DESIGN OF WIND TUNNEL (FLUID FLOW ANALYSIS)

NELTON KOO CHWEE YANG

A thesis submitted in fulfillment of the requirements

for the award of the Degree of

Bachelor of Manufacturing Engineering

Faculty of Manufacturing Engineering

UNIVERSITI MALAYSIA PAHANG

JUNE 2012

EXAMINER'S APPROVAL DOCUMENT

I certify that the thesis entitled "Design of wind tunnel (Fluid Flow Analysis)" by Nelton Koo. I have examined the final copy of this thesis and in my opinion; it is fully adequate in terms of scope and quality for the award of the degree of "Bachelor of Manufacturing Engineering". I herewith recommend that it be accepted in fulfillment of the requirements for the degree of Bachelor of Manufacturing Engineering.

Name of Examiner: Signature:

Institution:

SUPERVISOR'S DECLARATION

"I hereby declare that I have checked this thesis and in my opinion, this thesis is adequate in terms of scope and quality for the award of Degree of Bachelor of Manufacturing Engineering (Manufacturing)"

Signature : ………………………………………………..

Name of Supervisor : MR. SHAIFUL HAKIM

Position :

Date : JUNE 2012

STUDENT'S DECLARATION

I hereby declare that this thesis entitled "Design of Wind Tunnel (Fluid flow analysis)" is the results of my own research except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature	:	……………………………..
Name	:	NELTON KOO CHWEE YANG
ID Number	:	FA08052
Date	:	JUNE 2012

Special Dedication to my beloved father and mother, my respected supervisor,

Mr Shaiful Hakim, and my family members for their constant encouragement and motivation and also to my friends who have support and inspired me.

ACKNOWLEDGEMENTS

The special thanks go to my helpful supervisor, Mr Shaiful Hakim.The supervision and support gave truly help the progression and smoothness of the final year project. The co-operation is much indeed appreciated.

My grateful thanks to my team member Hidayah, Norlizaa, and Husna. A big contribution and hard worked form them during the twelve week is very great indeed. All projects would be nothing without the enthusiasm and imagination from them.

Great deals appreciated go to the contribution of my faculty, Faculty of Manufacturing Engineering (FKP). I am also would like to thankful the Coordinator of final year project, Prof Madya Dr Ajisman Apren, and all the staff in the FKP office that patient in helping us complete this project.

I hope this research project will be helpful for those who need reference in the field of Manufacturing with Aerospace background. Last but not least, I would like to express my gratefulness to all of them who involve directly or indirectly in the completion of my final year project. Thank you.

ABSTRACT

A low speed wind tunnel was design and fabricate. The project covered the process of design and fabrication of the small wind tunnel. In completing this project, a computer aided drawing (CAD) called Solid Work is use to design the wind tunnel. Experiment conducted after build the wind tunnel to find drag coefficient of a sphere. The drag force on a sphere in an air stream was measured at various free stream velocities below 100 m/s. This was done in a low speed wind tunnel using an integral balance system to measure the drag force and a Pitot tube and to measure the velocity. The raw data were processed according to classical equations of fluid mechanics which define the drag coefficient. An impression of fluid field flow around a sphere is also capture using white smoke. Method of analysis the flow in test section was shown by using strings. The experimental results are compared to published results over the range tested.

ABSTARK

Terowong angin subsonik direka dan dibina. Projek ini merupakan proses reka bentuk dan fabrikasi terowong angin. Rekaan terowong angin dilukis dengan bantuan computer aided drawing (CAD) iaitu Solid Work. Experimen dijalankan untuk mencari pekali rintangan sfera diuji di terowong angin ini. Daya ringtangan suatu sfera diukur pada variasi kelajuan bawah 100 m/s. Ia boleh diuji melalui terowong angin dengan menggunakan sistem imbangan bagi mengukur daya ringtangan dan tiub pitot digunakan untuk mengukur kelajuan. Data diambil dan diproses dengan rumus cecair mekanik bagi mendpatkan nombor pekali ringtangan. Gambaran pegerakan udara pada sfera ditangkap dengan menggunakan kamera. Asap digunakan untuk melihat pegerakan udara pada sfera. Kaedah tali benang juga digunakan untuk menganalisis pegerakan udara di dalam kebuk ujian terowong angin. Keputusan experimen dibandingkan dengan keputusan theori di buku.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

F_D	Drag force in N
C_D	Drag coefficient
Re	Reynolds number
D	Diameter of sphere in m
ρ	Density of air in kg/m^3
u_∞	Velocity of air stream in m/s
P	Atmospheric pressure in N/m^2
Δp	Pressure difference in manometer in N/m^2
Δh	Difference in heights of liquid in manometer in mm
T	Atmospheric temperature in Kelvin
μ	Viscosity of air in kg/m-hr
ρ_o	Density of oil in manometer in kg/m^3

CHAPTER 1

INTRODUCTION

1.1 WIND TUNNEL

1.1.1 History

Discovery and development by experimental means has been its lifeblood, extending all the way back to George Cayley. In 1804, he built a whirling arm apparatus for testing aerofoil. This was simply lifting surface (aerofoil) mounted on the end of long road, which was rotated at some speed to generate a flow of an air over the aerofoil. In modern aerospace engineering, the workhorse for such experiments has been predominantly the wind tunnel, so much such that today most aerospace industries, government and university laboratories have a complete spectrum of wind tunnels ranging from low subsonic to hypersonic speeds.[19]

In other side, the enormous advances in computer technology both computer hardware and numerical methods have possible to model the fluid flow more accurate. The setup of model can be performed easily. This question comes up, whether theoretical calculation could one day substitute wind tunnel tests altogether. At present the computational method can surely predict the simple models such as real flow in wind tunnel still requires a large time consuming therefore it is still difficult and expensive. In addition the computational method still provides the calculation of drag not satisfactory compare to the wind tunnel results. Thus, for aeronautical developments wind tunnel testing will remain predominant in foreseeable future. [19]

However, besides for aerospace applications, the experimental techniques using wind tunnel also have broad applications in many other branches of science and engineering such in automotive and architecture. For automotive applications, the experimental techniques in wind tunnel may be used to predict aerodynamic characteristics of designed racing cars and other high performance vehicles. For architecture applications, the wind tunnel may be used to simulate the pollutant condition happened around buildings in the city. [19]

The scientists work in experimental techniques of wind tunnel is somewhat different than of engineers. Scientists are engaged in fundamental research, engineers make experiments in course of the project. Their detailed investigations are aiming at the improvement of basic knowledge about the physics of fluid flow. The results may provide the foundation of new theories, or are used to evaluate or scrutinize existing theories or supplement them. Quite often they don’t appear have any direct bearing on practical engineering problems. For engineering purposes test techniques to be employed during project work must be proven standard, reliable, quick to use and possibly cheap. The test results must be provide comprehensive information such as the aerodynamic characteristics of a tested aerodynamic configuration. Details of the flow are studied just with the particular purpose of deducing improvements for the configuration. In general the test results must be comprising a proof, that the design targets are being met and safe. [19]

1.1.2 Wind tunnel

A wind tunnel is a device designed to generate air flows of various speeds through a test section. Wind tunnels are typically used in aerodynamic research to analyze the behaviour of flows under varying conditions, both within channels and over solid surfaces. Aerodynamicists can use the controlled environment of the wind tunnel to measure flow conditions and forces on models of aircraft as they are being designed. Being able to collect diagnostic information from models allows engineers to inexpensively tweak designs for aerodynamic performance without building numerous fully-functional prototypes. In the case of this project, the wind tunnel will serve as an educational and research tool to analyze basic flow principles. [2]

The wind tunnel provides great benefits for aerodynamic tests compared to free flight testing, that is:

i. Specified flow condition such as Mach number and incidence can be achieved sustained much easier in a wind tunnel.
ii. Dangerous, uncontrollable flight condition may safely investigate in wind tunnel.
iii. Data acquisition and processing is simpler with direct connection to ground based equipment.

The main disadvantage of wind tunnel is that it is seldom possible to reproduce the condition of full scale motion exactly. This is mainly due to the use of scaled models for reason of tunnel cost and power consumption. [2]

1.1.3 Fluid flow

For centuries, fluid flow researchers have been studying fluid flows in various ways, and today fluid flow is still an important field of research. The areas in which fluid flow plays a role are numerous. Gaseous flows are studied for the development of cars, aircraft and spacecrafts, and also for the design of machines such as turbines and combustion engines. Liquid flow research is necessary for naval applications, such as ship design, and is widely used in civil engineering projects such as harbour design and coastal protection. In chemistry, knowledge of fluid flow in reactor tanks is important; in medicine, the flow in blood vessels is studied. Numerous other examples could be mentioned. In all kinds of fluid flow research, visualization is a key issue [3]

1.2 PROBLEM STATEMENT

This paper will focus primarily on the fabrication process of small scale wind tunnel, flow visualization analysis on an object and calculation of drag coefficient of an object through experiment.

1.3 OBJECTIVES

i) To develop a small scale wind tunnel for educational and research purpose.
ii) To get an impression of fluid flow around a scale model of a real object.
iii) To calculate the drag coefficient of object design.

1.4 PROJECT SCOPES

i) To find the design fundamental for a small wind tunnel.
ii) Make the research for small wind tunnel background and contruction.
iii) To find the best material to be used and estimate the cost for model construction.
iv) To study the flow visualazation of an object design.
v) To determine the drag coefficient of an object design.

CHAPTER 2

LITERATURE REVIEW

2.1 FLUID MECHANICS TERMINOLOGY

Drag is the component of force acting on a body that is projected along the direction of motion. Both shear forces and pressure induce drag on a body in motion. Shear forces, known as skin friction drag, are more significant in streamlined objects, while the pressure drag is more significant in blunt objects [14] Figure 2.1 shows the net drag force acting on a cylinder.

