

High Speed Subsonic Wind Tunnel Design

A project present to
The Faculty of the Department of Aerospace Engineering
San Jose State University

in partial fulfillment of the requirements for the degree
Master of Science in Aerospace Engineering

By

Jordan Michael Towels- Moore

May 2014

approved by



Dr. Nikos Mourtos
Faculty Advisor



San José State
UNIVERSITY

HIGH SPEED SUBSONIC WIND TUNNEL DESIGN

A Project Thesis
Presented to
The Aerospace Engineering Academic Faculty

By:

Jordan Michael Towles-Moore

In Partial Fulfillment
of the Requirements for the Degree
Masters of Science in the
School of Aerospace Engineering

San Jose State University
May December, 20143

|

HIGH-SPEED SUBSONIC WIND TUNNEL DESIGN

Approved by:

Dr. Nikos Mourtos, Advisor
School of Aerospace Engineering
San Jose State University

~~Dr. Nik Djordjevic
School of Aerospace Engineering
San Jose State University~~

~~Dr. Periklis Papadopoulos
School of Aerospace Engineering
San Jose State University~~

Date Approved: 02/04/2013

TABLE OF CONTENTS

List of Figures.....	5
Abstract.....	6
Chapter 1 - Introduction.....	6
Background.....	6
Problem Definition.....	7
Requirements.....	9
Advantages/Disadvantages.....	9
Breakdown.....	9
Initial Design.....	11
Chapter 2 - Aerodynamic Analysis.....	12
Continuous Flow.....	13
Pressure Variation in the tunnel.....	13
Head Losses.....	18
Turning Vanes.....	22
Screens.....	26
Boundary Layer Analysis.....	26
Chapter 3- Test Section Flow Quality.....	28
Blockage/Boundary Correction.....	28
Chapter 4 – Fan Design.....	31
Fan Straightener.....	32
Flow Power.....	34
Fan Power.....	35
Chapter 5- Design Break-Down.....	36
Balance.....	39
Cost Analysis.....	39
Future Steps.....	40
Appendix A.....	41
Matlab Coding.....	41
Headlosses.....	42
Boundary Layer.....	43
Function Call.....	45

Blockage/ Corrections.....	45
Works Cited.....	50
List of Figures.....	4
Abstract.....	5
Chapter 1.....	5
Introduction.....	5
Background.....	5
Problem Definition.....	6
Solution.....	6
Requirements.....	8
Advantages/Disadvantages.....	8
Breakdown.....	9
Initial Design.....	10
Chapter 2.....	11
Aerodynamic Analysis.....	11
Continuous Flow.....	11
Pressure.....	12
Head Losses.....	17
Flow Power.....	21
Turning Vanes.....	22
Screens.....	26
Fan Power.....	26
Balance.....	27
Chapter 3.....	27
Design Break-down.....	27
Cost Analysis.....	30
Future Steps.....	31
Appendix A.....	32
Matlab Coding.....	32
Headlosses.....	33
Works Cited.....	35

List of Figure

Figure 1- Diagram of a circulating wind tunnel.....	5
Figure 2- Diagram of an open circuit wind tunnel.....	6
Figure 3- Wind tunnel component breakdown.....	10
Figure 4-Initial design for wind tunnel.....	11
Figure 5- Close-up view of a section of the wind tunnel.....	11
Figure 6- Diagram of area changes of wind tunnel.....	12
Figure 7- Pressure change vs. Velocity distribution.....	13
Figure 8- Pressure change distribution throughout main stages of the wind tunnel.....	14
Figure 9- Energy losses.....	16
Figure 10- Pressure losses due to friction.....	18
Figure 11- Preliminary design of diffuser section.....	21
Figure 12- CFD image of corner with no vanes.....	23
Figure 13- CFD of a corner with vanes.....	24
Figure 14- CFD of corner with optimized conditions.....	25
Figure 15 - Power graph for power requirement.....	27
Figure 16- George Palmer's original closed circuit wind tunnel design (Purdue University).....	28
Figure 17- Example test section door for a 4x6 wind tunnel. (Purdue University).....	29
Figure 18- Full scale dimensioned design of wind tunnel.....	30
YFigure 1- Diagram of a circulating wind tunnel (Wikipedia).....	6
Figure 2- Diagram of an open circuit wind tunnel (Science Buddies).....	7
Figure 3- Wind tunnel component breakdown.....	11
Figure 4-Initial design for wind tunnel.....	12
Figure 5- Close-up view of a section of the wind tunnel.....	12
Figure 6- Diagram of area changes of wind tunnel.....	13
Figure 7- Pressure change vs. Velocity distribution.....	14
Figure 8- Pressure change distribution throughout main stages of the wind tunnel.....	15
Figure 9- Energy losses.....	17
Figure 10- Energy Ratio vs. Volumetric Flowrate.....	18
Figure 11- Pressure losses due to friction.....	19
Figure 12- Diffuser head losses vs length.....	20
Figure 13- Diffuser head losses vs length (2).....	21
Figure 14- Diffuser head losses vs length (3).....	21
Figure 15- Preliminary design of diffuser section.....	22
Figure 16- CFD image of corner with no vanes.....	23
Figure 17- CFD of a corner with vanes.....	24
Figure 18- CFD of corner with optimized conditions.....	25
Figure 19- Velocity profile across the wall of the test section.....	27
Figure 20- Velocity profiles for different stations along test section.....	28
Figure 21- Reference for variable Lambda (Rae, 1984).....	29
Figure 22- NACA 0012 Airfoil (Sankar, 1997).....	30
Figure 23 - Area ruling for nacelle section of the wind tunnel.....	32
Figure 24- Flow power graph for power requirement.....	35
Figure 26- Example test section door for a 4x6 wind tunnel. (Purdue University).....	38

