AERO-THERMAL-FLUIDS LABORATORY

ME 43600

The City College of New York
Department of Mechanical Engineering
Steinman Hall 233
160 Convent Avenue
New York City, NY 10031

Last revision by O. Goushcha on January 24, 2019

TABLE OF CONTENTS

Chapter 1

INTRODUCTION

1.1 Preface

Welcome to the ME 43600, Aero-thermal-Fluids Laboratory. The overall goals of this course for engineers are shaped by a variety of applications in Mechanical engineering, which include theoretical and practical concepts in fluid mechanics, heat transfer, thermodynamics and aerodynamics. Accordingly the following key objectives have been adopted in this laboratory.

1. To demonstrate principles, phenomena, practices and systems taught in fluid mechanics, heat transfer, thermodynamics, and aerodynamics.
2. To foster appreciation for engineering measurements and their uncertainties.
3. To demonstrate the use of advanced techniques and instrumentation in carrying out specific experimental tasks.
4. To allow hands-on experience with experimental setups, instrumentation, and procedures in a safety concern environment.
5. To foster appropriate oral and written professional communications.

Your cooperation is needed in achieving the above-mentioned objectives. Prepare adequately before each laboratory session, obey the safety rules during experimentation and follow closely the recommended procedures.

1.2 Safety in the laboratory

Safety is the important priority in all laboratory session. This is the reason why it is being discussed first in this manual.

Laboratory work may involve the operation of equipment that can cause injury. Every precaution must be taken to ensure safety.

Do not proceed with your work if you have any doubts as to the operation of any instrument. Consult the instructor to assist you. Do not wear loose clothing (ties, long sleeves) or jewelry. Long hair must be tied or covered. Be especially careful of rotating parts, power lines and heated surfaces.

Do not switch ON power or any equipment before you have had a chance to read its operational manual or have asked your instructor. When you finish your experiment switch OFF carefully all equipment and instrumentation, if some equipment is left ON for a long time we run an unnecessary risk with the potential for an accident. Always remember that negligence may cause fatalities.

1.3 Course organization and conduct

Each experiment in this course is designed to present distinct theoretical and practical topics. Prior to performing an experiment, students are prepared for it by reviewing the underlying theory and understanding its purpose. Every student should acquire the fundamental knowledge necessary to conduct the experiment, including the operation of the experimental apparatus, measuring instruments, data acquisition systems and related software. The collected data should be analyzed to determine the uncertainty in the results by estimating errors in the measured quantities. Finally, each student will submit a report in which the experimental study is described and presented.

The Aero-thermal-fluids Laboratory course consists of six experiments. Each experiment is scheduled for two consecutive classes (6 hours). The first week (3 hours) is devoted to data collection and the subsequent meeting to computation and analysis of data. In some cases the second meeting can be used to repeat the experiment in the case the original data is insufficient.

Students are divided into groups of three or four. Each group will work together to perform an experiment assigned to them by the instructor. Data from the experiment is shared with each group member. The processing of data can be done in groups of at most two students. The final experiment report is individual to each student.

The first two sessions (6 hours) are dedicated to reviewing relevant theories associated with the experiments and familiarization with the instrumentation and data acquisition systems.

1.4 Troubleshooting

Most of the present experimental setups can be considered as complicated systems consisting of many electronic or mechanical components. It is almost certain that one or more of these components will break down or malfunction at a given time, causing considerable disruption of your experimental work. In this case, report the problem to your instructor or technician. Your instructor may ask you to return a set of commonsense diagnostics to pinpoint the location of the trouble and try to remedy it. An engineer should be able to identify the major cause of the problem and fix it as soon as possible. This is not a straightforward procedure and some training and prior experience is required. The laboratory environment and the existing complicated instrumentation provide an excellent opportunity to acquire some experience in the art of troubleshooting.

1.5 Experimental Report

Your laboratory experience and the results of your data analysis should be described and presented in the final report. Use Times New Roman font in size 10 for text. Use Equation Editor or any similar suitable software in order to type the equations; *do not copy and paste them from the class manual*. Plot as many data as possible in one graph but avoid any overcrowded look. Do not leave gaps in the text more than what is equivalent to two lines. A report with large characters and many gaps is aesthetically unacceptable. In the report you should use the double column format and incorporate the graphs accordingly in the proper position or put them all together at the end of the report. The format of the report is shown as follows:

Experiment Name
ME 436 Aerothermal Fluids Laboratory
Author Name
Experiment #
(e.g Report 2)

Date of Submission
Mechanical Engineering Department
The City College of New York, USA

Abstract (8pts)
This is a very brief, self-contained summary of the report. Only by reading its abstract, one should be able to decide whether it is useful or not to read the rest of the paper. Write *one or two sentences* each for the following (do not include these headings, write in paragraph format). **1) Subject:** Description of the phenomenon investigated. **2) Purpose:** What is the end goal of this study?. **3) Methods:** The *essentials* (not the details) of methods used for the experiment, what are you measuring to get the goal result? **4) Results:** A summary of results obtained, what *numbers* for your final values did you get? **5) Conclusions:** What conclusions can you draw from all the numbers, comparisons, trends…etc from the results you gave before? Are they expected? Are they satisfactory? Use no more than 200 words. No references to figures, tables or citations are done. Complicated equations should be avoided.

Introduction (12 pts)
First describe the physical phenomena as concise as possible. Describe what we want to study. In order to study this phenomena, describe what data we need to collect. Provide formulas that show how we can obtain our result if we collect above mentioned data. No need to show formulas for everything, just the main ones. Cite the reference you have found them in. When giving references use the reference number in brackets [1]. Use your own words and do not copy sentences from this manual or other textbook.

Experimental Setup and Procedure (15pts)
Now that you have described the data you need to collect, explain what tools we need to obtain these data. Start by describing in detail the major apparatus. Next, describe the sensors used and their location to collect data. *Include a figure* of the apparatus and annotate each component. Describe in paragraph format the procedure that you have followed. Be specific and mention what values for certain parameters you have used.

Results (15 pts)
In this section present your results. You should reference and describe each figure and table that you present in Appendix A. For each figure describe what is plotted, what the axis are, and what the data trend shows.

Conclusions (5pts)
Based on the Results, what is the major conclusion about the phenomenon that you studied? Was the experiment successful? What would you change?

List of References
All the references cited [1] in the report should be listed here. Use the following format:

[1] Holman, J.P. *Experimental Methods for Engineers* , 7th ed., McGraw-Hill, 2001.
[2]…etc.

Appendix A (20pts)

Place the plots and tables you referred in the text here. Keep double column structure unless the plots must be placed bigger for visual clarity. Refer to the "plot and table formatting" section.

Appendix B (20pts)

Show a sample of all calculations for *one data point* you (MATLAB) have made to process your raw data. After each calculation show the calculation for propagating uncertainty. *Use Equation Editor!* Start at the very beginning, usually this mean by converting all units to SI standard. Here is an example:

$$x_m = 0.0254 \cdot x_{in} = 0.0254 \cdot 2.1 = 0.053\ m$$

$$u_{x_m} = \left(\left(\frac{\partial x_m}{\partial x_{in}} u_{x_{in}}\right)^2\right)^{\frac{1}{2}} = 0.0254 \cdot u_{x_{in}}$$
$$= 0.0254 \cdot 0.1$$
$$= \pm 0.00254\ m$$

$$P = \rho g\ x_m \sin\alpha = 1.23 \cdot 9.8 \cdot 0.053 \cdot \sin 5$$
$$= 0.056\ Pa$$
$$u_P = \left(\left(\frac{\partial P}{\partial x_m} u_{x_m}\right)^2\right)^{\frac{1}{2}} = \rho g\ \sin\alpha\ u_{x_m}$$
$$= 1.2 \cdot 9.8 \cdot \sin 5 \cdot 0.00254$$
$$= \pm 0.0027\ Pa$$

Any sample integrals are shown with just a result:

$$Q = 2\pi \int r\, u_r\, dr = 0.54 \frac{m^3}{s}$$

Appendix C
Place the tabulation of raw data here
Appendix D
It contains computer programs if available.

The reports will be typed on a computer and submitted as a paper copy. Typographical errors and improper use of language may deduct points from your score.
Do not copy and paste any material (sentence, equation, plot, diagram etc) from any reference. Make your own sentences and diagrams. Learn how to produce simple diagrams in your office software and use them. Once you make such diagrams, you can save them and modify if you want to use again. If you think that reproducing a plot or diagram is absolutely impractical you can use it by **giving the appropriate reference**.

The text part of your report may not exceed two pages. Appendices A and B may not exceed three pages in total.

1.6 Formatting of plots

A major part of this course deals with technical writing. Clearly presenting your results in figures is crucial in writing your report. A sample of a well formatted plot is shown in Figure 1. Follow these guidelines in order to receive full credit.

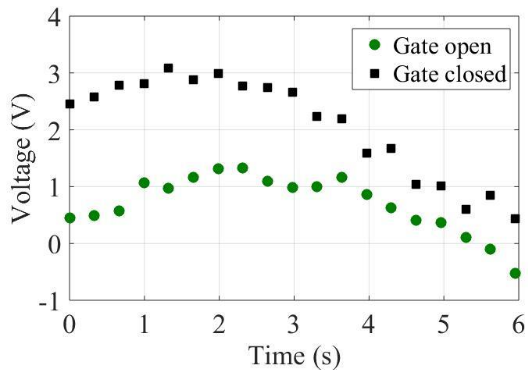

Figure 1: Sample of a formatted plot.

1) Font name and size have to be consistent with the rest of the text. (2pts per plot)
2) Each data series is presented with distinct color or marker. Legend describes each data series (unless only one data series is shown). (2pts per plot)
3) Plot is not cluttered and each series can be seen distinctly. Sometimes it is useful to reduce number of data series presented to show results clearly. (2pts per plot)
4) Unique figure caption is placed at the bottom of each plot. (2pts per plot)
5) Size of the figures is uniform. (2pts per plot)

A MATLAB code will be provided to assist you with formatting your figures.

Chapter 2

UNCERTAINTY ANALYSIS

How much can I trust my data, anyway?
If you want to measure the distance between your apartment and this classroom, how would you do it? You could drive there, and look in the change of your car's odometer. This would give you an answer in tenths of a mile, but you would probably agree that the real distance could be several hundred feet different. If you really want to know the answer badly, you could hire a surveyor to triangulate between the buildings. He might tell you that the distance is 0.723 miles, but warn you that the answer might be off by a few feet in either direction. Frustrated that no one will give you a straight answer; you decide to measure the distance directly with the ruler, foot by foot. After a day's worth of tedious labor, you get a result of 0.7236 miles (3288 feet and 9 inches). Would you believe that you were off by an inch? How about a tenth of an inch?

The point is that is not possible to measure things completely accurately. We are limited by our capacity of error, by the quality of our tools, and ultimately by our inability to control the state of what we are measuring. Since the scientific method demands that we compare our theories and models to what is really happening, we need to at least guess what the size of the errors in our measurements are.

These guesses are called uncertainties. The first step in taking data is always to estimate the uncertainty of your measurements. This process is an even mixture of common sense and technical knowledge. Before we start to uncertainty analysis, let's first get some understanding on the error analysis, which is an essential technical tool.

2.1 Definition of error

Measurements and calculations are a vital part of our real world engineering activities. Both however are associated with errors. Error, $\delta,$ is the difference between a measured or assigned value and the true value is not known a priori.

$$x_{measured} = x_{true} \pm \delta \tag{2.1}$$

Often a measurement is subjected to multiple errors $\pm\delta_i$ from various sources (each source is labeled with index i).

$$x_{measured} = x_{true} \pm \delta_1 \pm \delta_2 \pm \delta_3 \ldots$$

These errors can come from various sources in the experimental setup and are split into two categories. Errors that do not change value in the measurement period are called *bias* or *systematic* error ($\pm\beta_i$). Errors that do fluctuate in time during the measurement are called *random* error ($\pm\epsilon_i$).

During experimental measurements both x_{true} and δ are unknown. Therefore, an uncertainty interval $\pm u$ can be prescribed around $x_{measured}$ to cover possible variations in $x_{measured}$ due to δ to include x_{true}.

$$x_{true} = x_{measured} \pm u \tag{2.2}$$

A relative uncertainty is defined as a ratio of uncertainty to the measured value

$$\frac{\pm u}{x_{measured}} \tag{2.3}$$

Relative uncertainty is unitless and is often expressed in percent.

2.2 Uncertainty in measurements

2.2.1 Systematic error uncertainty

In this laboratory you will be collecting data using several analogue measuring devices whose value does not change during the measurement period. The only source of uncertainty is therefore due to systematic error. This uncertainty is usually specified by the manufacturer. If not, then it is customary to estimate the uncertainty as a *half of the minimum division of the readout.*

For example, a pressure gauge is shown in the Figure 2. The minimum division of the readout is 0.5psi. Therefore, the uncertainty of this pressure gauge is 0.25psi. The mean measurement and the uncertainty are:

Figure 2: Typical pressure gauge

$$P = 5.5\,psi$$

$$u_P = 0.25\,psi$$

This measurement can be written as:

$$\frac{u_P}{P} = \frac{0.25\,psi}{5.5\,psi} = 0.046 = 4.6\%$$

Or

$$P + u_P = 5.5 \pm 0.25 psi$$

2.2.2 Random error uncertainty

Random error is encountered when the value of the measurement changes in the time it takes to collect data. For example, consider a temperature measurement using a thermocouple. Thermocouple is a very sensitive device that will pick up small variations in temperature. Therefore, the reading will fluctuate slightly in time. To get a good measurement, we need to collect N data points $(x_1, x_2, \ldots, x_N)$ and take an average of those values $\bar{x}$.

The uncertainty of such instruments consists of two parts. First one is the uncertainty due to systematic error, which is either specified by the manufacturer of the sensor or by the half of the minimum division on the readout. We will call this uncertainty $\pm u_b$. The second one, $\pm u_\epsilon$, is due to the natural fluctuations of data over time. We can combine the two uncertainties using sum of squares into one uncertainty for the fluctuating measurement:

$$u = \sqrt{(u_b)^2 + (u_\epsilon)^2}$$

To estimate of the fluctuating uncertainty u_ϵ, we use standard deviation of the signal:

$$s_x = \sqrt{\frac{\sum_{n=1}^{N}(x_n - \bar{x})^2}{N-1}}$$

Assuming the measured signal has a Gaussian distribution around the mean (this is a good assumption based on central limiting theorem for measurement with multiple sources of error such as systematic and random) we can use the standard deviation as a measure of the random uncertainty. Consider data containing infinite number of samples, with standard deviation σ. If these data have a Gaussian distribution (described in more detail in Section 2.3), 68.3% of all data will lie within $\pm 1\ \sigma$ of the mean value. 95% of all data lie within $\pm 1.96\ \sigma$ of the mean value. Therefore, we can prescribe with 95% confidence that

$$u_\epsilon = \pm 1.96\sigma$$

This formulation will give an uncertainty with 95% confidence if we took infinitely many points. In reality, we are taking a finite number of measurements with standard deviation s_x and the mean $\bar{x}$. Question is now close is $\bar{x}$ to μ, mean of the parent population? We can take multiple samples each with N data point in the Gaussian-distributed signal and compute multiple mean $\overline{x_i}$. Each mean will be different. Their standard deviation is related to the standard deviation of the parent population by:

$$\sigma_{\bar{x}} = \frac{\sigma}{\sqrt{N}}$$

The uncertainty associated with the sample mean is then:

$$u_\epsilon = \pm 1.96\sigma_{\bar{x}} = \pm 1.96\frac{\sigma}{\sqrt{N}}$$

The problem now, however, is that realistically we cannot measure infinite number of samples, so σ is unknown. Instead, we will have a finite sample with standard deviation s_x that DOES NOT equal to σ . Moreover, the distribution of finite sample is not exactly Gaussian. Instead we need to utilize *Student t distribution*.

$$u_\epsilon = \pm t_{95}\frac{s_x}{\sqrt{N}}$$

Values for t_{95} distribution are listed in the table below and depend on confidence value (we usually use 95%) and the number of samples written as degree of freedom $\nu = N - 1$. Note that for infinite number of samples for 95%, the value for t_{95} is 1.96, which corresponds to $\pm 1.96\sigma$ seen in Gaussian distribution.

Table 1: The t Distribution[1]

	C				
ν	0.900	0.950	0.990	0.995	0.999
1	6.314	12.706	63.657	127.321	636.619
2	2.920	4.303	9.925	14.089	31.598
3	2.353	3.182	5.841	7.453	12.924
4	2.132	2.776	4.604	5.598	8.610
5	2.015	2.571	4.032	4.773	6.869
6	1.943	2.447	3.707	4.317	5.959
7	1.895	2.365	3.499	4.029	5.408
8	1.860	2.306	3.355	3.833	5.041
9	1.833	2.262	3.250	3.690	4.781
10	1.812	2.228	3.169	3.581	4.587
11	1.796	2.201	3.106	3.497	4.437
12	1.782	2.179	3.055	3.428	4.318
13	1.771	2.160	3.012	3.372	4.221
14	1.761	2.145	2.977	3.326	4.140
15	1.753	2.131	2.947	3.286	4.073
16	1.746	2.120	2.921	3.252	4.015
17	1.740	2.110	2.898	3.223	3.965
18	1.734	2.101	2.878	3.197	3.922
19	1.729	2.093	2.861	3.174	3.883
20	1.725	2.086	2.845	3.153	3.850
21	1.721	2.080	2.831	3.135	3.819
22	1.717	2.074	2.819	3.119	3.792
23	1.714	2.069	2.807	3.104	3.768
24	1.711	2.064	2.797	3.090	3.745
25	1.708	2.060	2.787	3.078	3.725
26	1.706	2.056	2.779	3.067	3.707
27	1.703	2.052	2.771	3.057	3.690
28	1.701	2.048	2.763	3.047	3.674
29	1.699	2.045	2.756	3.038	3.659
30	1.697	2.042	2.750	3.030	3.646
40	1.684	2.021	2.704	2.971	3.551
60	1.671	2.000	2.660	2.915	3.460
120	1.658	1.980	2.617	2.860	3.373
∞	1.645	1.960	2.576	2.807	3.291

[a]Given are the values of t for a confidence level C and number of degrees of freedom ν.