The drag force is often non-dimensionalized as a function of Reynolds number. This is then referred to as the drag coefficient (Eqn2.1). Similarly, the pressure acting on each differential element of an object may be normalized by the dynamic free stream pressure $\frac{1}{2}\rho v_{\infty}{}^{2}$ to obtain the pressure coefficient (Eqn2.2). This quantity may also be rewritten as the reduced pressure coefficient (Eqn2.3).

$$C_D = \frac{F_D}{\frac{1}{2}\rho v^2 A} \tag{2.1}$$

$$C_P = \frac{\Delta p}{\frac{1}{2}\rho v^2} \tag{2.2}$$

$$C_P = \frac{q}{q_\infty} \tag{2.3}$$

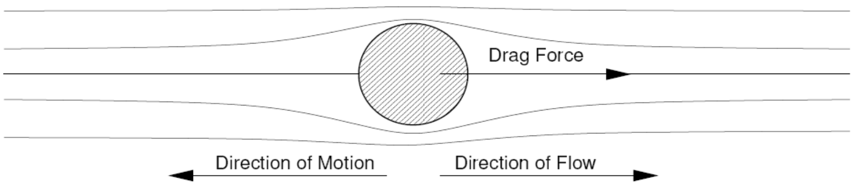


Figure 2.1: Drag force on sphere

2.2 TESTING PARAMETER

2.2.1 Flow consideration

In wind tunnel testing, the real configurations such an aircraft and its components are usually scaled down to become the corresponding small size model that can be installed in test section of wind tunnel. In order to obtain similar flow condition in the wind tunnel as free flight at full scale, it requires more than just to ensure geometrical similarity of the model. One also has to take care that forces produced by flow are in the same relationship to each other. [14]

In fluid dynamics there exist a number of similarity parameters. They represent the relation of the various forces in a fluid flow such as:

i. Inertia forces = ρV^2
ii. Viscous forces = $\mu V/I$
iii. Compression forces = ρa^2
iv. Gravity forces = ρgI

All forces here are taken per unit area. Similarity parameters are formed by relating the various forces to the inertia force. They include Reynolds number, Mach number and Froud number.[2]

2.2.2 Reynolds number

The Reynolds (Re) number is a quantity which engineers use to estimate if a fluid flow is laminar or turbulent. This is important, because increased mixing and shearing occur in turbulent flow. [2]

The Reynolds number is calculated using mean velocity, pipe diameter, density, and viscosity, and is valid for any fluid. The Reynolds number is also dependent upon the geometry of the pipe, as well as the roughness of the walls. Analysis of the Reynolds number using the dimensionless forms of the Navier Stokes equations reveals that the Reynolds number is really a ratio of inertial forces to vicious forces. As of yet, no successful analytic methods for determining Reynolds numbers have been developed due largely to the difficulty associated with predicting turbulent flow, and so Reynolds numbers for flow through pipes or around immersed objects must be determined experimentally. We define Reynolds number as

$$Re = \frac{\rho L_{char} V}{\mu} \tag{2.4}$$

The L_{char} is the length characteristic of geometry and V is a velocity appropriate for the flow. Osborn Reynolds identifies this parameter in1883 as being important in fluid mechanics. If Re < 2100 it is said to be in laminar and Re > 4000 is in turbulent state. [2]

2.2.3 Mach number

This number relates the compressibility to the inertia forces. Its similarity is important when noticeable variations of density and temperature occur at high flow velocities.[9] The Mach number is defined as:

$$\text{Ma} = \frac{V}{a} \tag{2.5}$$

2.2.4 Froud number

This number relates the gravity to the inertia forces. Its similarity is important when gravity forces are involved. This is important for model drop tests. It is also interesting when waves occur such when the hydrodynamic analogy is used to simulate supersonic waves.[9] The Froude number is defined as:

$$F_r = \frac{V}{\sqrt{gI}} \tag{2.6}$$

When the two flow parameters such as Mach number and Reynolds number and the objects are geometrically similar, the flows will be dynamically similar and the results from investigating one flow should be transferable to the other. [2]

For example suppose want to improve the design of a golf ball. You want the ball to have as small a drag force as possible. If the golf ball 0.04 meter in diameter, travel at 25m/s and at sea level air, then it has Reynolds number of 68,200 based on its diameter and its travelling at Mach number of 0.06. You can make a large model of a golf ball, say 0.2 in diameter and tested it in your wind tunnel as long as you match the Reynolds number. Mach number is so low and can be ignore, there is no significant compressibility effects in the flow field. In this case you should you should run your wind tunnel with a free stream velocity of 5m/s to match Reynolds number. Under these conditions the boundary layer on the golf ball and the wake behind it will be perfectly to scale. If the golf ball in flight has a boundary layer of 0.001 meter thick at one point, then the model will have boundary layer of 0.005 meter thick its corresponding point. You are free to test all sort of dimple design and the drag coefficients measure will be the sane drag coefficient the ball would experience at 25 m/s. [2]

2.2.5 Boundary layer

Boundary layers are regions of fluid located immediately adjacent to an immersed object or wall in which flow velocities are governed by viscous forces. Drag forces and most of the heat exchange experienced by the object are due to fluid in this region. Boundary layers typically begin as a very thin region of laminar flow that thickens with increasing Reynolds numbers and then gradually transitions to a turbulent layer flowing over a viscous sub layer. Flow outside of the boundary layer is independent of Reynolds number criteria. Figure 2.2 shows boundary layer of a flat plate. [21]

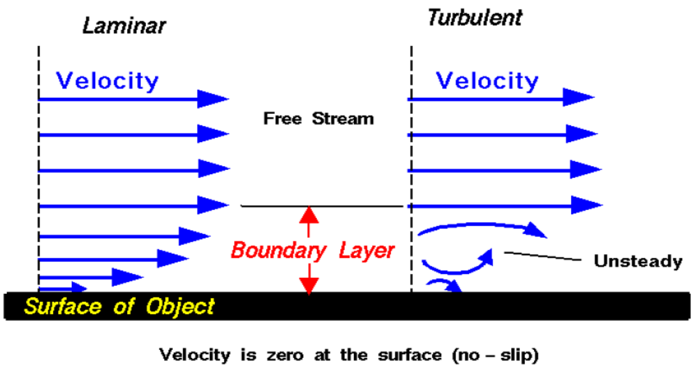


Figure 2.2: Velocity profile for boundary layers along the wall

2.3 WIND TUNNELS

Wind tunnel is a device used to investigate an interaction between solid body flows in wind tunnel can be performed in term of:

i. Monitoring physical flow phenomenon such as laminar, tuberlent and separation flows, vortex and shock wave.
ii. Measuring aerodynamic quantities such as pressure, skin friction, lift, drag and moments.

In order to monitor the flow phenomenon and measure aerodynamic quantities, engineers require measuring equipments and measurement techniques. One experimental aerodynamic problem can be solved by some different measurement techniques. In addition, for a special problem of experimental aerodynamic sometimes requires a specific wind tunnel construction. [9]

The aerodynamic problems can be distinguished in two matters:

i. External aerodynamics, that is solid body immersed in the flow such as around wings or aerofoils.
ii. Internal aerodynamics that is flow moving inside the body such as ducts, pipes and turbines.

2.4 CLASSIFICATION OF WIND TUNNEL

2.4.1 Based on speed range

The most appropriate classification of wind tunnels is by the speed range they cover. The classification of wind tunnels based on the speed range includes:

i. Low speed wind tunnel
The flow velocity in low subsonic wind tunnel is of the Mach number range of zero till 0.3. Viscous and inertial forces are dominant while compressibility effects are negligible.
ii. High speed wind tunnel
The designation high speed usually includes high subsonic, transonic and low supersonic regimes, so that the range of the flow velocity for high speed wind tunnel is of Mach number between 0.3 and 1.4. Here, in principle, compressibility effects are of dominant importance. However, viscous effects also play an important part in particular when shock boundary layer intersection leads to flow separation.
iii. Supersonic wind tunnel
The flow velocity in supersonic wind tunnel is the range of Mach number of 1.4 till 5.0. Compressibility effects are dominant. The pressure disturbance raises in the flow field propagating downstream.
iv. Hypersonic wind tunnel
The flow velocity in hypersonic wind tunnel is of Mach number above 5.0. It is desired to allow real gas effects to occur. This requires that besides the high Mach number in test section also high total temperatures are provided.

> The high temperatures, which are linked with high pressures, yield vibration of the gas molecules, possibly causing dissociation and ionization. These are dominant features of hypersonic flows where the gas can no longer be treated as an ideal gas. [9]

With increasing Mach number the tendency to intermittent operating wind tunnels linked with an appropriate energy storage arrangement, becomes more and more compelling. However, for measurement of low subsonic flow, the continuously operating wind tunnel is more preferred. [9]

2.4.2 Based on flow circulation

The other wind tunnel classification based on flow circulation is divided into open circuit wind tunnel and closed circuit wind tunnel.

i. Open circuit wind tunnel
   Open circuit wind tunnel is first type of wind tunnel built. The tunnel is usually referred to as an Eiffel type. Such a wind tunnel consists of a nozzle, at test section, a diffuser and a driving unit. The principle work of this wind tunnel is a direct sucking of the atmospheric air lying outside of the wind tunnel brought into the tunnel settling chamber and continued to the end of the wind tunnel using a driving unit then the air is threw away to atmosphere. The position of driving units can be at the downstream end where the tunnel is operated as suction tunnel while otherwise it would be termed a blow down tunnel. The suction tunnel is more preferred in a design by a reason of airflow quality.

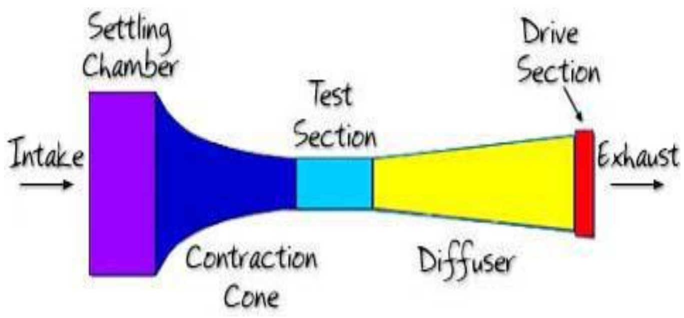


Figure 2.3: Open circuit wind tunnel

ii. Closed circuit tunnel

Closed circuit tunnel has been developed to reduce the amount of used energy. This tunnel is also called as Gottingen type. The principle work of this tunnel is by circulating the used airflow passing by the diffuser to the settling chamber using the connecting channel. The closed circuit tunnel consists of three types including single and double return. Of these, only the first is in general acceptance at present. In the double return arrangement, the particular air that scrapes along the wall of the return passages forms wakes in the centre of the jet and hence passes directly over the model. Unless the contraction ratio is large, this air is extremely tuberlent and tends to make the interpretation of the test data difficult. The fans are preferred attach on the connecting channel by reason of a protection from the model failure and of good from standpoint of fan efficiency.