<u>Table 1- Initial table for project breakdown.....</u>	<u>8</u>
<u>Table 2-Cost analysis for wind tunnel.....</u>	<u>40</u>

Table 1- Initial table for project breakdown.....	8
Table 2-Cost analysis for wind tunnel.....	31

Abstract

The focus of this project is the design of a subsonic wind tunnel with a test section X4 ft. tall by 6 ft. wideY by 10 ft. longZ and a maximum speed of XY255 m/s. The tunnel will be able to test _____large aerodynamic objects at Reynolds numbers ranging from from A266,821 to B6,803,935.

~~This thesis mainly focuses on the idea of taking a concept and following a process to turn it into an actual product. This system incorporates the processes involved in systems engineering, design engineering, and of course, aerospace engineering. Throughout the course of this thesis, a structured break down of a basic wind tunnel will be analyzed to create a design for the actual construction requirements of the tunnel. In doing so, a conceptual design will be made, followed by a breakdown of the aerodynamics, system head loss analysis and integration. Within the analysis of the tunnel, the main focus of research is the head loss. Within each section, the head losses will be considered when making the design criteria for the tunnel. Further plans involve dimensioning the tunnel based on the acquired information in the analysis followed by a cost analysis for the full construction of such a tunnel based on the analysis of its head losses.~~

Chapter 1 - Introduction

Introduction

Background

A wind tunnel is an apparatus designed to create a controlled situation of fast moving air in order to analyze the flow around different objects. At the same time, it is also used to determine measurements of lift and drag, as well as the forces acting on objects as a result of yaw, pitch, and roll. The first kind of wind tunnel was credited to have been built by Frank H. Wenham in 1871. It was the first invention that channeled smooth air into a test section to analyze lift and drag forces on an object. Wind tunnels come in many different sizes and can be applied to many different situations. They can be used to analyze a golf ball moving 50 mph through the air, or they can be used to get g-loads on a scaled model of a Boeing 787 making turns at 500 mph.

The two main types of wind tunnels that are used today consist of either an open circuit, wind-tunnel which takes ambient air, speeds it up and releases it, or a closed circuit wind tunnel, which keeps the air circulating as it picks up speed.

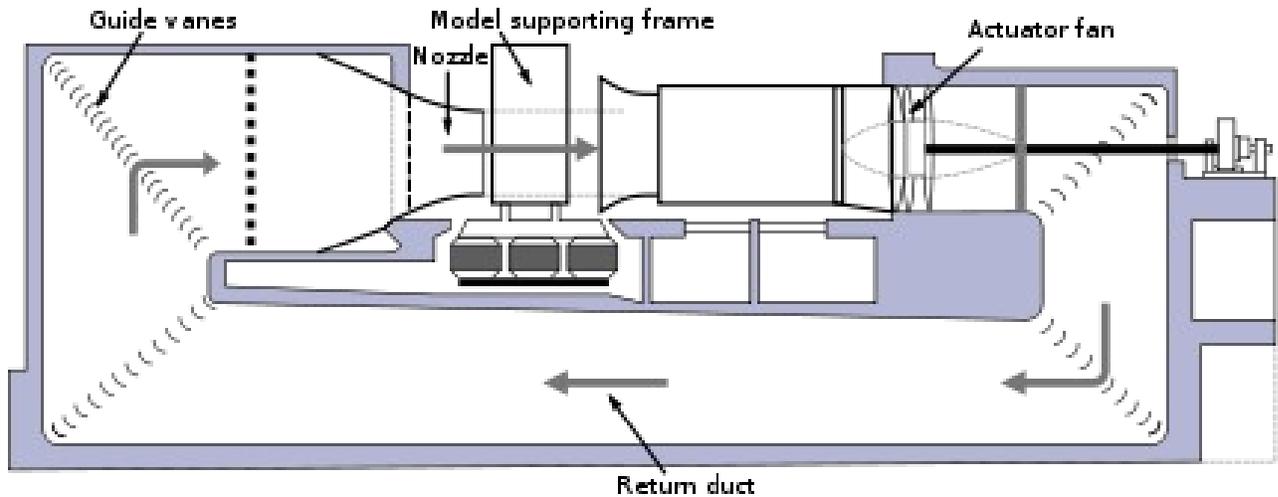


Figure 1- Diagram of a circulating wind tunnel (Wikipedia) [NEED REFERENCE](#)

