\bar{V}^2}{2g} .
\]

With all of these substitutions, Equation 4-3 simplifies to

\[
h - \frac{\alpha}{2} \frac{\bar{V}^2}{2g} = f \frac{\ell \bar{V}^2}{D 2g} + K_L \frac{\bar{V}^2}{2g}
\]

which can be solved for the volumetric flowrate in the tube.
Uncertainty Analysis

As part of the data analysis, an uncertainty analysis will be conducted for both the experimental and model volumetric flowrates.

For the experimental flowrate, assume that all measurements have negligible uncertainty except for the weight of water and the time measurement. Using the Uncertainty Propagation for Multiplication technique from Appendix B and Equation B-15, the uncertainty in the experimental flowrate is found from Equation 4-2 as

\[
\frac{u_Q}{Q} = \sqrt{\left(\frac{u_W}{W}\right)^2 + \left(\frac{u_t}{t}\right)^2}
\]

where \( u \) are the uncertainties in each of the corresponding measured quantities.

For the flowrate predicted using our model, assume that all variables have negligible uncertainty except for height of the water in the tank, friction factor (assume 10% fractional uncertainty), and entrance loss coefficient (assume 70% uncertainty). Using the General Uncertainty Propagation technique from Appendix B and Equations B-10 and B-11, the uncertainty in the flowrate predicted using our model is found as

\[
u_o = \sqrt{\left(\frac{\partial Q}{\partial h} u_h\right)^2 + \left(\frac{\partial Q}{\partial f} u_f\right)^2 + \left(\frac{\partial Q}{\partial K} u_K\right)^2}.
\]

To evaluate the partial derivatives, we substitute in \( Q \) from Equation 4-8. For the height of the water term we get

\[
\frac{\partial Q}{\partial h} = \sqrt{\frac{2g}{\alpha_2 + K_L + f(\ell/D)\left(\frac{\pi}{4} D^2\right)\frac{1}{2} h^{-1/2}}} = \frac{1}{2} \frac{Q}{h}.
\]

Evaluation of the remaining derivatives is left for you to complete as part of the lab.
III. Experiment

Equipment

- Rectangular tank with drain tube attached and sufficient water
- Ruler to measure height of water in tank
- Digital scale, rack, and bucket to weigh water collected
- Thermometer to record water temperature

Procedure

1. Using the digital scale, weigh the empty bucket (Figure 4-2). This weight must be subtracted from all weight measurements to calculate the water weight. Record the uncertainty of the digital scale.

2. Record entrance radius of curvature from the front of the tank (Figure 4-4).

3. Record tube inside diameter and tube length from the side of the tank. Estimate the uncertainty in both of these measurements.

4. Shut the drain valve. Shut all three overflow valves (Figure 4-5). Open the bypass valve in the lower tank or sump (Figure 4-6) and turn on pump to rapidly fill the upper tank. When the water is at the level of standpipe #14, shut the bypass valve. *DO NOT record data with the bypass valve open! DO leave the pump on!* When the level in the tank is steady, proceed to the next step.
5. Open the outlet of the tube with the rounded inlet (this is the only inlet condition you will use for this lab). Place the bucket in the discharge stream and record the initial time. Collect water in the bucket until it is almost full (about 4 minutes). The more water that is collected, the more accurate the flowrate measurement. While the water is collected, record the height of water in the tank using the ruler attached to the tank. Note that the zero mark on the ruler corresponds to the center of the tube. Record the uncertainty of the height and time measurements. Remove the bucket from the water stream and record the final time. Weigh the bucket and water using the digital scale. Subtract the weight of the empty bucket from the total weight and record the water weight. Record the temperature of the water in the bucket. Pour the water back into the sump.

6. Open the next lower overflow valve (Figure 4-7). Let the water in the tank reach a new steady height. Repeat step 5 for this new flowrate.
7. Repeat step 5 for the remaining two heights. There are a total of four flowrates that will be measured with this experimental setup.

8. After recording the last flowrate, shut the pump off and allow the tank to drain through the rounded inlet tube (DO NOT open the tank drain valve). During this final draining, record the range of heights of the fluid when transition occurs. Remember this is a transition from turbulent to laminar flow. Be sure to observe the water draining from the tube end and note the physical difference between the turbulent and laminar flow stream.

IV. Report

1. Calculate and tabulate the experimental data along with the recorded uncertainties. The table should include columns for height (in ft), weight, time, and calculated flowrate (in gpm). Include sample calculations with your table.

2. Calculate and tabulate the model predictions. The table should include columns for height (in ft), Reynolds number, friction factor, and flowrate (in gpm) obtained after the iteration has converged. Include sample calculations with your table.

3. Calculate and tabulate the percent uncertainty in the experimental flowrate and the flowrate predicted by the model for all four flowrates investigated. Include sample calculations with your table. What measurement contributes the most to the uncertainty in the experimental flowrate? What measurement contributes the most to the uncertainty in the flowrate predicted by your model.

4. Plot flowrate (gpm) versus tank height (ft) using your experimental flowrate (symbols and no lines) and the calculated flowrate predicted by our model (line with no symbols). Do the trends in this plot make sense? Why does the flowrate increase or decrease with water level height?
5. Add error bars (using the results of your uncertainty analysis) to the figure from step 4 for both the experimental and model data. Compare the difference between the experimental and theoretical flowrates. Based on the error bars, what is your conclusion about the agreement between the model and experiment?

6. Estimate the range of Reynolds numbers over which transition occurs between turbulent and laminar flow. Compare your measured transition Reynolds number to the expected transition Reynolds number of 2300. Explain why they are different or the same.

7. What is the physical difference between laminar and turbulent flow? Use your observations from the experiment to help answer this question.

8. What is the physical significance of the kinetic energy coefficient, $\alpha$? Based on your observations for turbulent and laminar flow, can you justify the values of $\alpha$ used for each flow regime.

V. References


I. Objectives

The objectives of this experiment are to design and conduct an experiment to measure the friction factor and minor losses in a ventilation system.

II. Introduction

As a practicing engineer you will be asked to design an experiment, to conduct an experiment, and to compare the results of an experiment to those predicted by an analytical model. Therefore, this experiment is different than the other experiments because you are being asked to develop and to perform an experiment. This assignment will develop your skills in experimental design by measuring the performance of the ventilation system shown in Figure 5-1. This experiment is a continuation of the axial fan performance test, section 7; therefore, you will need the results and procedure from that experiment to help you adapt the apparatus for your new measurements.

The ventilation system illustrated in Figure 5-2 is a typical system used in industrial or commercial applications. As a matter of fact, you should see similarities between the experimental...
apparatus and the actual ventilation system in the lab. The fan, ducting, and flow control dampers are typical for a building of this size and this system was assembled from components used for a building ventilation system. The objectives for this investigation are to determine the friction factor of the galvanized metal ducting (roughness of 0.0003 ft) and the pressure drops associated with the following minor losses: 1) a Y-fitting, 2) 180° bend, and 3) the VAV damper when fully open.

To begin, let us discuss the requirements of a properly designed experimental investigation. According to Holman [1], the requirements for an experiment can be summarized as follows:

III. Experiment

Step 1: Establish the Objective for the Experiment

An experiment is one of the most effective techniques to verify a design, model, or prototype. Experiments allow for measuring the performance of a system or component, verifying some physical model or theory, and establishing a physical correlation between variables (i.e., developing a new model). Also, experiments on actual systems are a valuable diagnostic tool to determine performance degradations. When you work on your senior project, you will be required to design an experiment to test your prototype. This assignment will prepare you for your senior project testing.

Figure 5-2 Axial fan experimental apparatus.
For this investigation, the objectives are as follows:

1) Experimentally measure the friction factor to an accuracy of ±10% for the 16-in galvanized metal duct at four fan rotational speeds: 1750 rpm, 1575 rpm, 1400 rpm, and 1320 rpm.
2) Compare the experimental friction factor to the theoretical value.
3) Measure the total pressure drop in the system. Calculate and measure the individual pressure drops that make up the total pressure drop including:
   a. Pressure drop across the “Y” fitting
   b. Pressure drop in length of straight ducting
   c. Pressure drop in the 180° bend fitting
   d. Pressure drop across the VAV damper when fully open.

Step 2: Establish Background Knowledge for the Study

What is the underlying theory for this system or phenomenon? Are there established relationships between variables (example: radiation is addressed by the Stefan-Boltzmann Law)? Are there any parameters (like the Reynolds number) that would help reduce the number of variables that you have to measure? And most importantly, has your particular study been done before? Is an experiment therefore needed? While the experiment you are designing has been well-documented, this experience is still valuable in evaluating the quality of the experimental apparatus. Many times experimental investigators perform simple experiments to verify the validity of their apparatus.

For this experiment, you should be asking what theoretical background is necessary to satisfy the objectives. What parameters/equations describe frictional pressure drop in a duct? What are the assumptions behind this theory? You should have a physical understanding of the phenomenon you are trying to measure.

Step 3: Commit Resources to the Experiment

Experiments are costly. In industry, budgetary, manpower, and time requirements would have to be set and approved. Steps 1 and 2 must be carefully considered to justify the need for the experiment. For this investigation, time is a limited resource which must be budgeted carefully. You have limited access to the equipment and cannot perform a large number of experiments. The same will be true for your senior project. Testing will be performed in a short period, so careful planning is essential. Since you will be performing this experiment four weeks after performing the axial fan performance test, you are limited by time. Therefore good time management will be needed to complete this assignment.

Step 4: Plan the Experiment

In this step of the experimental design sequence, you will perform the theoretical analysis to determine the measured variables and the predicated uncertainty in these variable measurements. You are not designing the physical experiment yet – instead, you are establishing the measurements you will be taking. These include the following steps:
a. **Identify the primary variables** that must be measured (force, strain, flowrate, pressure, temperature, etc.). Establish any parameters (similarity variables like Reynolds number or drag coefficient) that would simplify the collection of data or reduce the amount of data collected.

b. **Establish the range of values** over which you will be measuring the primary variables. This requires setting up data reduction calculations to ensure that an adequate range of values are collected.

c. Perform an **uncertainty analysis** on the experiment. This includes establishing the desired accuracy in the calculated results, and then determining how accurate the primary measurements must be to satisfy the desired accuracy. This is an important step in experiment design, because you can identify which measurements must be the most accurate (and are likely the most costly). Another benefit of uncertainty analysis appears when you are verifying a theoretical model. If the actual data is off from the model by more than you expect, this discrepancy can be identified immediately if the uncertainty analysis is complete.

The predicted uncertainty will dictate the required instrument accuracy. Again your goal is to accomplish the objectives established in Step 1. Based on the design of the system shown schematically in Figure 5-2 and with data from your axial fan performance test, you need to:

1) Predict the friction factor that you expect in the duct,
2) Determine the experimental measurements needed to calculate the experimental friction factor,
3) Determine the minimum uncertainty needed in this measurement so the friction factor has an uncertainty of ±10%.
4) Determine how the minor losses will be measured/calculated.

![Figure 5-3 Differential pressure gages.](image-url)
As part of your theoretical analysis, let us review what is available in the current experimental apparatus. The ventilation system in Figure 5-2 is divided into two parallel 100-ft ducts both connected to the axial fan through a ‘Y’-fitting. At the end of the ‘Y’-fitting, 43-ft of straight ducting is attached followed by a 180° bend. An additional 41-ft of straight ducting is attached to the end of the 180° bend followed by the VAV damper. Prior to the 180° bend, an end cap is provided. By opening this end cap, the duct system length can be reduced (eliminates the 180° bend, the 41-ft of ducting, and the VAV damper).

Currently, nine Pitot-static tubes are installed in the ducting system. These are also indicated on Figure 5-2. Pitot-static tube #1 is located prior to the ‘Y’-fitting at the outlet of the fan. Pitot-static tubes 2, 4, 6, and 8 are located in the right duct, while Pitot-static tubes 3, 5, 7, and 9 are located in the left duct. The Pitot-static tubes are all connected to the differential pressure meters shown in Figure 5-3. Pitot-static tube #1 is connected to the center gage, labeled fan static pressure and currently only measures the static pressure at location 1. Pitot-static tubes 2, 4, 6, and 8 can be connected to the gage labeled right duct by opening the corresponding ball valves located beneath the gage as shown in Figure 5-4. For each Pitot-static tube there are two ball valves. For example, for Pitot-static tube #2, the ball valves are labeled 2TP and 2SP. 2TP is connected to the total pressure tap while 2SP is connected to the static pressure tap. When both ball valves are open, the differential pressure meter displays the difference between the total pressure and the static pressure otherwise known as the dynamic pressure (or velocity).