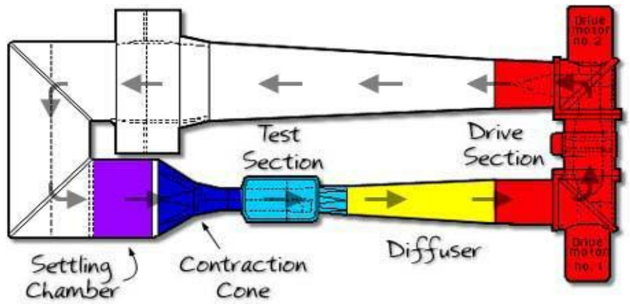


Figure 2.4: Closed circuit wind tunnel

The question as to which tunnel preference should be given depends on the weight attached to various arguments.

The open circuit tunnel is certainly the cheapest to build. Lack of space for the return leg may dictate choice of an open layout. The losses due to the kinectic energy rejected into the quiet surrounding air are detrimental. The problem that dust from the surrounding may be brought into the test section can easily be avoided by putting filter in front of intake. [14]

The closed circuit tunnel design is for most purposes the most useful arrangement. Particularly with respect to power economy it is the preferable layout. On the other hand, additional construction and maintenance cost are detrimental. Also, precautions must be taken that disturbances and noise from the fan have decayed before the flow entering into the test section. This problem of the closed circuit tunnels is that a cooler may become necessary. The losses in the tunnel flow result in an equivalent increase in heat energy which in fact is equal to the work performed by driving unit. [14]

2.5 BASIC COMPONENT OF LOW SUBSONIC WIND TUNNEL

In order to carry out its function obtaining aerodynamic experimental data, the subsonic wind tunnel composes several components. The test section is one of the important components where the test models to be located. The flow quality in the test section is strongly influenced by other components of the wind tunnel. The components of open circuit tunnel include and entrance cone, a settling chamber, a nozzle or contraction cone, a test section and a diffuser. While, the closed circuit tunnel has additional components to circulate the flow such corner, a return passage, breather and some control devices. [14]

2.5.1 Test Section

Test section of wind tunnel is where the model is located and tested. The flow from contraction cone comes into the test section expected in smooth or less turbulence condition. The flow condition in front of the test model in the test section will affect to the result of aerodynamic measurement of test model. Therefore any obstacle lies between end of contraction cone and test model ought to be removed. [9]

The test section consists of cross section, attached windows on a given side wall and model support. The cross sectional shape can be square, rectangular, rectangular with tempered corner, circular, circular with flats in the sides and floor, elliptical and octagonal. The test cross section area is the most fundamental consideration in selecting the test section shape. The cross section area directly affects the required power. Another consideration is combinations of aerodynamic, manufacturing and utility. The square and rectangular cross sectional are usually selected by reasons of ease manufacture and better window placement for viewing the model and ease of wing tip model attachment. For a given area of the square and rectangular cross section, the greatest width section can install the largest span model and thus be able to simulate the high Reynolds number flows. [9]

For bigger wind tunnel, instalment of a test model on test section often need a lot of time. This problem can be overcome by producing two exchangeable test sections. When the first test section is used for testing the model, at the same time the other test section can be prepared to install the subsequent tested model. The replacement of each test section can be performed using test section cart. [9]

2.5.2 Contraction Cone

Contraction cone is utilised to accelerate flow from the settling chamber to test section. The flow acceleration is produced by changing the area of contraction cone from large area at the inlet to small area at outlet of contraction cone. The flow velocity in the settling chamber is usually in the range of 5 to10 m/s. While the require flow velocity in the test section is a large enough depending on wind tunnel type and testing requirements.[14] The matters should be considered in the contraction cone including:

i. Outgoing flow from the contraction cone should be steady and uniform
ii. Separation flow should be avoided

iii. Boundary layer thickness releases from end of contraction cone should be thin as possible
iv. Contraction cone length is made short as possible.

2.5.3 Settling Chamber and Screens

Settling chamber is a room in the wind tunnel for conditioning the airflow to be calm. The airflow velocity in the settling chamber is decelerated by making the cross section area if the settling chamber be greater than that of other wind tunnel components. Based on a continuity principle for incompressible flow the increased cross section area will decrease the velocity. The irregular flows due to swirl, low frequency pulsation and turbulence can be suppressed in this room. More reduction of the irregular flows can be treated using screens and honeycombs. [9]The physical mechanism of screen and honeycomb to reduce the irregular flows can be explained as:

i. Screen and honeycomb have a characteristic as resistance that its quantity is equal to be quadratic of the flow velocity. When the local velocity is greater than the average flow velocity, the screen and honeycomb will produce higher drag.
ii. A screen is effective as a resistance when the flow velocity vector is parallel to flow direction, while honeycomb is effective as a resistance when the flow velocity vector is normal to flow direction.

Screens are far more useful than honeycombs for adjusting flow, and a fairly large percentage of modern tunnels employ fine mesh screens in the settling chamber to even the flow. The screen is normally made of woven metal wires to form square mesh. A typical installation might have one or two screens, low turbulence might have more than two screens. The effect of screen can improve the velocity distribution in entrance cone to become a more smooth flow so that the flow quality in the test section is better. [9]

2.5.4 Diffuser

A diffuser is a section installed at the end of test section. This section has a function of converting the kinetic energy of fluid at the downstream end of test section into a pressure potential energy as efficiently as possible. The adverse pressure gradient on the diffuser may result into separation boundary layer, severely degrading its efficiency. At this time, the separation flow can be avoided. The efficiency usually drops to become 80-90 %. [9]

2.5.5 Driving unit

For a steady flow in wind tunnel, the power input from the driving unit must be equal to the losses the flow encountering in the circuit. The losses related to skin friction, flow separation, turbulence are linked with dissipation of energy kinetic into heat. In closed wind tunnel the heat must be removed by heat exchanger. In case of an open tunnel this is avoided but additional losses occur due to rejection of kinetic at the diffuser exit.[14]

The powers require will depend on:

i. The efficiency of the driving system since only part of the driving power will be communicated to the airstream.
ii. The magnitude of the losses cause by friction between component surface and air.

2.6 WIND TUNNEL TESTING

Establishing a clear procedure is important to provide the students with maximum safety and efficiency, as the wind tunnel measurement is a very tedious work. A general suggestion is given here to be adopted and modified following the local needs.

2.6.1 Wind Tunnel testing preparation

The steps usually performed during preparation of wind tunnel testing include:

i. Defining the types of test will be performed, the kind of measurement technique will be adopted, the equipments should be used in the testing and the kind of testing data are extracted for analysis.
ii. Checking all required equipments are ready to use and in good condition.
iii. Checking all calibration curves of new equipment before a test.
iv. Checking the model that will be installed in test section. For pressure measurements, make sure that each plastic tube has been connected to pressure orifices on the model and no leakages.
v. Checking the wind tunnel motor and controller can be operated well.
vi. Being clear in all instruction.

2.6.2 General testing procedure

i. Install the tested model on the test section of wind tunnel properly.
ii. Check the test section wall has been closed tightly to avoid the flow leak.
iii. Determine the atmospheric condition by reading testing room data such as ambient pressure, temperature and humidity. The room data reading is accomplished before and after wind tunnel running.
iv. Prepare data chart and log book.
v. Switch on the wind tunnel motor to produce airflow in the wind tunnel. The airflow speed can be controlled by the rpm controller. At the first run the rpm controller should be turned to zero.
vi. Measure a free stream dynamic pressure by measuring the pressure references and by using the free stream calibration chart.
vii. Compute the free stream static pressure from the free stream dynamic pressure and total pressure.
viii. Take the testing data based on the measurement technique used.
ix. After the first run has been made, it should be checked thoroughly against expected results by making an appropriate graphics.

x. After all data has been taken, the wind tunnel motor is switched off.

2.7 TESTING MODEL

In order to provide accurate results of the wind tunnel testing, the model should be designed and manufactured to the highest possible standard of accuracy. The design of wind tunnel models is largely dictated by the data desired. The manufacturing process of the models with high accuracy is usually by numerically controlled machines. In addition, the inspection of the model geometry is usually done mainly on the machines. The accuracy achieved for the wind tunnel models is generally in the order to plus and minus 0.02mm. In earlier days quite often models are made from wood. Today, stainless steel is preferred material and aluminium is sometimes used for low speed models. For two dimensional model a combination of aluminium resin construction is preferred. [20]

2.7.1 Aerofoil models

The best aerofoil is meant to provide some detailed information about the flow field and especially to investigate the two dimensional aerodynamic characteristics of aerofoils. This is achieved by surface pressure measurements. In order to obtain the two dimensional aerofoil aerodynamic characteristic, the model is usually designed as wing section model with two end sides attaching on the wind tunnel walls. The pressure orifices are usually located at mid-span and near the wall. The pressure orifices at mid-span are utilized to measure the two dimensional aerofoil characteristics. While pressure orifices located near wall is used to investigate the effect three dimensional flows due to the side wall boundary layer. [20]

2.7.2 Aircraft models

The models of aircraft to be tested in wind tunnel are desires to be as large as possible. There are a variety of reasons for such desire, Reynolds similarity is improved with large models, representation of geometrical details is more accurate and their influence on the aerodynamics of the model configuration is more reliable to measure.

There is also more space the internal balance, for pressure tubes and for an integrated pressure scanning system, as well as for partial load measurements. Even a remote system for positioning the various control surface of an aircraft may be installed if the space is available. [20]

On the other hand, large and complicated models are expensive and even more so appropriate wind tunnel required. The final choice will be made with respect to the test requirements and wind tunnels available. For a given wind tunnel, restriction of model size results from the necessity to limit wall interference. [20]

Test requirement for complete aircraft models may comprise the following measurements:

i. Total forces and moments, in particular its variation with model incidence and free stream Mach number.
ii. Partial loads on aircraft components, including loads on the control surfaces and the root bending moment.
iii. Surface pressure distribution, in particular for a number of span wise wing cross sections, for surroundings of the engine and selected sections on the fuselage.

Beside that, the test may include flow visualisation.