2.2.3 Propagating uncertainty

Once you have collected your data and associated uncertainties, you will be processing these data using formulas. The error propagates from the raw data to the result of the formula. To illustrate this consider an experiment where a velocity of a car is calculated by measuring a distance, d, that it has travelled and a time it took to travel that distance, t. Both the distance and time measurements have uncertainties. The speed can be calculated using a formula:

$$V = \frac{d}{t}$$

We can use the chain rule to write the change in velocity as:

$$u_V = \frac{\partial V}{\partial d} u_d + \frac{\partial V}{\partial t} u_t = \frac{1}{t} u_d - \frac{d}{t^2} u_t$$

[1] Coleman, Hugh W., and W. Glenn Steele. *Experimentation, Validation, and Uncertainty Analysis for Engineers*. Wiley, 2018.

The left hand side is a relative uncertainty, which depends on several quantities. It depends on uncertainties of distance and time, u_d and u_t. Also, it depends on the measurement of distance and time themselves. Therefore, we can write the general expression for propagating uncertainty of a function $y(x_1, x_2, \dots x_n)$ with uncertainties $u_{x_1}, u_{x_2}, \dots u_{x_n}$ as:

$$u_y = \frac{\partial y}{\partial x_1} u_{x_1} + \frac{\partial y}{\partial x_2} u_{x_2} + \cdots + \frac{\partial y}{\partial x_n} u_{x_n}$$

There is a problem with the derived expression for uncertainty. We have allowed for both a negative and positive value of the uncertainty $\pm u$. This adds ambiguity to the propagating uncertainty as we have two solutions: one for negative and one for positive uncertainty of each variable. Therefore, instead of using the algebraic sum we are using the root mean squares sum:

$$u_y = \sqrt{\left(\frac{\partial y}{\partial x_1} u_{x_1}\right)^2 + \left(\frac{\partial y}{\partial x_2} u_{x_2}\right)^2 + \cdots + \left(\frac{\partial y}{\partial x_n} u_{x_n}\right)^2}$$

2.3 Probability Density Function (PDF)

The following section explains why the uncertainty of the fluctuating time series is related to the standard deviation.

The significance of the pdf $f(x)$ is that $f(x)dx$ is the probability that a random error value x' is in the interval $(x, x+dx)$, written as:

$$prob(x \le x' \le x + dx) \equiv P((x \le x' \le x + dx) = f(x)dx \quad (2.4)$$

This is an operational definition of *f(x)* since *f(x)dx* is unitless (it is a probability), then f(x) has units of inverse random value units, e.g., 1/cm or 1/s or 1/cm^2, depending on the units of *x*. Figure 2.1 shows a typical pdf *f(x)* and illustrated the interpretation of the probability of finding the random value in *(x+dx)* with the area under the curve *f(x)* from *x* to *(x+dx).*

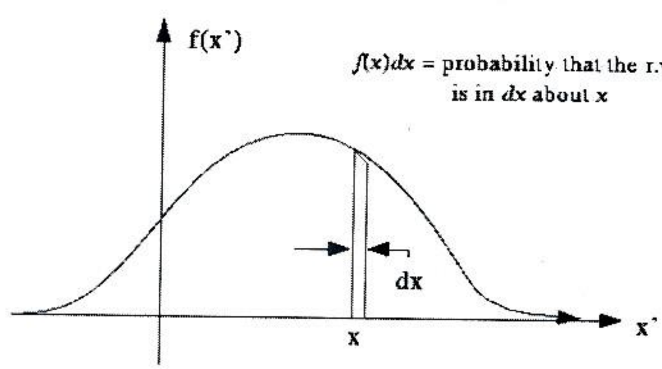

Figure 2.1. *Typical probability Distribution function*

We can also determine the probability of finding the random value somewhere in the finite interval [a, b]:

$$prob(a \le x \le b) \equiv P(a \le x \le b) = \int_a^b f(x)dx \qquad (2.5)$$

which, of course, is the area under the curve *f(x)* from *x=a* to *x=b*

2.3.1 Gaussian distribution

The Gaussian distribution is a pdf which has two free parameters: The mean and the standard deviation. The probability of finding a measurement in the range [x, x+dx] is equal to the area under the curve in that range. The curve is normalized to have a total area of 1 which is why its amplitude is not also a free parameter. Notice also that the distribution is symmetric which means that an error is equally likely to occur in either direction. Just to edify the interested, here is the equation which describes this curve:

$$G(x) = \frac{1}{\sigma\sqrt{2\pi}} \exp\left[\frac{-(x - x_m)^2}{2\sigma^2}\right] \qquad (2.6)$$

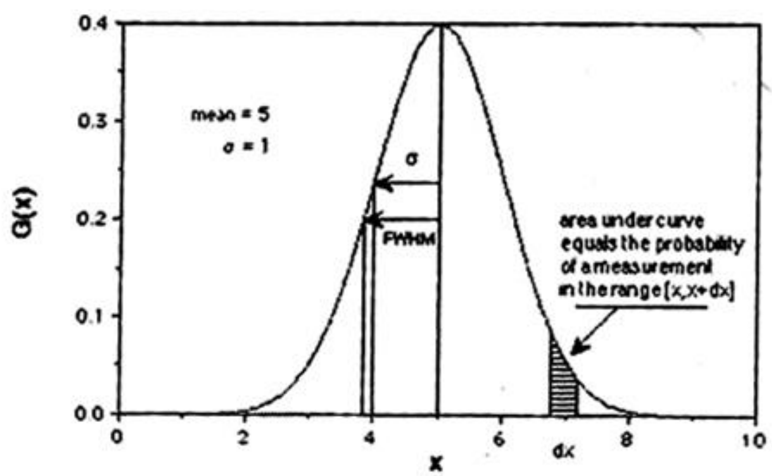

Figure 2.2. *Gaussian distribution*

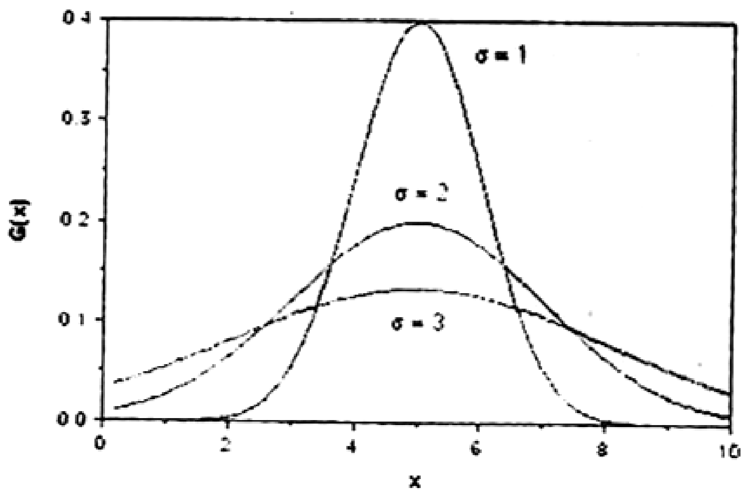

Figure 2.3. *Effect of changing σ*

From Figure 2.3 we can see that the standard deviation σ describes the width of the distribution; a higher standard deviation means that you are more likely to find large errors. The mean x_m lies on the axis of symmetry of the distribution. These two parameters completely determine the shape of the curve and are used to describe the results of your measurements. Another common way of describing the width of the distribution is by using the "full width at half maximum" or FWHM which is equal to 1.17σ and is easier to figure out from a plot. By integrating all or part of the Gaussian curve, we can make precise statements about how probable it is that our results are correct.

The standard deviation measures the width of the distribution and is used as a measure of uncertainty. Any single measurement has a 68.3% chance of falling within one standard deviation of the true value. Often data is given error bars of $\pm\sigma$, meaning that the true value has a 68.3% chance of falling in the stated range. The probability comes from integrating the curve between in the range $[x_m + \sigma, x_m - \sigma]$.

This form of the Gaussian is most likely to describe your data, but it is certainly not the only reasonable choice. Small changes in the mean or standard deviation will still produce curves which reproduce your data well. It stands to reason, then, that there are also uncertainties on these parameters. The uncertainty on the mean is called the standard deviation of the mean, and is given by $\sigma_m = \frac{\sigma}{\sqrt{N}}$. If you repeated the whole experiment several times, you would expect to get a slightly different mean each time, right? In fact, the distribution of means would also be Gaussian, and σ_m would be the standard deviation of that set.

The uncertainty on the standard deviation can also be calculated, but it is not used very often.

Once you have worked out all these numbers, what do you do with them? In general, you tell the rest of the world that your answer is the mean of your measurements, and the uncertainty of your answer is the standard deviation of the mean. Anyone reading your results will have a clear idea how precise your measurement was and how confident you are that the true values lies in that range (68.3%).

Scientists often want to present a range with a higher confidence level. Fortunately, this is a simple matter of integrating the Gaussian between different limits. Giving error bars of $\pm 2\sigma_m$ corresponds to a 95% chance that the true value is in the stated range, and error bars of $\pm 3\sigma_m$ corresponds to a 99.7% confidence level. Often people will use multiples of and probability interchangeably when discussing their results.

Chapter 3

DIGITAL DATA ACQUISITION

Most of the signals directly encountered in science and engineering are continuous: light intensity that changes with distance; voltage that varies overtime; drag on a wing that depends on the speed, etc. Before a computer can analyze an analog signal, the signal must be converted into something that the computer can understand. The signal at a given time must therefore go through some kind of process to transform the signal into a number representation of the original signal value. Due to the limitations of computers, the representation of a random analog signal has to be finite; thus digital formatting is different from its continuous counterpart in two important respects: it is sample and it is quantized. Both of these restrict how much information a digital sign can contain.

3.1 The sampling theorem

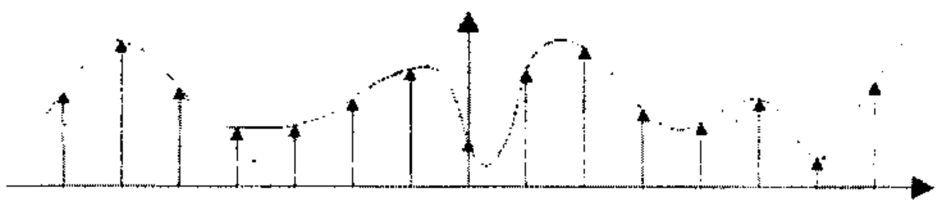

Figure 3.1. *Sampling of analog signal*

Sampling means that the signal must be made discrete in time as illustrated by Figure 3.1. If an analog signal is transformed to numbers at a fixed rate the signal can be represented by a sequence of numbers. The sampling rate of the analog signal determines the accuracy of the digital representation (at this point we ignore the actual number representation).

Such that proper sampling is required to maintain desired accuracy during analog to digital conversion. Suppose you sample a continuous signal in some manner; if you exactly reconstruct the analog signal from the samples, you must have done the sampling properly. Even if the sampled data appears confusing or incomplete, the key information has been captured if you can reverse the process.

Figure 3.2 to Figure 3.5 show several sinusoids before and after digitization. The continuous line represents the analog signal entering the ADC, while the cross is the digital signal leaving the ADC. In figure 3.2, the analog signal is a constant DC value, a cosine wave of zero frequency. Since the analog signal is a series of straight lines between each of the samples, all of the information needed to reconstruct the analog signal is contained in the digital data. According to our definition, this is a proper sampling.

The sine wave shown in figure 3.3 has a frequency of $1/2\pi$ Hz while the sampling rate is 2Hz. such that there are $4\pi \approx 12$ samples taken over each complete cycle of the sinusoid. This situation is more complicated than the previous case, because the analog signal cannot be constructed by simply drawing straight lines between the data points.

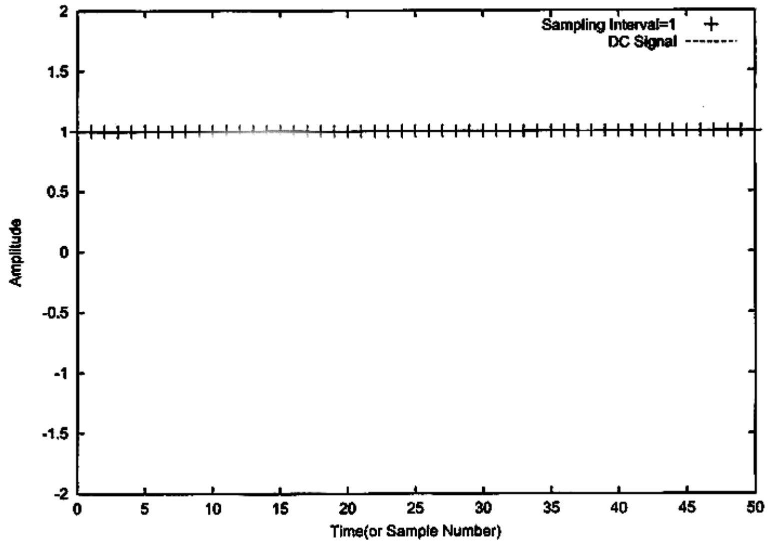

Figure 3.2.

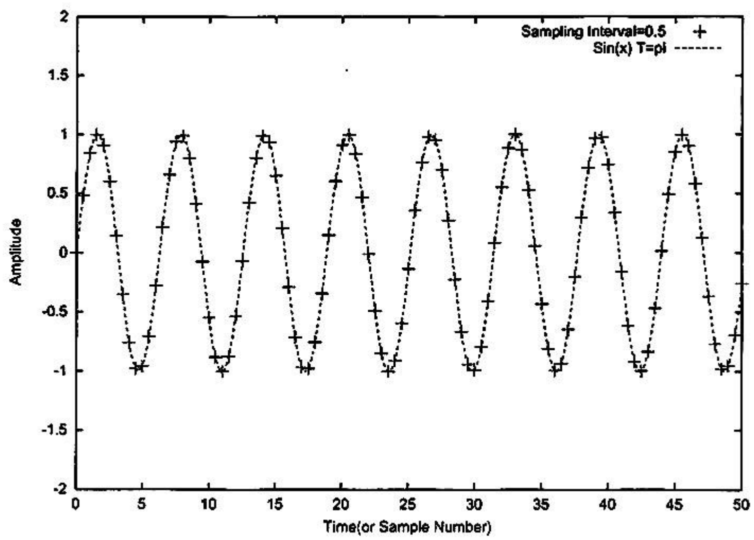

Figure 3.3

Do these samples properly represent the analog signal? The answer is yes, because no other sinusoid or combination of sinusoids will produce this pattern of samples (within the reasonable constrains listed below). These samples correspond to only one analog signal, and therefore the analog signal can be exactly reconstructed. Again, an instance of proper sampling.

In Figure 3.4 the sampling rate drops to 0.5 Hz. This results in only $\frac{0.5}{1/2\pi} \approx 3.1$ samples per sine wave cycle. Here the samples are so sparse that they do not even appear to follow the general trend of the analog signal. Do these samples properly represent the analog waveform? Again, the answer is yes, and for exactly the same reason. The samples are a unique representation of the analog signal. All of the information needed to reconstruct the continuous waveform is contained in the digital data. Obviously, it must be more sophisticated than just drawing straight lines between the data points. As strange as it seems, this is proper sampling according to our definition.

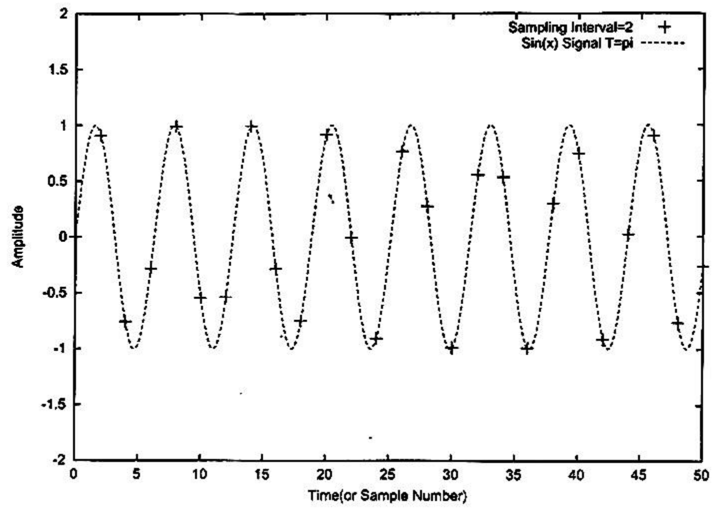

Figure 3.4

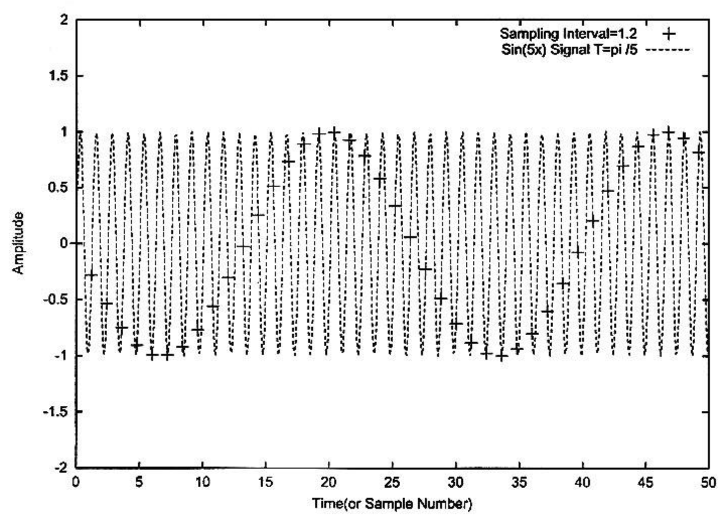

Figure 3.5

In figure 3.5, the analog frequency is pushed higher to $5/2\pi$ Hz, while our sampling rate is $1/1.2 \approx 0.8333$ Hz. So now we took a mere $\frac{1/1.2}{5/2\pi} \approx 1.05$ samples per sine wave cycle. Do these samples properly represent the data? No, they do not! The samples represent a different sine wave from the one contained in the analog signal. In particular, the original sine wave of 0.796 frequency misrepresents itself as a sine wave of 0.04 Hz frequency in the digital signal. This phenomenon of sinusoids changing frequency during sampling is called aliasing. Just as a criminal might take on an assumed name or identity (an alias), the sinusoid assumes another frequency that is not its own. Since the digital data is no longer uniquely related to a particular analog signal, an ambiguous reconstruction is impossible. There is nothing in the sampled data to suggest that the original analog signal had a frequency of 0.796 Hz rather than 0.04 Hz. The sine wave has hidden its true identity completely; the perfect crime has been committed! According to our definition, this is an example of improper sampling.