**Step 5: Design and Build the Experiment**

With your theoretical analysis complete, you should now know which variables are to be measured and to what accuracy you need that measurement. Is the current instrumentation on the experiment sufficient to obtain quality results? If not, what changes do you recommend? Do you need additional instrumentation? If so, what additional or different instruments do you need?
For this class you are actually not designing and building the apparatus, but you may need to make temporary modifications. Prior to designing your experiment, you must perform the analysis and determine the accuracy of instrumentation needed to accomplish the objectives. Building an experiment with no expectation of outcome will only lead to failure.

**Step 6: Run the Experiment**

At this point, a detailed experimental procedure should be written. This procedure should document how the data will be collected so accurate measurements can be obtained. Preliminary operation of the experimental apparatus to test your procedure and collect some preliminary data may be conducted. This data will be compared to the analysis already conducted and changes to the procedure or measurements should be implemented. With these changes implemented, conduct the experiment using the procedure. **Prior to running the experiment, show the instructor your data sheet and procedure to have it checked off. This should be done no later than the week prior to your scheduled lab.**

**Step 7: Analyze the Data**

Analyze the data. This includes all data reduction, plotting, and any other analysis that you perform in order to draw meaning from the experiment. Compare the data to the theoretical models. Discuss all observations and offer an explanation for these observations. This discussion should be based on quantitative measurements not conjecture. Only discuss what your experimental results present. Trends in the data need to be discussed after including uncertainty analysis. What are your conclusions?

**IV. Report**

**Step 8: Organize, Discuss, and Publish the Results**

The written report is a summary of Steps 1 through 7. Discuss in your introduction the objectives of the experimental investigation. Present your theoretical analysis. Discuss your experimental measurement procedure including uncertainty. Present your results. Compare your results to the theoretical analysis. Finally in your conclusion summarize the experiment and the conclusions. The report should be written in the same format as the other group reports.

**V. References**

I. Objectives

The objectives of this experiment are to (1) measure the boundary layer profile for flow over a flat plate using the apparatus shown in Figure 6-1 and use this data to calculate boundary layer thicknesses, drag, and drag coefficient, (2) estimate each of these quantities using a solution to the boundary layer equations for laminar flow and correlations for turbulent flow, and (3) compare the measured data to the values predicted by the models to determine if the flow is laminar, turbulent, or transitional for the conditions tested.

II. Introduction

Due to the no-slip condition, the fluid velocity relative to the surface must go from zero at the surface to the free-stream velocity, $U$, far away from the body. Thus, there is a thin fluid region near the solid surface with high velocity gradients called the boundary layer. Figure 6-2 shows a two-dimensional boundary layer for uniform flow (constant velocity parallel to the surface) over
a flat plate. The velocity gradients result in shear stresses inside the boundary layer while a nearly inviscid flow exists in the free-stream flow. These shear stresses determine the magnitude of the drag (force that opposes motion of a body in a fluid) due to friction acting on the body. Thus, understanding the nature of boundary layer flow is necessary for determining frictional drag and for designing bodies that minimize drag.

![Figure 6-2 Schematic of a boundary layer for uniform flow over a flat plate.](image)

Boundary layers are typically classified as either laminar, turbulent, or transitional. The type of boundary layer will have a significant impact on the frictional drag. For uniform flow over a flat plate with a smooth leading edge, a laminar boundary layer will form at the leading edge and continue along the plate in the streamwise direction until the Reynolds number, \( Re_x = \frac{U x}{\nu} \), reaches approximately \( 5 \times 10^5 \) where \( x \) is the distance from the leading edge as shown in Figure 6-2 and \( \nu \) is the kinematic viscosity. In practice, it is not always clear if a leading edge and plate are smooth enough to maintain laminar flow. Other factors, such as turbulence in the free-stream flow, three dimensional effects, or vibrations, can impact the transition from a laminar to turbulent boundary layer.

The boundary layer thickness is approximately zero at the leading edge and increases along the plate in the streamwise direction. The growth rate of a laminar boundary layer is less than that of a turbulent boundary layer. There are three ways typically used to define the thickness of the boundary layer. First, the boundary layer thickness, \( \delta \), is defined as the location where the boundary layer velocity is 99% of the free-stream velocity. Second, the displacement thickness, \( \delta^* \), is defined as the distance the plate would have to be displaced for the mass flow rate of a flow at the free-stream velocity to be the same as that for the actual boundary layer. For incompressible flow this can be calculated using

\[
\delta^* = \int_{0}^{\infty} \left(1 - \frac{u}{U}\right) dy
\]

(6-1)

where \( u(y) \) is the boundary layer velocity profile and \( y \) is the distance away the plate perpendicular to the streamwise direction as shown in Figure 6-2. Third, the momentum thickness, \( \Theta \), is defined as the distance the plate would have to be displaced for the momentum flux of a flow at the free-stream velocity to be the same as that for the actual boundary layer. For incompressible flow this can be calculated using

6-2
\[ \Theta = \int_0^\infty \frac{u}{U} \left( 1 - \frac{u}{U} \right) dy. \] (6-2)

The momentum thickness can be used to find the drag due to friction, \( F_D \), acting on the plate from the leading edge up to the location where \( \Theta \) is known by

\[ F_D = \rho \ b \ U^2 \Theta. \] (6-3)

where \( \rho \) is the fluid density and \( b \) is the width of the plate (perpendicular to the direction of the free-stream flow). This relationship was introduced by T. von Karman in 1921 (Ref. 1). It is derived by first applying the momentum equation for a finite control volume to an incompressible two-dimensional boundary layer resulting in the momentum integral boundary layer equation which is valid for both laminar and turbulent flows. Second, the definition of \( \Theta \) is used to obtain Equation 6-3. Thus, from our momentum balance we find that the drag force is equal to the loss of momentum in the boundary layer due to the no-slip condition at the solid surface. In terms of the dimensionless drag coefficient, \( C_D \), this becomes

\[ C_D = \frac{F_D}{\frac{1}{2} \rho \ b \ L \ U^2} = \frac{2 \Theta}{L} \] (6-4)

where \( L \) is the stream-wise distance from the leading edge to where \( \Theta \) is known.

**Experiment**

For this experiment, boundary layer flow at a measured location along a flat plate will be investigated using the apparatus shown in Figure 6-1. The flat plate has a smooth and a rough side. In addition, a roughness element can be added to intentionally trip the flow to turbulent. The wind tunnel cross section is constant to provide a nearly uniform free-stream velocity and insignificant free-stream pressure gradient. The velocity profile, \( u(y) \), shown in Figure 6-2 will be measured using a Pitot-tube for at least one specified free-stream velocity. The experimental data will then be used to estimate the boundary layer, displacement, and momentum thicknesses, drag coefficient, and total drag force.

A numerical integration technique must be used to solve Equations 6-1 and 6-2 for the displacement and momentum thicknesses from the discrete experimental data. The easiest method is to use the trapezoidal rule. This method will be illustrated using the following integral

\[ \int_{x_1}^{x_2} f(x) \, dx \quad \text{where} \quad f(x) = 3 - 4 \left( x - 2 \right)^2, \ x_1 = 1.5, \ \text{and} \ x_2 = 2.5. \] (6-5)

Solving this definite integral for the exact solution gives the following result

\[ \int_{x_1}^{x_2} f(x) \, dx = \int_{1.5}^{2.5} \left[ 3 - 4 \left( x - 2 \right)^2 \right] \, dx = \left[ 3x - \frac{4}{3} \left( x - 2 \right)^3 \right]_{1.5}^{2.5} = 2.667. \] (6-6)
This integral will now be solved numerically using the trapezoidal rule. Recall the definite integral in Equation 6-5 represents the area under the curve \( f(x) \). This area is divided into many trapezoids where the area of each trapezoid, or partial integral, is approximated as

\[
\frac{1}{2} \left[ f(x_i) + f(x_{i+1}) \right] (x_{i+1} - x_i).
\]

Each area is then summed to get the total area under the curve. Table 6-1 shows the result of the numerical integration which represents the sum of the partial integrals. As you can see the numerical integration is within 1% of the exact solution. A smaller interval between the values of \( x_i \) will result in smaller trapezoids and a more accurate result.

<table>
<thead>
<tr>
<th>( x_i )</th>
<th>( f(x_i) )</th>
<th>Partial Integral</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.50</td>
<td>2.00</td>
<td></td>
</tr>
<tr>
<td>1.60</td>
<td>2.36</td>
<td>0.218</td>
</tr>
<tr>
<td>1.70</td>
<td>2.64</td>
<td>0.250</td>
</tr>
<tr>
<td>1.80</td>
<td>2.84</td>
<td>0.274</td>
</tr>
<tr>
<td>1.90</td>
<td>2.96</td>
<td>0.290</td>
</tr>
<tr>
<td>2.00</td>
<td>3.00</td>
<td>0.298</td>
</tr>
<tr>
<td>2.10</td>
<td>2.96</td>
<td>0.298</td>
</tr>
<tr>
<td>2.20</td>
<td>2.84</td>
<td>0.290</td>
</tr>
<tr>
<td>2.30</td>
<td>2.64</td>
<td>0.274</td>
</tr>
<tr>
<td>2.40</td>
<td>2.36</td>
<td>0.250</td>
</tr>
<tr>
<td>2.50</td>
<td>2.00</td>
<td>0.218</td>
</tr>
<tr>
<td>Sum:</td>
<td>2.660</td>
<td></td>
</tr>
</tbody>
</table>

**Model**

The governing equations in differential form that can be used to solve for the velocity distribution in the boundary layer are conservation of mass and the Navier-Stokes equations (or momentum balance equation for a Newtonian fluid)

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) &= 0 \\
\rho \frac{\partial \mathbf{V}}{\partial t} &= -\nabla p + \rho \mathbf{g} + \mu \nabla^2 \mathbf{V}
\end{align*}
\]  

(6-7)

where \( \rho \) is density, \( t \) is time, \( \mathbf{V} \) is velocity, \( p \) is pressure, \( \mathbf{g} \) is gravitational acceleration, and \( \mu \) is viscosity. In 1908, H. Blasius solved this system of non-linear second-order partial differential equations for incompressible, two-dimensional, laminar boundary layer flow (shown in detail in Ref. 2 and briefly outlined here). The first step is to make two assumptions for a thin boundary layer: (1) velocity is much greater in the streamwise than cross-flow direction (or \( u >> v \)) where \( u \) and \( v \) are the \( x \) and \( y \) components of velocity, respectively, for the coordinate system shown in...
Figure 6-5) and (2) gradients are much greater normal than tangent to the plate \( \partial / \partial x \ll \partial / \partial y \). This reduces the above system of equations to the following boundary layer equations along with the corresponding boundary conditions:

\[
\begin{align*}
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \\
\frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= \frac{\mu}{\rho} \frac{\partial^2 u}{\partial y^2} \\
u(x=0) = v(x=0) = v(x=\infty) = 0, \quad u(x=\infty) &= U
\end{align*}
\]

(6-8)

For the second step, these equations are further simplified by introducing a dimensionless similarity variable:

\[
\eta = \frac{U}{\sqrt{v x y}}
\]

(6-9)

where \( v = \mu / \rho \) is kinematic viscosity and by using the stream function where:

\[
\psi = \sqrt{v x U} \cdot f(\eta).
\]

(6-10)

This reduces Equation 6-8 to a single non-linear third-order ordinary differential equation along with the corresponding boundary conditions:

\[
2 \frac{d^3 f}{d\eta^3} + f \frac{d^2 f}{d\eta^2} = 0
\]

(6-11)

\[
f(\eta=0) = \left. \frac{df}{d\eta} \right|_{\eta=0} = 0, \quad \left. \frac{df}{d\eta} \right|_{\eta=\infty} = 1.
\]

Because this equation is non-linear, it must be solved using a numerical integration technique. The solution is shown in Table 6-2 and is called the Blasius solution. From this solution the boundary layer thickness, \( \delta \) (the location where the boundary layer velocity is 99% of the free-stream velocity or \( \partial f / \partial \eta = u/U = 0.99 \)) corresponds to \( \eta \approx 5 \). Using this result, the column for \( y/\delta = \eta/5 \) has been added to Table 6-2. In addition, this velocity profile is used to determine the relationships for the remaining boundary layer thicknesses. These are summarized in Table 6-3.

For turbulent flow where there is a high degree of mixing, the boundary layer equations cannot be solved analytically or numerically without introducing an additional model for the fluctuating velocities. For this class we will use experimental results that have been used to develop correlations which are curve fits to experimental data. A widely used correlation for the boundary layer velocity distribution often called the power law equation is
\[ \frac{u(y)}{U} = \left( \frac{y}{\delta} \right)^{1/n} \quad \text{for} \quad 0 \leq y \leq \delta \]  

(6-12)

where \( n = 7 \) for most boundary layers (Ref. 1). From this profile, application of the momentum integral equation, and additional experimental results, correlations for boundary layer thickness, displacement thickness, and momentum thickness can be derived (shown by Example 9.6 in Ref. 1). The results are given in Table 6-3.