2.8 MEASUREMENT TECHNIQUES

2.8.1 Pressure measurement

Surface pressure measurements as well as static and total load head measurements in the flow field are very common in wind tunnel testing. The local pressures are directly related to the local flow velocity or Mach number. Thus the pressure measurement provides detailed information about the flow field. [14]

For a measurement of two dimensional aerofoil aerodynamic characteristic, the model is usually designed as a wing section model with two end sides attaching on the wind tunnel walls. The pressure orifices are usually located at mid-span and near the wall. The pressure orifices at mid-span are utilized to measure the two dimensional aerofoil characteristics. While the pressure orifices located near the wall is use to investigate the effect three dimensional flows due to sidewall boundary layer. [14]

The pressure orifices are connected to tubes that will be gathered into a bundle. The bundle will be led to a scanivalve. From scanivalve the pressure can be measured using a fluid manometer or digital pressure indicator. For a modern measurement, the scanivalve are combined with pressure transducers and connected to a data acquisition unit and controlled by a computer. The measured pressures from the pressure orifices on the model surfaces are static pressures. [14]

The static pressure data can be integrated to obtain the lift and pitching moment, and their coefficients. The static pressure data along the model surface cannot be used for calculating the drag. The drag is measured by applying the principles of energy conservation. A model in a uniform flow domain causes a reduction of the uniform energy flow due to surface friction. This is marked by a reduction of the total energy (momentum) of the uniform flow behind the model. The momentum of the flow downstream the model is measured using a total wake rake. [14]

2.8.2 Force measurement

Force and moment may be obtained by integrating the measured pressure distribution. For three dimensional models like aircraft configurations, the forces and moments are usually measured directly by the use of balance system.[9] There are two methods using a balance system, that is by internal balance system of external balance system:

i. Internal balance

The internal balance is, almost exclusively, strain gauge balances. The strain gauges are used for measuring, for three force components and three moments, that is normal, axial, and side forces, pitch, roll and yaw moments.

ii. External balance

External balance is used in low speed wind tunnels in connection with floor mounted strut support system or wire suspension from the roof. The advantages of external balances provide better accuracy than internal balances. The disadvantage of the external balances is the comparatively large interference of the associated strut mounting.

2.9 WIND TUNNEL INTERFERENCES

The flow field around the model in the wind tunnel generally differs from that around the full scale configuration even when Mach and Reynolds similarities are ensured. The factors producing these discrepancies arise from various causes as the following.

2.9.1 Test Section Wall Interferences

Test section wall interferences are associated with the fact that at the test section boundaries a constraint is imposed upon the flow. If the test section has rigid plain walls the constraint is that the flow has to run parallel to the walls in its immediate vicinity. The wall constraints distort the flow field around the tested model. The precise nature of the distortion is of complex character. The effect is usually subdivided in a number of independent components whose effects are assumed to be additive. The major affects are termed blockage, buoyancy, downwash, and streamline curvature. [3]

Blockage is related to the displacement of the model and its wake provided to the flow. It results into a lateral expansion and bulging of the streamlines which is hindered by plain solid wall or furthered by a constant pressure boundary condition of a free jet. In the first case the velocity is reduced. The increased velocity in closed test section wind tunnel is equivalent to an increase in dynamic pressure, augmenting all forces and moments at a given angle of attack. [3]

Buoyancy originates from a gradient in static pressure along the tunnel axis. This may already be present in the empty test section due to the thickening of the wind tunnel wall boundary layer. It is also may be due to the cross flow characteristic of the ventilated wall. In fact it is also a problem of protected wall configurations. In case a negative longitudinal pressure gradient exists the model is drawn downstream. Thus the buoyancy mainly affects the drag. [3]

Downwash or lift interference is related to the alteration of the flow deflection produced by the test section boundaries. This type of interference would occur even if the modelled as a lifting line or a vortex double. These interference effects are important for lifting aircraft models with a tail. They affect the lift as well as the pitching moment. [3]

Streamline curvature is related to a deterioration of the shape of the streamline in flow field. Its effect on the model characteristics may be represented by a variation of downwash along the tunnel axis. Such effects are highly undesirable because they allow no correction to measure the data. In fact it corresponds to change in model chamber. [3]

2.10 FLOW VISUALISATION

Flow visualisation use for obtaining a qualitative local or global picture of the flow. It may also be employed for acquiring qualitative measurements without introducing probes. Some techniques are confined to the visualisation of surface flow in a pattern, other provides visual information about the flow in a cross section, but there are also technique yielding a global picture of the flow structure. The main techniques will be explained in the following. [16]

2.1.1 Wool tuft technique

Wool tuft technique is a very simple and adequate mean to obtain information about local flow condition on aerodynamic surfaces. The length of tufts depends on the size of tested model. The tufts are fixed on the aircraft or on the model by cello type. By the flow they are aligned in the local flow. Separating flow and high tuberlent is indicated by more or less pronounced oscillation of the tuft. [16]

2.1.2 Oilflow technique

In this technique the surface model is coated with a mixture of light oil and white titanium dioxide or fluorescent powder. A drop of oleic acid is added as dispersing agent. The airstream causes the oil to flow along the model surface. Streaky deposits of the pigment are showing the direction in which the skin friction acts. [16]

In the case of delta wing the size of the leading edge vortex, the vortex core position and the vortex intensity may be deduced from the oilflow picture. The size of the vortex is given by the distance between leading edge and the attachment line. The separation line indicates the position of the vortex core and the direction of the oil streaks produced by the vortex is indicative of the circulation intensity. Thus the oilflow pictures may be used for quantitative evaluation. [16]

2.1.3 Smoke technique

The smoke technique requires low tuberlent flow in order to avoid destabilisation of the smoke filaments and their subsequent mixing with the airflow. Thus this technique was mostly employed for flow at very low speeds. However if a particular care is taken by use of screens in the settling chamber and a large contraction ratio, extension of this technique even to supersonic speed is possible. The smoke may be produced from vaporisation of mineral oil, from vaporisation of substance containing bromides or chlorides or form burning wood, paper or tobacco. The vaporisation of oil probably the technique often used. The smoke is introduced by a rake of pipes placed into settling chamber. The contraction of the flow in the nozzle also reduces the diameter of the smoke filaments so that comparatively large rake pipe diameters yield still tiny smoke filaments. A laser screen may be used to illuminate the smoke lines to provide a more contrast picture. [16]

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

The activities from the initiation to the finalization of this project involve some these processes:

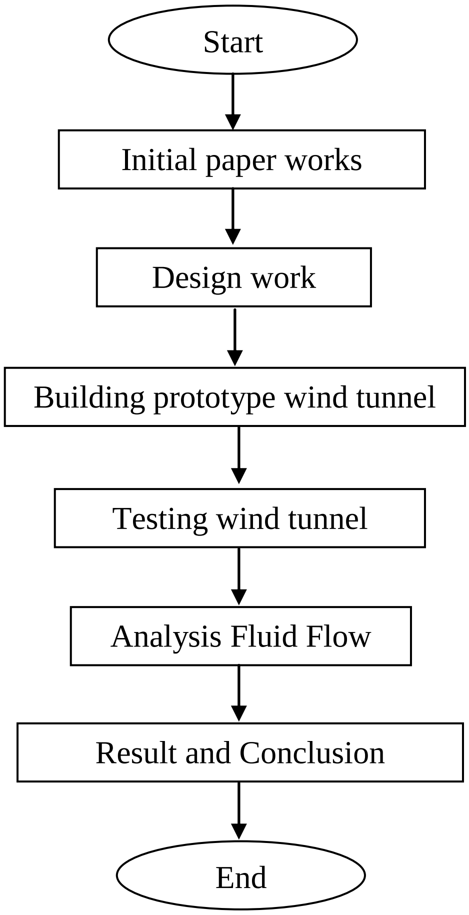


Figure 3.1: Research methodology flow chart

3.2 INITIAL PAPER WORKS

This includes the search for the entire text materials, books about the design of Wind tunnel from library to internet. Regular Supervisor's guidance is most important at this stage as this is the beginning. Then design processes for separate parts started. This includes rough sketches of different parts, small calculations.

3.3 DESIGN WORK

Design work refers to complete design of each component. Full calculation considering each and every aspect of wind tunnel component is carried out. From this we can have a detailed drawing of each chamber. This it-self is the complete design. In this step we choose the materials for each section.

Table 3.1: Specification of wind tunnel

Components	Design specification
Settling chamber	Dimension: 900mm x 900mm x100mm
Contraction section	Ratio :9 Dimension: 900mm x 900mm (large area) : 300mm x 300mm (small area) Length : 500mm
Test section	Length : 300mm x300mm x700mm
Diffuser	Angle :4.3 degree Dimension: 600mm x 600mm Length : 1220 mm

3.4 BUILDING THE PROTOTYPE WIND TUNNEL

After basic research the early development of our wind tunnel seems to break down into five major components. These consist of the settling chamber, compression cone, test section, diffuser, and power source or fan. Most sources indicate that the test section is the most important part of the wind tunnel and should be designed first, based on specific needs and Reynolds numbers, so that the rest of the wind tunnel can be constructed accordingly to meet the specifications determined by the test section.

3.4.1 Contraction cone

One of the larger and more critical aspects of the wind tunnel is the contraction cone. The most difficult part about this section is deciding what shape to make it. There were two main options we had. First, there was the more traditional approach of a curved contraction, a profile with an ‘S’ shape to it. The other option was to make flat walls in the shape of a trapezoid, which would serve the same purpose. The underlying goal of the contraction cone is to transfer from a larger area to the smaller area of the test section in a sense a large square to a smaller square. The contraction cone serves many purposes in the overall scheme of the wind tunnel. The contraction cone increases the efficiency of the system by giving the fan a larger pool to pull air from it is easier to pull air through the tunnel. Also by starting off at a larger area the velocity of the air is much lower and more ideal for the use of screens and honeycomb to straighten the airflow. Problems that can occur in this section include separation of air and an increased boundary layer. All of these factors were considered in making the final design, however, it was ease of construction, and total cost that also played a large factor in the final decision. A curved contraction cone would be much harder to construct, so the flat design was chosen for its ease of construction. First a plywood is cut into four equal trapezium size according to design and it is assemble in cone shape. The figure 3.1 below shows our flat design of contraction cone. Four equal trapezium ply woods are glued together by using wood glue and leave for a day to dry.