This line of reasoning leads to the sampling theorem. Frequently this is called the Shannon sampling theorem, or Nyquist sampling theorem, after the authors of 1940 papers on the topic. The sampling theorem indicates that a continuous signal can be properly sampled, only if it does not contain frequency components above one-half of the sampling rate. For instance, a sampling rate of 2000 samples/second requires the analog signal to be composed of frequencies below 1000 cycles/second. If frequencies above this limit are present in the signal, they will be aliased to frequencies between 0 and 1000 cycles/second, combining with whatever information that was legitimately there.

Two terms are widely used when discussing the sampling theorem: The Nyquist frequency and the Nyquist rate. Unfortunately, their meaning is not standardized. To understand this, consider an analog signal composed of frequencies between 0 Hz and 3 KHz. To properly digitize this signal it must be sampled at 6000 samples/sec (6 KHz) or higher. Suppose we chose to sample at 8000 samples/sec (8 KHz), allowing frequencies between 0 Hz and 4 KHz to be properly represented. In this situation there are four important frequencies: 1). The highest frequency in the signal, 3 KHz; 2). Twice this frequency, 6 KHz; 3). The sampling rate, 8 KHz; and 4). One half the sampling rate, 4 KHz. Which of these four is the Nyquist frequency and which is the Nyquist rate? It depends who you ask! All of the possible combinations are used. Fortunately, most authors are careful to define how they are using the terms. In our class, they are both used to mean double the maximum signal frequency, i.e. 6 KHz here.

To aliasing phenomenon is an inherent consequence of the discrete sampling process. To illustrate this mathematically, consider a simple periodic signal which can be described by a one-term Fourier series:

$$y(t) = C.Sin(2\pi ft + \varphi) \tag{3.1}$$

Suppose y(t) is sampled at a time increment Δt so that its discrete time signal is given by:

$$y(p\Delta t) = C.Sin(2\pi\, fp\Delta t + \varphi) \qquad \text{p=0,1,2,…} \tag{3.2}$$

Now using the identity, sin$x$ = sin (x+$2\pi k$), where k is any integer, we can rewrite $y(p\Delta)$ as:

$$y(p\Delta t) = C.Sin\left(2\pi.f.p\Delta t + 2\pi.k + \varphi\right) = C.Sin\left[2\pi\left(f + \frac{m}{\Delta t}\right)p\Delta t + \varphi\right] \tag{3.3}$$

where m=0,1,2,…,(and hence mp=k is an integer). This implies that for any value of Δt, the frequencies, f and $f+m/\Delta t$ must be indistinguishable. Thus, all frequencies given by $f+m/\Delta t$ are the aliasing frequencies of f. However, by adherence to the sampling theorem criterion, all $m \geq 1$ will be eliminated from the sampled signal and thus, this ambiguity between frequencies is avoided. The same discussion can be applied to a general Fourier series:

$$y(p\Delta t) = \sum_{n=1}^{\infty} C_n Sin\left(2\pi nfp\Delta t + \varphi\right) \qquad (3.4)$$

which can be rewritten as:

$$y(p\Delta t) = \sum_{n=1}^{\infty} CSin\left[2\pi n\left(f + \frac{m}{\Delta t}\right)p\Delta t + \varphi\right] \qquad (3.5)$$

Estimating the proper sampling rate is in itself a challenge, and is often chosen by trial and error. With signals that exhibit known behavior proper sampling rate can be readily found: It must be more than twice the maximum frequency in the signal. However the signals that show unusual characteristics, sampling determination can become very difficult. When the sampling rate is too high for a particular signal it will yield correlated but redundant data. This will add unnecessary labor to future calculations using these samples. On the other hand if too few samples are taken (That is, the sampling rate is smaller than the Nyquist frequency) aliasing error in frequency reading will arise.

3.2 Quantization

Quantization is the process of converting an analog signal to a digital representation which is normally performed by an analog-to-digital converter (A/D converter or ADC). Since the magnitude of each data sample must be expressed by some fixed number of digits, only a fixed set of levels are available for approximating the infinite set of the continuous data. No matter how fine the scale is, a quantizer has to round the signal into one of the predefined set of values. A quantized signal is illustrated in Figure 3.6

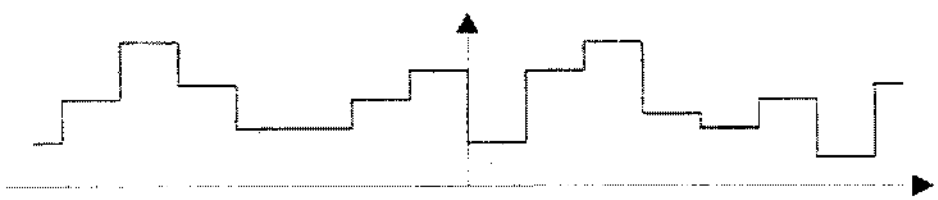

Figure 3.6

In the following we consider only quantizers defined with uniformly spaced quantization levels. Using an n-bit binary word for representing the signal, the voltage range is divided into 2^n equal spaced levels. In this case the step size of the quantizer is given by:

$$LSB = \frac{2A}{2^n} \qquad (3.6)$$

where ±A is the full-scale amplitude of A/D converter. In general, the quantized signal X[n] will be different from the original signal x[n]. The difference between these signals is called the quantization error e[n].

$$e[n] = X[n] - x[n] \tag{3.7}$$

The quantization error is limited in size by:

$$-\frac{1}{2}LSB < e[n] < \frac{1}{2}LSB \tag{3.8}$$

The resolution for a 12-bit A/D converter with ±5-volt input is 10 volts/ 2^{12} = 2.44 mV/bit. We should always be careful with the resolution when we are interested in measurements of fluctuations although the quantizing error may be relatively unimportant as compared to other sources of error inherent in the data processing and data acquisition systems.

3.3 Data acquisition system in lab

3.3.1 A/D Hardware

In some experiments, a data acquisition system will be utilized to obtain temperature readings or fluid velocity profiles (with a pitot tube and a pressure transducer). This section describes briefly the presently used data acquisition system. The

National Instruments data acquisition boards are used in the several of the experiments. Specifically two boards are used: NI PCI-6052E & NI PCI-6033. They have the following specifications:

NI PCI-6052E
Sampling rate 333 KS/s, 16-bit, 16 Analog Input Multifunction

NI PCI-6033
Sampling rate 100 KS/s, 16-bit, 64 Analog Input Multifunction

PICO TC-08
Sampling Rate 10S/s, 8 Channel Thermocouple Input

3.3.2 A/D Software

A custom-made data acquisition software has been written by Mr. Minwei Gong which is self-explanatory. This software has been installed in all data acquisition systems in the ME-436 lab.

3.3.3 A/D Essentials

The most important factors when sampling a signal are dynamic range, input range, and sampling rate. The dynamic range of the data acquisition card is the number of discernible levels that the A/D converter can assume. The number of bits output of the A/D converter determines the dynamic range: namely, there are 2^N available levels for a N-bit A/D converter. For example, a 12-bit A/D converter has a dynamic range of 2^{12} or 4096. It is important to utilize as much of the available dynamic range as possible. For example, if the input range of a 12-bit data acquisition card is set for 0 to 10 V, but the input signal varies only between 0 to 100mV, only 1 percent (0.1 V divided by 10 V) of the possible dynamic range, or only about 41 of the possible 4096 levels are utilized. Signals that do not utilize the full dynamic range of the A/D converter may be poorly represented (in terms of resolution) at the output. In this example, one should either amplify the signal before entering the data acquisition system, or adjust the input range of the converter to something more in line with the expected data. For example, the input range of many data acquisition cards can be set from 0 to 1 V, even in this case, only about 10.

Another potential problem with digital acquisition is clipping. If a voltage lies beyond the input range of the A/D converter, the signal will be clipped. For example, an A/D converter with an input range of 0 to 10V will not be able to distinguish voltages above 10V from voltages equal to 10V. Likewise, any negative voltages will be indistinguishable from a zero volt signal.

Significant measurement errors called aliasing errors are also possible if the waveform is not sampled at high enough frequency. To avoid aliasing, the minimum sampling rate for a waveform must be more than twice the maximum frequency of the measured signal. This restriction is called the Nyquist criterion. Signal aliasing occurs when waveforms are sampled at frequencies below the Nyquist frequency. Aliased signals appear to have frequencies (and possibly even waveform shapes) that differ from those of the actual signal. In practice, waveforms should be sampled at least five times the Nyquist frequency, if possible to ensure adequate representation of the signal.

Chapter 4

Measurement Devices

4.1 Fluid Pressure Measurement

Fluid pressure measurement is an essential process in experimental fluid mechanics. In a stationary fluid the pressure is exerted equally in all directions and is referred to as the static pressure. In a moving fluid, the static pressure is exerted on any plane parallel to the direction of motion. The fluid pressure exerted on a plane right angle to the direction of flow is greater than the static pressure because the surface has, in addition, to exert sufficient force to bring the fluid to rest. The additional pressure is proportional to the kinetic energy of fluid; it cannot be measured independently of the static pressure.

When the static pressure in a moving fluid is to be determined, the measuring surface must be parallel to the direction of flow so that no kinetic energy is converted into pressure energy at the surface. If the total pressure is to be determined, the measuring surface must be aligned perpendicular to the direction of flow such that all kinetic energy is converted into pressure energy.

4.1.1 Manometer

Manometer is a fluid pressure measurement device which consists of a tube containing one or more liquid of different specific gravities.

In using a manometer, generally a known pressure (Which may be atmospheric) is applied to one end of the manometer tube and the unknown pressure to be determined is applied to the other end.

In some cases, however, the difference between pressures at ends of the manometer tube is desired rather than the actual pressure at the either end. A manometer to determine this differential pressure is known as differential pressure manometer.

There are several different configurations for manometers; the simplest one is the U shaped manometer, which is shown in figure 4.1.

Equating the pressure at the level XX' (pressure at the same level in a continuous body of fluid is equal) for the left hand side:

$$P_x = P_1 + \rho g(a+h) \tag{4.1}$$

For the right hand side:

$$P_{x'} = P_2 + \rho g a + \rho_m g h$$
(4.2)

Since $P_x = P_{x'}$

$$P_1 + \rho g(a + h) = P_2 + \rho g a + \rho_m g h \tag{4.3}$$

$$P_1 - P_2 = \rho_m g h - \rho g h \tag{4.4}$$

$$P_1 - P_2 = (\rho_m - \rho) g h \tag{4.5}$$

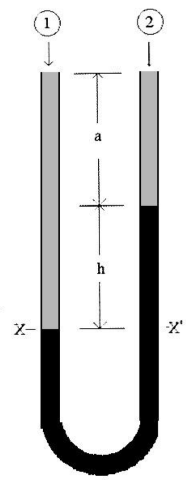

Figure 4.1. *A simple U tube manometer*

The maximum value of P_1-P_2 is limited by the height of the manometer. To measure larger pressure differences we can choose a manometer with higher density, and to measure smaller pressure differences with accuracy we can choose a manometer fluid which is having a density closer to the fluid density. There is also an alternative way that we can use an inclined manometer showed in figure 4.2. We can see that for such kind of configuration, if we read l from the manometer, then h in equation 4.5 could be expressed as $h = l\ Sin\alpha$, thus equation 4.5 becomes:

$$P_1 - P_2 = (\rho_m - \rho) g h = (\rho_m - \rho) g l.Sin\alpha \tag{4.6}$$

In the case of aerodynamics measurement, since the density of the fluid (usually oil) is much larger than the density of the air, $\rho_m >> \rho$, we can ignore the effect of ρ and rewrite the above equation as:

$$P_1 - P = \rho_m g l.Sin\alpha \tag{4.7}$$

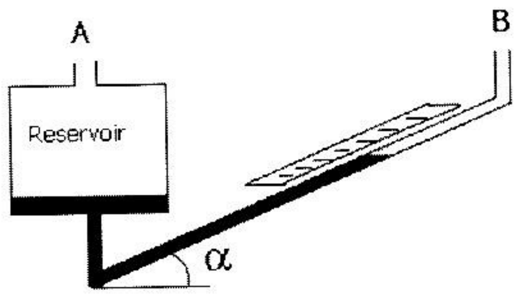

Figure 4.2. *An inclined Manometer*

4.1.2 Pitot tube

The Pitot tube is a device to measure the local velocity along a streamline. The pitot tube has two tubes: one is static tube (b), and another is impact tube (a). The opening of the impact tube is perpendicular to the flow direction. The opening of the static tube is parallel to the direction of the flow. The two legs are connected to the legs of the manometer or equivalent device for measuring small pressure differences. The static tube measures the static pressure, since there is not velocity component perpendicular to its opening. The impact tube measures both the static pressure and impact pressure (due to kinetic energy). In terms of heads the impact tube measures the static pressure head plus the velocity head.

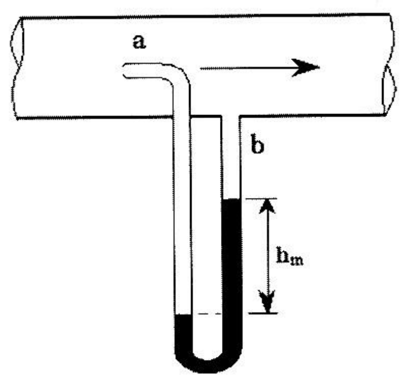

Figure 4.3. *Pressure readings from a pitot tube*

The reading (h_m) of the manometer will therefore measure the velocity head, and $v^2/2g$ = Pressure head measured indicated by the pressure measuring device.

$$\frac{v^2}{2} = \frac{\Delta P}{\rho} \tag{4.10}$$

$$v = \sqrt{\frac{2\Delta P}{\rho}} \quad (4.11)$$

Pressure difference indicated by the manometer ΔP is given by:

$$\Delta P = h_m(\rho_m - \rho)g \quad (4.10)$$

$$v = \sqrt{\frac{2h_m(\rho_m - \rho)g}{\rho}} \quad (4.11)$$

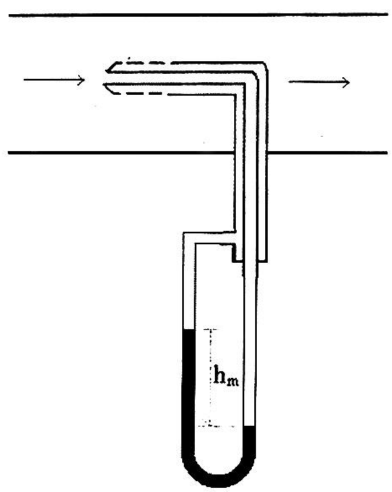

Figure 4.4. A *pitot static tube.*

The pitot-static tube consists of two concentric tubes arranged parallel to the direction of flow; the impact pressure is measured on the open end of the inner tube. The end of the outer concentric tube is sealed and a series of orifices on the curved surface give an accurate indication of the static pressure. For the flow rate not to be apparently disturbed, the diameter of the instrument must not exceed about one fifth of the diameter of the pipe. An accurate measurement of the impact pressure can be obtained using a tube of very small diameter with its open end at right angles to the direction of the flow; hypodermic tubing is convenient for this purpose.

The pitot tube measures the velocity of only a filament of liquid, and hence it can be used for exploring the velocity distribution across the pipe cross-section. If, however, it is desired to measure the total flow of fluid through the pipe, the velocity must be measured at various distances from the walls and the results integrated. The total flow rate can be calculated from a single reading only if the velocity distribution across the cross-section is already known.

4.1.3 Venturi meter

In venturi meter the fluid is accelerated by its passage through a converging cone of angle 15^o-20^o. The pressure difference between the upstream end of the cone and the throat is measured and provides the data necessary to compute the rate of flow. The fluid is then slowed down in a cone of smaller angle (5^o-7^o) in which large proportion of kinetic energy is converted back to pressure energy.

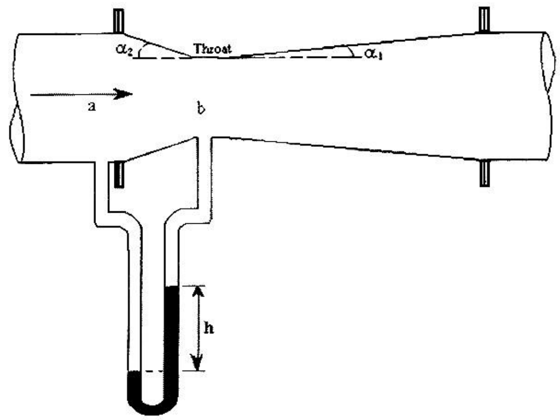

Figure 4.5. *Venturi meter*

The attraction of this meter lies in its high-energy recovery so that it may be used where only a small pressure head is available, though its construction is expensive. To make the pressure recovery large, the angle of downstream cone is small, so boundary layer separation is prevented and friction minimized. Since separation does not occur in a contracting cross-section, the upstream cone can be made shorter than the downstream cone.

The basic equation for the venturi meter is obtained by writing the Bernoulli equation for incompressible fluids between the two sections a and b. Friction is neglected, the meter is assumed to be horizontal.

If v_a and v_b are the average upstream and downstream velocities, respectively, and ρ is the density of the fluid,

$$v_b^2 - v_a^2 = \frac{2(P_a - P_b)}{\rho} \tag{4.12}$$

The continuity equation can be written as,

$$v_a = \left(\frac{D_b}{D_a}\right)^2 v_b = \beta^2 v_b \tag{4.13}$$

where:
D_a = Diameter of pipe.
D_b = Diameter of throat of meter.

β = Diameter ratio, D_b/D_a.

If v_a is eliminated from the equation 4.12, we can get:

$$v_b = \frac{1}{\sqrt{1-\beta^4}}\sqrt{\frac{2\left(P_a - P_b\right)}{\rho}} \tag{4.14}$$

This equation applies strictly to the frictionless flow of non-compressible fluids. To account for the small friction loss between locations *a* and *b*, this equation can be corrected by multiplying the right hand side by an empirical factor C_v which is less than one. This coefficient C_v is determined experimentally.

For a well-designed venturi, the constant C_v is about 0.98 for pipe diameters of 2 to 8 inch and about 0.99 for larger sizes.

4.1.4 Volumetric flow rate

The velocity through the venturi throat usually is not the quantity desired. The flow rates of practical interest are the mass and volumetric flow rates through the meter.

Volumetric flow rate is calculated from,

$$Q_{ideal} = A_b v_b \tag{4.15}$$

if we neglect viscous losses. To correct for the viscous losses, the ideal volume flow is multiplied by the correction factor C_v

$$Q_{actual} = Q_{ideal} C_v$$

The Mass flow rate can then calculated as $\dot{m} = \rho Q$ where ρ is the density of the flowing fluid.