**Table 6-2** Blasius solution for laminar flow over a flat plate.

| \( \eta \) | \( \frac{U}{\sqrt{V_x} y} \frac{y}{\delta} \) | \( \frac{\partial f}{\partial \eta} = \frac{u}{U} \) | \( \eta = \frac{U}{\sqrt{V_x} y} \frac{y}{\delta} \frac{\partial f}{\partial \eta} = \frac{u}{U} \) |
|-----------------|-----------------|-----------------|
| 0.00            | 0.00            | 0.0000          | 2.80            | 0.56            | 0.8115          |
| 0.20            | 0.04            | 0.0664          | 2.80            | 0.56            | 0.8115          |
| 0.40            | 0.08            | 0.1328          | 3.00            | 0.60            | 0.8460          |
| 0.60            | 0.12            | 0.1989          | 3.20            | 0.64            | 0.8761          |
| 0.80            | 0.16            | 0.2647          | 3.40            | 0.68            | 0.9018          |
| 1.00            | 0.20            | 0.3298          | 3.60            | 0.72            | 0.9233          |
| 1.20            | 0.24            | 0.3938          | 3.80            | 0.76            | 0.9411          |
| 1.40            | 0.28            | 0.4563          | 4.00            | 0.80            | 0.9555          |
| 1.60            | 0.32            | 0.5168          | 4.20            | 0.84            | 0.9670          |
| 1.80            | 0.36            | 0.5748          | 4.40            | 0.88            | 0.9759          |
| 2.00            | 0.40            | 0.6298          | 4.60            | 0.92            | 0.9827          |
| 2.20            | 0.44            | 0.6813          | 4.80            | 0.96            | 0.9878          |
| 2.40            | 0.48            | 0.7290          | 5.00            | 1.00            | 0.9916          |
| 2.60            | 0.52            | 0.7725          | 6.00            | -               | 0.9990          |

**Table 6-3** Summary of boundary layer relationships for uniform flow over a flat plate.

<table>
<thead>
<tr>
<th>boundary layer thickness</th>
<th>Laminar</th>
<th>Turbulent</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \delta ) ( x ) = \frac{5.0}{\sqrt{\text{Re}_x}} )</td>
<td>( \delta ) ( x ) = \frac{0.370}{\text{Re}_x^{1/5}} )</td>
<td></td>
</tr>
<tr>
<td>displacement thickness</td>
<td>( \delta^* ) ( x ) = \frac{1.721}{\sqrt{\text{Re}_x}} )</td>
<td>( \delta^* ) ( x ) = \frac{0.0463}{\text{Re}_x^{1/5}} )</td>
</tr>
<tr>
<td>momentum thickness</td>
<td>( \Theta ) ( x ) = \frac{0.664}{\sqrt{\text{Re}_x}} )</td>
<td>( \Theta ) ( x ) = \frac{0.0360}{\text{Re}_x^{1/5}} )</td>
</tr>
</tbody>
</table>
III. Experiment

Equipment

TecQuipment AF10 Airflow Bench
AF14 Boundary Layer Apparatus
Boundary Layer Pitot Tube
Dwyer Micro Manometer
Flat plate of width $b = 5.00$ cm and length $L = 27.5$ cm
Portable Pitot-Static Tube with Oil Manometer

Preliminary Calculations

1. Verify with the instructor how the flat plate should be installed; with the smooth or rough side on the same side as the Pitot tube and if you should “trip” the boundary layer by attaching a “roughness element” to the plate.

2. Verify with the instructor which $x$-location(s) to use for the Pitot tube measurements. The distance of the Pitot tube from the leading edge can be set to 11.5 cm, 16.5 cm, 21.5 cm, or 26.5 cm depending on which notch in the plate is used with the setscrew.

3. Verify with the instructor which flowrate(s) to use for your lab. Calculate the dynamic pressure that will be measured by the Pitot tube for the highest flowrate and determine the range required for the manometer in inches of H$_2$O.

4. Calculate the Reynolds number for each flowrate and $x$-location.

5. Use the Blasius solution for laminar flow and a correlation for turbulent flow to estimate the boundary layer thickness for the flowrate(s) and location(s) specified by the lab instructor.

6. Use the estimated boundary layer thickness to select the $y$-locations for your velocity measurements across the boundary layer. Your first measurement will be at an average distance of 0.25 mm from the plate surface due to the Pitot tube thickness of 0.50 mm. Your measurement strategy should consider if your flow is more likely to be laminar or turbulent for your flow conditions. Also, collect more data in the region with highest velocity gradients. Try to get a total of about 20 velocity measurements across your boundary layer.

7. Use the Blasius solution for laminar flow and a correlation for turbulent flow to estimate velocity at the first measurement location ($y = 0.25$ mm) for the flowrate(s) and location(s) specified by the lab instructor. Calculate the dynamic pressure that will be measured by the Pitot tube for each velocity and determine the minimum error required for the manometer in inches of H$_2$O for 5% uncertainty in the velocity measurement.
**Procedure**

1. Record the atmospheric pressure and temperature.

2. Install the flat plate in the test section using the setscrew on the back of the channel with the specified side (smooth or rough) on the same side as the Pitot tube and to the specified distance from the leading edge.

3. Verify that the Pitot tube is installed in the micrometer positioning mechanism. Position the Pitot tube approximately halfway between the flat plate and the wall. Connect the Pitot tube to the incline oil manometer using flexible tubing.

4. Verify the damper is shut and turn on the fan. Set the free-stream velocity to the specified magnitude by opening the damper and using the incline oil manometer reading.

5. Position the Pitot tube at the plate surface. Record the initial micrometer reading.

6. Based on the uncertainty analysis from step 5 and the required minimum error for your flow conditions, either leave the Pitot tube connected to the incline oil manometer or connect the Pitot tube to the micro-manometer (Figure 6-5). If necessary, review the operating instructions of the micro-manometer and if you still have questions ask the instructor. *Note that the micrometer on the instrument only measures half of the vertical displacement.*

![Figure 6-5 Micro-manometer.](Image)
7. Record the Pitot tube pressure from the manometer. Use the micrometer to move the probe to the next position and record the next pressure. You are measuring very small pressure changes. Allow sufficient time for the pressure to respond. Continue this procedure until you have traversed through the boundary layer and into the free stream. Do not expect an absolute uniform velocity in the free stream. Consider you have reached the free stream when several successive velocity readings change by less than about 1 to 2%. Make several measurements in the free stream.

8. The lab instructor may instruct you to repeat the experiment (steps 2-7 of the Procedure) for additional location(s) or flowrate(s).

9. Shut the damper and turn off the wind tunnel.

IV. Report

Repeat the following items for each velocity profile measured:

1. Tabulate absolute distance from the plate (mm), Pitot tube pressure from the manometer (mm H₂O), calculated pressure (Pa), and calculated stream-wise velocity (m/s) from the Pitot tube measurements. Use the data in your table to estimate the value of the boundary layer thickness \( \delta \) (where the velocity reaches approximately 99% of the free-stream velocity).

2. Use the data from your experimental measurements in step 1 to tabulate the normalized distance from the plate \( y/\delta \), normalized stream-wise velocity \( u/U \), \( (1-u/U) \), \( (u/U)(1-u/U) \), and the partial integrals needed to apply the trapezoidal rule to determine the displacement and momentum thicknesses. Note that for the integration you need to include the area between the plate (at zero velocity) and the first reading. Also, both of the functions being integrated go to zero outside the boundary layer, so you only need to include partial integrals up to \( \delta \). Sum the partial integrals to obtain the two thicknesses. Include sample calculations with your table.

3. Tabulate the normalized distance from the plate \( y/\delta \), \( u/U \) for laminar flow (from the Blasius solution), and \( u/U \) for turbulent flow (calculated using the power law profile).

4. Tabulate the experimentally determined boundary layer, displacement, and momentum thicknesses along with all three values predicted for your conditions by the Blasius solution for laminar flow and by the turbulent flow correlations. How do each of the boundary layer thicknesses compare with the expected values? If there are differences, explain the differences based on experimental observations and physical considerations. Include sample calculations with your table.

5. Add a column to the table from item 4 that calculates the drag coefficient from the momentum thickness using Equation 6-4. Comment on how the drag coefficients compare. Calculate the drag force in N using Equation 6-3 from your experimental data. Comment on the magnitude of the drag force and if it seems reasonable.
6. Plot normalized stream-wise velocity $u/U$ versus normalized distance from the plate $y/\delta$ for your experimental data (use symbols and no line), laminar flow (solid line with no symbols), and turbulent flow (dashed line with no symbols). Compare and contrast the experimental data with the predictions for laminar and turbulent flow. Is the experimental boundary layer laminar, turbulent, or in transition?

7. How does your Reynolds number compare with the transition Reynolds number for flow over a flat plate? How would you expect your flow conditions (such as smooth versus rough surface) to affect your flow regime?

V. References


I. Objectives

The objectives of this experiment are (1) to measure the drag for a cylinder in a uniform flow stream by two indirect methods, (2) to compare these results to experimental data to determine if each method is accurate, and (3) to visualize the flow in a turbulent wake.

II. Introduction

Flow around a cylinder appears in many engineering applications under a wide range of conditions such as flow around bridge pylons, buildings, or golf club shafts. As a result, many researchers have studied these flows extensively. They have found that the flow around a cylinder exhibits a wealth of behaviors, such as flow separation and complex oscillating wake features, that can form the basis of a detailed understanding of fluid mechanics (Ref. 1).
In this lab, we will investigate the flow around a cylinder in the wind tunnel under ideal conditions. The cylinder is inserted in the wind tunnel with its axis perpendicular to the flow which produces an approximately uniform flow upstream of the cylinder. In addition, the flow is approximately two-dimensional at the cylinder center on the plane perpendicular to its axis because the length is large compared to the diameter and the effects of the surrounding walls are negligible. We will determine the drag on our cylinder by two methods; (1) from calculations that use the measured pressure distribution along the cylinder surface and (2) from calculations that use the measured velocity distribution in the wake behind the cylinder (Ref. 2). Each method has its limitations and advantages. To understand these we need to know more about the characteristics of two-dimensional uniform flow over a cylinder.

Two-Dimensional Uniform Flow Over a Cylinder

The flow pattern and drag (force on an object parallel to flow) for a cylinder is generally dependent on the Reynolds number defined as

\[ Re_D = \frac{U D}{\nu} \]  

(7-1)

where \( U \) is the free-stream velocity, \( D \) is the diameter of the cylinder, and \( \nu \) is the kinematic viscosity of the fluid. For a cylinder the characteristic dimension is the diameter and for our lab the fluid is air. Physically, the Reynolds number represents the ratio of inertial forces to viscous forces. Thus, for higher Reynolds number flows perturbations in the flow or velocity fluctuations are more likely to result in additional mixing because viscous forces, or friction, will not be sufficient to stop the flow.

At different ranges of \( Re_D \), the flow pattern and drag behavior for a cylinder can take on different forms as shown in Figure 7-1. In particular, we will consider the flow pattern that exists about the cylinder for \( 1 \times 10^3 < Re_D < 2 \times 10^5 \). At the front of the cylinder there will be a stagnation point with zero velocity and therefore maximum pressure. At this point the pressure, \( p \), is equal to the stagnation pressure, \( p_0 + \frac{1}{2} \rho U^2 \) where \( p_0 \) is the static pressure in the free stream, and \( \rho \) is the density. In terms of the dimensionless pressure coefficient defined as

\[ c_p = \frac{p - p_0}{1/2 \rho U^2} \]  

(7-2)

at the stagnation point \( c_p = 1 \). To either side of the stagnation point the flow accelerates about the forward face of the cylinder resulting in a drop in \( p \) and \( c_p \) on the surface of the cylinder. A boundary layer is formed and it remains laminar for a smooth surface at low enough \( Re_D \). This laminar boundary layer increases in thickness as the flow continues over the surface of the cylinder until the point of maximum thickness of the cylinder. The fluid comprising the boundary layer cannot follow the contour of the cylinder due to the adverse pressure gradient on the back and lifts off or separates from the surface. Downstream of this point is a broad, turbulent wake resulting in a low and approximately uniform pressure on the aft face of the cylinder. The wake is unstable and sheds vortices asymmetrically and periodically.
For \( Re_D > 2 \times 10^5 \) the flow field and pressure distribution about the cylinder changes. The stagnation point is still at the leading edge of the cylinder. However, as the flow accelerates about the forward face of the cylinder the boundary layer that is initially laminar transitions to turbulent. A turbulent boundary layer has greater momentum and can withstand the adverse pressure gradient better as the flow continues to the back face of the cylinder. This delay of separation results in a wake of reduced size.