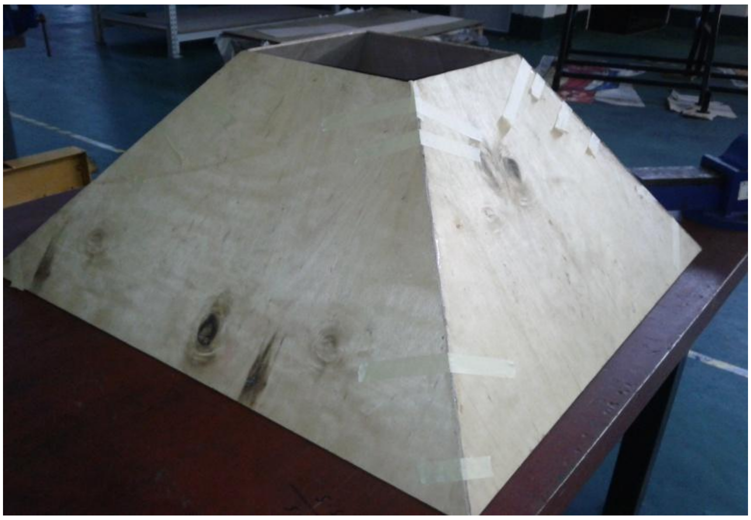


Figure 3.2: Contraction cone

3.4.2 Test section

The next section of the wind tunnel is the test section. A test section area of 300 x 700 mm (Figure 3.3) was chosen because it allowed a wide range of wind velocities, and it was also convenient to the length of wood we had on hand. The only other part of the test section is the Plexiglass or clear material needed to view the experiment. The only purpose of the Plexiglass is to view the object, so it would seem reasonable that only one side is needed to look in. However, other considerations such as the amount of light that will then get in or the background in viewing smoke needs to be looked into. The Plexiglass of a thickness 3mm is cut into four by using band saw with a dimension of 300mm x700mm and are glued together by using chloroform glue. Figure 3.3 shows a complete assembles of test section.

Figure 3.3: Test section

3.4.3 Diffuser

Following the air through the test section the next large component required is the diffuser section. This connects the end of the test section with the fan and goes from a smaller area to a slightly bigger one. Research has noted that the angle of expansion should not be more than 5 degrees, which fits the 4.3 degrees in the current design. Although the shape of this section is best when square is blended with a circle it is very difficult to construct. For something that is more easily built the square attaches to 4 triangles, whose tips connect to the fan shroud. (Figure 3.4)

Figure 3.4: Diffuser

3.4.4 Fan

The final aspect in our wind tunnel design and construction is the fan. We have decided on and purchased an industrial strength fan that has a power of 1.1 kW (Figure 3.5). However, most experiments require a wide range of wind velocities, so the need to control a wide range of speeds is very important. The idea is to use a dimmer switch to interrupt the power leg to the fan motor and thus control its speed. This worked out very well and produced fine adjustments in the speed of the fan. More importantly, though, it produced no signs of damage to the motor or switch. The figure 3.5 shows industrial fan that use in this wind tunnel.

Figure 3.5: Fan unit

3.4.5 Wind tunnel assembly

The wind tunnel is assembling together with springs connected between the test section, contraction and diffuser (Figure 3.6). At the joint of each part, rubber gasket is used to make sure that no flow losses at connection (Figure 3.7). Rubber gaskets are cut with same size as the joint which is 300x300mm. Using the plastic glue gun, the rubber gasket is glued to the connectors. It has to be glued at one part only. After assembling all three part of wind tunnel, the wind tunnel then connected to the fan. The connection to the fan is not screwed. The fan and the wind tunnel are just clamped together.

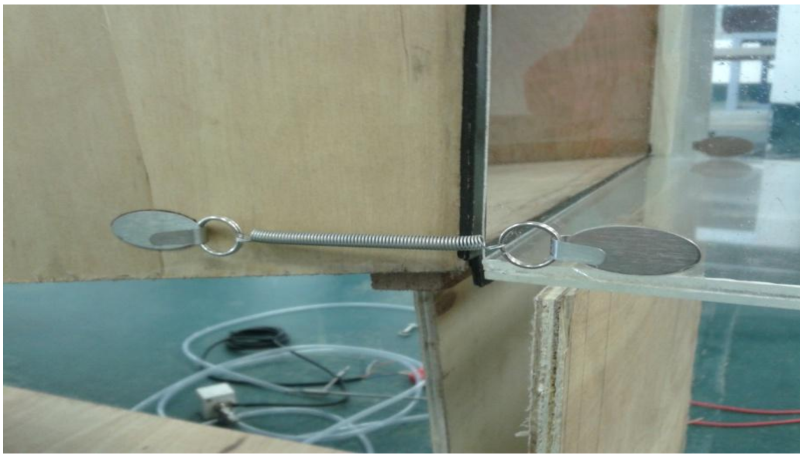


Figure 3.6: Spring connector

Figure 3.7: Installation of rubber gasket

Figure 3.8: Complete assemble of wind tunnel

3.5 TESTING THE WIND TUNNEL

Now the prototype wind tunnel is tested if the flow inside it at various rpm of fan is within the range of design. To do that Pitot probe is attached at different points (Figure 3.8). Velocity profiles are drawn and matched with the designed profiles. If the values come approximately same then the design is good. Also the type of flow and characteristics of flow inside of the tunnel is studied in detail.

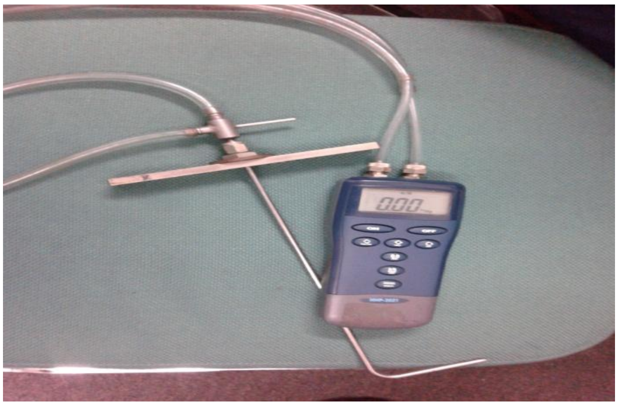


Figure 3.9: Pitot tube

3.6 FLUID FLOW ANALYSIS

3.6.1 Flow impression in test section

Before conduct any experiment in this small wind tunnel, the flow in test section must be check whether it is laminar or tuberlent. In order to that, three methods can be used that is using Pitot tube to test the pressure in the test section, if the pressure is same at any point in test section than the flow in the test section is laminar. The second method is using white smoke to see the flow in test section, straight line smoke shows the flow is laminar. In this analysis, I have use string method to check the flow in the test section. A number of white strings are attached around the test section using tape and the air flow in the test section can be viewed through the string that flows in test section. Wind of different speed range of 4m/s to 60 m/sis tested in this analysis. If all the strings are in straight line than the flows is laminar. If the string does not parallel with the air flows than it is tuberlent.

3.6.2 Flow impression of a sphere

To get an impression of fluid pattern around a sphere, smoke technique is used. The smoke is produced from vaporisation of mineral oil. In this experiment, smoke generator (Figure 3.9) is use to generate white smoke to a ping pong ball.

Once the wind tunnel start, smoke is generated by smoke generator and an impression of fluid pattern around the ping pong ball is capture.

Figure 3.10: Smoke generator

High speed camera is use in this experiment in order to see the flow field at the object.

3.6.3 Drag of sphere

In this experiment, a 40 mm ping pong was use as a test specimen. Before start the experiment, the atmospheric pressure and temperature were recorded at the beginning of data collection.

The drag force of the specimen mounting stand was measured using the force balance and recorded on the data sheet. The ping pong ball test specimen was then mounted. The test section is then check to make sure wall has been closed tightly to avoid the flow leak.

Once the installation is ready, motor of wind tunnel is switch on with low speed. The speed of motor is adjusted by a controller (Figure 3.10). The drag force was measured and recorded on the data sheet. The free stream static pressure from the free stream dynamic pressure and total pressure is measure by a Pitot tube which connected to digital pressure sensor.

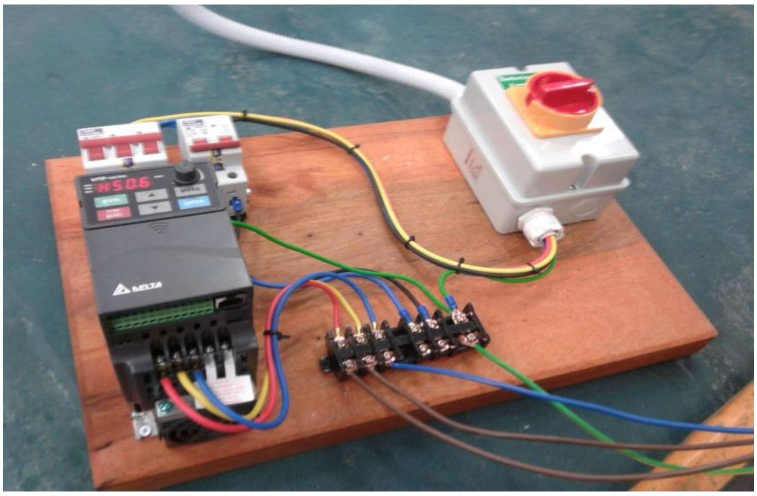


Figure 3.11: AC inverter use to control the current frequency

After taken the pressure and drag force data.The following equations was used to reduce the raw data:

i. Calculation of air density.

$$\rho = \frac{P}{RT} \tag{3.1}$$

where R is the gas constant for air.

ii. Calculation of velocity pressure from manometer reading.

$$\Delta P = \rho_o \Delta h g \tag{3.2}$$

iii. Calculation of free stream air velocity.

$$u_\infty = \sqrt{\frac{2 g_c \Delta P}{\rho}}. \tag{3.3}$$

v. Calculation of Reynolds number.

$$\mathrm{Re} = \frac{u_\infty D \rho}{\mu}. \tag{3.4}$$

vi. Calculation of drag coefficient, $C_{D.}$

$$C_D = \frac{8}{\pi}\left(\frac{F_D}{D^2 \rho u_\infty^2}\right) \tag{3.5}$$

CHAPTER 4

RESULTS AND DISCUSSION

4.1 SPECIFICATION OF WIND TUNNEL

4.1.1 Results

Table 4.1: Wind tunnel specification

Components	Design specification
Settling chamber	Dimension: 900mm x 900mm x100mm
Contraction section	Ratio :9 Dimension: 900mm x 900mm (large area) : 300mm x 300mm (small area) Length : 500mm
Test section	Dimension : 300mm x300mm x700mm
Diffuser	Angle :7 degree Dimension: 600mm x 600mm Length : 1220 mm

The wind tunnel had been designed with a selected dimension which has been determined before the fabrication phase takes over.