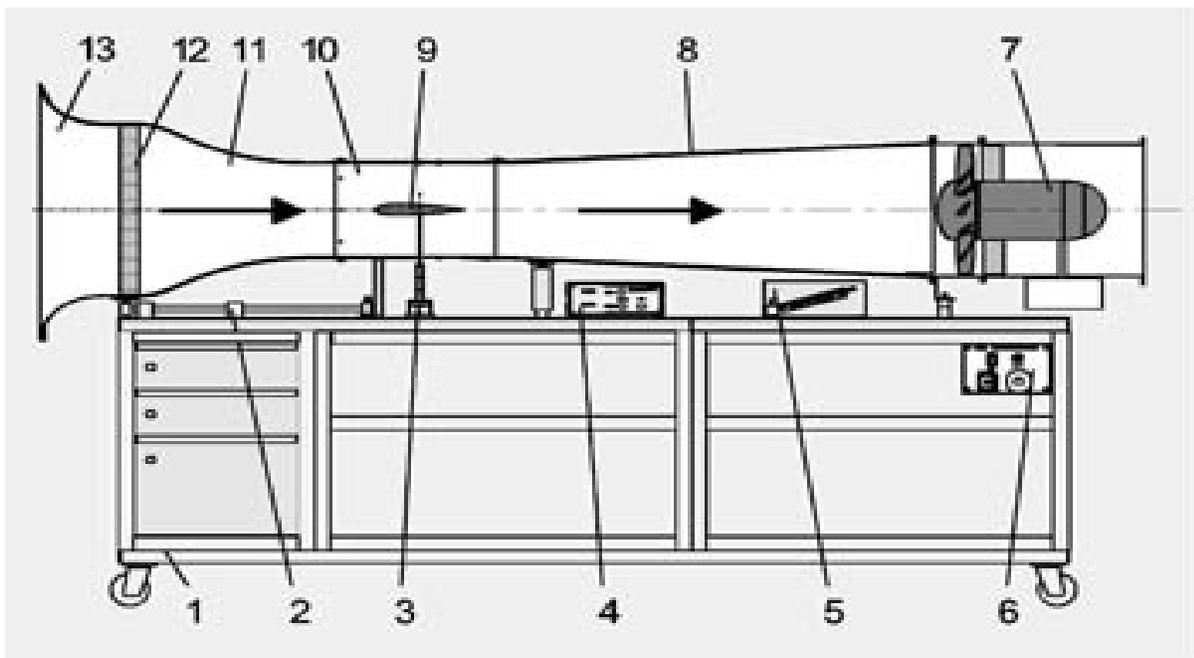


Figure 2- Diagram of an open circuit wind tunnel (Science Buddies) [NEED REFERENCE](#)

In context with this proposal, a wind tunnel can be useful in determining the forces that are applied to objects, as long as the proper equipment is installed in the test section of the tunnel. For a modern wind tunnel today, a balance is placed in the test section of the tunnel that is very sensitive to forces acting on it.

Problem Definition

San Jose State University has a small wind tunnel that is fully functional, but is used for specific tests for students within the university related to research and analysis of small objects. The wind tunnel is also an open circuit wind tunnel inside of a lab room, and therefore cannot run at high subsonic speeds. There are wind tunnels at the nearby location of NASA Ames research center, but the cost to use the wind tunnel for small tests is [very high not cost affective.](#)

Moreover, it is difficult to schedule test time in government wind tunnels and frankly a hassle to organize. Hence, it is proposed to design and build This university has a need for a fully functional a high subsonic speed wind tunnel, that is capable of testing doing full analysis on large objects and can be used by both aerospace and mechanical engineering students for research, experimentation, and analysis. There are also several aerospace companies in the surrounding area that ~~c~~would find the use of such a wind tunnel to do analysis quite useful and cost effective. A properly decent sized, functional wind tunnel stationed at San Jose State University would be an asset to the AE Program school and to aerospace companies in the area companies around, as it can bring the university both profit and recognition.

The goal of this Solution

As a thesis project, my plan is to initially provide the university with the design of such a wind tunnel, to use for its students and researchers as well as to rent out to companies for profit if need be. The specifications of this proposed wind tunnel are as follows: ~~ject consist of the following for development:~~

- 4'x6' test area cross section
- Vented closed circuit tunnel
- Floored balance (400 lb. capable)
- Motor supply for max test area airspeed of approx. 450 mph

Development

To develop this wind tunnel of this magnitude, I will use my background of aerospace structures, dynamics, design, and systems integration to break down the construction of this wind tunnel into steps that will be taken to organize how it will be designed. The following is a list of goals to be researched and steps in the design developed before the whole tunnel as a system can be integrated. ~~of the wind tunnel~~

1. Initial design of test section
 - a. Structural design
 - b. Materials selection
 - c. Balance integration
2. Initial design of full section (WHAT DO YOU MEAN FULL SECTION?)
 - a. Structural design
 - b. Aerodynamic design
3. Full Aerodynamic analysis of fluid of within the tunnel
 - a. Ensuring Continuous laminar flow through the test section (Carlone, 2008) ~~section~~
 - b. Turning the flow around corners
 - c. Diffusion of the high speed flow
4. Motor analysis for integration
 - a. Power requirements
 - b. Power regulation analysis for motor
 - c.
 - d. Energy requirements
5. Power regulation analysis for motor
6. Balance

- a. Design for interchangeability
- b. Design for test analysis
- c. Integration into test section
- 7. Full systems integration
- 8. Cost analysis
- 9. Report on full initial design
- 10. Contract
 - a. Funding

Table 1- Initial table for project breakdown

Project Breakdown				
WEEK 1-4	WEEK 5-7			WEEK 8
Init. Design/Lit Search	Aerodynamic Analysis			Structural Breakdown
Project Breakdown				
WEEK 10	WEEK 11	WEEK 12-13	WEEK 14-15	WEEK 16
Full sys Int.	Cost Analysis	Report	Contract Attempt	
Project Breakdown				
WEEK 17-20	WEEK 21-24	WEEK 25-28	WEEK 29-32	
Power Analysis	Comer Analysis	Design	Cost/Report	

Requirements

In obtaining information about wind tunnels, one would need to look into the requirements and needs of the customer. For this project, the customer consists of the students, faculty, and staff of San Jose State University. The needs of the customer would consist of a large test section that can be modified for any different kind of test for larger scaled test objects. The university has a smaller tunnel that works very well for small tests that the wind tunnel was specifically designed for. The tunnel that will be designed will be for custom tests whereas it can be modified to accommodate a large range of projects.