The two flow patterns described above have dramatic effects on the drag exerted on the cylinder. The total drag is the sum of two mechanisms, friction drag and pressure drag. Friction drag, \( D_f \), is due to shear stresses at the surface and is greater for turbulent boundary layers than for laminar boundary layers due to enhanced turbulent mixing. Pressure drag, \( D_p \), is the component of drag due to the imbalance of pressures on the front and back faces of the cylinder. For dimensionless drag we define the total drag coefficient as

\[
C_D = \frac{F_D}{\frac{1}{2} \rho U^2 A}
\]  

(7-3)  

where \( F_D \) is the total drag and \( A \) is the projected area normal to the flow (for a cylinder \( A = D L \) where \( L \) is the length).

The initial case where \( 1 \times 10^3 < Re_D < 2 \times 10^5 \) is called the sub-critical range and is characterized by high drag with an approximately constant drag coefficient of around 1.2 to 1.3. For the sub-critical condition the large turbulent wake results in a large pressure imbalance where the pressure drag makes up approximately 86% of the total drag at \( Re_D = 10^3 \) and 98% of the total drag at \( Re_D = 10^5 \). For \( Re_D > 4 \times 10^5 \) the cylinder is in the super-critical range and is characterized by a greatly reduced drag coefficient of approximately 0.3 at \( Re_D = 4 \times 10^5 \). The total drag decreases because the reduced pressure drag is much more significant than the slight increase in friction drag due to the higher shear stresses in the turbulent boundary layer. This behavior for \( C_D \) as a function of \( Re_D \) is presented in Figure 7-2. The transition from the sub-critical to super-critical range (shown as a significant dip in the figure) is commonly referred to as the drag crisis (Ref. 2). The actual \( Re_D \) at which the reduction in \( C_D \) occurs varies with the cylinder surface roughness (such as dimples on a golf ball) and the turbulence level in the surrounding flow stream. For example, it is well documented that a cylinder in a wind tunnel will typically enter the super-critical range at a much lower \( Re_D \) than in the free atmosphere due to the turbulence levels in the wind tunnel flow stream.

**Figure 7-2** Coefficient of drag versus \( Re_D \) for a cylinder in uniform flow.
Determination of Drag by the Surface Pressure Method

The pressure drag on a body can be measured by integrating the surface static pressure over the body. If we take a differential element of the cylinder surface shown in Figure 7-3

\[ ds = r \, d\theta \]  \hspace{1cm} (7-4)

where differential force per unit length due to the pressure exerted on the differential element is

\[ \frac{dF}{L} = p \, ds = p \, r \, d\theta . \]  \hspace{1cm} (7-5)

![Figure 7-3 Cylinder surface pressure integration variables.](image)

We can integrate this component of the force over the surface of the cylinder to obtain the drag force due to pressure per unit length

\[ \frac{F_{D,p}}{L} = \int_0^{2\pi} p \cos \theta \left( \frac{D}{2} \right) d\theta \]  \hspace{1cm} (7-6)

We can combine Equations 7-3 and 7-6 to obtain the drag coefficient for the cylinder due to the surface pressure distribution

\[ C_{D,p} = \frac{1/2 \int_0^{2\pi} p \cos \theta \, d\theta}{\frac{1}{2} \rho U^2} \]  \hspace{1cm} (7-7)

Incorporating the definition of the pressure coefficient given in Equation 7-2 we get
\[ C_{D,p} = \frac{1}{2} \int_{0}^{\pi} c_p \cos \theta \, d\theta \] (7-8)

where the free stream static pressure term goes to zero because \( p_\infty \) acts uniformly on all sides of the cylinder and contributes no net force. Finally, for a symmetric pressure distribution on the top and bottom sides of the cylinder Equation 7-8 can be reduced to

\[ C_{D,p} = \int_{0}^{\pi} c_p \cos \theta \, d\theta \] (7-9)

where Equation 7-9 can be evaluated using a numerical integration method such as the trapezoidal rule (see the Introduction to Lab 6).

**Determination of Drag by the Momentum Method**

Drag can also be determined by calculating the difference in the flow momentum upstream and downstream of the body. The change in momentum reflects the energy lost due to the presence of the body in the flow stream, and thus can be related to the drag force acting on the body under certain conditions. First, the flow must be two-dimensional with negligible spanwise variation in the momentum loss. This is a valid assumption for our lab except in the region nearest the wall. Second, there must not be significant rotational losses due to large-scale vortices. For the flow behind a cylinder this assumption is not generally valid. In particular, for ranges of Reynolds number where there is a large oscillating wake, and the velocity profile is measured within this region, this method will significantly underestimate the total drag. This method can still be used to accurately calculate the drag if the wake velocity profile is measured far enough downstream where the large-scale vortices have been damped out. For a cylinder, 30 diameters downstream is typically sufficient.

To calculate the drag, consider the wind tunnel longitudinal section shown in Figure 7-4. Recall the momentum equation in integral form for a finite control volume

\[ \frac{\partial}{\partial t} \int_{CV} \rho \mathbf{V} \, dV + \int_{CS} \rho \mathbf{V} \cdot \hat{n} \, dA = \sum \mathbf{F} \] (7-10)

where \( t \) is time, \( \mathbf{V} \) is velocity, \( \rho \) is density, \( \mathcal{V} \) is volume, \( A \) is area, \( \hat{n} \) is the outward unit normal for area, and \( \mathbf{F} \) are forces. We will use the control volume of height \( H \) shown in Figure 7-4. We will assume the only significant longitudinal force on the body is the reaction force, \( R \), necessary to hold the cylinder in place and that it is the negative of the drag force, thus neglecting changes in the static pressure in the streamwise direction. Assuming steady flow the momentum equation for our control volume in the x-direction reduces to

\[ -\rho U^2 L H + L \int_{y=0}^{H} \rho V_x^2 \, dy + U \, \dot{m}_3 = -F_D \] (7-11)
where we have also assumed \( V_1 = U \) is constant (for the uniform flow upstream) and \( \dot{m}_3 \) is the mass flowrate out the top and bottom of the control volume (at \( y = 0 \) and \( H \)).

\[
\frac{\partial}{\partial t} \left( \int_{CV} \rho \, dV \right) + \int_{CS} \rho \, \mathbf{V} \cdot \mathbf{n} \, dA = 0 .
\]  
(7-12)

For steady state and our control volume we get

\[-\rho U L H + L \int_{y=0}^{H} \rho V_2 \, dy \left. + \dot{m}_3 = 0 \right. \]  
(7-13)

which can be solved for \( \dot{m}_3 \), substituted into Equation 7-11, and simplified to get

\[ F_D = L \int_{y=0}^{H} \left( U - V_2 \right) \rho V_2 \, dy . \]  
(7-14)

Substituting this into Equation 7-3 we get

\[ C_D = \frac{2}{D} \int_{y=0}^{H} \left( 1 - \frac{V_2}{U} \right) \left( \frac{V_2}{U} \right) \, dy \]  
(7-15)

which can be numerically integrated to get \( C_D \).
III. Experiment

Equipment
Cal Poly ME Wind Tunnel Facility
brass cylinder with 1.50 in diameter and single pressure tap
Pitot-static probe
x-y traverse
MKS Baratron® Series 200 pressure transducer

Preparations for Test (Completed by Instructor)

1. Energize the 230V power supply for wind tunnel. Turn on wind tunnel instrumentation system (Figure 7-6) and allow to warm up for at least 15 minutes.

2. Ensure that the zero and span for the pressure transducer located on the instrumentation system front panel (Figure 7-6) have been properly calibrated to display differential pressure from 0.00 to 10.0 in H₂O on the pressure transducer display.

3. Ensure that the Pitot-static probe is mounted correctly in the traverse and that the traverse is properly connected to the back panel of the instrumentation system (Figure 7-7).

4. Ensure that the zero and span for the traverse are set correctly. The x-direction (streamwise position) should be zero at the entrance of the test section. The y-direction (vertical position) should be zero at the bottom of the test section. The span for the x-axis and y-axis should be set to display the position in inches. Verify by checking that the length of the test section reads $x = 48.0$ in and the height of test section reads $y = 24.0$ in.

Figure 7-6 Instrumentation system front panel.
5. Ensure that the total and static pressure tubes from the Pitot-static probe are connected to the total and static pressure barbed fittings on the back panel of the instrumentation system (Figure 7-7). Port 0 on the pressure selector (Figure 7-6) corresponds to the Pitot-static probe reading.

6. Ensure that the brass cylinder is installed horizontally in the wind tunnel test section and oriented such that the pressure tap is facing forward. Connect the pressure tube from the cylinder to the brown tube (corresponds to Port 2 on the pressure selector in Figure 7-6) on the pressure bulkhead connector on the back panel of the instrumentation system (Figure 7-7).

**Procedure**

1. As with any piece of equipment, the condition of the equipment and safety of operation must be checked before operating. First verify, along with the instructor, that the inlet area of the wind tunnel is free from obstructions and foreign objects and the honeycomb flow straightener is clean and undamaged. Inform the lab instructor of any issues and document your observations. Next ensure that there are no foreign objects in the wind tunnel. This completes the most basic of checks before operation of a wind tunnel and must be performed every time the wind tunnel is operated.

2. Record the atmospheric pressure, $p_{amb}$, and temperature, $T_{amb}$. Record the initial value of the pressure gauge, $p_{zero}$. This is the zero reading for all channels that will be needed to offset all of your pressure readings. Use the data sheet at the end of this section.
3. Select Basic Parameters Mode on the motor speed controller panel by pressing the dial, <enter>, on the keypad shown in Figure 7-8.

4. If necessary, set the adjustment increment to 1Hz increments by pressing the F3 function key. Finer increments of 0.1Hz are possible by selecting the F4 key.

5. Turn the dial clockwise to increase the inverter frequency setting to 20Hz.

6. Press the dial, <enter>, to return to the Top View Mode.

7. Press Run.

8. Allow the tunnel to settle, or reach steady state, (approximately 2 minutes).

Surface Pressure Measurements

9. Locate the Pitot-static probe horizontally halfway between the test section entrance and the cylinder axis. Verify the streamwise location is at approximately \( x = 5 \) in. Raise the Pitot-static probe to halfway between the cylinder axis and the upper wall. Verify the vertical location is at approximately \( y = 18 \) in. Note that the traverse and Pitot-static probe should be located away from the cylinder when taking pressure measurements from the cylinder pressure tap to minimize interference.

10. Record the Pitot-static tube pressure (or wind tunnel dynamic pressure) using Port 0 on the pressure selector (Figure 7-6).

11. Rotate the cylinder pressure tap such that it is oriented at 0°. Check its alignment by verifying that the pressure reading from the cylinder pressure tap (Port 2 on pressure selector) is highest at 0° and drops off uniformly when rotated by the same amount in the positive and negative direction (use \( \theta = 5° \) and 10° for this test). If it is not, either loosen the lock screw and rotate the cylinder so that the pressure tap is facing forward or record the amount that it is off and use this adjustment when setting your angles below.
12. Take cylinder differential pressure measurements \((p - p_0)\) for a full 180° in 10° increments. We will assume that the measurements are the same on the top and bottom.

13. Confirm the wind tunnel dynamic pressure has not changed.

**Wake Profile Pressure Measurements**

14. Position the traverse 5 diameters downstream \((x \approx 17.5 \text{ in})\) and 4.0 in below \((y \approx 8.0 \text{ in})\) the axis of the cylinder.

15. Wait for the Pitot-static pressure readings to stabilize. Record the Pitot-static pressure (Port 0 on Pressure Selector) and the actual \(y\)-axis position.

16. Repeat Step 15 for a range of traverse heights above the tunnel lower wall in 0.5 in increments to 4.0 in above the axis of the cylinder \((y \approx 16.0 \text{ in})\).

17. Repeat Steps 14-16 for a traverse position 10 diameters downstream of the axis of the cylinder \((x \approx 25.0 \text{ in})\).

18. Press Stop to turn off the wind tunnel.

**Flow Visualization**

19. Locate the Pitot-static probe streamwise at the rear of the test section \((x \approx 36 \text{ in})\) and vertically halfway between the cylinder and the upper wall of the test section \((y \approx 18 \text{ in})\).

20. Insert the rod with yarn tufts through the Pitot-static probe slot opening in the top of the wind tunnel. Position the rod at various locations around the cylinder to observe the streamlines in the flow. In particular, check how uniform the flow is upstream, the flow over the front of the cylinder, and the size and nature of the wake. If desired, observe the effect of flowrate on the above by increasing the fan speed to 40 and 60 Hz.

**IV. Report**

1. Calculate the upstream velocity in the wind tunnel and the corresponding Reynolds number for your experiment from the Pitot-static tube measurements.

2. Tabulate the cylinder angle, cylinder differential pressure measurement, and pressure coefficient.

3. Plot pressure coefficient versus cylinder angle (symbols connected by straight lines). Do the trends for the pressure coefficient make sense (compare to figure in your textbook)? Based on your measurements what do you think the wake looks like? Does this agree with your expectations based on your calculated Reynolds number?