4.2 Temperature Measurement

There are a wide variety of temperature measurement devices available. All of them determine the temperature by sensing some change in the characteristics of the physical make-up of the temperature-measuring device. Six basic types of device used are thermocouples, resistance temperature devices (RTD's), infrared radiators, bimetallic devices, liquid expansion devices, and change of state devices. Among these six devices, thermocouples are the most widely used and are also the used in our laboratory.

4.2.1 Thermocouples

Thermocouples are the most commonly used temperature sensors. A thermocouple is created when two dissimilar metals touch and the contact point produces a small open circuit voltage as a function of temperature. This thermoelectric voltage is known as Seebeck voltage, named after Thomas Seebeck, who discovered the phenomenon in 1821. The voltage is non-linear with respect to temperature; however for small changes,

$$V = S * T \tag{4.16}$$

Where S is the Seebeck coefficient. However, S changes with temperature, causing the output voltages to be non-linear over the operating range. By measuring a thermocouple's voltage, you can calculate temperature.

Thermocouples are designated by capital letters that indicate their composition according to American National Standards Institute (ANSI) conventions. For example, J-type thermocouple is made from iron as one conductor and constantan (copper-nickel alloy) as another, K-type thermocouple is made from Chromel and Alumel. Figure 4.7 shows a typical thermocouple measurement configuration with reference junctions, and where:

$$V_{out} = S * (T - T_{ref}) \tag{4.17}$$

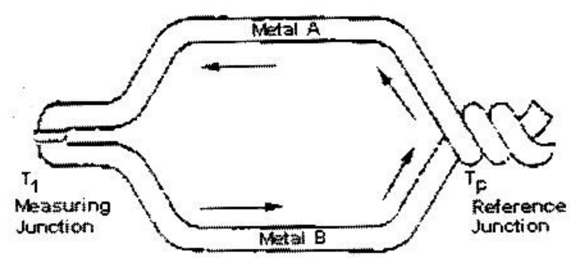

Figure 4.6. *Seebeck effect*

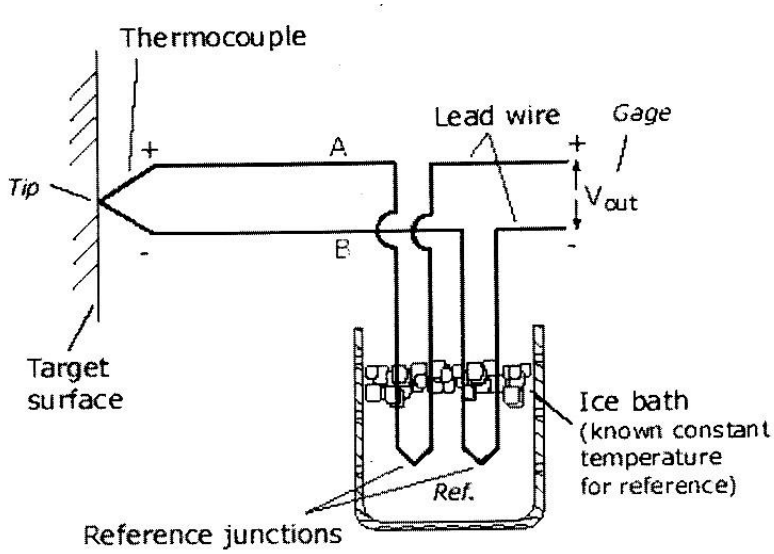

Figure 4.7. *Thermocouple measurement configuration.*

Chapter 5

LIFT & DRAG MEASUREMENTS IN THE WIND TUNNEL

5.1 Objectives

1. Calculate coefficient of list and drag on a NACA0012 airfoil using measured pressure distribution
2. Calculate coefficient of drag using momentum deficit analysis

5.2 Theoretical Background

Any body of any shape when immersed in a fluid stream will experience forces and moments from the flow. These forces and moments are due to only two basic sources:

1. Pressure distribution over the body surface.
2. Shear stress distribution over the body surface.

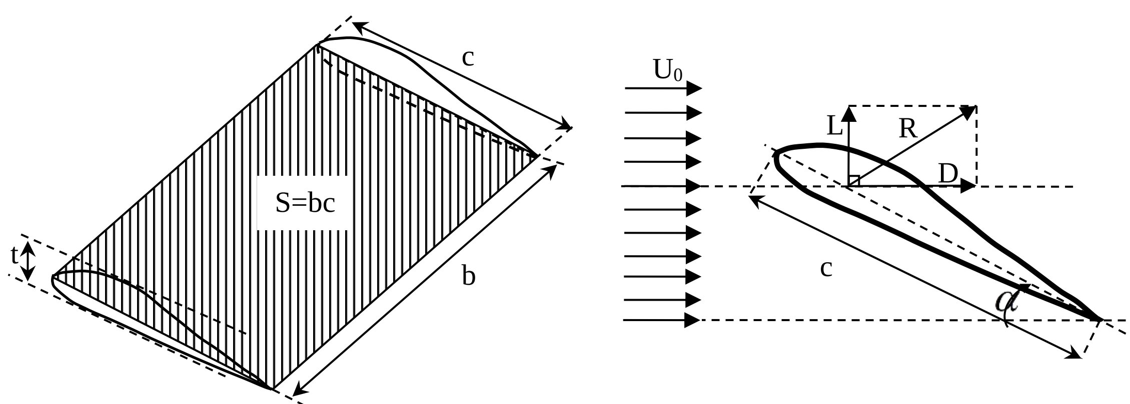

Figure 5.1. *Airfoil and wing nomenclature.*

The incoming uniform flow is called “freestream”, its magnitude is called “freestream speed” and usually shown by U_0 (or U_∞); its direction is called “freestream direction”. The line connecting the leading edge and the trailing edge is called “chord” and its length is shown by *c*. The length of the wing is called “span” and shown by “b”. The angle between the chord and the freestream direction is called “angle of attack” and shown by α . Both the pressure P and shear stress τ have dimensions of force per unit area. In non-dimensional forms pressure is expressed by the pressure coefficient C_p and shear stress is expressed by the skin friction coefficient c_f. These coefficients are defined as:

$$C_P = \frac{P - P_{ref}}{\frac{1}{2}\rho_0 U_0^2} \tag{5.1}$$

and

$$c_f = \frac{\tau}{\frac{1}{2}\rho_0 u_0^2} \tag{5.2}$$

P_{ref} is a reference pressure which is usually taken as the static pressure far upstream, and $½\rho(u_0)^2$ is the dynamic pressure far upstream. The component D of the resultant force R acting over the body, in freestream direction is called drag (Figure 5.1). The component L is the lift force and it is perpendicular to the freestream. The coefficient of this force, called drag coefficient is defined as

$$c_L = \frac{L}{\frac{1}{2}\rho_0 u_0^2 S} \tag{5.3a}$$

$$c_D = \frac{D}{\frac{1}{2}\rho_0 u_0^2 S}$$

(5.3b)

where S is a reference area which is usually one of three types:

1. Cross-sectional or frontal area: The body area as seen from the stream; suitable for stubby (blunt) bodies, such as spheres, cylinders, cars, missiles, projectiles and torpedoes.

2. Planform area: The body area S as seen from above; suitable for wide flat bodies such as wings (Fig. 5.1).

3. Wetted area: Customary for surface ships and barges.

The name of the game in our days is drag reduction. Car and aircraft manufacturers will pay a lot for 1% reduction in C_D. Drag formation is a very complicated phenomenon which is still not well understood. It is usually accompanied by flow separation the consequences of which cannot accurately estimate the usually low-pressure distribution in the separated region. The difference between the high pressure in the front stagnation region and the low pressure in the rear separated region causes a large drag contribution called pressure drag. This is added to the friction drag to yield the total drag that is:

$$C_D = C_{D,pressure} + C_{D,friction} \tag{5.4}$$

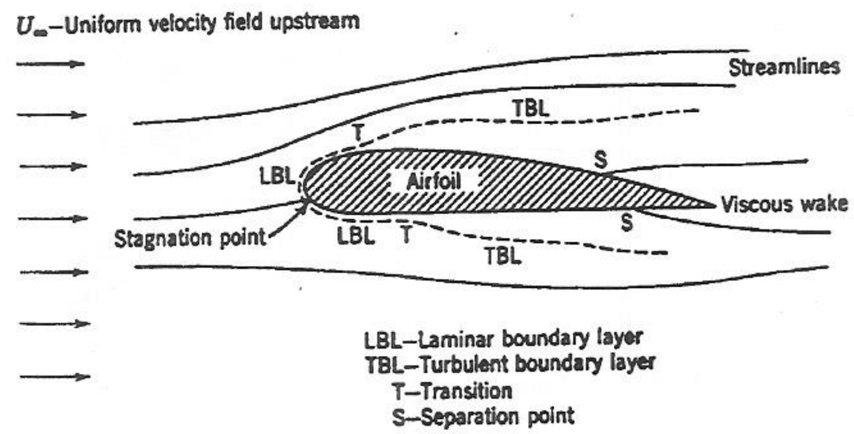

Figure 5.2. *Details of viscous flow around an airfoil.*

To illustrate the dramatic effect of flow separation and the subsequent C_D let us consider the case of flow past an airfoil as shown in Figure 5.2.

The viscosity of the real fluid and the consequent development of the boundary layers produces lift, as well as drag on the airfoil. Over the front part of the body, the flow is laminar, that is, smooth and proceeds in streamlines roughly parallel to the surface, but at some stage, a transition takes place more or less rapidly to what is blown as turbulent flow. Laminar flow is very vulnerable to adverse pressure gradient that on the rear of the airfoil, flow separation occurs. The fluid that was in the boundary layers on the body surface forms the viscous wake behind the separation points. There is an important relation between the drag on the airfoil and a property of the wake called the momentum thickness. The relation can be found by an integral analysis.

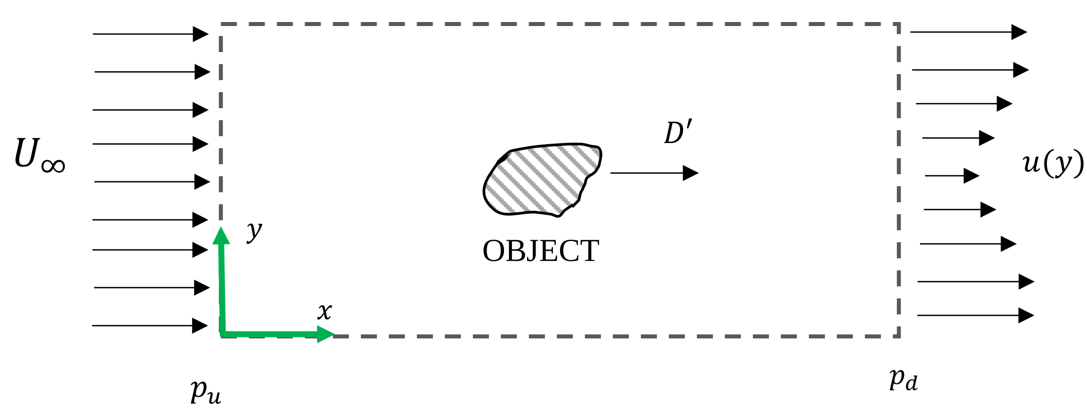

Figure 5.3. *The calculation of boundary layer drag*.

Consider the fixed control region, ABCD, shown in Figure 5.3. It is rectangular, one unit in depth. The upstream flow into the region is uniform with value u_o and at the wake on the downstream boundary the flow has a profile of $u=u(y)$, v is unknown. Because the volume flow through the downstream boundary is less than the entering through the upstream boundary, there must also be an outflow across the sides of the control region. The velocity on the sides is $u=u_o$. We denote the mass flow across both sides as $\int \rho u(y)dy$, and begin the analysis with the integral continuity equation for steady flows.

The drag per unit span of the airfoil, *D'* must be balanced by an equal and opposite force per unit span acting on the fluid. We equate *D'* to the force per unit span resulting from the pressures on *AB* and *CD* plus the net rate of flux of momentum in the negative *x* direction across the sides of *AB* and *CD*. Hence we have:

$$D' = \int_A^B (P_u + \rho u_o^2)\, dy - \int_C^D (P_d + \rho u^2)\, dy$$

5.3 Experimental Set-up

The experimental set-up consists of a wind tunnel with 300 mm by 300mm working section, powered by 5 kW electrical motor, which can produce 36 m/s maximum speed. The motor is controlled by a digital control inverter for variable power output. The air enters the tunnel through a carefully shaped inlet being covered by a protective screen.

The airfoil model to be tested is a NACA 0012 with a span of 297mm and a chord of 152mm with pressure tapping. It is mounted on the tunnel and can be rotated about its axis.

5.4 Report

What you need to present:

1. Velocity profiles for the airfoil wake for various α (and Re if applicable)

   Steps you need to follow:

   a. Import the wake data files using load MATLAB function. The file is organized in the following way:

Row 1	Pitot tube position above wind tunnel floor in mm
Row 2	Pitot tube pressure difference in Pa
Row 3	Static pressure upstream in Pa
Row 4	Static pressure downstream in Pa

   b. Convert the Pitot tube pressure profile of the wake to the velocity profile, $u(y)$,using Pitot tube equation

   c. **Plot** velocity in the wake versus the height in the wind tunnel

d. From the plot, deduce the mean free stream velocity, U_o, by looking at the flow speed outside the wake for zero angle of attack. Keep U_o constant for a given wind tunnel speed setting.

e. Compute the drag per unit depth ($D'_{(Momentum\ Deficit)}$) for various α from known upstream and downstream static pressure, P_u and P_d and upstream and downstream velocity, U_o^2 and $u(y)$ (use `trapz` MATLAB function):

$$D'_{(Momentum\ Deficit)} = -\left(P_d\, H + \int_d \rho u(y)^2\, dy \right) + (P_u\, H + \rho U_o^2 H)$$

Note P_d and P_u are constant for each α. You may take only the first value for upstream and downstream static pressure from each file. Integrate the control volume from $y = 100\ mm$ to $y = 170\ mm$, hence $H = 70mm$.

f. Compute the coefficient of drag:

$$C_D = \frac{D'_{(Momentum\ Deficit)}}{\frac{1}{2}\rho U_o^2 c}$$

where c is the chord length ($c = 152\text{mm}$)

2. C_P vs chord length plots for various α (and Re if applicable)

Steps you need to follow:

a. Import pressure distribution data files using `load` MATLAB function. The file is organized in the following way:

Row 1	x-location of pressure tap in mm
Row 2	y-location of pressure tap in mm
Row 3	Pressure at the corresponding pressure tap Pa

b. Using pressure distribution data, convert pressures along airfoil into coefficient of pressures using:

$$C_P = \frac{P - P_u}{\frac{1}{2}\rho U_o^2}$$

Note: U_o and P_u are free stream velocity and upstream static pressure from previous part.

c. **Plot** C_P vs chord length (which is x-coordinate) for several α.

3. C_L vs α for various α. You may choose the layout and number of plots to present results clearly.

   Steps you need to follow:

   a. Compute the normal force per unit depth N' by integrating the pressure distribution over the airfoil in the x −coodrinate (use trapz MATLAB function)

   $$N' = -\int_0^c P\,dx$$

   b. Compute the axial force per unit depth A' by integrating the pressure distribution over the airfoil in the y −coodrinate

   $$A' = -\int_0^c P\,dy$$

   c. Compute the lift force per unit depth L'

   $$L' = N' \cos\alpha - A' \sin\alpha$$

   d. Compute C_L:

   $$C_L = \frac{L'}{\frac{1}{2}\rho U_o^2 c}$$

   Note U_o is the free stream velocity and c is the chord length ($c = 152\text{mm}$)

   e. **Plot** C_L vs α

4. $C_{D\,(Pressure\ Distribution)}$ vs α. You may choose the layout and number of plots to present results clearly.

   a. Compute the drag force per unit depth D'

   $$D' = N' \sin\alpha + A' \cos\alpha$$

   b. Compute C_D:

   $$C_D = \frac{D'}{\frac{1}{2}\rho U_o^2 c}$$

   Note U_o is the free stream velocity and c is the chord length ($c = 152\text{mm}$)

   c. **Plot** $C_{D\,(Pressure\ Distribution)}$ versus α **and** $C_{D(Momentum\ Deficit)}$ versus α on the same plot.

5. **Plot the error bars** associated with the uncertainty of the pressure measurement on the C_P vs chord length.
6. In Appendix B, show the sample of all calculations done in MATLAB.

Chapter 6

VISCOUS FLOW IN A PIPE

6.1 Introduction

The term pipe flow is generally used to describe the flow through round pipes, ducts, nozzles, sudden expansions and contractions, valves and other fittings. Pipe flows belong to a broader class of flows, called internal flows, where the fluid is completely bounded by solid surfaces. In contrast, in external flows, such as a flow over a flat plate or an airplane wing, only part of the flow is bounded by a solid surface. In this experiment we will limit our study to the flow through round pipes with a venturi tube.

6.2 Theory

When gas or liquid flows through a pipe, there is a loss of pressure in the fluid, because energy is required to overcome the viscous or frictional forces exerted by the pipe walls on the moving fluid. The pressure loss in pipe flows is commonly referred to as head loss. The frictional losses are referred to as major losses (h_f) while losses through fittings, etc, are called minor losses (h_d). Together they make up the total head losses (h_l) in the pipe flow. Hence:

$$h_l = h_f + h_d \tag{6.1}$$

When written in terms of volumetric flow rate, this equation defines the characteristic curve for viscous flow in a pipe.

6.2.1 Minor losses

Pipe systems often include inlets, outlets, flanges and other pipe fittings in the flow that create eddies resulting in head losses (termed as minor losses) in addition to those due to pipe friction. The minor head losses can be expressed as:

$$h_d = K\frac{V^2}{2} \tag{6.2}$$

where K is the loss coefficient and must be determined experimentally for each situation.

Another common way to express minor head loss is in terms of frictional (major) head loss through an equivalent length, L_e, of a straight pipe. In this form, the minor head loss is written as:

$$h_d = f\frac{L_e}{D}\frac{V^2}{2g} \tag{6.3}$$

Loss coefficients, K, and equivalent lengths can be found in a variety of handbooks; representative data for limited fittings is available in most undergraduate Fluid Mechanics texts.