As a result of the needs of the customer, the requirements will consist of a range of information for different aspects of the wind tunnel. When dealing with the test section of the wind tunnel, the air moving through it will need to be laminar. The floor of the test section will need to be able to hold up a substantial amount of weight, for the use of custom projects that can be large enough to fit within the test section. Another requirement for the wind tunnel would be the ability to gather specific information as to the lift, drag, and yaw forces that are put on an object. With these basic requirements, a preliminary design for the test section of the wind tunnel can begin. For this design, different aspects of a wind tunnel will be considered.

Advantages/Disadvantages

For the first consideration, the type of wind tunnel that will be made is taken into account. The two choices are either the open circuit or the closed circuit wind tunnel. There are both advantages and disadvantages to having these wind tunnels.

For a closed circuit, energy can be recovered in some ways. A result of this can conclude in a smaller motor required for operation. Since a closed circuit is closed, it also provides protection from exhausted air expelled from the fan, as well as protection from harmful weather conditions. A disadvantage of this same type of tunnel is the fact that because it is a full circuit, it is much more expensive to build. Another disadvantage is that because it is closed, it circulates the air. As the air moves through the wind tunnel, it tends to get hotter from all the friction that it encounters. This can provide a problem in calculating lift, because as the temperature of the air increases, the amount of lift that an object can get will change.

When considering an open circuit wind tunnel, an advantage would be that they are a lot less expensive to build. A disadvantage is the fact that for the case of this wind tunnel being for indoor use, it will become a problem when being run at high speeds (Vertical Wind Tunnel Corporation, n.d.).

When taking into consideration the space that is available to place a wind tunnel, a closed circuit would be a good choice for an indoor environment. If the tunnel were placed outdoors, weather conditions would become a major complication to the design of the tunnel. Heating can be a hindrance to experimentation in circulating tunnels, but it can also be avoided through refrigeration devices. Overall, the circulating wind tunnel would be the better choice to use.

Breakdown

System Breakdown

To design a basic closed circuit wind tunnel, there are a few main sections that the full system must have. For this design, the main subsystems of the wind tunnel will consist of a drive section, a settling section, a contraction cone section, a test section and a diffuser section. (Carlone, 2008)

Drive Section

The drive section consists of the components that pulls the air through the test section or pushes it through the contraction cone. For high speed tunnels, specifically supersonic flow, pressurized air or different gases are used to provide the flow of air through the tunnel. For the case of this project, it will consist of a fan that will either be powered by an electric motor or a pneumatic pump (NASA Observatorium, 1995).

Settling Section

The settling chamber of a wind tunnel basically rids the airflow of turbulent eddy's that can show up from turning the flow around corners, or intercepting objects on the inside of the tunnels surface. Generally, it consists of a wire mesh followed by a length of honeycomb structure that straightens the flow out right before it enters the contraction cone (NASA Observatorium, 1995).

Contraction Cone

The contraction section of a wind tunnel simply reduces the space that the air has to move through from a large volume to a specified smaller volume. Converging the total area of the

tunnel in this manner will obviously speed the flow of air up to allow for a faster flow of laminar air into the test section (NASA Observatorium, 1995).

Test Section

The test section is where all the analysis is taken. At this point in a wind tunnel, there is an even flow of laminar air flowing through the tunnel. Because of the contraction section, the flow at this point is also the fastest moving air in the whole tunnel. In this section, calculations of lift, drag, and yaw can be made. Beyond that, with the proper equipment, visuals of flow can be made at this point with proper gasses (NASA Observatorium, 1995).

Diffuser Section

After the air leaves the test section, it needs to be diffused to reduce airspeed for recirculation and to reduce operating power and cost. To prevent big pressure jumps, the flow area needs to come back to its original area slowly. In order to do this, the diffuser section is generally a lot longer than the contraction cone (NASA Observatorium, 1995).

Component Breakdown

Just as the system of the wind tunnel is broken down into subsystems, these subsystems are broken down into components. This is a step further to analyzing the full system as a whole. The graph below shows a system component breakdown of the wind tunnel.