The CAD drawing has been done using Solid Work software. The drawing of 2 dimensional (2D) and 3 dimensional (3D) of the wind tunnel is done resulting the drawing of desired wind tunnel proportional with the geometrical dimension and limitation that has been considered. The limitation is such as size of each wind tunnel component,cost of production and performance fan suitable to this low speed wind tunnel. The design of wind tunnel from Solid Work is shown in figure 4.1.

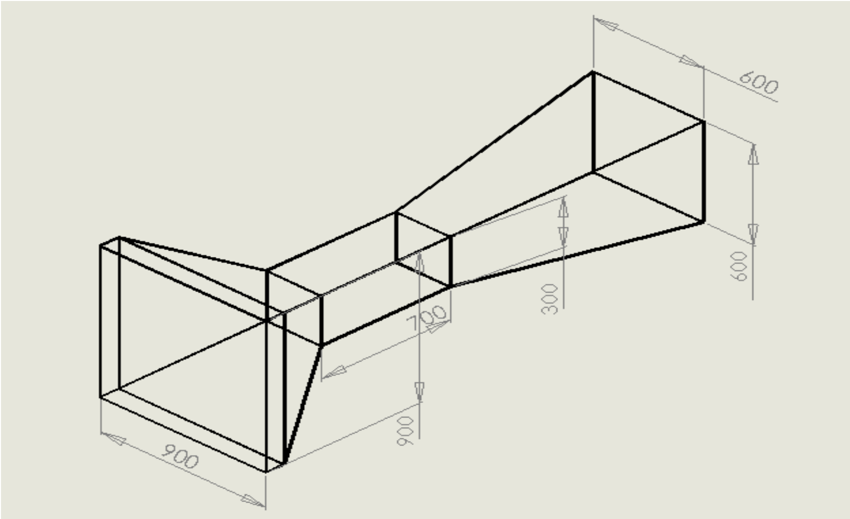


Figure 4.1: Design wind tunnel from solid work (Dimension in mm)

The studies have been made to determine the suitable dimension of this open wind tunnel. All the factors was considered than come out with this dimension:

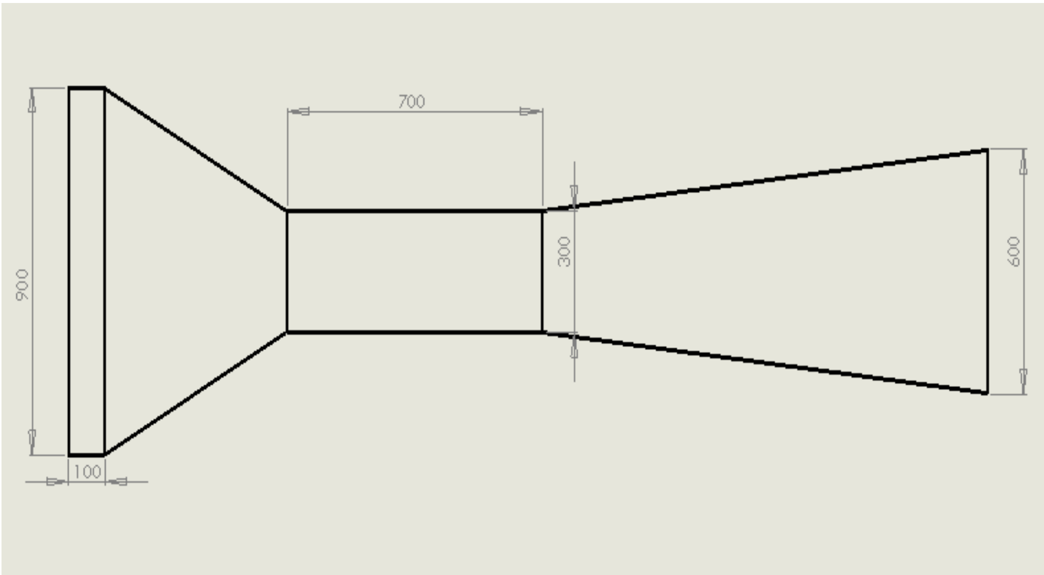


Figure 4.2: Side view of wind tunnel (Dimension in mm)

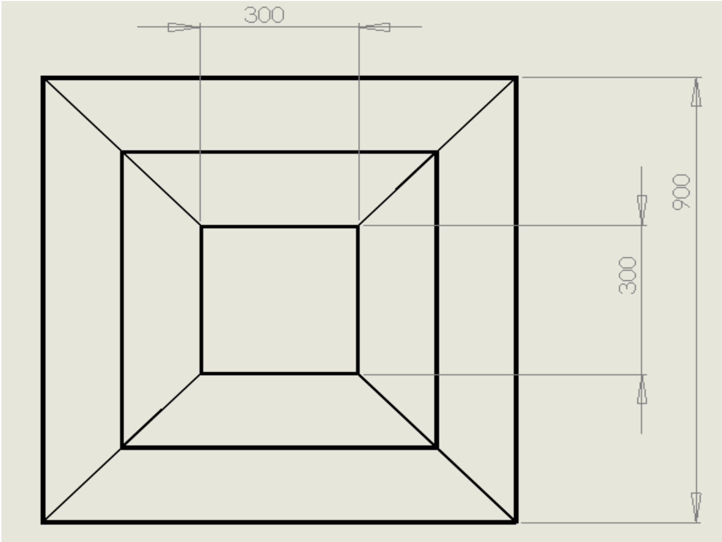


Figure 4.3: Front view of wind tunnel (Dimension in mm)

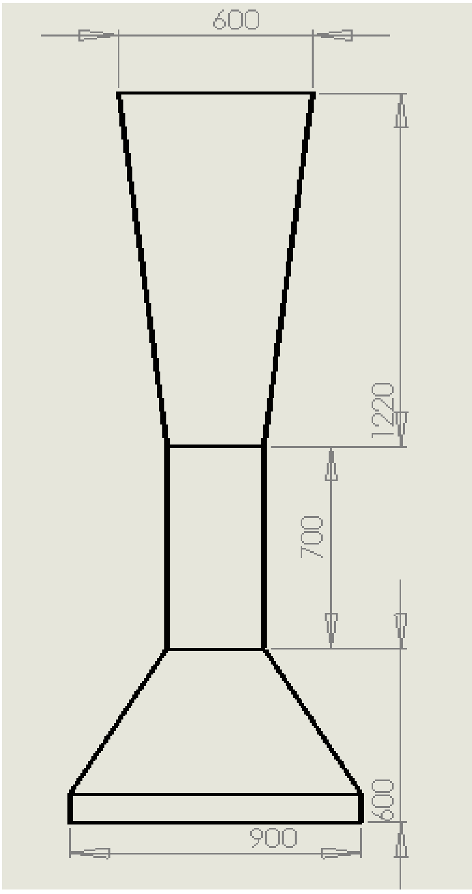


Figure 4.4: Top view of wind tunnel (Dimension in mm)

4.2 FLUID FLOW IN THE TEST SECTION

The fluid flow in the test section is tested using many white string. The flows is test in speed range of 4 m/s to 60 m/s and the movement of the string is observed. Based on the picture capture, it can be observed that the flows in the testion section is tuberlent.

Table 4.2: Flow observation in test section with different air speed

Digital manometer pressure reading, Δ*P* (Pa)	**Free stream air velocity, u_∞ (m/s)**	**Flow of air in test section**
1	4.11	Tuberlent
8	11.63	Tuberlent
16	16.44	Tuberlent
31	22.89	Tuberlent
47	28.18	Tuberlent
75	35.6	Tuberlent
99	40.9	Tuberlent
130	46.87	Tuberlent
158	51.67	Tuberlent
200	58.14	Tuberlent

Figure 4.5: Strings show tuberlent flow in test section at air speed of 23 m/s

Figure 4.6: Strings show tuberlent flow in test section at air speed of 58 m/s

4.21 Discussion

From the results, it can be observed that the flow in test section in the range air speed of 4 m/s to 60 m/s are tuberlent. This results is due to the irregular flows due to swirl, low frequency pulsation and turbulence air that goes in into the settling chamber. In order to make the flows straight, the settling chamber must be long enough and honeycomb can be used in the entrance of settling chamber.

4.3 FLUID FLOW AROUND AN OBJECT

In this research, smoke is used to visualize streamlines and flow pattern about the model. White smoke produced by smoke generator delivered through a wand. This method would be ideal as it would demonstrate the best laminar flow over sphere. Smoke genarator is used to generate smoke at settling chamber. The smoke is used to generate flow profile around the test object. The fluid flow field around the sphere is capture by using camera. The figure 4.6 shows the smoke travel around the object.

Figure 4.7: Flow of air around sphere without smoke

Figure 4.8: Flow of air around sphere with smoke impression

4.3.1 Discussion

From the flow pattern, we can say that the method of using white smoke would be ideal as it would demonstrate the best laminar flow over an sphere as indicated on the website of the Low Speed Wind Tunnel Testing site in San Diego.

4.4 DRAG COEFFICIENT CALCULATION

In this experiment, a sphere drag coefficient is calculated. During the experiment the temperature before and after is taken. Table below shows the initial condition of the experiment.