6.2.2 Frictional losses

Knowledge about the velocity distribution is important if we are interested to evaluate the shear stress in a fluid flow. In a fully developed, axis-symmetric pipe flow (see Figure 6.1), one expects that the axial velocity (the only velocity component) at some distance *r* from the pipe centerline, *u=u(r),* be the same whatever the direction in which r is considered. However, the shape of the velocity profile is different according to the type of flow, i.e., laminar or turbulent.

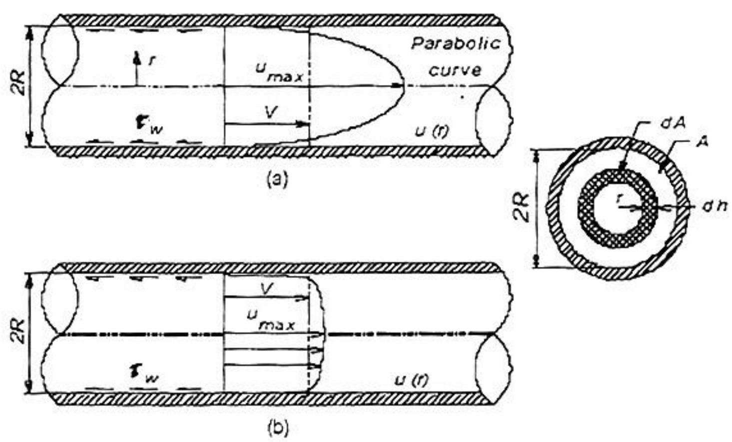

Figure 6.1. *Velocity distributions for fully developed flow in a pipe flow. (a)Laminar flow; (b) Turbulent flow.*

Under the assumption of fully developed laminar flow (Re<2000) (see section 4.2.3 for details), Analytical solution for governing equations of the flow (Navier-Stokes equations) can be obtained. The Reynolds number for pipe flow is defined as:

$$\mathrm{Re} = \frac{VD}{\nu} \tag{6.4}$$

where:
V is the pipe average (bulk) velocity,
D is the pipe diameter, and
ν is the kinematic viscosity of the fluid.

A typical exercise of viscous-flow analysis leads to the following distribution of the velocity in the pipe:

$$\frac{u(r)}{u_{\max}} = \left(1 - \frac{r^2}{R^2}\right) \tag{6.5}$$

Where *u(r)* is the axial velocity and *R* is the inner radius of the pipe. Thus, the velocity profile for the laminar pipe flow (also termed the Hagen-Poiseuille flow) is a paraboloid falling to zero at the wall and reaching a maximum at the axis. It can be shown that for this flow the average velocity over the pipe cross-section, *V*, (defined as *V=Q/A*, where *Q* is the flow discharge and *A* is the pipe cross-section area) is one-half the maximum velocity namely.

$$V = \frac{1}{2} u_{\max} \tag{6.6}$$

For turbulent pipe flows (Re>2000), the fluid flow is so complex that there is not exact solution of the Navier-Stokes equations. Extensive experimental analysis led to the well-known logarithmic law that gives the velocity distribution in a turbulent boundary-layer flow. Use of the logarithmic law in a pipe flow (accurate all the way to the center of the pipe) leads to the following expression for the velocity distribution in the pipe.

$$\frac{u(r)}{u_0} = \frac{1}{k} \ln \frac{(R-r)u_0}{\nu} + B \tag{6.7}$$

where the quantity $u_0 = \left(\frac{\tau_w}{\rho}\right)^{1/2}$ is termed as fraction velocity, τ_w is the wall shear stress, and ρ is the density of the fluid. *k* and *B* are dimensionless constants having the approximate values of 0.41 and 5.0 respectively. Equation 6.7 may be rearranged in the form:

$$\frac{u(r)}{u_{\max}} = \left(1 - \frac{r^2}{R^2}\right)^{1/n} \tag{6.8}$$

where n is a parameter dependent on the flow Reynolds number and can be taken as 7 for this case. Integration over the pipe cross-section of equation 6.7 and use of the equation 6.8 gives an expression for the velocity distribution:

$$V = \frac{2n^2}{(n+1)(2n+1)} u_{\max} \tag{6.9}$$

It can be noted that the V/u_{max} ratio for the turbulent pipe flow is much larger than the value of 0.5 predicted for the laminar pipe flow. Thus, a turbulent velocity profile is very flat in the center and drops off sharply to zero at wall (see Figure 6.1).

Once the velocity distribution (*u=u(r)*) has been established, the total volumetric discharge can be calculated as:

$$Q = \int_A u dA = 2\pi \int_0^R u r dr \tag{6.10}$$

where $dA = 2\pi r dr$ corresponds to a circumferential ring as shown in figure 6.1, and A is the total cross-section of the pipe. The velocity distribution in a pipe is directly linked to the distribution of the shear stress. The shear stress, τ, is zero at the center of the pipe and increases linearly to a maximum at the pipe wall, τ_w. Combining the integral momentum equation with dimensional analysis considerations, one can obtain an expression for the pipe head loss (i.e., Darcy-Weisbach equation):

$$h_f = f \frac{L}{D} \frac{V^2}{2g} \tag{6.11}$$

where:
f is the Darcy's friction factor,
L is the length of the pipe over which the loss occurs,
h_f is the head loss due to viscous effects, and
g is the gravitational acceleration.

One of the most famous and useful figures in fluid mechanics is Moody Chart that provides the friction factor for pipe flows with smooth and rough walls (see figure 6.2). The friction factor in general depends on the Reynolds number *Re* and the relative roughness *k/D*. Values of *k* for standard pipe types are listed in standard fluids and hydraulic texts. For large enough values of *Re*, the friction factor is solely dependent on the relative roughness.

6.2.3 Pipe entrance

The characteristic curve presented by equation 6.1 is valid for fully developed flows, excluding the entrance region. At the entrance the flow is nearly inviscid, but the subsequent growth of the viscous boundary layer increases the radial core velocity and retards the velocity near the pipe wall. At a finite distance from the entrance the boundary layer mergers at the center of the pipe and the inviscid core disappears. The tube flow is then completely viscous, and after additional adjustment at $x=L_e$, the flow becomes fully developed.

Equivalent Roughness for New Pipes

	Equivalent Roughness, ε	
Pipe	**Feet**	**Millimeters**
Riveted steel	0.003–0.03	0.9–9.0
Concrete	0.001–0.01	0.3–3.0
Wood stave	0.0006–0.003	0.18–0.9
Cast iron	0.00085	0.26
Galvanized iron	0.0005	0.15
Commercial steel or wrought iron	0.00015	0.045
Drawn tubing	0.000005	0.0015
Plastic, glass	0.0 (smooth)	0.0 (smooth)

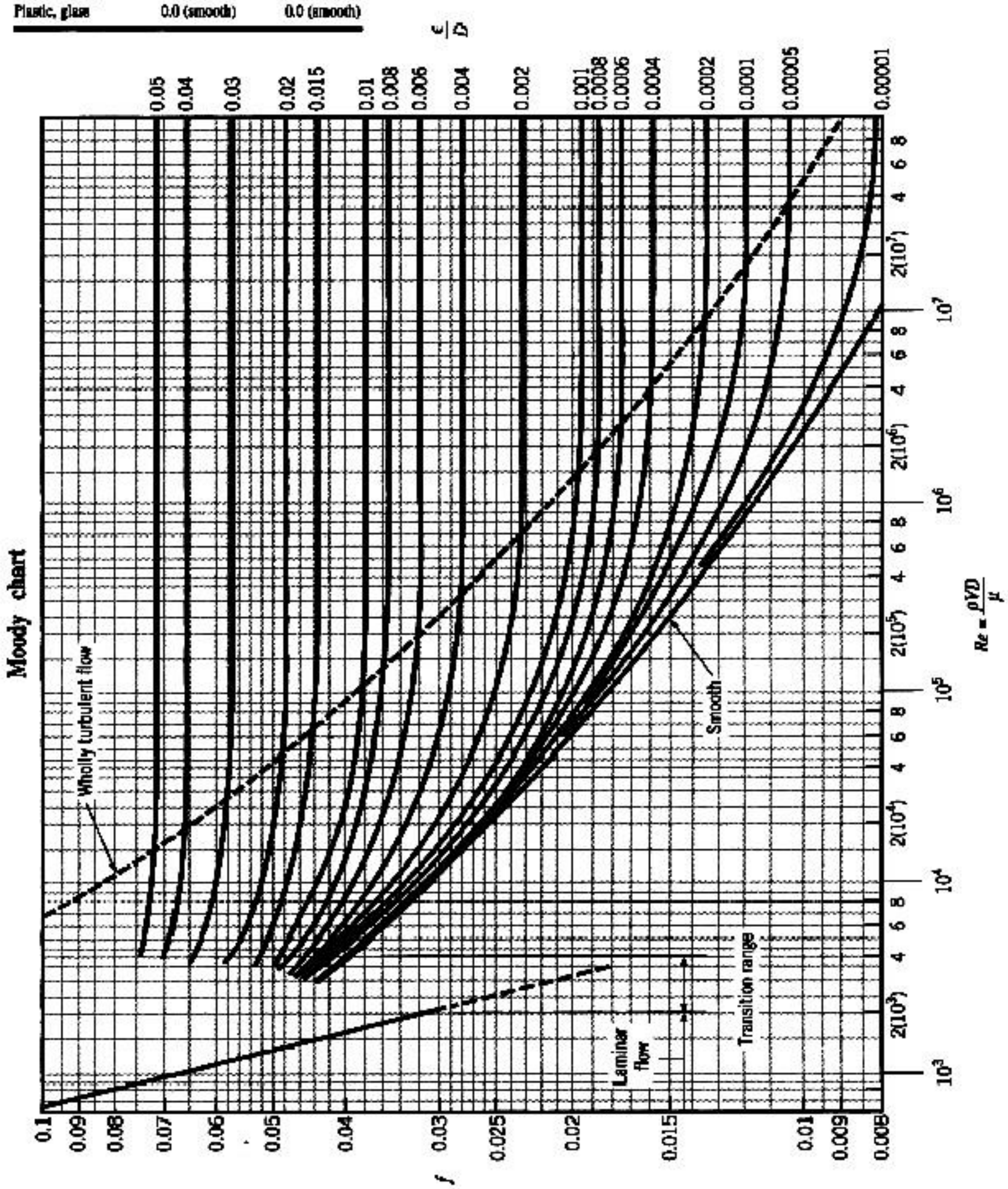

Figure 6.2. Moody chart for pipe friction with smooth and rough walls.

For $x > L_e$, the velocity profile is only a function of r, and the pressure gradient is constant. The entrance length for laminar flow is given by:

$$\frac{L_e}{D} = 0.06\,\mathrm{Re} \qquad (6.12)$$

And, for turbulent flow is given by:

$$\frac{L_e}{D} = 4.4\,\mathrm{Re}^{1/6} \qquad (6.13)$$

The transition Reynolds number is about 2000, but may vary depend on the actual flow conditions.

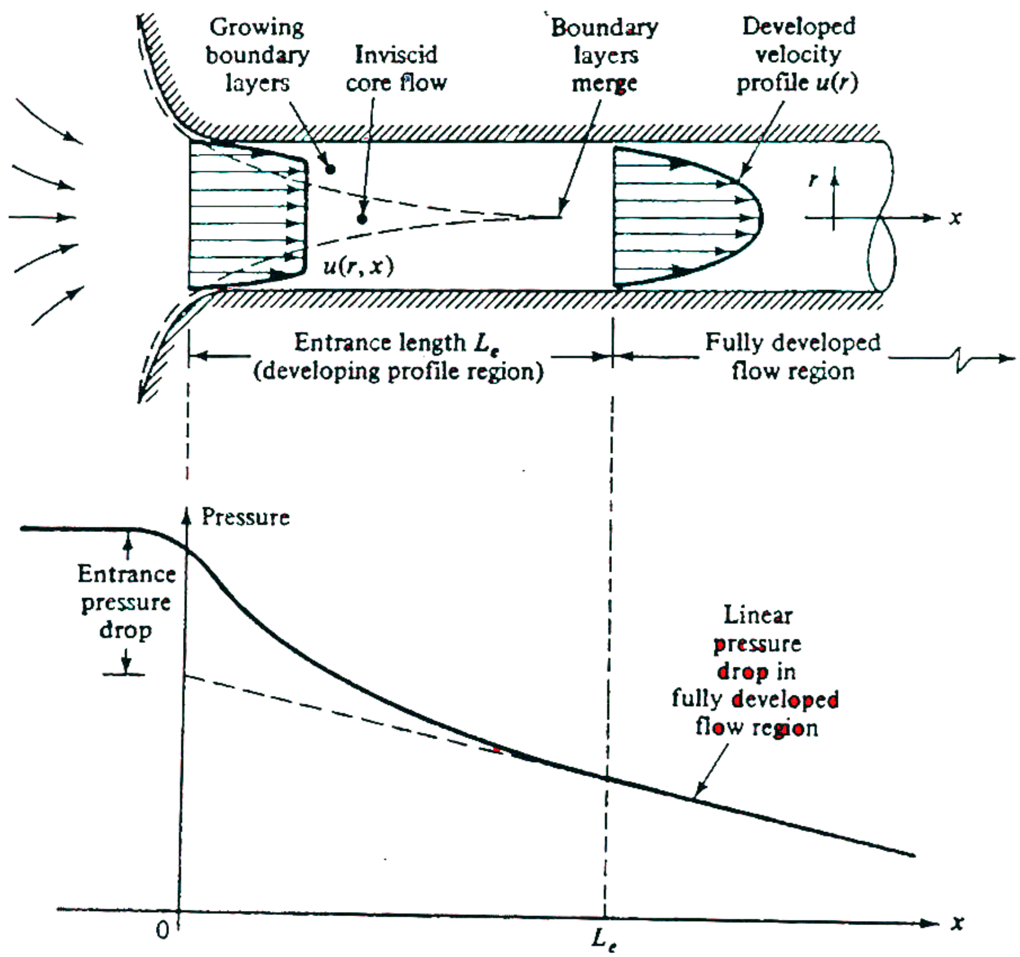

Figure 6.3. *Entrance flow*

6.3 Objective

1. Observe the pressure trop in a pipe with viscous flow through it
2. Calculate the flow rate in the pipe using venturi tube
3. Calculate the flow rate by integrating the velocity profile measured using a Pitot tube

6.4 Experimental Setup

The experimental system consists of a blower, a Plexiglas tube, and a venturi meter, (Figure 6.4). A Pitot tube and a static pressure tap on the side of the pipe are used to measure the velocity profile. An inclined manometer is used for pressure measurements.

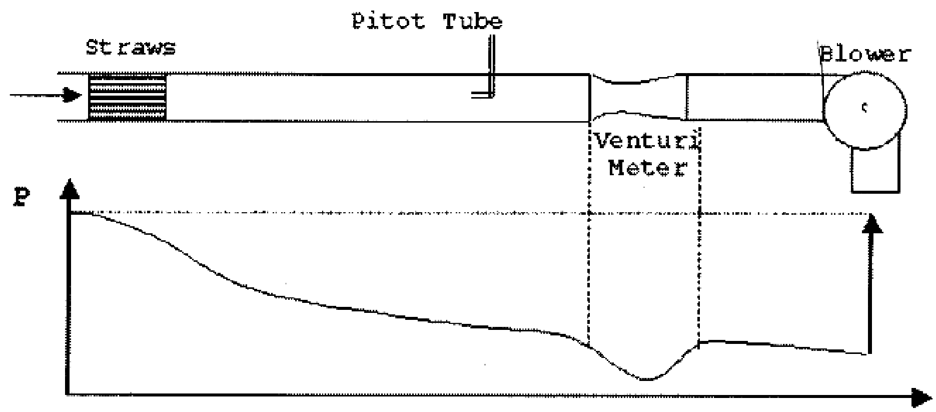

Figure 6.4. *Experimental apparatus and pressure diagram*

Air enters the pipe through a flow management section (straws). The main pipe section has an inside diameter of 138 mm. Downstream, a Pitot tube is located for scanning the total pressure across flow in the pipe. After the Pitot tube, a venturi meter ($D_1 = 138mm$ and $D_2 = 90mm$) is used for measuring the flow rate. Finally the pipe is attached to a centrifugal blower with a dynamometer. The speed of the blower is controlled by a variac and can be measured by a digital tachometer.

Some number of wall pressure taps is available along the tube for the measurement of static pressure. This pressure is rather small and an inclined manometer, which is very sensitive to small pressure changes, can measure it.

6.5 Experiment procedure

Before starting the experiment, measure the locations of all pressure taps and the distances between them. Make sure the inclined manometer and dynamometer are calibrated to read zero.

Switch on the blower keeping the variac at zero output; you can now increase the flow by changing the variac setting. Once the air flows through the pipe, take the following measurements:

1. Level the inclined manometer before every reading and select the inclination angle to obtain good resolution without allowing the manometer fluid to overflow. Measure the static pressure drops along the pipe for various motor speeds. Do not spill the manometer liquid in the Tygon tubes!
2. Read the torque and rpm of the motor.
3. Use your data from the venturi meter to calculate the bulk (average) velocity and the flow rate, and the Reynolds number. In these calculations use the C_v= 0.97 for the discharge coefficients.
4. Measure the velocity profile by traversing the pitot tube across the pipe. When the Pitot tube is physically touching the pipe wall it measures $y=d/2$ where d is the diameter of the Pitot tube.
5. **5.** Repeat these steps for one other variac settings.