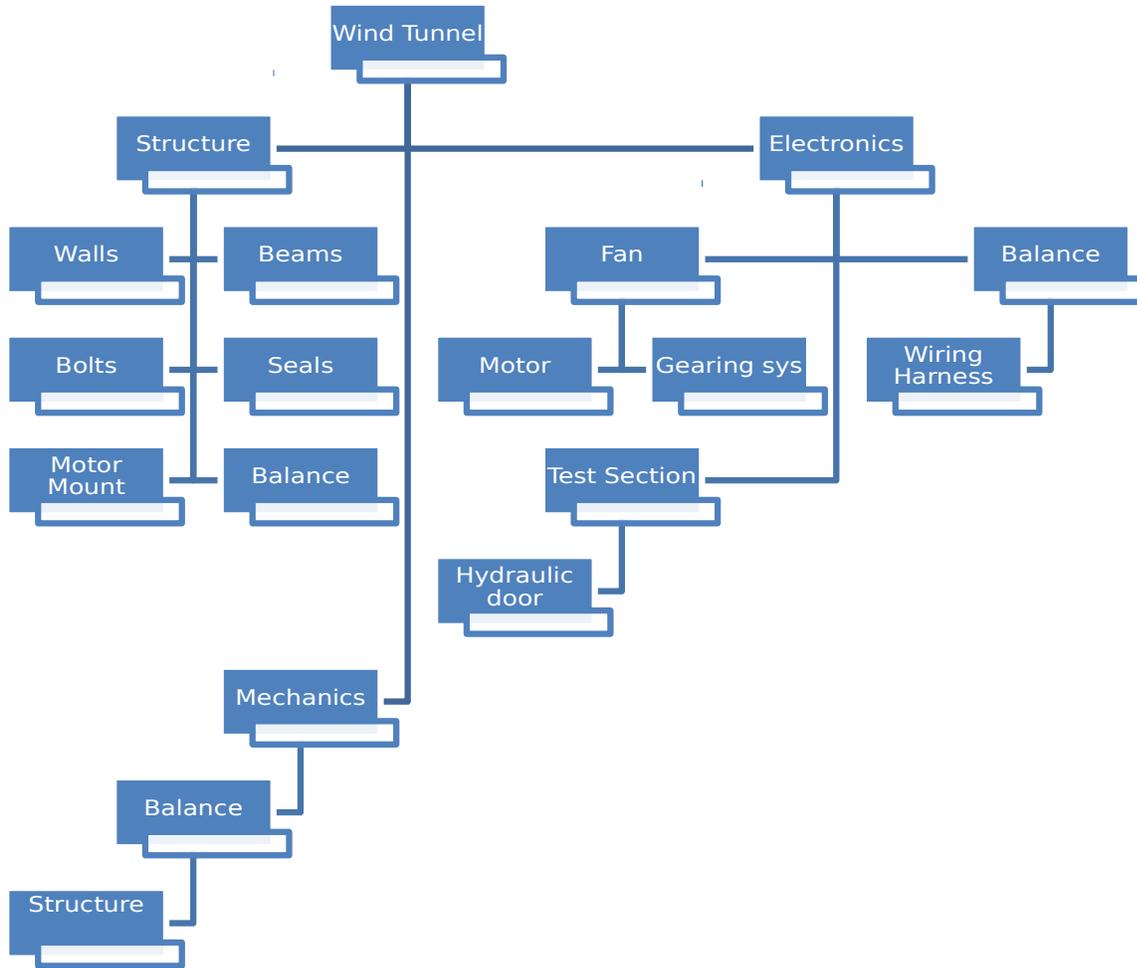


Figure 3- Wind tunnel component breakdown

Initial Design

The first step to designing the wind tunnel consists of an initial preliminary design that can be modified to fit the specifications of the initial requirements. The figures below show a general outlay of how the wind tunnel will look.