Table 4.3: Initial condition of experiment

Diameter of sphere (m)	**Temperature (K)**	**Density of air (Kg/m^3)**	**Viscosity of air (Pa.s)**
0.04	300	1.161	18.6×10^{-6}

4.4.1 Results

Table 4.4: Drag coefficient of ping pong ball

Free stream air velocity, u_∞ (m/s)	**Drag force (N)**	**Drag Coefficient, C_D**	**Reynolds number, Re**
2	0.0007	0.24	4994
4	0.0025	0.21	9987
6	0.0057	0.22	14980
8	0.0099	0.21	19974
10	0.0152	0.21	24968
12	0.0217	0.21	29961
14	0.0297	0.21	34955
16	0.0382	0.20	39948
18	0.0495	0.21	44942
20	0.0608	0.21	49935

4.4.2 Graph

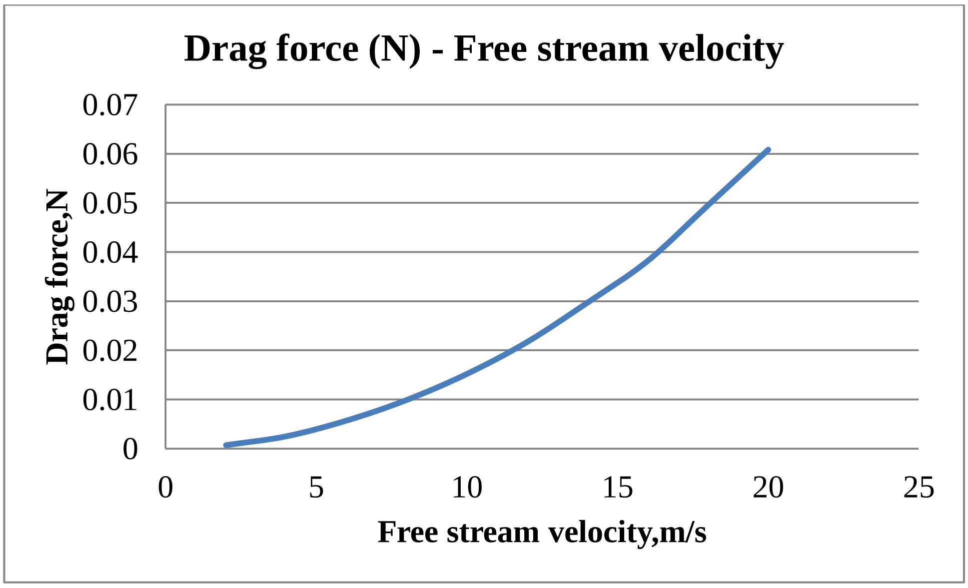


Figure 4.9: Drag versus air velocity graph

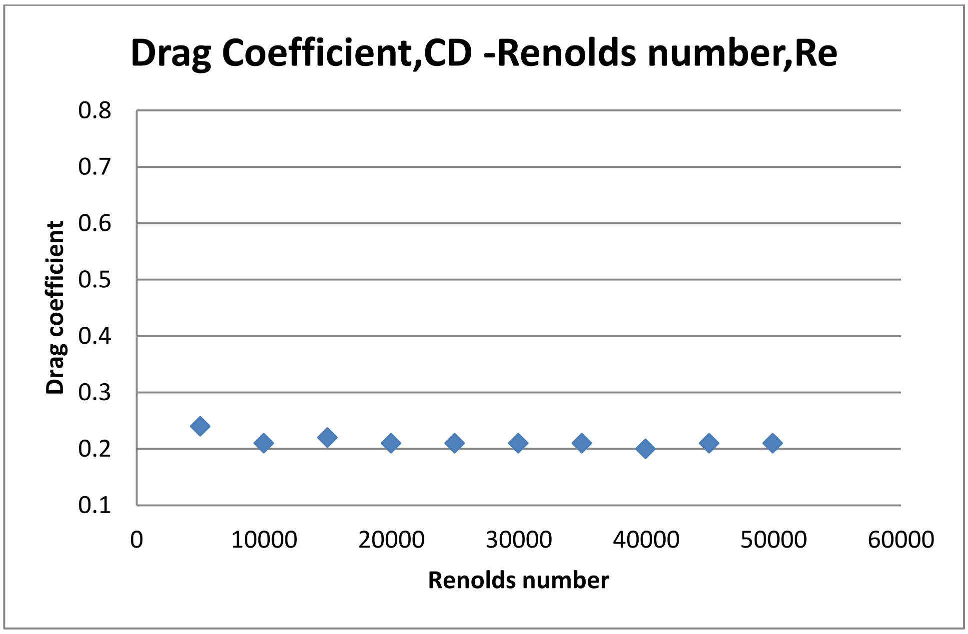


Figure 4.10: Drag coefficient versus Renolds number

4.4.3 Discussion

Form the graph of drag versus velocity, the drag force of ping pong ball generally increase if the speed of air velocity increases. The results are calculated using simulation in Solid Work data show that the drag coefficient for ping pong ball of size 40 mm is 0.21.

From the graph of drag coefficient versus Reynolds number, over the speed range tested, the drag coefficient generally decreases as the Reynolds number increases. Similar results are reported in figure 4.8 (Reference). Owing to the limitations of the low speed wind tunnel used in this experiment, comparison of results is possible over only a single decade ($10^2 \leq Re \leq 10^3$).

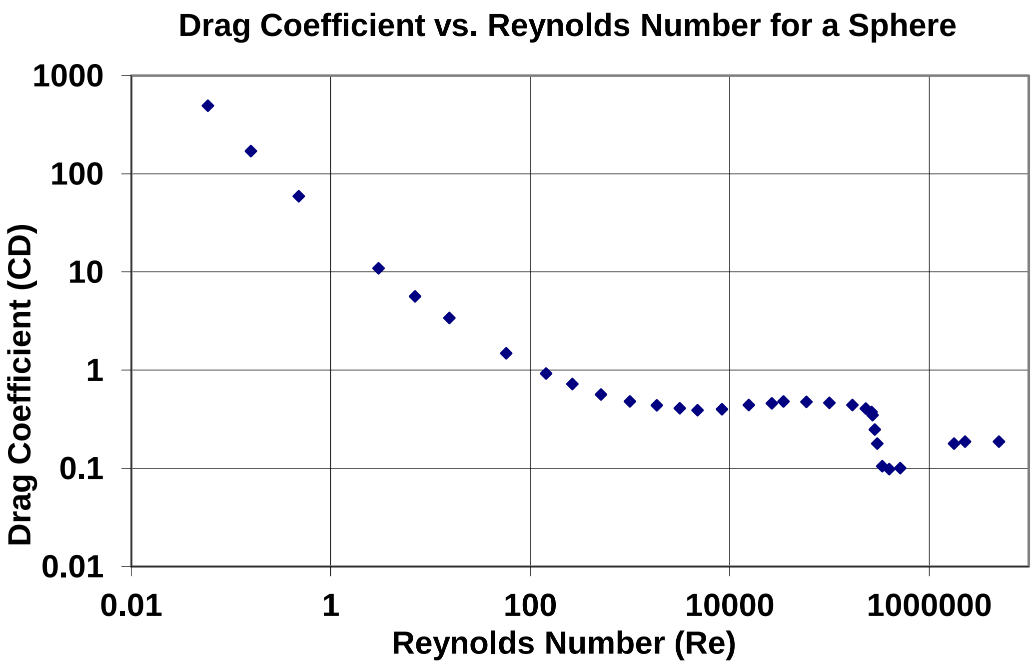


Figure 4.11 : Experimental values of drag coefficient vs Reynolds number for a sphere

CHAPTER 5

CONCLUSION

5.1 CONCLUSION

This final year project consists of three objectives which the first is design and fabricating a wind tunnel in 300mm x 300mm x 700mm test section. The conclusion of the wind tunnel design is that the design produces an ideal geometrical condition that may not be achieve in practise. It is because some factors such as time to fabricate, human factors, not enough tool and some other limitation.

Second objective is to get an impression of fluid flow around the object. It can be conclude that the method of using white smoke would be ideal as it would demonstrate the best laminar flow over an sphere as indicated in reference. Besides, many strings can be attached to wall of test section in order to analyze the flow of air in the test section.

The third objective is to calcualte coefficient of drag of a sphere in low speed wind tunnel. The results from Solid Work simulation show that the drag coefficient for a ping pong ball of size 40 mm is 0.21. Besides, the results obtained agree with other published results at the higher range of velocities used.

5.2 RECOMMENDATION

To make the wind tunnel more perfect, few recommendations have been made for future improvement. The inner surface had to be smoothend more by using epoxy mixture. To reduce air losses from the connection of component, thinner rubber gasket has to be used.

The smoke flow in this wind tunnel cannot be seen clearly due some limitation such as improper lighting and the smoke does not travel around the test object. Recomendation in future, a proper wand must for smoke generator must be located near to the test object. On the other hand, the capture of the flow pattern around the sphere must be use high speed camera in order to see the flow pattern lines clearly. Instead using wire mesh, it is the best to use straw as a honeycomb on settling chamber to make the straight air flows in test section. The settling chamber must long enough so that the swril and tuberlence air from outside can be straighten when entering the settling chamber.

The drag coefficeint of a ping pong ball cannot be determine experimentally due to the reason of lack balance system in the wind tunnel. This is due to limitation of time and budget to buy material. Futher research on building balance system to determine the drag force of an object must be carry out and some budget must be invest for this research.

REFERENCES

1 B. Bienkiewicz et al., “Active modeling of large-scale turbulence,” J. Wind Eng. Ind. Aerodyn. 13,1983, 465-476.

2 Deborah A. Kaminski and Michael K.Jensen, “ Introduction to Thermal and Fluid Engineering,” John Wiley & Sons,Inc, 2005,pp. 450-471.

3 ad-www.larc.nasa.gov/tsab/tetruss/images/joined.html, December 1, 1998.

4 Wolkovitch, Julian, Application of the Joined Wing to Cruise Missiles: Final Report for Phase I, Contract N00014-79-C-0953, Rancho Palos Verdes, CA: Aeronautical Consultant Associates, November 1980 (AD-A096450).

5 Wolkovitch, Julian, “The Joined Wing: An Overview”, Journal of Aircraft, Vol. 23, No. 3, 1986, pages 161-178.

6 “Aeronautical Systems Tests Progressive Joined-Wing Aircraft”, www.lmco.com/files2/lmtoday/9707/aero.html, posted August 7, 1997.

7 Fleischmann, Shirley. *Fluid Mechanics EGR 365 Designed-Based Fluids Lab.* Grand Rapids: Grand Valley State University, 2005.

8 Frits H. Post and Theo van Walsum, “Fluid flow visualization*”*, Delft University of Technology, The Netherlands, 1993.