4. Tabulate the streamwise position, vertical position, normalized vertical position, \(y/D\), Pitot-static probe pressure, velocity, and normalized velocity, \(u/U\), for both streamwise locations. Let \(y/D = 0\) correspond to the center of the cylinder.
5. Plot normalized velocity, \( u/U \), versus normalized vertical position, \( y/D \), (symbols connected by straight lines) for both streamwise locations on one plot. Do the trends for the velocity distribution in the wake make sense? Describe how the wake changes in the vertical and streamwise direction.

6. Calculate the drag coefficient for the cylinder from the surface pressure measurements using the surface pressure method and numerical integration. Include sample calculations and tables with your results.

7. Calculate the drag coefficient for the cylinder from the wake velocity profile measurements for each downstream survey using the momentum method and numerical integration. Include sample calculations and tables with your results.

8. Tabulate your calculated drag coefficients from items 6 and 7. Add to your table the value estimated using Figure 7-2 and your calculated Reynolds number. Do your drag coefficient values agree? If not, what are the possible sources for the differences for each case?

9. Based on your flow visualization, describe the nature of the flow distribution around the cylinder and the wake.

V. References


# DATA SHEET
## CYLINDER SURFACE PRESSURE

<table>
<thead>
<tr>
<th>$\theta^\circ$</th>
<th>$p - p_0$ (in H$_2$O)</th>
<th>NOTES/COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td></td>
<td></td>
</tr>
<tr>
<td>40</td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td></td>
<td></td>
</tr>
<tr>
<td>60</td>
<td></td>
<td></td>
</tr>
<tr>
<td>70</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80</td>
<td></td>
<td></td>
</tr>
<tr>
<td>90</td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>110</td>
<td></td>
<td></td>
</tr>
<tr>
<td>120</td>
<td></td>
<td></td>
</tr>
<tr>
<td>130</td>
<td></td>
<td></td>
</tr>
<tr>
<td>140</td>
<td></td>
<td></td>
</tr>
<tr>
<td>150</td>
<td></td>
<td></td>
</tr>
<tr>
<td>160</td>
<td></td>
<td></td>
</tr>
<tr>
<td>170</td>
<td></td>
<td></td>
</tr>
<tr>
<td>180</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## DATA SHEET
### WAKE VELOCITY PROFILE

<table>
<thead>
<tr>
<th>Date:</th>
<th>Engineers:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$p_{amb}$:</th>
<th></th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>$T_{amb}$:</th>
<th></th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>$p_{zero}$:</th>
<th></th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>$x$ (in)</th>
<th>$y$ (in)</th>
<th>Pitot-static (in H$_2$O)</th>
<th>NOTES/COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>17.5 in</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>


## DATA SHEET
### WAKE VELOCITY PROFILE

<table>
<thead>
<tr>
<th>$x$ (in)</th>
<th>$y$ (in)</th>
<th>Pitot-static (in H$_2$O)</th>
<th>NOTES/COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>25.0 in</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
8. **Flow Through a Converging-Diverging Nozzle**

![Figure 8-1 Compressible flow apparatus.](image)

### I. Objectives

The objectives of this experiment are to (1) measure the pressure distribution in a converging-diverging nozzle for a range of airflow velocities, (2) compare these measurements to one-dimensional models for incompressible and compressible flow, and (3) demonstrate the phenomenon of choking.

### II. Introduction

All the other experiments performed in this lab assume incompressible flow. This is a reasonable assumption as long as the Mach number remains less than 0.3 (Ref. 1). The Mach number is defined as

\[
Ma = \frac{V}{c}
\]  

(8-1)

where \(V\) is local flow velocity and \(c\) is speed of sound. Physically, it represents the ratio of compressible to inertial forces, thus compressible effects are significant at higher Mach numbers.
The speed of sound in water at 20°C and atmospheric pressure is 1,481 m/s (or 3,313 mph) which is extremely fast, thus all our experiments with water are correctly modeled as incompressible. The speed of sound in air at 20°C and atmospheric pressure is 343 m/s (or 768 mph) which is significantly lower because air is more easily compressed. At 103 m/s (or 230 mph) we must begin to consider compressible effects. Again, for all of our other labs with air it is reasonable to assume the flow is incompressible. For this lab, the Mach number will exceed 0.3 in the throat of the nozzle and compressible effects must be considered.

Experiment

In this experiment, a converging-diverging nozzle is attached to the inlet of a four-stage centrifugal compressor as shown in Figure 8-1. The compressor draws in air and accelerates it through the inlet and throat as shown in Figure 8-2. As the rotational speed of the compressor is increased, the pressure at the nozzle outlet decreases (while the room air remains at atmospheric pressure). The increase in pressure gradient forces more air to be sucked through the nozzle. For low velocities, where Mach number stays below 0.3 in the throat, compressible effects are negligible. At higher pressure gradients and velocities compressible effects become significant. Finally, when the Mach number reaches one in the throat the flow velocity is at the speed of sound. Now, if the outlet pressure is reduced further it can no longer be sensed upstream of the throat because the pressure wave cannot travel faster than the speed of sound and it gets washed downstream. As a result the mass flowrate cannot increase any further. This is called "choked" flow. We can observe the choked phenomena by listening to the compressor through the nozzle before and after the flow is choked.

![Figure 8-2](image_url) Converging-diverging nozzle picture and schematic where \( d_1 = d_3 = 2.54 \text{ cm} \) and \( d_2 = 1.0 \text{ cm} \).
We will begin this experiment by measuring the pressure and temperature in the room. The compressor is then run at a sequence of increasing speeds up to and past the choked flow condition. For each flowrate the pressure at the inlet, throat, and outlet are recorded. In addition, we will make observations about the flow before and after choking.

Experimental Mass Flowrate Calculation

The mass flowrate for the experiment will be calculated from the measured pressure drop across the inlet contraction and by using a simple model. Because the inlet velocity is much lower than the throat velocity (even when the flow is choked) this section of the flow is modeled as incompressible. The relationship used for this calculation is developed by applying the energy equation for incompressible, steady flow to a control volume with a single inlet and exit

\[
\left( \frac{p_1}{\rho g} + \frac{V_1^2}{2g} \right) - \left( \frac{p_2}{\rho g} + \frac{V_2^2}{2g} \right) = h_L
\]

(8-2)

where \( p \) is pressure, \( \rho \) is density, \( g \) is acceleration due to gravity, \( \alpha \) is the kinetic energy coefficient, \( \bar{V} \) is average velocity, \( z \) is elevation, \( h_L \) is the head loss term, and subscripts 1 and 2 indicate the inlet and exit, respectively. We will define our control volume as the air starting from the room and extending to the nozzle inlet. Assuming the air in the room is at atmospheric or stagnation pressure (\( p_1 = p_0 \)) and zero velocity (\( \bar{V}_1 = 0 \)), uniform flow at the inlet (\( \bar{V}_2 = \bar{V}_{in}, \alpha_2 = 1 \)), constant elevation (\( z_1 = z_2 \)), and only minor losses for the contraction this reduces to

\[
\frac{p_0}{\rho_0 g} - \left( \frac{p_{in}}{\rho_0 g} + \frac{V_{in}^2}{2g} \right) = K_L \frac{V_{in}^2}{2g}
\]

(8-3)

where \( \rho_0 \) is the density of the air at room temperature and pressure and \( K_L = 0.08 \) is the minor loss coefficient for the inlet contraction. Solving for velocity we get

\[
V_{in} = \sqrt{\frac{2(p_0 - p_{in})}{\rho_0 (1 + K_L)}}
\]

(8-4)

Finally, the mass flow rate is calculated as

\[
\dot{m} = \rho_0 V_{in} A_{in} = A_{in} \sqrt{\frac{2 \rho_0 (p_0 - p_{in})}{(1 + K_L)}}
\]

(8-5)

Experimental Throat Mach Number Calculation

The throat Mach number is calculated from the experimental mass flow rate obtained above, measured throat pressure, and measured room pressure and temperature by applying conservation of mass from the inlet to the throat. From Equation 8-1 the throat Mach number is
where the subscript \( th \) indicates conditions in the throat. To calculate the throat velocity, we apply conservation of mass from the inlet to the throat (again assuming steady uniform flow in the throat) to get

\[
\dot{m} = \rho_{th} V_{th} A_{th} .
\]  

(8-7)

The speed of sound in the throat is calculated from the following relationship for an ideal gas

\[
c_{th} = \sqrt{k R T_{th}}
\]  

(8-8)

where \( k \) is the specific heats ratio, \( R \) is the gas constant, and \( T \) is temperature. The air temperature at the throat is eliminated by using the ideal gas law

\[
p_{th} = \rho_{th} RT_{th} .
\]  

(8-9)

Combining Equations 8-6 through 8-9 we get

\[
Ma_{th} = \frac{\dot{m}/A_{th}}{\sqrt{k \rho_0 p_0 (\rho_{th}/\rho_0) (p_{th}/p_0)}} .
\]  

(8-10)

Next, we need to determine the density in the throat. We could measure the temperature in the throat and calculate it using the ideal gas law. Alternatively, we will assume the flow is isentropic and calculate it using the following isentropic flow relationship for an ideal gas

\[
\frac{\rho_{th}}{\rho_0} = \left(\frac{p_{th}}{p_0}\right)^{1/k} .
\]  

(8-11)

This is a reasonable assumption for the contraction where there is a favorable pressure gradient. Substituting Equation 8-11 into 8-10 we get

\[
Ma_{th} = \frac{\dot{m}/A_{th}}{\sqrt{k \rho_0 p_0 (p_{th}/p_0)^{1/k}}} .
\]  

(8-12)

Model

We will use two models that apply conservation of mass and momentum to predict the throat pressure and Mach number for our converging-diverging nozzle geometry for a specified mass flow rate. The first will assume incompressible flow and the second will account for compressible effects. Both will assume steady one-dimensional flow (or uniform flow at each cross section which requires gradual area changes), ideal gas behavior, and isentropic flow from the
room air up to the throat. By comparing both models to the experimental data we will consider the range of conditions for which the above assumptions are valid.

Incompressible Flow Model

For our momentum balance we use the Bernoulli equation

$$\frac{p}{\rho} + gz + \frac{V^2}{2} = \text{constant} \tag{8-13}$$

which is valid for steady, incompressible, inviscid flow along a streamline. First, we apply the Bernoulli equation along a streamline at constant elevation from the room air to inside the nozzle

$$\frac{p_0}{\rho_0} + \frac{V_0^2}{2} = \frac{p}{\rho} + \frac{V^2}{2} \tag{8-14}$$

where for our incompressible model we use the density of the room air, $\rho_0$. Also, assuming the room air is stagnant ($V_0 = 0$) and solving for the normalized pressure we get

$$\frac{p_0}{p} = 1 + \frac{1}{2} \left( \frac{\rho_0}{\rho} \right) V^2 \tag{8-15}$$

To write this in terms of the Mach number use Equation 8-8 for the speed of sound to get

$$\frac{p_0}{p} = 1 + \left( \frac{k}{2} \right) Ma^2 \tag{8-16}$$

This equation can be used to determine the stagnation pressure from the local pressure and Mach number at any location in the nozzle. For our case, apply this equation to the throat to get

$$\frac{p_0}{p_{th}} = 1 + \left( \frac{k}{2} \right) Ma_{th}^2 \tag{8-17}$$

Next, we need to apply conservation of mass. This has been done as part of the experimental Mach number calculation. For the incompressible case, use Equation 8-10 with $\rho_0 = \rho_{th}$ to get

$$Ma_{th} = \frac{\dot{m}/A_{th}}{\sqrt{k p_0 p_0 \left( p_{th}/p_0 \right)}} \tag{8-18}$$

Finally, substitute this into Equation 8-17 to get the normalized throat pressure as
Compressible Flow Model

For our momentum balance we will start with an equation from the derivation of the Bernoulli equation before it has been integrated along the streamline, where density has not yet been assumed constant, given as

\[
\frac{p_{th}}{p_0} = 1 - \frac{(\dot{m}/A_{th})^2}{2 \rho_0 p_0}.
\]  

(8-19)

which is valid for steady and inviscid flow. Next, we need a relationship for density to integrate this equation. Using Equation 8-11 for an ideal gas undergoing an isentropic process and again assuming constant elevation we get

\[
\rho_0 \left( \frac{p}{p_0} \right)^{\frac{1}{k-1}} dp + g \; dz + \frac{1}{2} \left( \frac{V^2}{2} \right) \; dV = 0
\]

(8-20)

which can be integrated from \( p \) to \( p_0 \) by separating variables

\[
\frac{p_0^{\frac{1}{k}}}{\rho_0} \int_p^{p_0} p^{\frac{1}{k}} dp + \int_{y=0}^{y=0} \left( \frac{V^2}{2} \right) dV = 0
\]

(8-22)

and evaluated to obtain

\[
\left( \frac{k}{k-1} \right) \left( \frac{p_0^{\frac{1}{k}}}{\rho_0} \right) \left[ p_0^{(k-1)/k} - p^{(k-1)/k} \right] - \frac{V^2}{2} = 0.
\]

(8-23)

Finally, we solve this relationship for the pressure ratio in terms of the Mach number using Equation 8-8 for the speed of sound and after a bit of algebra we get

\[
\frac{p_0}{p} = \left[ 1 + \left( \frac{k-1}{2} \right) Ma^2 \right]^{\frac{k}{(k-1)}}.
\]

(8-24)

This equation can be compared to Equation 8-17 for incompressible flow. From here the steps to find the throat Mach number and pressure ratio are similar to the incompressible case. As before we apply this equation to the throat to get

\[
\frac{p_0}{p_{th}} = \left[ 1 + \left( \frac{k-1}{2} \right) Ma_{th}^2 \right]^{\frac{k}{(k-1)}}.
\]

(8-25)
Next, we need to again apply conservation of mass. This time we will use Equation 8-12 from the experimental Mach number section because it accounts for the compressibility of the air by assuming isentropic flow

\[
Ma_{th} = \frac{\dot{m}/A_{th}}{\sqrt{k \rho_0 P_0 (p_{th}/P_0)^{(k+1)/k}}}.
\] (8-12)

At this point we would like to substitute Equation 8-12 into Equation 8-25 to get a single equation to solve for the pressure ratio like we did for the incompressible case. However, because these nonlinear equations are more complicated we cannot get a closed form solution. To get a solution we must solve Equations 8-12 and 8-25 simultaneously.