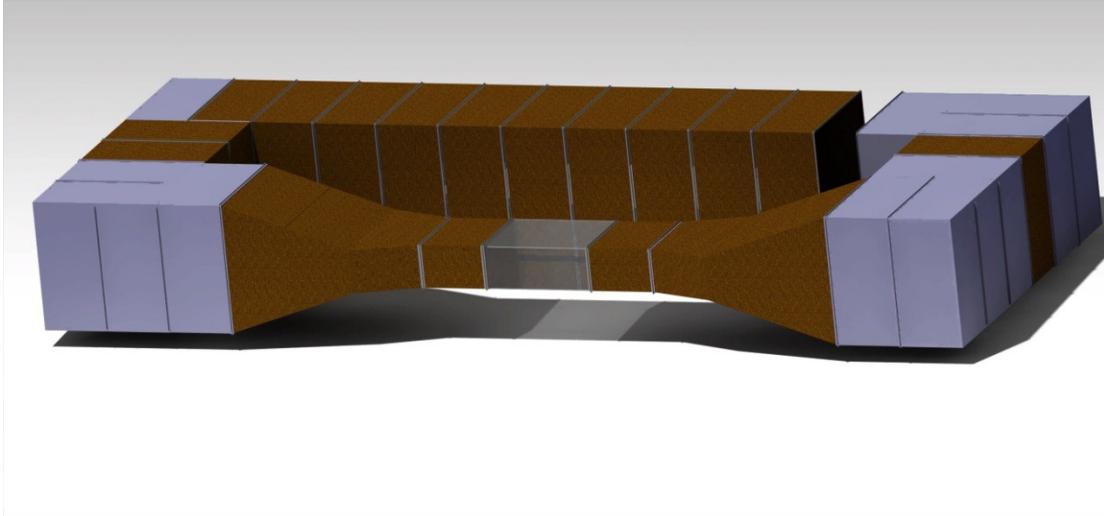


Figure 4-Initial design for wind tunnel

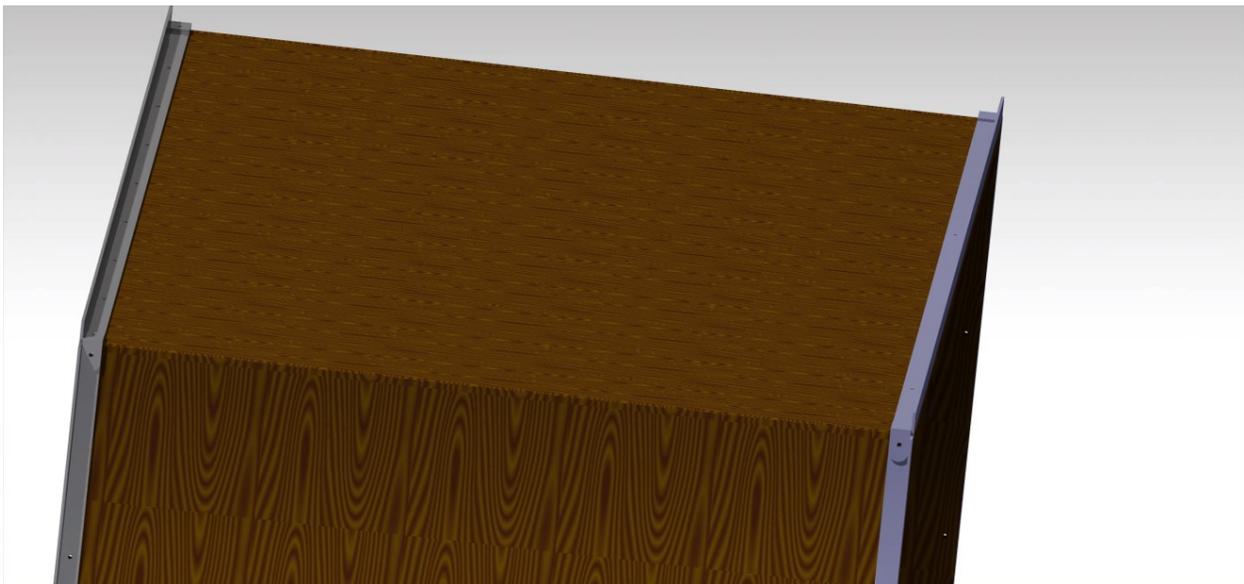


Figure 5- Close-up view of a section of the wind tunnel

Chapter 2 ~~TITLE?~~ - Aerodynamic Analysis

Aerodynamic Analysis

For this section of the project, I will utilize aerodynamic analysis to find a basic design that will allow the wind tunnel to be efficient, reliable, and feasible within a realm that is accomplishable. If you are going to call this subsection Aerodynamic Analysis then you need to perform an aerodynamic analysis in this section. For an aerodynamic analysis, I will start by looking into the velocity and pressure distributions in the tunnel. I will then look at the head losses throughout the tunnel, focusing on the location where the most losses occur. Next, I will cover a boundary layer analysis on the test section of the tunnel. From that, I will look into the

amount of blockage that will occur in that section due to the boundary layer and object being tested (mainly a simple airfoil), and finally boundary corrections.

Continuous Flow

The idea for the design of this wind tunnel is to have a continuous ~~(is there a possibility for a discontinuous flow regime?)~~ flow regime throughout the wind tunnel. The reason for this in this project is to allow the tunnel to be run indoors without causing a physical disturbance to the surrounding area. For this portion of the project, the closed circuit wind tunnel will be considered for the analysis of the pressure, drag, and steady flow analysis of the wind tunnel.

Pressure Variation in the tunnel

When looking at the pressure throughout the tunnel, the main focus is changes between the reservoir section and the test section, and the test section and the diffusion section. The following analysis consists of the pressure changes in the test section as the velocity of the air in the tunnel increases to the max speed. This analysis was done to ensure that the amount of pressure that will be built up at maximum velocity is feasible.

The analysis started with Bernoulli's equation for ideal flow.

$$1. \quad p_1 + \frac{1}{2} \rho V_1^2 = p_2 + \frac{1}{2} \rho V_2^2$$

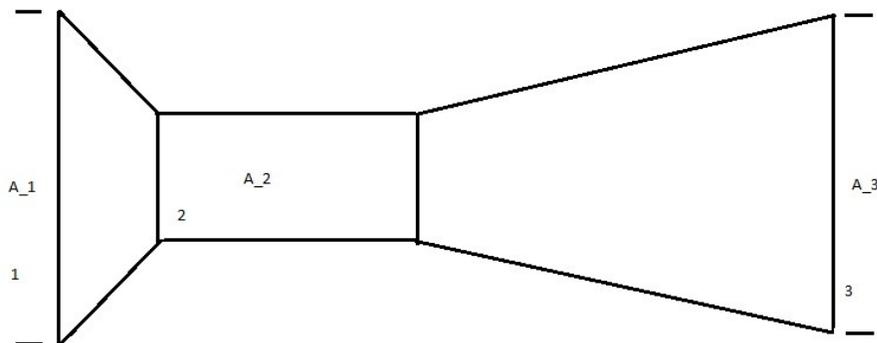


Figure 6- Diagram of area changes of wind tunnel

Assumptions were made for the analysis. The velocity in the test section of the wind tunnel was assumed to be Mach = .75, which is approximately 255 m/s. The initial pressure the reservoir section of the tunnel was assumed to be 1 atmosphere and the density was that of normal sea-level conditions.

For the first analysis, the pressure was taken for a range of velocities up to the max speed that the airspeed is expected to achieve. With this, the pressure at position 2 in the image above was solved for.

$$2. \quad p_2 = p_1 + \frac{1}{2} \rho (V_1^2 - V_2^2)$$

Where $V_1 = \frac{A_2 V_2}{A_1}$, and $A_1 = 144 \text{ m}^2$ where the ratio of A_1 to A_2 is 6.