9 Ganzer, U, ”Experiment techniques in Aerodynamic,” Lecturer Series for presentation at Bandung Institute of technology,1894.

10 H. E. Donley, “The Drag Force on a Sphere,” (1991). Available at http://www.ma.iup.edu/projects/CalcDEMma/drag/drag.html (21 March 2002).

11 H.W. Coleman and W.G. Steele, “ Experimentation and Uncertainty Analysis for Engineers,” John Wiley & Sons, Inc., New York, 1989.

12 Munson, Young, and Okiishi. *Fundamentals of Fluid Mechanics, Fifth Edition.* Hoboken: John Wiley & Sons, Inc

13 Potter, M.C., & Wiggert, D.C. (2002). *Mechanics of Fluids, Third Edition.* Wadsworth,Group Brooks/Cole, Pacific Grove, CA.

14 Pope, A, Harper,J.J, ”Low Speed Wind Tunnel Testing,” John Wiley and Sons,Inc,1996, page 183- 189.

15 Reneau, L. R., Johnson, J. P. & Kline, S. J. 1976 Performance and design of straight, two-dimensional di_users. Trans. ASME, J. Basic Engineering 89, 141-156.

16 Rouse, H. (1946). *Elementary Mechanics of Fluids.* John Wiley & Sons, Inc., New York, NY

17 Scheiman, J. & Brooks, J. D. 1981 Comparison of experimental and theoretical turbulence reduction from screens, honeycomb and honeycomb-screen combinations.JAS 18, 638-643.

18 Saniei, N. (2007). *Personal communication (lecture notes).* Los Angeles, CA: Loyola Marymount University

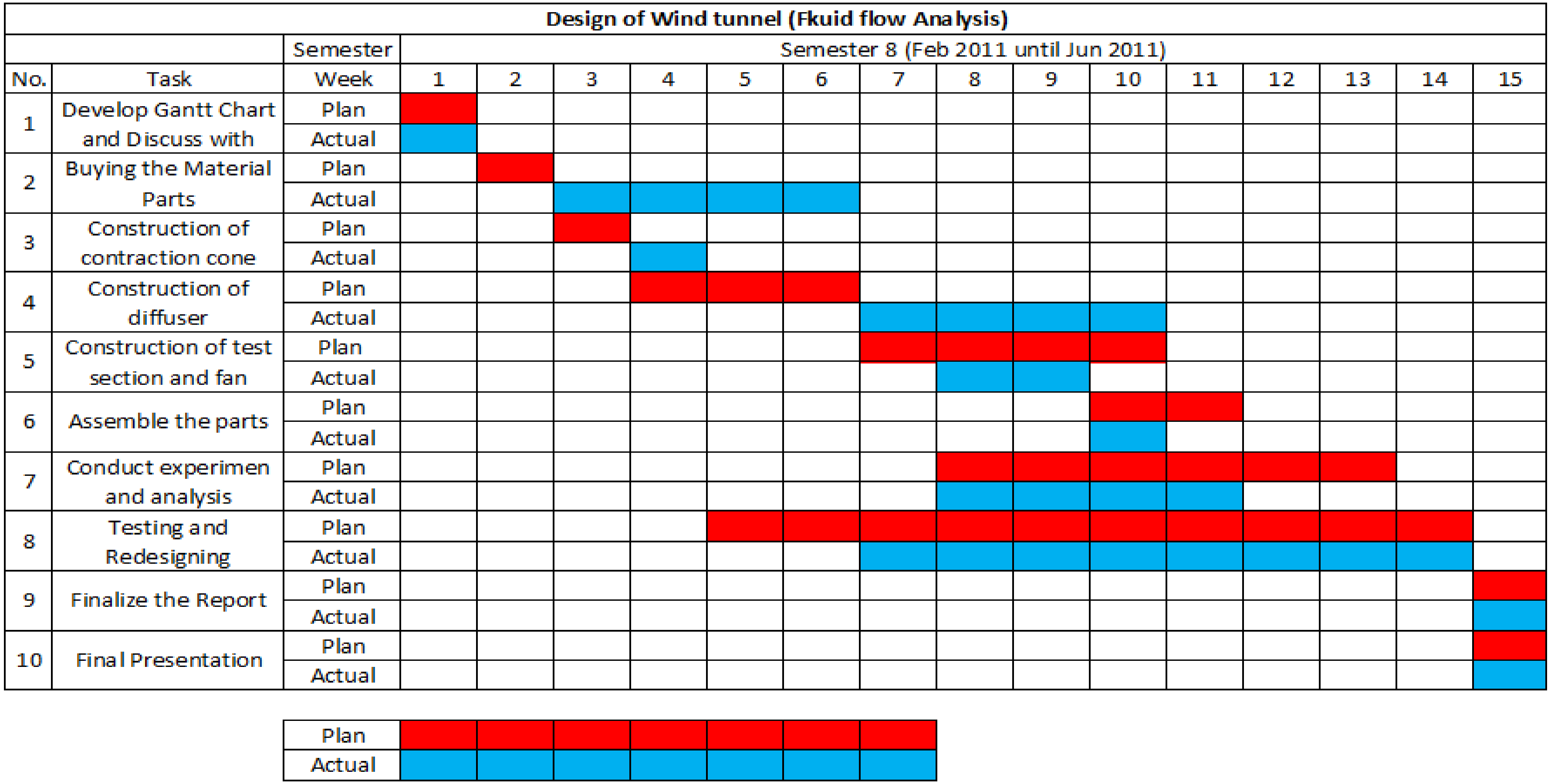
19 R. L. Daugherty and J. B. Franzini, *Fluid Mechanics with Engineering Applications*. McGraw-Hill, New York, 1965, pp. 309-405.

20 Saniei, N., & Es-Said, O. (2007). *Laboratory Manual, MECH 341 Mechanical Engineering Lab 1.* Department of Mechanical Engineering. Los Angles, CA: Loyola Marymount University.

21 The boundary layer .NASA Glenn Research Center.http://www.grc.nasa.gov/WWW/k-12/airplane/dragco.html

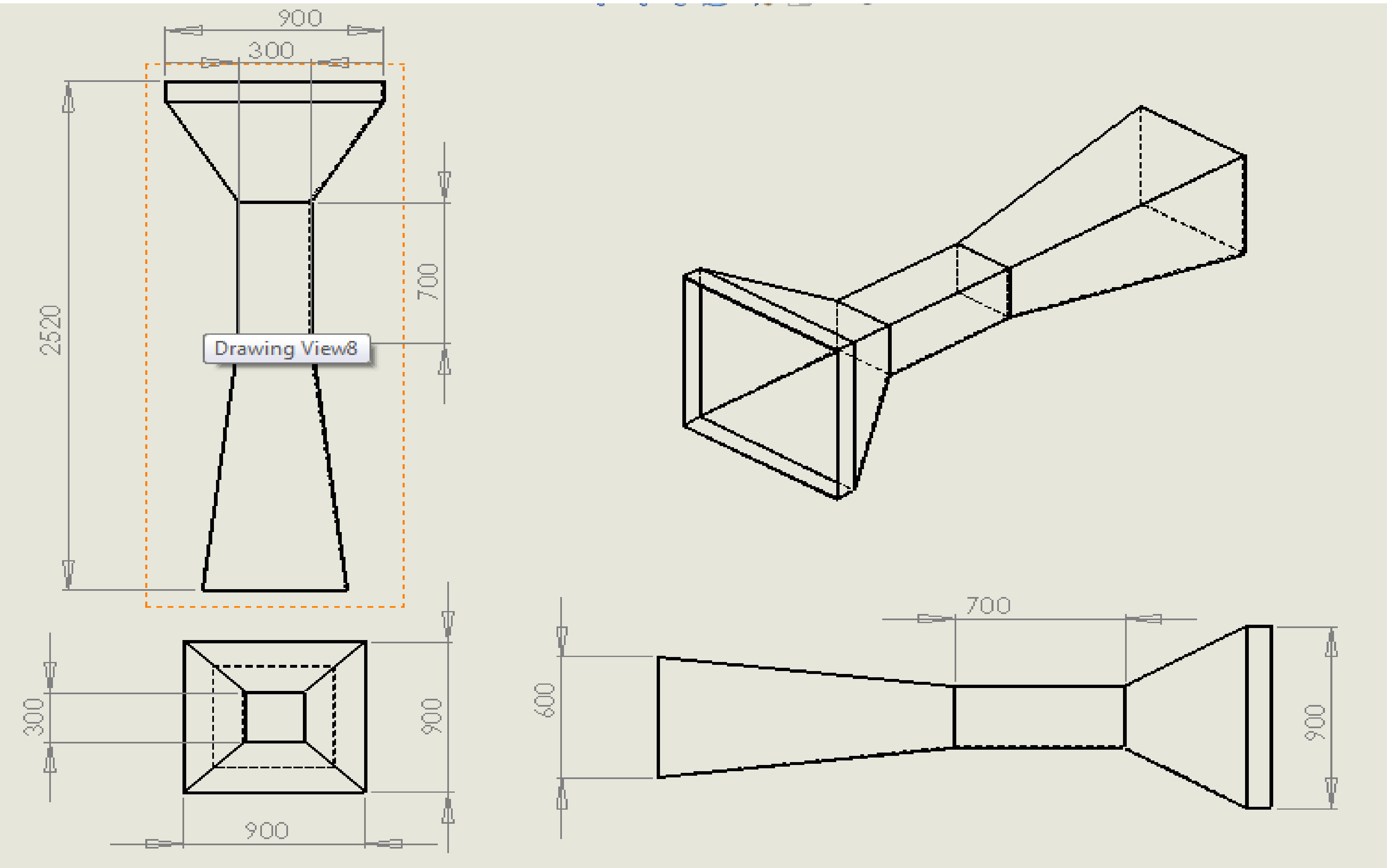
APPENDIX A

Gantt chart



APPENDIX B

Isometric drawing of Wind tunnel



APPENDIX C1

Fabrication of Small Wind Tunnel

Figure 1: Measuring and cutting ply wood for building contraction cone.

Figure 2: Assembling components of wind tunnel.

APPENDIX C2

Analysis of fluid flow in test section

Figure 3: Analysis of fluid flow in test section using string

Figure 4: Measuring pressure in test section using Pitot tube and digital manometer