In addition to the above relationships, we will also use our model to predict the normalized throat pressure and mass flow rate for the choked condition. For sonic velocity in the throat we have \( Ma_{th} = 1 \) which when substituted into Equation 8-24 gives

\[
\frac{P_0}{P_{th}} = \left(\frac{k+1}{2}\right)^{k(k-1)} = 1.893.
\] (8-26)

where \( k = 1.40 \) was used for air at standard temperature and pressure. The mass flow rate for the choked condition can be calculated from the conditions at the throat from

\[
\dot{m}_{\text{choked}} = \rho_{th} c_{th} A_{th} = \rho_{th} \sqrt{k \frac{P_{th}}{\rho_{th}}} A_{th}
\] (8-27)

where the speed of sound relation has again been used. Using the isentropic relation from Equation 8-11 once more for the throat density we get

\[
\dot{m}_{\text{choked}} = A_{th} \sqrt{k \rho_0 \left(\frac{P_{th}}{P_0}\right)^{(k+1)/k}} P_{th}
\] (8-28)

and then substitute in Equation 8-26 for the pressure ratio and after some algebra get

\[
\dot{m}_{\text{choked}} = A_{th} P_0 \sqrt{\frac{k}{RT_0}} \left(\frac{2}{k+1}\right)^{(k+1)/2(k-1)} = 0.6847 A_{th} \frac{P_0}{\sqrt{RT_0}}
\] (8-29)

where \( k = 1.40 \) was again used. Equation 8-29 is the maximum mass flow rate that can be sustained in the converging-diverging nozzle based on our model assumptions. For a given throat area, atmospheric pressure, atmospheric temperature, and throat pressure the mass flow rate at the choked condition can be predicted.
III. Experiment

Equipment

Compressible flow apparatus with converging-diverging nozzle

Preparations for Test (Completed by Instructor, students should NOT perform)

1. Prior to turning on the apparatus, verify that the speed control knob is fully counterclockwise and reading zero. Starting the apparatus with the speed above zero will cause the circuit breaker to trip.

2. Verify that the emergency stop button, located on top of the display panel, is pulled up.

3. Turn on the apparatus by switching on the on/off switch located on the back of the display panel (Figure 8-3).

4. Verify that the tube for the throat pressure tap is connected to the lowermost socket (-side of the differential transducer) labeled P1 on the display panel.

5. Verify that the tube for the outlet pressure tap (closest to the compressor) is connected to the lowermost socket labeled P2 on the display panel.

6. Verify that the tube for the inlet pressure tap is connected to the lowermost socket labeled P3 on the display panel.

Procedure

1. Record atmospheric pressure, $p_0$, and temperature, $T_0$ on the data sheet.
2. Verify that the pressure tap tubes on the duct are connected to the display panel as follows: inlet to P3, throat to P1, and outlet to P2. All three tubes should be connected to the lower tap which corresponds to the negative side of a differential pressure transducer while the positive top tap is left open to the atmosphere. Thus, a positive display value for each transducer corresponds to *pressure below atmospheric* or *vacuum pressure*. Note the units are indicated on the panel for each gage.

![Figure 8-4 Display panel.](image)

3. **Caution**: Keep small items well away from the inlet of the duct to avoid them being sucked into the compressor. Also, keep your hands away from the outlet of the compressor and away from the moving belt.

4. Increase the compressor speed by turning the speed control knob on the display panel clockwise (Figure 8-4). Increase the speed until the pressure reading on the outlet pressure P2 reads 0.5 kPa. You should notice the air flow into the duct increasing as the compressor speed increasing. When data is steady, record all three pressure readings on your data sheet.

5. Continue the process of increasing P2 by increments of 0.5 kPa until P2 reaches 12.5 kPa. Record all three pressure readings at each speed. While collecting data be sure to note in the comments at what pressure the air flow noise coming from the duct stops. This is an audible indication of choked flow and sonic conditions at the throat.

6. After taking the last reading, slowly turn the speed control knob fully counterclockwise to stop the compressor.
IV. Report

1. Tabulate the three measured experimental pressures and calculated outlet pressure ratio, \( p_{\text{out}}/p_0 \), throat pressure ratio, \( p_{\text{th}}/p_0 \), inlet pressure ratio, \( p_{\text{in}}/p_0 \), experimental mass flowrate, \( \dot{m} \), and experimental Mach number at the throat, \( Ma_{\text{th}} \). Before calculating the pressure ratios be sure to convert vacuum pressure to absolute pressure. Include sample calculations with your table.

2. Calculate and tabulate \( p_{\text{th}}/p_0 \) and \( Ma_{\text{th}} \) (for a range of \( \dot{m} \) that spans your measured values) using the incompressible and compressible flow models. For the compressible flow model you will need to solve two non-linear equations simultaneously. This can be done by using the "Solver" in Excel, built in functions in MATLAB©, EES©, a programmable calculator or many other non-linear equation solvers (Ref. 2-4). Your lab instructor will recommend which method to use. Include sample calculations (and/or your program) with your table.

3. Plot \( p_{\text{th}}/p_0 \) and \( p_{\text{in}}/p_0 \) versus \( p_{\text{out}}/p_0 \) for your experimental data (symbols connected with a straight line). How does the experimental pressure vary between the inlet, throat, and outlet? Explain how this makes sense in terms of 1-D compressible flow theory. Discuss if there is evidence of viscous effects and choked flow.

4. Plot \( p_{\text{th}}/p_0 \) versus mass flowrate for your experimental data (symbols connected with straight lines), incompressible flow model (dashed line with no symbols), and compressible flow model (solid line with no symbols).

5. Plot \( Ma_{\text{th}} \) versus mass flowrate for your experimental data (symbols with straight lines), incompressible flow model (dashed line with no symbols), and compressible flow model (solid line with no symbols).

6. For both plots, explain the trends in your data and if they make sense. Explain the differences between experimental values and values predicted by your models. At what Mach number does the incompressible model begin to differ from the experimental data significantly? Is the agreement for the compressible flow model acceptable? If not which assumptions might be affecting the agreement?

7. For choked conditions, calculate \( p_{\text{th}}/p_0 \) and the mass flowrate predicted by the compressible flow model. Compare to your values calculated from your experimental measurements and your observations of the choking phenomena.

V. References


## DATA SHEET
### COMPRESSIBLE FLOW EXPERIMENT

<table>
<thead>
<tr>
<th>Date:</th>
<th>Engineers:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### p<sub>0</sub>: |

### T<sub>0</sub>: |

<table>
<thead>
<tr>
<th>No.</th>
<th>Throat Pressure P&lt;sub&gt;1&lt;/sub&gt; (kPa-v)</th>
<th>Outlet Pressure P&lt;sub&gt;2&lt;/sub&gt; (kPa-v)</th>
<th>Inlet Pressure P&lt;sub&gt;3&lt;/sub&gt; (Pa-v)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
9. WATER TURBINE PERFORMANCE TEST

I. Objectives

The objectives of this experiment are to (1) measure the performance characteristics of a hydraulic turbine, (2) calculate the Best Efficiency Point (BEP), and (3) calculate the specific speed of the turbine.

II. Introduction

Machines that extract energy from a fluid are called turbines. When liquid water is the fluid, the device is called a hydraulic turbine. Figure 9-1 shows an experiment from the previous fluid mechanics lab that was used to measure the performance of hydraulic turbine and calculate its specific speed. In the foreground of this picture is a weir used to measure flow rate. The mechanical engineering department is currently in the process of building a new test stand for a hydraulic turbine and this lab is unavailable for this quarter.
10. AXIAL FAN PERFORMANCE TEST

Figure 10-1 Axial fan.

I. Objectives

The objectives of this experiment are to obtain a set of performance curves for the axial fan shown in Figure 10-1 and to verify the fan laws by illustrating similarity.

II. Introduction

Axial fans are often used in ventilation systems. Modern designs can deliver large volumes of air against high resistances with high efficiency. One deficiency of an axial fan is its high rotational speed can generate significant noise (Ref. 1). To minimize energy consumption, it is desirable to select a fan for your application that will operate near its maximum efficiency or best efficiency point (BEP). Performance curves can be used to help identify operating conditions at BEP and other efficiencies for a particular fan (or pump). Performance curves typically include actual head rise (or energy delivered to the fluid), $h_a$, power driving the shaft (or brake horsepower), $W_{shaft}$, and overall efficiency, $\eta$, versus the volumetric flowrate, $Q$ (Ref 2.). These curves are usually obtained experimentally on a test stand similar to the one in our lab. Once performance curves have been obtained for one set of conditions, scaling laws can be used to predict the performance at different conditions such as higher operating speed or for a different size impeller.
Experiment

A schematic of the test stand used to obtain performance curves for the axial fan is shown in Figure 10-2. During operation, air at atmospheric pressure is sucked into the fan through the metal screen shown in Figure 10-1. The air is accelerated by the moving blades and exits the fan with a significant increase in velocity and a modest increase in pressure. It then moves through a duct to a Y-fitting, splits into two U-shaped legs before exiting back into the room via two variable air volume (VAV) dampers, all of which serve as a resistance to the flow and drop the pressure back to atmospheric. The magnitude of the flow resistance for the Y-fitting and U-shaped legs is investigated in another lab. The VAV dampers are boxes containing butterfly valves that can be adjusted using a pneumatic control system, thus changing the resistance of the system to airflow. The volumetric flowrate can also be varied by changing the rotational speed of the fan with a variable frequency drive (VFD). There are nine Pitot-static probes installed on the test facility for measuring the air velocity at the locations indicated in Figure 10-2. For this experiment, performance curves will be obtained at two rotational fan speeds using the VFD. The volumetric flowrate will be varied at both of these speeds using the VAV dampers.

![Figure 10-2 Axial fan test stand.](image)

Volumetric Flowrate

The volumetric flowrate through the fan, $Q_{\text{fan}}$, will be calculated from center-line velocities measured using the Pitot-static tubes at the end of the two U-shaped legs (locations 8 and 9 in Figure 10-2). By conservation of mass for an incompressible fluid, all of the air that enters the fan must exit from the two legs (assuming negligible leakage) giving

$$Q_{\text{fan}} = Q_8 + Q_9$$

where subscripts 8 and 9 indicate flowrates at the two Pitot-static tube locations. From our definition of average velocity, $\bar{v}$, we can write this as

$$Q_{\text{fan}} = \bar{v}_{\text{fan}} A_{\text{fan}} = \left(\bar{v}_8 + \bar{v}_9\right) A_{\text{duct}}$$

10-2
where \( A \) is area, the subscript \( \text{fan} \) corresponds to the fan with diameter \( D_{\text{fan}} = 21.2 \) in, and the subscript \( \text{duct} \) corresponds to the U-shaped legs with diameter \( D_{\text{duct}} = 16.0 \) in. Note that the Pitot-static tube will measure the center-line velocity, \( V_c \), instead of average velocity. Because we are measuring the velocity at the end of a long straight section of pipe at high flowrates it is reasonable to assume fully developed turbulent flow and approximate the average velocity by

\[
\frac{V}{V_c} = \frac{2n^2}{(n+1)(2n+1)}, \quad n = -1.7 + 1.8 \log \text{Re}_{V_c}
\]

where the Reynolds number is defined as \( \text{Re}_{V_c} = \rho V_c D_{\text{duct}} / \mu \), \( \rho \) is density, and \( \mu \) is viscosity. This correlation accounts for the shape of the turbulent velocity profile. Note that you need to evaluate Equation 10-3 separately for both the left and right U-shaped legs. Finally, recall that the Pitot-static tube will measure the dynamic pressure and that you will need to calculate the velocity using Equations 1-5 and 1-6 (which uses water density instead of air density).