6.6 Report

What you need to present:

1. Pressure drop along the length of the pipe (for each fan speed if applicable):
   Steps you need to follow:
   a. Covert the pressure measured in $in\ H_2O$ to Pa.
   b. **Plot** P versus x distance along the pipe
2. A table comparing flow rate measured using a Venturi tube to Flow rate measured using a Pitot tube.
   Steps you need to follow:
   Flow rate measured using Venturi Tube:
   d. Find two pressure measurements: one right before the Venturi tube and one in the middle of the Venturi tube. The difference between them, $\nabla P_{Venturi}$, is the change of pressure through the Venturi tube
   e. Find the mean flow speed in the pipe using the Venturi tube equation based on the diameter change in the pipe:

$$U_{mean} = \sqrt{\frac{2}{\left(\frac{d_a}{d_b}\right)^4 - 1}\left(\frac{\nabla P_{Venturi}}{\rho}\right)}$$

   Note: d_a is the pipe diameter and d_b is the Venturi tube diameter
   f. Multiply the area by the mean speed to get the flow rate, $Q_{Venturi}$

   Flow rate measured using Pitot tube
   a. Find the Pitot tube pressure profile, $P_{Pitot}(r)$,by converting the manometer column readings obtained using a Pitot tube across the pipe to pressure using procedure described in Problem a)
   b. Find to the velocity profile across the pipe, $u(r)$, by converting the pressure profile, $P_{Pitot}(y)$, to velocity profile using a Pitot tube equation

c. Compute the flow rate using Pitot tube:

$$Q_{Pitot} = 2\pi \int_{r=0}^{r=wall} u\, r\, dr$$

d. Construct a **Table** listing $Q_{Venturi}$ and Q_{Pitot} (for each fan speed if applicable)

3. Theoretical and experimental velocity profiles in the pipe
   Steps you need to follow:
   a. **Plot** $u(r)$ versus r calculated in previous part
   b. Calculate the theoretical velocity profile based on the maximum velocity measured using the Pitot tube, $u_{\max} = \max(u(r))$ and the radius of the pipe, R:

   $$u_{theoretical}(r) = u_{\max}\left(1 - \frac{r^2}{R^2}\right)^{\frac{1}{n}}$$

   Note: $n = 7$ for fully turbulent case. Choose n that best matches your data.
   c. **Plot** $u_{theoretical}(r)$ versus r on the same plot as in part a
4. In Appendix B, show the sample of all calculations done in MATLAB.
5. In Appendix B, compute the uncertainty of the flow rate using Venturi tube, $Q_{Venturi}$ and uncertainty of the velocity profile $u(r)$. Add error bars to the velocity profile plot and mention the uncertainty of flowrate in the report.

Chapter 7

HEAT EXCHANGER

7.1 Introduction

A heat exchanger is a device that transfers heat from one fluid to another, usually separated by walls. The heat transfer coefficient of each fluid is determined by the geometry of the flow passages, fluid flow rates, temperatures, and other fluid properties. If the two streams flow across one another at right angles, the heat exchanger is of the cross flow type. If the two streams flow in parallel directions, the heat exchanger is of the in-line type.

The in-line class of heat exchanger can be further subdivided in two. If both fluids flow in the same direction, the heat exchanger is of the parallel flow type, and if the fluid streams flow in opposite directions, it is of the counterflow type. Schematic diagrams and temperature differences of the two types are given in Figure 7.1.

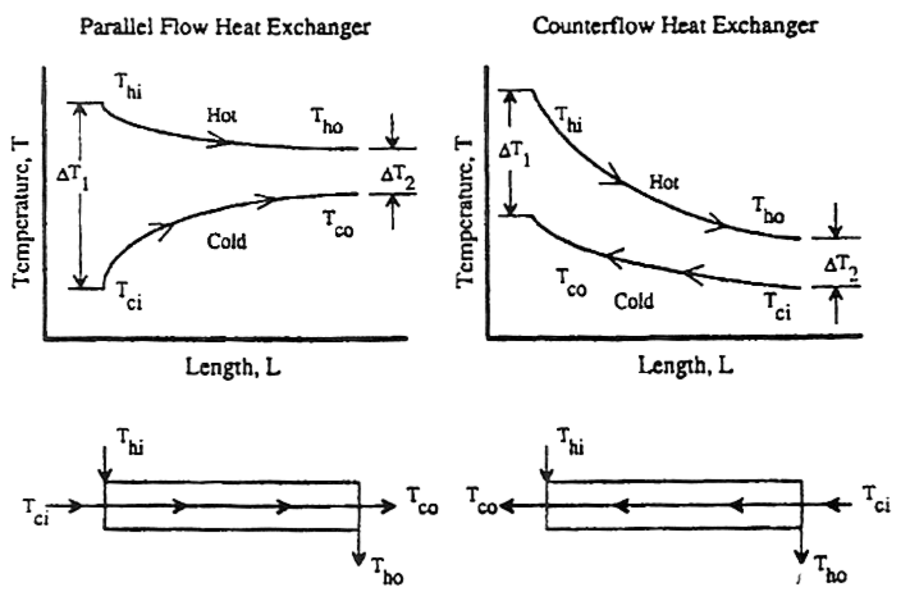

Figure 7.1. *Parallel flow and counterflow Heat exchangers*.

7.2 Theoretical Background

The overall heat transfer coefficient is, simply stated, as a measure of the total thermal resistance to heat transfer between two fluids or surfaces. Using the equation:

$$\dot{q} = UA(\Delta T) \tag{7.1}$$

where U is the overall heat transfer coefficient. We can see that the heat transferred between two points is a function of the temperature difference, the area through which the heat passes, and the overall heat transfer coefficient. Employing an energy balance between the hot and cold fluids, assuming no change of phase, constant specific heats, and minimal fouling, we get from the fluid enthalpies, the total heat transferred between the fluids:

$$\dot{q}_h = \dot{m}_h c_{p,h} (T_{h,i} - T_{h,o}) \tag{7.3}$$

$$\dot{q}_c = \dot{m}_c c_{p,c} (T_{c,o} - T_{c,i}) \tag{7.4}$$

In case of no losses (insulation losses, conduction through the pipes, radiation…etc) conservation of energy requires $\dot{q}_h = \dot{q}_c = \dot{q}$. But they are not, so you may calculate $\dot{q} = \frac{\dot{q}_h + \dot{q}_c}{2}$ **if** $\dot{q}_h$ **and** $\dot{q}_c$ **are reasonably close**.

We would like to have an equation relating the temperature difference between the hot and cold fluids, but since the difference is dependent upon the position in the heat exchanger, we instead use the rate equation shown below:

$$\dot{q} = UA(\Delta T_m) \tag{7.5}$$

For parallel flow heat exchanger, the heat transferred from the hot fluid to the cold fluid is:

$$\dot{q} = UA(\Delta T_{lm}) \tag{7.6}$$

Where,

$$\Delta T_{lm} = \frac{\Delta T_2 - \Delta T_1}{\ln(\Delta T_2 / \Delta T_1)} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1 / \Delta T_2)} \tag{7.7}$$

where ΔT_{lm} is the logarithmic Mean Temperature Difference (LMTD) and

$$\Delta T_1 = T_{h,i} - T_{c,i} \tag{7.8}$$

$$\Delta T_2 = T_{h,o} - T_{c,o} \tag{7.9}$$

For counterflow exchangers, the above becomes:

$$\Delta T_1 = T_{h,i} - T_{c,o} \tag{7.10}$$

$$\Delta T_2 = T_{h,o} - T_{c,i} \tag{7.11}$$

It should be noted that the arithmetic mean $\Delta T_m = (\Delta T_1 + \Delta T_2)/2$ is always greater than the LMTD. One can also see that $T_{c,o}$ can be greater than $T_{h,o}$ only during counterflow. Note that if the inlet and outlet temperatures of one of the pipes are very close or the same, then Eqn. 7.7 will be ill-conditioned. Then you will need to use

$$\Delta T_{lm} = \Delta T_1 \text{ if } \Delta T_2 \cong 0$$

or

$$\Delta T_{lm} = \Delta T_2 \text{ if } \Delta T_1 \cong 0$$

The effectiveness of a heat exchanger is the ratio of the heat transfer rate to the maximum possible heat transfer rate:

$$\varepsilon = \frac{\dot{q}}{\dot{q}_{\max}} \tag{7.12}$$

The maximum heat transfer rate is given by:

$$\dot{q}_{\max} = C_{\min}\left(T_{h,i} - T_{c,i}\right)$$

(7.13)

Where C_{min} is equal to $C_c = m_c c_{p,c}$ or $C_h = m_h c_{p,h}$, the heat capacity of the cold or hot fluid respectively, whichever is smaller. The ratio C_{min}/C_{max} is called the heat capacity ratio, C_R. The parameter UA/C_{min} is called the number of heat transfer units, or NTU. Consequently, the effectiveness ε can be expressed in terms of NTU and C_R.

Correlation equations can also be used to find the overall heat transfer coefficient. Using the equation:

$$U = \frac{1}{\dfrac{1}{h_i} + \dfrac{1}{h_o}} \tag{7.14}$$

Where “i” stands for “inner tube”, “o” stands for “outer tube” and

$$h = Nu_D \frac{k}{D} \tag{7.15}$$

$$Nu_D = 0.023 \mathrm{Re}_D^{\frac{4}{5}} \mathrm{Pr}^{0.4} \tag{7.16}$$

$$\mathrm{Re}_D = \frac{4\dot{m}}{\pi \mu D} \tag{7.17}$$

for turbulent internal flow. Note that D is the “effective diameter” ($D_{eff} = 4A/P$, P is the “wet perimeter”) of the tube of interest, not necessarily its physical diameter. See the related section on Prof. Jiji’s heat transfer book!

7.3 Objectives

1. Calculate the overall heat transfer using LMTD method
2. Calculate the heat transfer using energy conservation method

7.4 Experimental Setup

The diagram in figure 7.2 illustrates the layout of the heat exchanger assembly in the laboratory, which is of the concentric tube or "double pipe" type. By opening and closing certain valves, we can achieve a counterflow or parallel flow condition. Thermocouples are located at the entrance and exit of each tube of the heat exchanger, and flow meters monitor the flow rate in each tube of the heat exchanger. One thermocouple meter provides the readout for the above 4 temperatures, as well as the ambient temperature. The dial marked "OFF", "1", "2". "3", and "4" switches among the four temperatures. When the dial is set to "1" or "2", the reading are for the inlet or outlet of hot water respectively. When the dial is set to "3" or "4", the readings are dependent upon the direction of the cold water flow, i.e., parallel or counterflow. It will be apparent from the readings, which readings are for the inlet cold water, or for the outlet cold water.

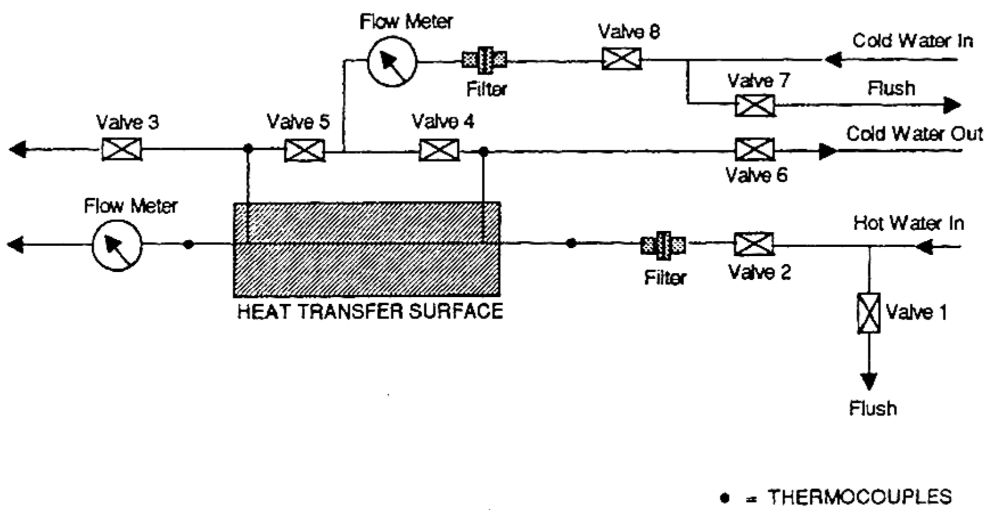

Figure 7.2. *Heat exchanger experimental setup*.

7.5 Procedure

1. First switch ON the power supply to the instruments.
2. Check whether all flow meters and thermocouples are working properly with flow on.
3. Calibrate both flow meters by taking six flow measurements.
4. Select one flow rate by adjusting the cold and hot water valves.
5. Turn ON the heater and allow sufficient time for steady state conditions.
6. Set up a parallel flow (or counterflow) heat exchanger by opening and closing the following valves:

Table 7.1. *Valves' configuration for counter flow or parallel flow*.

Parallel flow	**Valve number**	**Counter flow**
C	1	C
O	2	O
O	3	C
O	4	C
C	5	O
C	6	O
C	7	C
O	8	O

C: valve closed; O: Valve open

7. Select one of the flow rates (cold or hot flow) to be changed along your experiment. The other flow rate will remain constant all the time. Start from the smallest mass flow to take the required readings for the selected flow.
8. Without changing the flow rate reverse the direction of the flow in the outer tube to produce counterflow. This is possible by manipulating the valves according to the configuration (Table 7.1). Take the required readings
9. Increase the flow rate of the selected flow and repeat the same measurements for a total of six mass flow rates for the selected flow.
10. Make sure that you have recorded all physical dimensions of the heat exchanger, as printed on its board.

7.6 Report

1. Find an equation that converts digital flow meter readings to physical units
    a. Convert all volume flow rates from $\frac{L}{s}$ to mass flowrates in $\frac{kg}{s}$ using. Note that

    $$1\,\dot{V}_{\frac{L}{s}} = 1\dot{m}_{\frac{kg}{s}}$$

    b. Plot the calibration data for the *cold* flow meter with digital flow rate units on the x-axis and mass flow rate in SI units on the y-axis.
    c. Once the plot window opens, go to *Tools>Basic Fitting*. Select "Linear" and check "Show equations", click "Close". Write down the equation.
    d. Repeat for the *hot* flow meter

    e. Construct matrices and/or vectors in MATLAB with your data
    f. Convert all you flow rates to mass flow rates using equation found in previous part.

2. <u>Heat transfer Q using LMTD method:</u>

a) For each flow configuration (parallel and counter flow) and each mass flow compute Re_D, Nu_D, h, UA, Q using the following correlation equations:

$$Re_D = \frac{4\dot{m}}{\pi \mu D_{eff}}$$

Where

$\dot{m}$ is the mass flow rate

μ is the viscosity of working liquid:

$$\mu = 1.002 \times 10^{-3} \frac{Ns}{m^2}$$

D_{eff} is the effective diameter of pipe defined as:

$$D_{eff\ HOT} = \frac{4A}{P} = \frac{4\,\pi\, r_{hot}^2}{2\,\pi\, r_{hot}} = 2\, r_{hot}$$

$$D_{eff\ COLD} = \frac{4A}{P} = \frac{4(\pi\, r_{cold\ \ outer}^2 - \pi\, r_{hot}^2)}{2\,\pi\, r_{hot} + 2\,\pi\, r_{cold\ \ outer}} = 2\,(r_{cold} + r_{hot})$$

Where $r_{HOT\ outer}$ in the radius of the inner tube ($r_{HOT} = 0.0048\ m$)

$r_{COLD\ outer}$ in the radius of the outer tube ($r_{COLD} = 0.0095\ m$)

b) Next, calculate Nusselt number:

$Nu_{D\ HOT} = 0.023\ Re_D^{\frac{4}{5}} Pr^{0.3}$ for cooling, so hot water flow.

$Nu_{D\ COLD} = 0.023\ Re_D^{\frac{4}{5}} Pr^{0.4}$ for heating, so cold water flow.

Use $Pr = 6$

c) Finally, calculate the heat transfer coefficient

$$h = \frac{Nu_D k}{D_{eff}}$$

Where

k – conductivity of water ($k = 0.64 \frac{W}{mK}$)

$D_{eff} -$ effective diameter.

d) Compute the product of the overall heat transfer coefficient and area using correlation formula:

$$U_{CORR}A = \left[\frac{1}{h_{HOT}A_T} + \frac{1}{h_{COLD}A_T}\right]^{-1}$$

Where:

A is the surface area defined as:

$$A_T = \frac{1}{2}(2\pi L r_{HOT} + 2\pi L r_{COLD})$$

L is the length of the heat exchanger ($1.6891m$)

$r_{HOT\ outer}$ in the radius of the inner tube ($r_{HOT} = 0.0048\ m$)

$r_{COLD\ outer}$ in the radius of the outer tube ($r_{COLD} = 0.0095\ m$)

e) Compute heat transferred:

$$Q = U_{CORR}A\ \ \Delta T_{lm}$$

ΔT_{lm} is the temperature (in Kelvin so convert all temperatures to correct units) defined as:

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln\left(\frac{\Delta T_1}{\Delta T_2}\right)}$$

Where for parallel regime:

$$\Delta T_1 = T_{h\ inlet} - T_{c\ inlet}$$

$$\Delta T_2 = T_{h\ outlet} - T_{c\ outlet}$$

And for counter flow regime:

$$\Delta T_1 = T_{h\ inlet} - T_{c\ outlet}$$

$$\Delta T_2 = T_{h\ outlet} - T_{c\ inlet}$$

f) **Plot** Q vs *cold* mass flow rate for both parallel and counter flow configurations on the same plot. Label this plot as “Flow rate using LMTD method”.

3. <u>Heat transfer Q using energy conservation method:</u>
    a. Find the heat transfer for hot and cold liquid for counter and parallel flow:

$$Q_{HOT} = \dot{m}_{HOT} C_p \; (T_{HOT\ \ inlet} - T_{HOT\ outlet})$$
$$Q_{COLD} = \dot{m}_{COLD} C_p \; (T_{COLD\ outlet} - T_{COLD\ \ inlet})$$

    Where:

    C_p is specific heat at constant pressure $\left(C_p = 4.178 \times 10^3 \frac{J}{kg\ K}\right)$

    b. Find the overall heat transfer for parallel and counter flow:

$$Q_{PARALLEL} = Q_{HOT\ PARALLEL}$$

$$Q_{COUNTER} = Q_{HOT\ COUNTER}$$

    C_p is specific heat at constant pressure $\left(C_p = 4.178 \times 10^3 \frac{J}{kg\ K}\right)$

4. Plot $Q_{PARALLEL}$ and $Q_{COUNTER}$ using LMTD method and $Q_{PARALLEL}$ and $Q_{COUNTER}$ using energy conservation method versus cold mass flow rate and for both parallel and counter flow configurations on the same plot.
5. In Appendix B, show the sample of all calculations done in MATLAB.
6. In Appendix B, compute the uncertainty of the $Q_{PARALLEL}$ computed using energy conservation method. Add error bars to the appropriate plot.

Chapter 8

CONSTANT CROSS-SECTION FIN

8.1 Introduction

Extended surfaces (or fins) are usually mounted on heated surfaces to improve cooling by increasing the rate of heat transfer. A large variety of shapes are used for extended surfaces (annular fins, spines, straight fins, etc.). Fins are generally thin and long. They are made of high conductivity materials so that the corresponding Biot number is small. This condition permits considerable simplification of the heat transfer analysis based on the assumption that the fin temperature varies only along the fin and is independent of the transverse direction. Figure 8.1 shows a schematic of a horizontal fin, heated at one end.