The results of this analysis can be seen in the graph below.

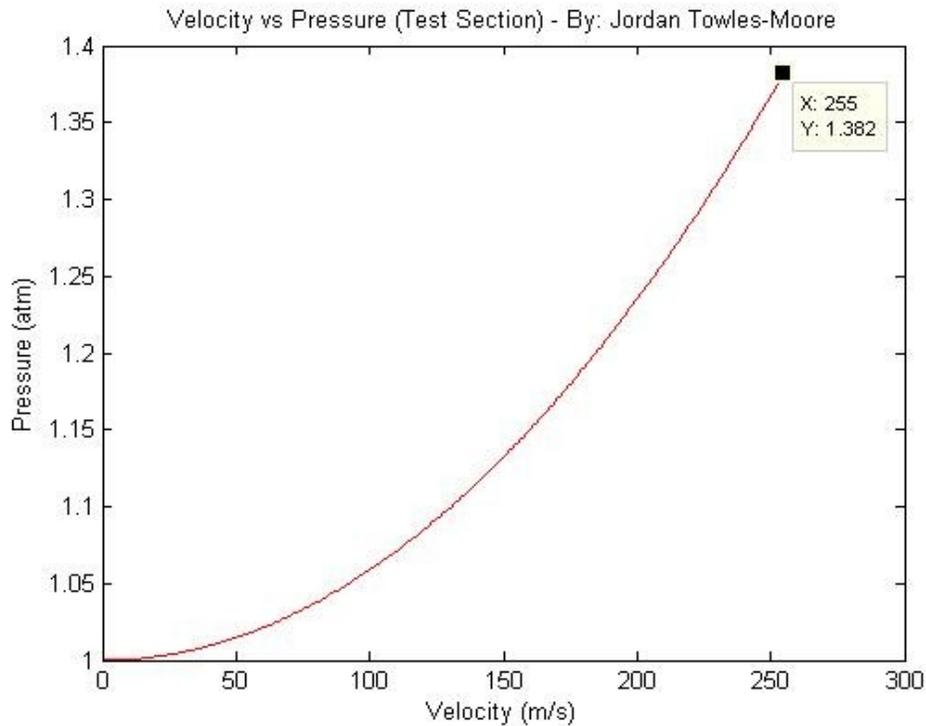


Figure 7- Pressure change vs. Velocity distribution

Next, a distribution of the pressure difference versus the distance through the tunnel from the reservoir area to the diffuser area was made. A result of the calculations for the pressure of the max speed with respect to the area changing throughout the wind tunnel can be seen in the figure below.

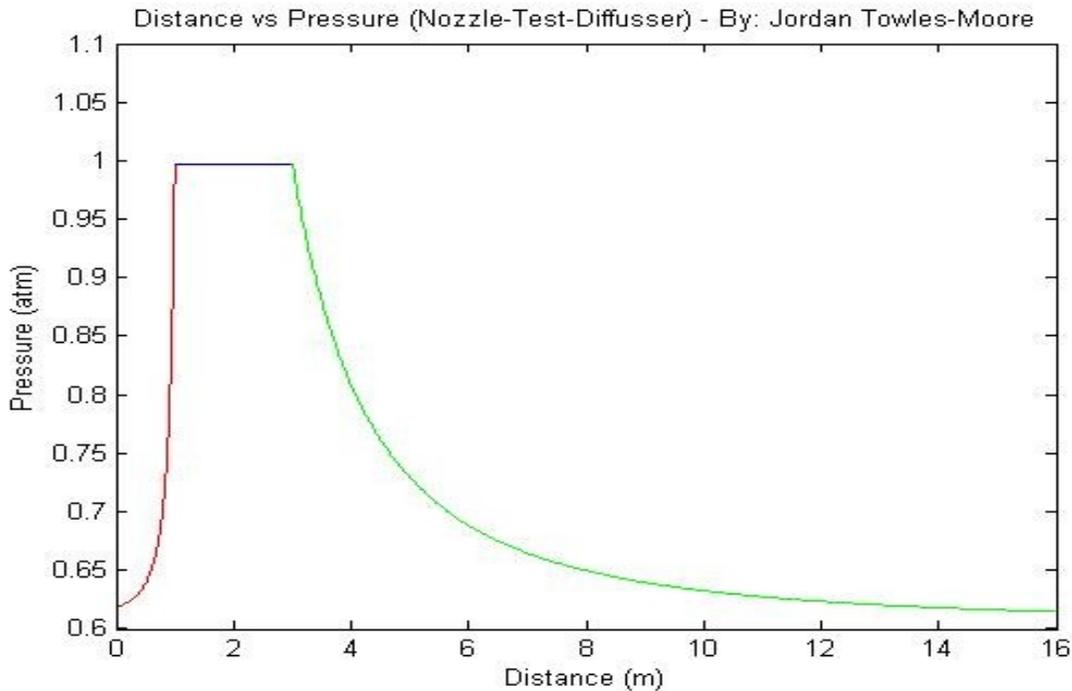


Figure 8- Pressure change distribution throughout main stages of the wind tunnel

Based on the numerical analysis from the graph above, the diffuser section needs to decrease slowly to prevent unsteady, turbulent flow back through the tunnel.