**Actual Head Rise**

We will calculate the actual head rise, \( h_a \), from the static pressure rise across the fan, our calculated flowrate above, and a simple flow model. The relationship used for the flow through the fan is developed by applying the energy equation for incompressible, steady flow to a control volume with a single inlet and exit

\[
\left( \frac{p_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 \right) - \left( \frac{p_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 \right) = h_L - h_a
\]

where \( p \) is pressure, \( \rho \) is density, \( g \) is acceleration due to gravity, \( \alpha \) is the kinetic energy coefficient, \( V \) is average velocity, \( z \) is elevation, \( h_L \) is the head loss term, and subscripts 1 and 2 indicate the inlet and exit, respectively. We will define our control volume to extend from the still air in the room to the fan exit. Thus, the inlet control surface (1) is the ambient air in the room (where \( p_1 = p_{\text{atm}} \) and \( V_1 = 0 \)) and the outlet control surface (2) is the low pressure, high velocity air leaving the fan (where \( \alpha_2 = 1 \)). If we also assume constant elevation and account for the minor loss due to the fan screen we get

\[
\frac{p_{\text{atm}}}{\rho g} - \left( \frac{p_{\text{fan}}}{\rho g} + \frac{V_{\text{fan}}^2}{2g} \right) = K_L \frac{V_{\text{fan}}^2}{2g} - h_a
\]

where the subscript \( \text{fan} \) corresponds to fan exit conditions and \( K_L = 0.20 \) is the minor head loss coefficient for the fan screen. Solving for the actual head rise we get

\[
h_a = \frac{\Delta p_{\text{fan}}}{\rho g} + \left( 1 + K_L \right) \left( \frac{V_{\text{fan}}^2}{2g} \right)
\]

10-3
To evaluate Equation 10-6 we will use our measured static pressure rise across the fan and \( P_{fan} \) from the previous section. Finally, we will use British Gravitational units for these calculations. This results in the actual head gain having units of feet of air which we will need for later calculations. However, for our plots it is more common to use inches of H\(_2\)O. To convert units we have to use the hydrostatic equation

\[ \Delta p = \rho_{air} g h_{a,air} = \rho_{H_2O} g h_{a,H_2O} \]

which can be solved for the height for H\(_2\)O as

\[ h_{a,H_2O} = \left( \frac{\rho_{air}}{\rho_{H_2O}} \right) h_{a,air} \]

and then use the standard conversion to go from feet to inches.

**Brake Horsepower**

For this lab the fan’s rotational speed, \( \omega \), will be set using the VFD to two operating conditions. The VFD adjusts the voltage and current to the electric motor to maintain the predetermined set point. The VFD displays the current and voltage supplied to the electric motor. From this information the power used by the motor can be calculated. The electric motor uses 3-phase 460-Volt AC. The power to the motor is thus calculated as

\[ W_{in} = \sqrt{3} V_{LL} I_L \cos \theta \]

where \( V_{LL} \) is the RMS line-to-line voltage, \( I_L \) is the RMS line current, and \( \cos \theta = 0.75 \) is the power factor or the phase difference between the voltage and the current. We will use this power as an estimate for the brake horsepower, \( W_{shaft} \). The power supplied to the motor will be higher than the power supplied to the shaft due to inefficiencies in the motor.

**Efficiency**

The overall fan efficiency, \( \eta \), is calculated by

\[ \eta = \frac{\mathcal{P}_f}{W_{shaft}} \]

where \( \mathcal{P}_f \) is the power gained by the fluid. Thus, if all of the power put into the shaft is delivered to the fluid the efficiency would be 100\%. In reality the actual efficiency must always be lower than 100\% mainly due to friction and unwanted recirculation of the fluid. The power gained by the fluid can be easily calculated from the actual head gain using

\[ \mathcal{P}_f = \rho g Q_{fan} h_a \]
where the units for \( h_a \) should be feet of air for British Gravitational units and air as the operating fluid. Thus, we can use our results from the last three sections for \( Q_{\text{fan}} \), \( h_a \), and \( W_{\text{shaft}} \approx W_{\text{in}} \) to calculate the efficiency using Equations 10-10 and 10-11.

**Similarity Laws**

The similarity laws are a great tool for predicting pump performance at different operating conditions (typically where test data are not available). We will verify that they are reasonably accurate for predicting performance for a specific turbo-machine at different operating speeds. To do this we will scale the data obtained at our lower operating speed to the higher operating speed tested. We will then compare this scaled data to the actual data obtained during the higher operating speed test.

The scaling laws are developed by performing a dimensional analysis on the parameters that govern turbo-machinery performance. From our discussion above, the dependent variables that define performance are \( h_a \), \( W_{\text{shaft}} \), and \( \eta \). We already know that \( Q_{\text{fan}} \) and the shaft rotational speed, \( \omega \), are two of the independent variables. In addition, based on our experience we can add the size of the turbo-machine or impeller, \( D_{\text{fan}} \), and the air density, \( \rho \). Other parameters such as viscosity, surface roughness, and additional length scales can also be included, but at high Reynolds numbers and for geometrically similar pumps it is reasonable to neglect these variables. Thus, we get the following functional dependence for each of our parameters

\[
\begin{align*}
    h_a &= f_1(Q_{\text{fan}}, \omega, D_{\text{fan}}, \rho) \\
    W_{\text{shaft}} &= f_2(Q_{\text{fan}}, \omega, D_{\text{fan}}, \rho) \\
    \eta &= f_3(Q_{\text{fan}}, \omega, D_{\text{fan}}, \rho)
\end{align*}
\]  

(10-12)

Then, using the Buckingham Pi theorem to form dimensionless groups these can be rewritten as

\[
\begin{align*}
    C_H &= \frac{g h_a}{\omega^2 D_{\text{fan}}^2} = \phi_1 \left( \frac{Q_{\text{fan}}}{\omega D_{\text{fan}}^3} \right) \\
    C_{\phi'} &= \frac{W_{\text{shaft}}}{\rho \omega^3 D_{\text{fan}}^5} = \phi_2 \left( \frac{Q_{\text{fan}}}{\omega D_{\text{fan}}^3} \right) \quad \text{where} \quad C_Q = \frac{Q_{\text{fan}}}{\omega D_{\text{fan}}^3} \quad (10-13) \\
    \eta &= \frac{\rho g Q_{\text{fan}} h_a}{W_{\text{shaft}}} = \phi_3 \left( \frac{Q_{\text{fan}}}{\omega D_{\text{fan}}^3} \right)
\end{align*}
\]

By definition, if we match the independent parameter \( C_Q \) then the dependent parameters (in this case \( C_H \), \( C_{\phi'} \), and \( \eta \)) must also match. Note that the units must cancel for each dimensionless parameter. Thus, for British Gravitational units and air as the fluid operating fluid the units should be feet of air for \( h_a \), radians/s for \( \omega \), ft-lb/s for \( W_{\text{shaft}} \), and ft\(^3\)/s for \( Q_{\text{fan}} \).
III. Experiment

Equipment

Axial Fan Experimental Apparatus
Pressurized Air

Preparations for Test (Completed by Instructor, students should NOT perform)

1. Connect compressed air hose from supply to VAV controller.

2. Adjust the VAV pressure regulator knob for a pressure of 20 psia (Figure 10-3). Ensure that both the Right and Left VAV pressure regulator knobs are fully counterclockwise.

3. Unlock and shut the Main Disconnect (Figure 10-4).

4. Set the system to Drive. **Warning: Do not turn the system to Bypass! This will supply 100% power to the fan and could cause damage.**

Procedure

1. On the variable frequency drive (VFD), press HAND START (Figure 10-4). Press the + arrow until 75% is indicated in the upper right of the display. This corresponds to a rotational speed of 1320 rpm. The fan will slowly speed up to this value.

2. While the fan speeds up, complete the top of the two data sheets. There is one data sheet for each of the two speeds that will be performed. Also press DISPLAY MODE twice on the VFD (Figure 10-4) to display motor voltage.
3. Once the fan is at 1320 RPM, record the instruments listed below. Refer to Figure 10-5 for instrument location.

   a. Fan Static Pressure, $\Delta p_{\text{fan}}$, in inches of water from the static port only of the Pitot-static tube shown in Figure 10-6. For values greater than 2-in H$_2$O, use the manometer located above this gage. Verify before taking your first reading that the right hand valve located directly under the gage is open. For ball valves, the valve is open when the handle is inline with the tube.

   b. Motor Voltage, $V_{LL}$, in volts AC. The motor voltage is in the center of the VFD display (Figure 10-4).

   c. Motor Current, $I_{L}$, in amps AC. The motor current is at the top of the VFD display (Figure 10-4).

   d. Left duct Pitot-static tube reading in inches of water from the Pitot-static tube shown in Figure 10-7. Verify before taking your first reading that the #9 TP (total pressure) and #9 SP (static pressure) ball valves beneath the gage are open and that all the other valves are shut. For values greater than 0.5-in H$_2$O, use the inclined manometer located above the gages. This inclined manometer measures the same pressure difference; thus, make sure that the readings are approximately the same by adjusting the zero on the inclined manometer until both instrument readings match.

   e. Right duct Pitot-static tube reading in inches of water. Verify before taking your first reading that the #8 TP and #8 SP ball valves beneath the gage are open and that all the other valves are shut.
Figure 10-5 Differential pressure gages.

Figure 10-6 Pitot-static tube used to measure fan static pressure.

Figure 10-7 Pitot-static tube #9 used to measure centerline velocity in left duct.
4. Adjust the flowrate in the system by shutting the right VAV damper slightly. Do this by turning the pressure regulator knob for the right VAV damper clockwise (Figure 10-3). Note that this is not exact or linear. Adjust the regulator in small increments and watch for changes in duct velocity and static pressure. Allow the system to return to steady state before taking readings. As you shut the right damper you will notice an increase in velocity in the left duct as more airflow is forced through the left duct. Take the same readings as in Step 3. Repeat until the right damper is completely shut. You may have to reopen the duct and repeat the process to get sufficient data (about 4-5 data points).

5. Continue to reduce the flowrate by shutting the left VAV damper. Do this by turning the pressure regulator knob for the left VAV damper clockwise (Figure 10-3). Like the right VAV damper, adjust the regulator in small increments and watch for changes in duct velocity and static pressure. As you shut the left damper the fan will go through stall. You will notice this condition when the instrument readings become unsteady and the fan noise changes pitch. Continue to take the same readings as in Step 3 through the fan stall (making sure to note in the comments when it occurs) until the left duct velocity is less than 0.01-in H₂O. Try to get at least 5-6 data points.

6. Open both VAV dampers fully by turning both pressure regulator knobs fully counterclockwise. Increase the speed of the fan to 100% by pressing HAND START on the VFD display and then pushing the + arrow until 100% is displayed in the upper right hand corner.

7. Press DISPLAY MODE twice to show motor voltage. When rotational speed reaches 100% or 1750 RPM, repeat steps 3 through 6, recording data on the second data sheet.

8. Open both VAV dampers fully by turning both pressure regulator knobs fully counterclockwise. Decrease fan speed to 20% by pressing HAND START on the VFD display and then push the – arrow until 20% is displayed in the upper right hand corner.
9. When the fan speed reaches 20%, press the OFF STOP button on the VFD display to remove power to the fan. When the fan stops spinning, please tell the instructor so they can turn off the VFD.

IV. Report

1. Tabulate all of the collected experimental data.

2. Calculate and tabulate volumetric flowrate (ft³/min or cfm), actual head gain (in H₂O), motor power in (hp), and fan efficiency. Include sample calculations with your table.

3. Plot actual head gain (in H₂O) versus flowrate (cfm) for both rotational speeds on one figure. Use symbols for the data points with no line. Add a 3rd order polynomial curve fit (or trendline) for each data set. Explain “stall” and how it corresponds to your figure.

4. Plot motor power (hp) versus flowrate (cfm) for both rotational speeds on one figure. Use symbols for the data points with no line. Add a 3rd order polynomial curve fit (or trendline) for each data set.

5. Plot total efficiency versus flowrate (cfm) for both rotational speeds on one figure. Use symbols for the data points with no line. Add a 3rd order polynomial curve fit (or trendline) for each data set. Under what conditions would you prefer to operate this fan?