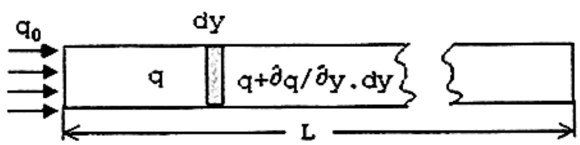

Figure 8.1. *Schematic of a horizontal fin.*

For a constant cross-section fin, the governing differential equation for steady state heat transfer is

$$\frac{d^2T}{dx^2} - \frac{hC}{kA}\left(T - T_\infty\right) = 0 \tag{8.1}$$

With two boundary conditions at the base of the fin, *x=0*, and at the tip of the fin, *x=L*.

where:
T is the temperature
T∞ is the ambient temperature,
x is the coordinate along the fin,
h is the heat transfer coefficient,
k is the thermal conductivity,
A is the fin cross-section area, and
C is the fin circumference.

The solution of the above equation provides the temperature distribution $T=T(x)$ along the fin. The result can be used to calculate the total heat loss from the fin as follows:

$$q_{loss} = -kA\left.\frac{dT}{dx}\right|_{x=0} \qquad (8.2)$$

But since we are not able to calculate $\frac{dT}{dx}$ very accurately, we will use

$$q_{loss} = \bar{h}\int_0^L (T - T_\infty)C dx \qquad (8.3)$$

to solve for $\bar{h}$**.** The last equation assumed negligible heat loss from the tip of the fin. Calculation of q_{loss} is explained later.

It may be interesting, however, to monitor the time dependent temperature distribution of the fin from the time the power is switched on until a steady state heat transfer equilibrium is reached after some time. The time dependent governing differential equation is

$$\frac{\partial^2 T(x,t)}{\partial x^2} - \frac{hC}{kA}\left[T(x,t) - T_\infty\right] = \frac{1}{\alpha}\frac{\partial T(x,t)}{\partial t} \qquad (8.4)$$

Where: α is the **thermal diffusivity** of the material. Find out this value for copper by yourself.

The initial and boundary conditions are:

1. $T(x,0)=T_\infty$ at any x.
2. $q_o'' = \frac{V^2}{RA}$ (Constant heat flux input at $x=0$ and $t>0$). V is the voltage input, and R is the resistance of the heating element. Note that R **is not constant**, it changes depending on your voltage setting. This is explained later.
3. $\frac{\partial T(L,t)}{\partial x} = 0$ (Negligible heat losses from the tip of the fin at $x=L$, $t>0$). I.e. $q(L,t)=0$

8.2 Objectives

1. Determine the unsteady and steady state temperature distribution along the fin and compare it with theoretical predictions.
2. To calculate the total heat transfer loss from the.

8.3 Apparatus

The experimental setup consists of a cylindrical cooper fin (12mm diameter), which is heated electrically at its base by a cartridge heater. Nine thermocouples (Type J) are embedded along the fin. The positions are shown in table 8.1:

Table 8.1. *Positions of thermocouples*.

Thermocouples	**Positions (mm)**
T (1)	x=65
T (2)	x=110
T (3)	x=180
T(4)	x=280
T (5)	x=394
T (6)	x=504
T (7)	x=605

A multimeter is provided for resistance and voltage measurements.

8.4 Procedure

1. Set the VariAC position to zero.
2. Make sure that all thermocouples and electrical contacts are secured and in place before turning on the power supply.
3. Turn ON the computer and start the Data Acquisition software, choose a sampling frequency of 1Hz. Choose your total sampling duration of 30 minutes.
4. Turn ON the power and increase the power supply to about **15 volts.** Use the multimeter to measure the exact voltage settings.
5. Plug in the power to heat the fin, also start the data acquisition at the same time.
6. After the data has been taken, save it.
7. Use an electrical fan to cool down the fin first, and then repeat measurements with one more power settings of about 40 VoltsIf you think all the data has been successfully acquired, switch OFF the power at the variac.

8.5 Report

What you need to present:

1. In your results, present the value of the experimental heat transfer coefficient $\bar{h}_{exp}$

   Steps you need to follow:

   a. Load data into MATLAB using *load* command
   b. The last column in the data file provides ambient temperature. Use the last data point in that column as your T_{amb}.
   c. Compute the mean experimental heat transfer coefficient, $\bar{h}_{exp}$ by integrating the steady state temperature along the fin:

$$q = \bar{h}_{exp} \int_0^L (T_{ss}(x) - T_{amb}) C \, dx$$

   Note: Solve the equation above for $\bar{h}_{exp}$, $T_{ss}(x)$ is the temperature distribution at steady state (last row of your data for thermocouples 1 through 7), C is the circumference of the fin (fin is 12mm in diameter)

d. **Indicate** the $\bar{h}_{exp}$ value in the report.

2. Use MATLAB to build a matrix of <u>theoretical</u> temperature values based on the computed $\bar{h}_{exp}$ values. This is done by computing the following equations:

$$T(x,t) = T_{amb} + \frac{q"_o}{k}\left\{\left[\frac{e^{m(x-2L)} + e^{-mx}}{m(1-e^{-2mL})}\right] - 2e^{-m^2\alpha t}\left[\frac{1}{2m^2L} + L\ \sum_{n=1}^{\infty}\frac{\cos\left(\frac{n\pi}{L}x\right)e^{-\frac{n^2\pi^2}{L^2}\alpha t}}{m^2L^2+n^2\pi^2}\right]\right\}$$

Where:

T_{amb} – ambient temperature (average of the temperatures at time $t = 0$)

$q"_o$ – power of the heater per unit area ($q"_o = \frac{V^2}{(A_cR)}$, V- supply voltage, R – heater resistance ($178\ \Omega$), A_c – fin cross section area)

k – thermal conductivity of copper ($k = 401\frac{W}{mK}$)

$m = \sqrt{\frac{hC}{kA_c}}$, h convective heat transfer coefficient, C is the circumference of the fin, A_c is cross-sectional area of the fin.

L – length of the fin ($L = 0.605m$)

α- thermal diffusivity ($\alpha = \frac{k}{\rho C_p}$, ρ for copper is 8960 $\frac{kg}{m^3}$, C_p specific heat for copper is $386\frac{J}{kg\ K}$)

t – time (s)

n - counter

3. Plot the <u>*steady*</u> theoretical and experimental temperature distribution in the fin in the same figure (for different voltage settings if applicable). Arrange your plots as to show results clearly, you may need more than one figure.

   Steps you need to follow:

   a. **Plot** $T_{ss}(x)$ temperature distribution at steady state (last row of your data) versus x coordinate of the fin
   b. **Plot** $T_{ss_{Theory}}(x)$ in the same figure

4. Plot the <u>*transient*</u> theoretical and experimental temperature distribution in the fin in the same figure for several thermocouples (for different voltage settings if applicable). Arrange your plots as to show results clearly, you may need more than one figure.

   Steps you need to follow:

   a. **Plot** $T(x,t)$ temperature distribution for some thermocouples (several columns of your data) versus t, time.

b. **Plot** $T_{theory}(x)$ in the same figure for the x location consistent with location of thermocouples plotted in experimental Plot.

5. Compute uncertainty of the temperature measurement based on 60 samples and add error bars to the $T_{ss}(x)$ plot.
6. In Appendix B, show the sample of all calculations done in MATLAB.

8.6 Theory

8.6.1 Problem statement

The initial temperature of a thin (straight pin) fin $0 \leq 1 \leq L$ of uniform cross-sectional area A_c is zero throughout, and the face at $x=L$ (the fin tip) is of negligible heat loss to the surrounding medium. Heat is supplied through the face at $x=0$ (the fin base) at a constant rate q_o (or equivalently, there is a constant flux of heat qo'') into the fin per unit area per unit time. Assume constant properties, negligible radiation exchange with surroundings, and uniform heat transfer coefficient over outer surface. The one-dimensional heat equation, in terms of temperature parameter $\theta(x,t) = T(x,t) - T_\infty$, has the form given by:

$$\frac{\partial^2 \theta}{\partial x^2}(x,t) - m^2\theta(x,t) = \frac{1}{\alpha}\frac{\partial \theta}{\partial t}(x,t) \tag{8.5}$$

Subject to the boundary conditions stated on the faces of the fin.

$$-kA_c \frac{\partial \theta}{\partial x}(0,t) = \dot{q}_0 \tag{8.6}$$

or

$$-k \frac{\partial \theta}{\partial x}(0,t) = q_0'' \tag{8.7}$$

and

$$\frac{\partial \theta}{\partial x}(L,t) = 0 \tag{8.8}$$

With the initial condition that

$$\theta(x,0) = 0 \tag{8.9}$$

The parameter m for the prescribed fin is given by

$$m^2 = \frac{hP}{kA_c} \tag{8.10}$$

Where the thermal conductivity of the fin material is k [W/m-ºC], the thermal diffusivity is $\alpha = \frac{k}{\rho . C_p}$ $\left[m^2 / s \right]$, the surrounding medium convection coefficient is h [W/m^2-ºC], the cross-sectional area is $A = \pi R^2$ [m^2], and the fin circumference is $P=2\pi R$[m].

8.6.2 Solution

From the verbal statement of the problem, we have non-zero homogeneous boundary values. With the physical facts, the required temperature function θ(x,t) is modified into two parts,

$$\theta(x,t) = U(x) + V(x,t) \tag{8.11}$$

Where U(x) is the steady-state temperature solution of equation 8.5, involving only x and satisfying the boundary conditions given by equation 8.7 and equation 8.8, and V(x,t) may be regarded as the transient temperature solution of equation 8.5.

Equation 8.5 to equation 8.8 become

$$\frac{d^2U}{dx^2}(x) + \frac{\partial^2 V}{\partial x^2}(x,t) - m^2\left[U(x) + v(x,t)\right] = \frac{1}{\alpha}\frac{\partial V}{\partial t}(x,t)$$
(8.12)

$$-k\left[\frac{dU}{dx}(0) + \frac{\partial V}{\partial x}(0,t)\right] = q_0'' \tag{8.13}$$

$$\frac{dU}{dx}(L) + \frac{\partial V}{\partial x}(L,t) = 0 \tag{8.14}$$

$$U(x) + V(x,t) = 0 \tag{8.15}$$

Suppose now that

$$\frac{d^2U}{dx^2}(x) - m^2 U(x) = 0$$
(8.16)

Subject to

$$-k\frac{dU}{dx}(0)=q_0'' \qquad \text{and} \qquad \frac{dU}{dx}(L)=0 \tag{8.17}$$

Then, the function V(x,t) must satisfy the conditions

$$\frac{\partial^2 V}{\partial x^2}(x,t)-m^2V(x,t)=\frac{\partial V}{\partial t}(x,t) \tag{8.18}$$

$$\frac{\partial V}{\partial x}(0,t)=0 \tag{8.19}$$

and

$$\frac{\partial V}{\partial x}(L,t)=0 \tag{8.20}$$

$$V(x,0)=-U(x) \tag{8.21}$$

A solution of equation 8.16 is of the form

$$U(x)=Ae^{mx}+Be^{-mx} \tag{8.22}$$

Where A and B are constants, obtained from the boundary conditions from equation 8.17 which are satisfied when

$$A=\frac{q_0''e^{-2mL}}{mk\left(1-e^{-2mL}\right)} \tag{8.23}$$

$$B=\frac{q_0''}{mk\left(1-e^{-2mL}\right)} \tag{8.24}$$

So, the steady-state temperature distribution in the fin is

$$U(x)=\frac{q_0''}{mk\left(1-e^{-2mL}\right)}\left[e^{m(x-2L)}+e^{-mx}\right] \tag{8.25}$$

The equivalent hyperbolic function of solution 8.25 could be readily verified as of the form:

$$U(x)=\frac{q_0''}{mk}\frac{\cosh\left(m(x-L)\right)}{\sinh(mL)} \tag{8.26}$$

The solution V(x,t) of 8.18 is obtained by the method of separation of variables. Suppose there exists a separated solution of the product form such that

$$V(x,t) = X(x)T(t) \tag{8.27}$$

Note that X is a function of x alone and T of t alone, and that X and T must be nontrivial (x≠0, T≠0). Since 8.27 must satisfy conditions 8.18, then

$$\frac{d^2X}{dx^2}(x)T(t) - m^2X(x)T(t) = X(x)\frac{1}{\alpha}\frac{dT}{dt}(t) \tag{8.28}$$

$$\frac{dX}{dx}(0)T(t) = \frac{dX}{dx}(L)T(t) = 0 \tag{8.29}$$

$$X(x)T(0) = -U(x) \tag{8.30}$$

Observe that the above equation is an ordinary differential equation that is variable separable, and may be expressed as

$$\frac{1}{X}\frac{d^2X}{dx^2} - m^2 = \frac{1}{\alpha}\frac{1}{T}\frac{dT}{dt} \tag{8.31}$$

Or conveniently

$$\frac{1}{X}\frac{d^2X}{dx^2} = \frac{1}{\alpha}\frac{1}{T}\frac{dT}{dt} + m^2 \tag{8.32}$$

Since the left side is a function only of x and the right side is a function only of t, the only way it can hold for all x and t is that each side must be a constant. There is a separation constant such that

$$\frac{1}{X}\frac{d^2X}{dx^2} = -\lambda^2 \tag{8.33}$$

And

$$\frac{1}{\alpha}\frac{1}{T}\frac{dT}{dt} + m^2 = -\lambda^2 \tag{8.34}$$

The constant is squared for later convenience.

Now 8.34 gives two ordinary differential equations:

$$\frac{d^2X}{dx^2}(x)+\lambda^2 X(x)=0$$
(8.35)

And

$$\frac{1}{\alpha}\frac{dT}{dt}(t)+\left(m^2+\lambda^2\right)T(t)=0 \tag{8.36}$$

The nature of the solutions the above equations will depend upon value given λ^2. The three types of solutions of 8.27, combining the separated solutions of 8.35 and 8.36 are:

For $\lambda^2 =0$:

$$V(x,t)=\left(c_1 x+c_2\right)c_3 e^{-m^2\alpha t} \tag{8.37}$$

For $\lambda^2 >0$:

$$V(x,t)=\left(c_4\cos\lambda x+c_5\sin\lambda x\right)c_6 e^{-\left(m^2+\lambda^2\right)\alpha t} \tag{8.38}$$

For $\lambda^2<0$:

$$V(x,t)=\left(c_7 e^{\lambda x}+c_8 e^{-\lambda x}\right)c_9 e^{-\left(m^2-\lambda^2\right)\alpha t} \tag{8.39}$$

It may be verified that the case where $\lambda^2<0$ requires that the constants be zeros. Hence 8.39 only gives the trivial solution and may be discarded.

Therefore, the solution of 8.37 that satisfy the boundary condition is

$$V(x,t)=c_0 e^{-m^2\alpha t} \tag{8.40}$$

With $\lambda^2 =0$, where $c_o = c_2c_3$. The subscript, 0, is to emphasize the value of separation parameter. Similarly, 8.38 is satisfied by 8.29 for the case where $\lambda^2 >0$, as

$$\frac{\partial V}{\partial x}(0,t)=\lambda\left(-c_4\sin 0+c_5\cos 0\right)c_6 e^{-\left(m^2+\lambda^2\right)\alpha t}=0 \tag{8.41}$$

We get

$$c_5=0 \tag{8.42}$$

From

$$\frac{\partial V}{\partial x}(L,t)=\lambda\left(-c_4\sin\lambda L\right)c_6 e^{-\left(m^2+\lambda^2\right)\alpha t}=0 \tag{8.43}$$

We get

$$\sin\lambda L=0 \tag{8.44}$$

Thus for non-zero solution

$$\lambda = \frac{n\pi}{L}, \qquad n = 1,2,.... \tag{8.45}$$

The value obtained from 8.45 is called the eigenvalue. Consequently, the corresponding eigenfunction is

$$\cos\left(\frac{n\pi}{L}x\right) \tag{8.46}$$

Thus

$$V(x,t) = c_n \cos\left(\frac{n\pi}{L}x\right)e^{-\left(m^2+\frac{n^2\pi^2}{L^2}\right)\alpha t} \tag{8.47}$$

Where $\lambda^2 > 0$ $c_n = c_4 c_6$. The subscript, n, denotes the multiplicity of λ.

The superposition principle applies and the generalized transient temperature solution may be expressed as the series

$$V(x,t) = e^{-m^2\alpha t}\left[c_0 + \sum_{n=1}^{\infty} c_n \cos\left(\frac{n\pi}{L}x\right)e^{-\left(\frac{n^2\pi^2}{L^2}\right)\alpha t}\right] \tag{8.48}$$

Where, in view of the last condition 8.30,

$$V(x,0) = -\frac{q_0''}{mk\left(1-e^{-2mL}\right)}\left[e^{m(x-2L)} + e^{-mx}\right] = c_0 + \sum_{n=1}^{\infty} c_n \cos\left(\frac{n\pi}{L}x\right) \tag{8.49}$$

Under the assumption that U(x) is pricewise smooth. The series in 8.49 is the Fourier cosine series representing –U(x) on the interval 0 < x < L, the coefficients are given by

$$c_0 = -\frac{1}{L}\int_0^L U(x)dx \tag{8.50}$$

And

$$c_n = -\frac{2}{L}\int_0^L U(x)\cos\left(\frac{n\pi}{L}x\right)dx \tag{8.51}$$

Then, when n=0

$$c_0 = -\frac{1}{L}\frac{q_0''}{mk\left(1-e^{-2mL}\right)}\left[e^{-2mL}\int_0^L e^{mx}dx + \int_0^L e^{-mx}dx\right] \tag{8.52}$$

And for n>0

$$c_n = -\frac{2}{L}\frac{q_0''}{mk\left(1-e^{-2mL}\right)}\left[e^{-2mL}\int_0^L e^{mx}\cos\left(\frac{n\pi}{L}x\right)dx + \int_0^L e^{-mx}\cos\left(\frac{n\pi}{L}x\right)dx\right] \qquad (8.53)$$

After integrating and simplifying:

$$c_0 = -\frac{q_0''}{m^2 kL} \qquad (8.54)$$

And

$$c_n = -\frac{2q_0''L}{k\left(m^2L^2 + n^2\pi^2\right)} \qquad (8.55)$$

Substitution of 8.54 and 8.55 into 8.48 completes the determination of transient term, i.e.,

$$V(x,t) = -\frac{2q_0''}{k}e^{-m^2\alpha t}\left[\frac{1}{2m^2L} + L\sum_{n=1}^{\infty}\frac{1}{\left(m^2L^2+n^2\pi^2\right)}\cos\left(\frac{n\pi}{L}x\right)e^{-\frac{n^2\pi^2}{L^2}\alpha t}\right] \qquad (8.56)$$

Finally, the formal solution of the temperature distribution of the fin of the form (6) is:

$$\theta(x,t) = \frac{q_0''}{k}\left\{\left[\frac{e^{m(x-2L)}+e^{-mx}}{m\left(1-e^{-2mL}\right)}\right] - 2e^{-m^2\alpha t}\left[\frac{1}{2m^2L} + L\sum_{n=1}^{\infty}\frac{\cos\left(\frac{n\pi}{L}x\right)e^{-\frac{n^2\pi^2}{L^2}\alpha t}}{m^2L^2+n^2\pi^2}\right]\right\}$$

(8.57)

So, summing up we obtain:

$$\theta(x,t) = U(x) + V(x,t)$$

$$T(x,t) = \theta(x,t) + T_\infty = U(x) + V(x,t) + T_\infty \text{ (unsteady temperature distribution)}$$

and since $V(x,t) \to 0$ as $t \to \infty$, the steady-state solution becomes

$$T_{steady}(x) = U(x) + T_\infty$$

Chapter 9

Pelton Wheel

9.1 Introduction

Impulse turbines are the oldest forms of hydraulic machines used for converting hydro energy to mechanical work. These are also the simplest hydraulic machines in terms of their transparent design, low maintenance and easy control. They are generally used at hydro power plants characterized by high heads and low discharges. Being a low specific speed machine their designs need not be that robust and complicated. The specific speed can, however, be increased by the addition of extra 'nozzles' when the need arises. Moreover, since these machines operate under atmospheric pressure, there is also no need of elaborate seal designs. Even the cavitation risk on them is very much limited as compared to other type of turbines. Because of these and other advantages impulse turbines have become the most widely used hydraulic machines for generating micro-hydro power all over the world.