6. Use the scaling laws to predict the head gain, motor power, and efficiency for the higher speed (1750 RPM) based on the low speed experimental data (1320 RPM). Plot the predicted values versus flowrate on the corresponding figures from items 3-5. Use symbols for each of the scaled data points. Include sample calculations and a table with your figures. Comment on the ability of the scaling laws to predict the actual behavior.

7. Plot dimensionless head rise coefficient, $C_H$, versus dimensionless flow coefficient, $C_Q$, for both rotational speeds on one figure. Have each axis start at 0. Note that both of these coefficients typically lie in the range of 0 to 0.1 for an axial fan. Use symbols for the data points with no line. Add a 3rd order polynomial curve fit (or trendline) for each data set. Include sample calculations and a table with your figures. Does the plot of $C_H$ versus $C_Q$ collapse onto a single line? Is this expected?

8. Demonstrate why the assumption of incompressible flow is reasonable.

V. References


DATA SHEET
AXIAL FAN PERFORMANCE TEST

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
DATA SHEET
AXIAL FAN PERFORMANCE TEST

<table>
<thead>
<tr>
<th>Date:</th>
<th>Engineers:</th>
</tr>
</thead>
<tbody>
<tr>
<td>RPM:</td>
<td></td>
</tr>
<tr>
<td>$\rho_{\text{amb}}$:</td>
<td></td>
</tr>
<tr>
<td>$T_{\text{amb}}$:</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>No.</th>
<th>Static Pressure (in H$_2$O)</th>
<th>Motor Voltage (Volts-AC)</th>
<th>Motor Current (Amps)</th>
<th>Left Flow (in H$_2$O)</th>
<th>Right Flow (in H$_2$O)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
APPENDIX A

TEXTBOOK READING


1. Sections 3.5-3.6
2. Sections 5.1-5.2
3. Sections 6.2-6.7
4. Sections 8.1-8.4
5. Sections 8.1-8.4
6. Sections 9.2
7. Sections 9.3
8. Sections 11.1-11.4
9. Sections 12.1-4, 12.8
10. Sections 12.5-7


1. Section 6.3
2. Sections 4.3-4
3. Sections 6.1-6.3, 6.7
4. Sections 8.1, 8.4-7
5. Sections 8.4-7
7. Sections 9.7
8. Chapter 12 and Sections 13.1-2
9. Section 10.4-5
10. Section 10.4-5
APPENDIX B
MEASUREMENT ERROR REVIEW

I. Introduction

All measurements have some error defined as the difference between the measured value and the true value of a variable. Uncertainty is an estimate of the error in a measured variable.

II. Minimum Error

The error in any measurement is at least the minimum error defined as plus or minus one-half the least count

\[
\text{minimum error} = \pm \frac{1}{2} \text{(least count)} \quad (B-1)
\]

where the least count is the resolution of the instrument or the smallest measurement division. This error is also referred to as resolution error.

Note that the error may be greater than the minimum error because of the nature of the reading. For example, you can measure your height with a scale that reads to the nearest 0.1 millimeter. However, can you measure your height that accurately with your eye? Sometimes your measuring device is more accurate than your measurement, therefore you should claim a higher uncertainty.

III. Instrument (or Calibration) Error

There may be significant error for your instrument reading due to errors in your calibration. In particular, the device you read can only be as accurate as the device used for calibration. Additional error can result from uncertainties in the calibration curve-fit parameters. Factors that can affect this error are variation in device behavior with operating temperature, hysteresis, and sensitivity to variations in local atmospheric pressure and gravity. Calibration uncertainties are typically available on the device or in their operating manual.

IV. Statistical Error

When measurements are fluctuating, a single measurement is not sufficient to calculate the uncertainty for the quantity being measured. An error due to statistical fluctuations, also called random error or precision error, should be estimated. To estimate the statistical error, or calculate the uncertainty, we will assume the following: (1) Gaussian or normal distribution for the experimental data. Thus, a histogram of many data points should look like a “bell curve.” (2) The variable being measured is continuous, like pressure or length, and not discrete, like the number of students. (3) The population (number of possible measurements) is infinite. For the infinite
population, we define the population mean as \( \mu \) and the population standard deviation as \( \sigma \). Each data sample will have a finite number of measurements, \( n \). The sample mean, \( \bar{x} \), is calculated using

\[
\bar{x} = \frac{1}{n} \sum_{i=1}^{n} x_i
\]  

and the sample standard deviation, \( s \), is calculated using

\[
s = \left[ \frac{1}{(n-1)} \sum_{i=1}^{n} (x_i - \bar{x})^2 \right]^{1/2}.
\]  

**Estimating \( \mu \) and Error for Known \( \sigma \)**

In the rare case when the population standard deviation is known, an estimate of the population mean and its uncertainty (the \( \pm \) value) with 95% confidence is

\[
\mu \equiv \bar{x} \pm \frac{1.96\sigma}{\sqrt{n}} \text{ (95% confidence)} \quad (B-4)
\]

and an estimate of an instantaneous value (or single value of the population), \( x_i \), and its uncertainty with 95% confidence is

\[
x_i \equiv \bar{x} \pm 1.96\sigma \text{ (95% confidence)} \quad (B-5)
\]

Note that Equations B-4 and B-5 are applicable to any size sample as long as \( \sigma \) is known or can be estimated.

**Estimating \( \mu \) and Error for Unknown \( \sigma \)**

Typically, measurements are taken and nothing is known about the standard deviation of the actual population. The standard deviation of the sample is the only information we have to calculate the uncertainty. A statistical analysis known as the Student t test can be employed to estimate the population mean and its uncertainty as

\[
\mu \equiv \bar{x} \pm \frac{t_s}{\sqrt{n}}
\]  

and to estimate an instantaneous value and its uncertainty as

\[
x_i \equiv \bar{x} \pm t_s
\]  

(B-7)
The value of $t$ varies with confidence level and the number of measurements, $n$. For 95% confidence, the values of $t$ are listed in Table 1 as a function of $n-1$. Note that as $n$ becomes large the value for $t$ becomes 1.960 as our estimate for $s$ is approaching $\sigma$. As expected, this causes Equations 6 and 7 to reduce to Equations 4 and 5, respectively.

<table>
<thead>
<tr>
<th>$n-1$</th>
<th>$t$</th>
<th>$n-1$</th>
<th>$t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12.706</td>
<td>8</td>
<td>2.306</td>
</tr>
<tr>
<td>2</td>
<td>4.303</td>
<td>9</td>
<td>2.262</td>
</tr>
<tr>
<td>3</td>
<td>3.182</td>
<td>10</td>
<td>2.228</td>
</tr>
<tr>
<td>4</td>
<td>2.776</td>
<td>15</td>
<td>2.131</td>
</tr>
<tr>
<td>5</td>
<td>2.571</td>
<td>20</td>
<td>2.086</td>
</tr>
<tr>
<td>6</td>
<td>2.447</td>
<td>30</td>
<td>2.042</td>
</tr>
<tr>
<td>7</td>
<td>2.365</td>
<td></td>
<td>1.960</td>
</tr>
</tbody>
</table>

### Estimating $\mu$ and Error for Unknown $\sigma$ and $s$

There are times when neither $\sigma$ nor $s$ are readily available, but you may still be able to estimate the uncertainty. For example, suppose you are estimating the mean level of water flowing through an open channel. Visually you observe that, despite the fluctuation in the water level, the level never fluctuates by more than $\pm$ 0.5 inch. You can then replace 1.96 $\sigma$ term in Equation 4 or 5 with 0.5 inch to get

$$\mu = \bar{x} \pm \frac{0.5 \text{ in.}}{\sqrt{n}}.$$  \hspace{1cm} (B-8)

As can be seen by the above, the error in estimating $\mu$ may still be reduced by taking a large sample of measurements and averaging them, since the uncertainty is still divided by $\sqrt{n}$.

### V. Total Error

The total error is the combination of all the individual errors in a measurement. The total uncertainty is calculated by taking the root-sum-square of all the individual uncertainties, $u_i$, including those due to minimum error, calibration errors, and statistical error as follows

$$\text{Total Error} \equiv \sqrt{u_1^2 + u_2^2 + u_3^2 + \cdots}.$$  \hspace{1cm} (B-9)

Note that if the measurement does not fluctuate this means that the statistical error is very small compared to the minimum error and the statistical error can be neglected. Conversely, if the measurement fluctuations are much greater than the minimum error the statistical error will dominate and the minimum error can be neglected.
VI. Uncertainty in Calculated Quantities

Once you have made measurements in the laboratory, you are often going to use these values to calculate other quantities. How do the uncertainties in your measurements affect the calculated result? We will use a technique called Uncertainty Propagation to estimate this uncertainty. We begin by defining a function \( R = R(x_1, x_2, \ldots) \), where \( x_1, x_2, \ldots \) are measured quantities. These measured quantities have uncertainties \( u_{x_1}, u_{x_2}, \ldots \). For the equations below, all measured quantities are assumed to be independent and the uncertainties are assumed to have the same confidence level.

General Uncertainty Propagation

The uncertainty in \( R \) can be determined by

\[
    u_R = \sqrt{u_{R,x_1}^2 + u_{R,x_2}^2 + \cdots},
\]

where

\[
    u_{R,x_i} = \frac{\partial R}{\partial x_i} u_{x_i}
\]

This technique will be illustrated by the example below.

**Given:** Consider the following function

\[
    F = \frac{AB^2}{C^{1/3}}
\]

where \( A \) is measured to be \( 30 \pm 5 \text{ kg} \), \( B = 1.5 \text{ m/s} \pm 5\% \), and \( C = 10.7 \pm 1.3 \text{ m}^3 \). Calculate \( F \) and its uncertainty.

**Solution:** First, the nominal value of \( F \) is found by substituting in nominal values of \( A, B, \) and \( C \)

\[
    F = \frac{AB^2}{C^{1/3}}
\]
To find the propagated uncertainty, first we find the relative errors, $u_{r,x}$, using Equation B-11:

$$u_{F,A} = \frac{\partial F}{\partial A} u_A$$

$$u_{F,B} = \frac{B^2}{C^{4/3}} u_B = F \frac{u_A}{A}$$

$$u_{F,C} = -\frac{1}{3} F \frac{u_C}{C}$$

Finally, substituting these values into Equation (9), we obtain:

$$u_F = \sqrt{u_{F,A}^2 + u_{F,B}^2 + u_{F,C}^2}$$
Therefore, the final answer is

\[ F = 31 \pm 6 \text{ N} \]

**Uncertainty Propagation for Addition and Subtraction**

This method applies the result of the *General Uncertainty Propagation* section to the special case when \( R \) is formed only by adding or subtracting all the \( x_i \)'s multiplied by constant coefficients, \( C_i \)'s. Thus, \( R \) has the special form

\[ R = R(x_1, x_2, \ldots) = C_1 x_1 + C_2 x_2 + \ldots \]  

(B-12)

The uncertainty in \( R \) can then be determined by

\[ u_R = \sqrt{(C_1 u_{x_1})^2 + (C_2 u_{x_2})^2 + \cdots} \]  

(B-13)

**Uncertainty Propagation for Multiplication and Division**

This method applies the result of the *General Uncertainty Propagation* section to the special case when \( R \) is formed only by multiplying or dividing all the \( x_i \)'s multiplied by a constant, \( C \). Thus, \( R \) has the special form

\[ R = C \ x_1^{m_1} x_2^{m_2} \ldots \]  

(B-14)

where the \( m_i \)'s are constant exponents. The uncertainty in \( R \) can then be determined by

\[ \frac{u_R}{R} = \sqrt{ \left( m_1 \frac{u_{x_1}}{x_1} \right)^2 + \left( m_2 \frac{u_{x_2}}{x_2} \right)^2 + \cdots} \]  

(B-15)

where each term is given in dimensionless form called the *fractional or percent uncertainty*. This method will be used to solve for the uncertainty for the same example given above.

**Solution:** To find the propagated fractional uncertainty substitute \( F \) into using Equation B-11

\[ \frac{u_F}{F} = \sqrt{ \left( \frac{u_A}{A} \right)^2 + \left( 2 \frac{u_B}{B} \right)^2 + \left( \frac{1}{3} \frac{u_C}{C} \right)^2} \]
\[
\frac{u_F}{F} = \sqrt{\left(\frac{5 \text{ kg}}{30 \text{ kg}}\right)^2 + (2 \times 0.05)^2 + \left(\frac{1}{3} \times \frac{1.3 \text{ m}^3}{10.7 \text{ m}^3}\right)^2} = 0.1985 = 19.85\%
\]

\[
u_F = 0.1985 \times 30.63 \text{ N} = 6.08 \text{ N}
\]

using the earlier result for \(F\). This is consistent with our earlier result for \(u_F\).