Impulse turbines use the total available head of water in form of kinetic energy of one or more jet(s) to run their runners. Movement of the water in runner passage takes place with a free surface contacting the ambient air, which implies that the energy available is extracted from the flow at atmospheric pressure. There is no pressure change across the runner blades. The flow in the runner only changes its direction, the magnitude of relative velocity remaining same. The work is done on the runner by the fluid due to the change in angular momentum and the motion of the vanes.

The modern Pelton wheel as shown in Figure 6.1 is a tangential partial turbine with double discharge bucket. In it one or more jets of water impinge on a wheel containing many curved buckets. The jet stream is directed inwardly, sideways and outwardly there, producing a force on the bucket which in turn results in a torque on the shaft. All the available head is converted to kinetic energy at the nozzle. As compared to other similar turbines it is the only most efficient turbine for high heads. It has a high efficiency of about 90% at the rated output and can maintain the same efficiency even under part load operation in the case of a multi jet design. It is suitable for both horizontal as well as vertical shaft arrangement and it can be equipped with multiple nozzles as per requirements. Apart from having all these advantages it is simple in design and cheap in construction, makes it is conventionally considered suitable for operating under very high heads in excess of 200 m and going up to 2000 m. Hence this device is the most commonly used impulse turbine in the world.

The Pelton Wheel used in our lab is Cussons p6240 Pelton Wheel. Although a model, it reproduces all the characteristics of full size machines and allows an experimental program to determine the performance of the turbine and to verify design theory as well.

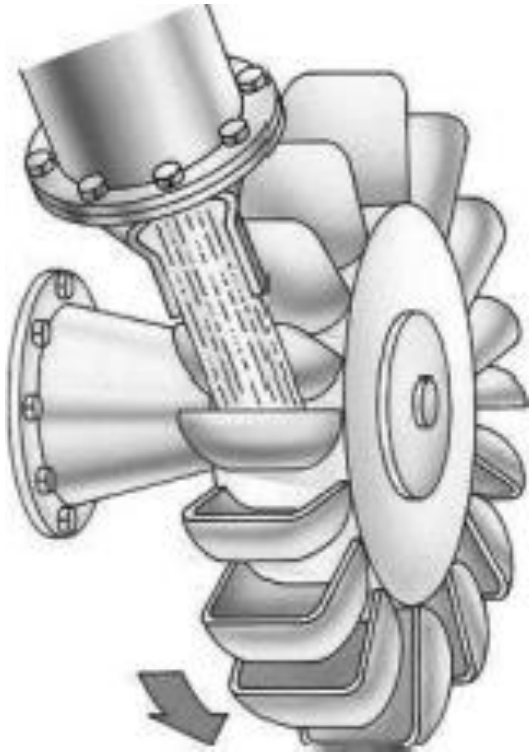

Figure 3: Pelton Wheel

Cussons P6240 Pelton Wheel consists of a Degener 4717 model Pelton Wheel mounted on a base plate and fitted with a friction dynamometer. The design of the Degener Pelton Wheel follows typical industrial practice with a horizontal shaft, single horizontal jets produced by a single nozzle fitted with a needle or spear regulator, and a wheel fitted with multiple (16) elliptical ridged buckets at a mean diameter of 100mm.

The nozzle is positioned in the same plane as the wheel and arranged so that the jet of water impinges tangentially onto the buckets. The nozzle and a single bucket are illustrated in Figure 6.2 below. Each "bucket" is divided by a "splitter" ridge which divides the jet into two equal parts. The buckets are shaped so as to prevent the jet to the preceding bucket being intercepted too soon. After flowing round the inner surface of the bucket, the fluid leaves with relative velocity almost opposite in direction to the original jet. The desired maximum deflection of the jet (180°) cannot be achieved without the fluid leaving bucket striking the following one, and so in practice the deflection is limited to approximately 155° (i.e. see figure 6.2 below 180 - 25°). The bottom of the casing is open to allow the water leaving the buckets to drain away. The front face of the casing is transparent acrylic allowing easy observation of the behavior of the water jet and assessment of exit angles.

The nozzle is controlled by movement of the spear regulator along the axis of the nozzle which alters the annular space between the spear and the housing, the spear being shaped so as to induce the fluid to coalesce into a circular jet of varying diameter according to the position of the spear. A static pressure tapping is provided to allow measurement of the inlet water pressure.

The friction dynamometer consists of a 60mm diameter brake wheel fitted with a fabric brake band. The brake band is tensioned by a weight hanger and masses. The fixed end of the brake band is secured to the support frame via a spring balance.

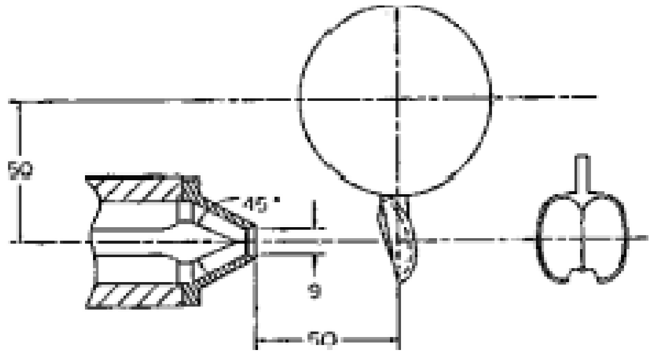

Figure 4: Figure 6.2: Detail of Pelton Wheel Buckets.

9.2 Theory
9.2.1 List of Symbols

A_r	area of incident jet	m^2
C	loss coefficient	
C_v	coefficient of velocity for jet	
F	force exerted on bucket	N
g	acceleration due to gravity	$9.806 m/s^2$
H	inlet head	m
k_1	frictional resistance coefficient	
$\dot{m}$	mass flow rate	kg/s
$\dot{Q}$	rate of flow of jet	m^3/s
R	radius of turbine rotor	m
U	tangential velocity of wheel	m/s
V_1	incident jet velocity	m/s
V_{r1}	incident jet velocity relative to bucket	m/s
V_{r2}	emergent jet velocity relative to bucket	m/s
$\dot{W}$	power output	Watts
η	wheel efficiency	%
θ	angle between incident and emergent jets	rads
ρ	density of water torque	kg/m3
τ	torque	Nm
ω	angular velocity	rads/s

9.2.2 Velocity Analysis

Consider a Pelton Wheel rotating in an anti-clockwise direction with an angular velocity ω due to the combined action of an incident water jet of velocity V_i and a clockwise resisting

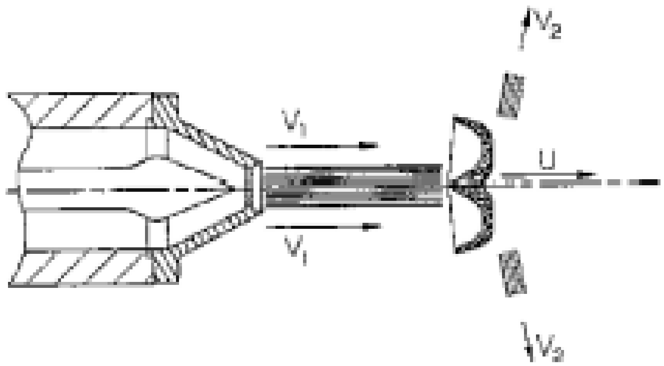

Figure 5: Absolute Velocities.

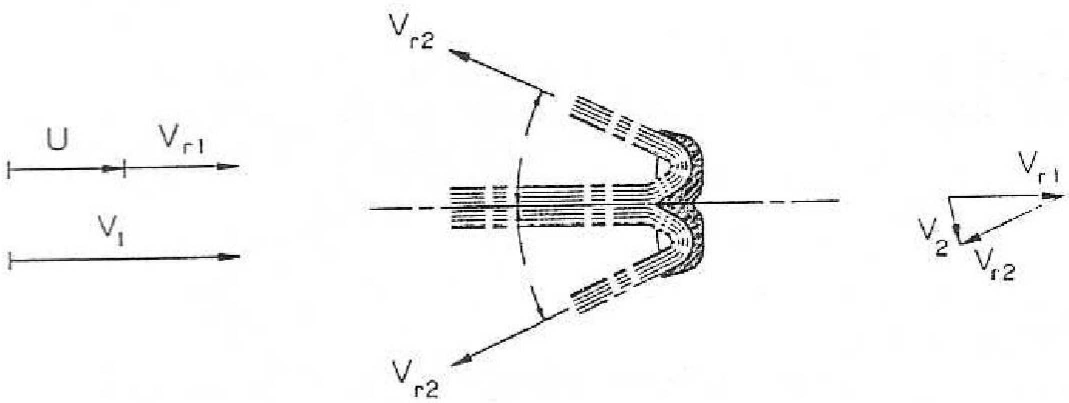

Figure 6: Velocities Relative to Bucket.

torque τ as shown in Figure 5. The velocity analysis will use a bucket on the Pelton Wheel as reference as shown in Figure 6.

The velocity of the incident jet relative to the bucket is given by

$$V_{r1} = V_1 - U = V_1 - \omega R$$

Since the incident and emergent jets are both exposed to atmospheric pressure, the magni-tude of the emergent jet velocity will be slightly less than the incident jet due to frictional resistance, which can be accounted for by introducing a frictional resistance coefficient k_1, such that

$$|V_{r2}| = k_1 V_{r1}$$

The jet is deflected so that the emergent jet is at an acute angle θ to the incident jet. The change in the component of velocity in the plane of the wheel (i.e. in the line of the incident jet) is:

$$\Delta V = V_{r1} + V_{r2}\cos\theta = V_{r1}(1 + k_1\cos\theta) = (V_1 - U)(1 + k_1\cos\theta)$$

If we define $c = k_1\cos\theta$, the above equation can be written as:

$$\Delta V = (V_1 - U)(1 + c)$$

9.2.2 Nozzle Flow Coefficient

The discharge through the nozzle from the inlet head H is given by

$$\dot{Q} = A_r V_1$$

Since the inlet velocity is

$$V_1 = C_v\sqrt{2gH}$$

then $\dot{Q}$ can be expressed in terms of Head H,

$$\dot{Q} = A_r C_v\sqrt{2g\ H}$$

9.2.4 Power Output

The force exerted on the bucket by the water jet can be obtained by using Newton's second law, which is the rate of the change of momentum between the incident and emergent jet in the plane of the wheel.

$$F = \dot{m}\Delta V = \dot{Q}\rho\Delta V$$

Then the torque acting on the shaft of the Pelton Wheel is:

$$\tau = FR = \dot{Q}\rho\Delta VR$$

and the power output is

$$\dot{W} = \tau\omega = \dot{Q}\rho\Delta VR\frac{U}{R} = \dot{Q}\rho\Delta VU$$

substituting for $\dot{Q}$ and ΔV

$$\dot{W} = A_r C_v\sqrt{2gH}\rho U\ (V_1 - U\ )(1 + k_1\cos\theta)$$

9.2.4Variation of Power Output With

For a given head and nozzle area the power output will be maximum if C_v and k_1 have their highest obtainable values while $\cos\theta = 1$and $U(V_1 - U\ )$is at its maximum.

$U(V_1 - U\ )$will be a maximum when $V_1\ =\ 2U$, that is, the incident jet velocity is at twice the Pelton Wheel bucket speed. For $\cos\theta = 1$, it yields $\theta = 0$, i. e. the incident jet is deflected backwards 180° to form the emergent jet.

With $V_1 =\ 2U$,i.e. $V_{r1} = U$ (and if ignoring friction across the buckets we also have $V_{r2} = U$) then the absolute velocity of the emergent jet in the plane of the wheel will be zero, all of the velocity head of the incident jet will be converted into useful work in the meantime the water will effectively fall off the trailing edge of the bucket.

Then the maximum power output is:

$$\dot{W}_{max} = \frac{\dot{Q}\rho V_1^2}{2} = \frac{A\rho V_1^3}{2} = \frac{A\rho C_v^3(2gH)^{\frac{3}{2}}}{2}$$

When the resisting torque is reduced, the wheel will accelerate, ΔV will decrease. The extreme case which would be $\Delta V = 0\ , U = V_1$, though it will not be obtainable in practice. As a consequence, both the torque and power output would be zero.

When the resisting torque is increased, the wheel can come to a stall. In such situation, $U = 0$ and the stall torque will be

$$\tau_{stall} = \dot{Q}\rho R\ V_1(1 + c)$$

9.2.6 Efficiency

The input hydraulic power to the Pelton Wheel is the product of the inlet pressure and flow rate.

$$\dot{W} = P\dot{Q} = \rho g\ H\ \dot{Q}$$

the efficiency of the Pelton Wheel is

$$\eta = \frac{\dot{W}}{\dot{W}_{in}} = \frac{\dot{Q}\rho U \, \Delta V}{\rho g H \dot{Q}} = \frac{U \Delta V}{gH}$$

Since the hydraulic power input depends only on the head and the nozzle area, it is independent of the Pelton Wheel speed. Such that efficiency 'T/ is directly proportional to the power output, power and efficiency will both achieve their maximum under the same conditions. In such kind of situation,

$$\eta_{max} = \frac{U \, \Delta V}{gH}$$

Substituting for ΔV,

$$\eta_{max} = \frac{U \, (\mathrm{V}_1 - U)(1 + c)}{gH}$$

under maximum power $V_1 = 2U$

$$\eta_{max} = C_v^2 \frac{1 + c}{2} = \frac{C_v^2 (1 + k_1 \cos\theta)}{2}$$

Thus under such kind of ideal situation, C_v, k_1 and cos 0 are all 1.0 and hence the ideal maximum efficiency is unity.

9.3 Experiment

9.3.1 Objective

In this lab the pelton wheel experiment will be carried out to investigate the performance of the machine for a various range of flow rates and rotational speeds. Two sets of parameters can be changed independently: the load applied to the friction brake and the position of the nozzle regulating spear. The position of the nozzle regulating spear controls the nozzle flow rate, while different friction brake load results in the change of turbine wheel rotational speed.

9.3.2 Experimental Procedure

The easiest way to organize the experiment is to set the nozzle regulating spear and conduct a number of measurement under different brake loadings. The turbine wheel speed is affected by the coefficients of friction between the band and the shaft pulley, which is subjected to change under different temperature, therefore we should not take readings before the turbine wheel speed is stabilized.

1. Switch on the Hydraulics Bench pump and fully open the bench regulating valve.
2. Fully open the spear regulator to produce maximum flow rate. Remove all weights including the weight carrier and unhook the friction band from the Pelton Wheel shaft. Measure the water flow rate and the free unloaded rotational speed of the Pelton Wheel by using a tachometer. Observe the emerging jet from the Pelton Wheel and make an assessment on the angle between the incident and emergent jets.
3. Replace the friction band around the shaft pulley and attached the weight carrier. Record the speed, the net load in the brake band (weights - spring balance reading), the volume flow rate, inlet pressure. Observe the emerging

jet and again make an assessment on the angle between the incident and emergent jet.

4. Add an additional weight to the weight carrier and wait for the speed to stabilize, record the measurements of speed, net load, flow rate, inlet pressure and assess the angle between the incident and emerging jets.
5. Repeat step 4 until the Pelton Wheel stalls. Record the measurements for each con-dition including the stalled condition.

Repeat procedure for difference settings of the spear regulator.

9.4 Report

What you need to present:

1. Plot of torque versus wheel speed for all valve pressure settings
   Steps you need to follow:
   a. Import all your data in MATLAB
   b. Convert everything into SI units:
      i. Volume flow $\left[\frac{m^3}{s}\right]$
      ii. Inlet Pressure $[\mathrm{Pa}]$
      iii. Wheel Speed $\left[\frac{\mathrm{rad}}{s}\right]$
   c. Calculate torque:
   $$T = (W - S)r$$
   Where, W is the brake load, S is the spring force, r is the brake wheel radius
   d. Plot T versus wheel speed
2. Calculate the power input, P_i
   $$P_i = PQ$$
   Where, P is the pressure, Q is the volume flow rate

3. Plot of power output, P_o, versus wheel speed for all valve pressure settings
   Steps you need to follow:
   a. Calculate power (N is the wheel speed):
   $$P_o = TN$$
   b. Plot P_o versus wheel speed

4. Plot of efficiency, η, versus wheel speed for all valve pressure settings
   Steps you need to follow:
   a. Calculate efficiency:

$$\eta = \frac{P_o}{P_i}$$

Where, P_o is the power output, P_i is the power input

b. Plot η versus wheel speed

5. In Appendix B, show the sample of all calculations done in MATLAB.
6. In Appendix B, compute the uncertainty of the efficiency. Add error bars to the appropriate plot.