When applying Bernoulli's theory to get the pressure, one is assuming an ideal flow. For this situation, there are a lot of losses for each individual section. The losses that will be considered are from the diffuser section, the compression section, the turning section, the screens, and the fan section. In order to analyze the head losses in the different sections of the tunnel, loss coefficients were found for each of the sections. The sum of the losses were calculated and placed in an energy equation to get a trend of the energy required for operation due to losses within the flow and around the walls. The following equations were used to obtain the loss coefficients. (Barlow, 1965)

$$3. \quad K_{test} = f \left(\frac{L_{test}}{D_h} \right)$$

This equation explains the losses in the test section. The losses tend to come from the sidewalls and the friction that is caused by them.

$$4. \quad K_{friction} = \frac{\left(1 - \left(\frac{1}{A_r^2} \right) \right) * f}{8 \sin(\theta)}$$

$$5. \quad K_{\text{expansion}} = .09623 - .004152 * \theta$$

The previous two equations are added together to give the losses in the diffuser section. The k friction obviously applies the all the friction forces that occur along the walls of the diffuser section. Keep in mind that the corner and screen coefficient are not accounted for in the coefficient calculation for the diffuser below.

$$6. \quad K_{\text{diffuser}} = K_{\text{friction}} + K_{\text{expansion}}$$

The three equations below are for the corner, screen and honeycomb section respectively. These losses will make up the remainder of the losses in the diffuser section.

$$7. \quad K_{\text{corner}} = .10 + \left(\frac{4.55}{\log(\Re)^{2.58}} \right)$$

$$8. \quad K_{\text{screen}} = .785 \left(\left(\frac{\Re}{241} \right) + 1 \right)^{-4} + 1.01$$

$$9. \quad K_{\text{honeycomb}} = .22 * (.3048^2)$$

The last section is the nozzle loss coefficient. This formula, as well as all the other formulas, was gathered from previous data collected and recorded over time.

$$10. \quad K_{\text{nozzle}} = .32 * f \left(\frac{L_n}{D_t} \right)$$

Where f = friction factor

L_{test} = Length of the test section

D_h, D_t = hydraulic diameter

A_r = Area Ratio

θ = diffuser angle

\Re = Reynold's Number

L_n = Length of the nozzle

The equations above will be generalized for the fastest flow and the power required for the flow will be calculated from there.

The losses will be given in the ratio of pressure losses to dynamic losses at the entrance to each section. For a typical local section, the following equation was used, where the delta H is the change in pressure required by the fan.

$$11. K_l = \frac{\Delta H_l}{q_l}$$

I will define a specific pressure required by the fan initially. But in order to do that, I need to define the time rate of energy loss, Delta E. After that, I will solve the rate of energy loss for each different section around the wind tunnel. Using the coefficients listed above.

So, by defining a specific test section area and velocity, I can obtain the power required for the tunnel.

$$12. \frac{1}{2} \rho_t A_t V_t^3$$

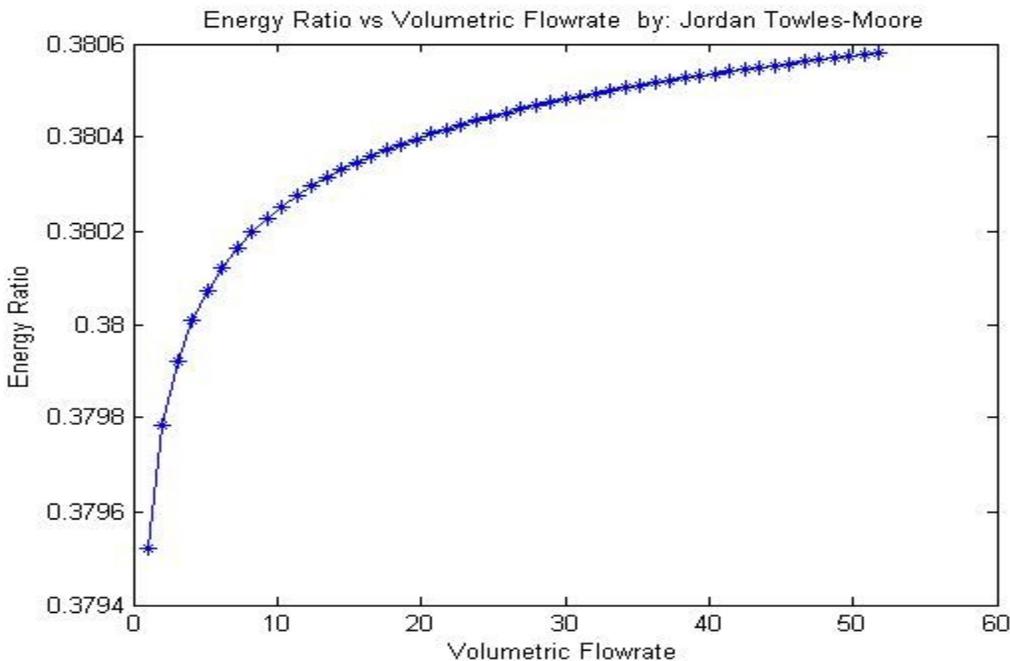


Figure 9- Energy losses

In the figure above, the energy ratio is plotted against the volumetric flow rate. This shows the amount of power required to maintain the flow when the head losses are accounted for.

For added security, I wanted to check this information with respect to the total losses on the system. Using the above equations, I obtained the energy ratio using the following equation:

$$13. E_R = \frac{1}{\sum K_i}$$

Where it includes the sum of all the loss coefficients. Below, a plot of the results of the two ratios can be seen.

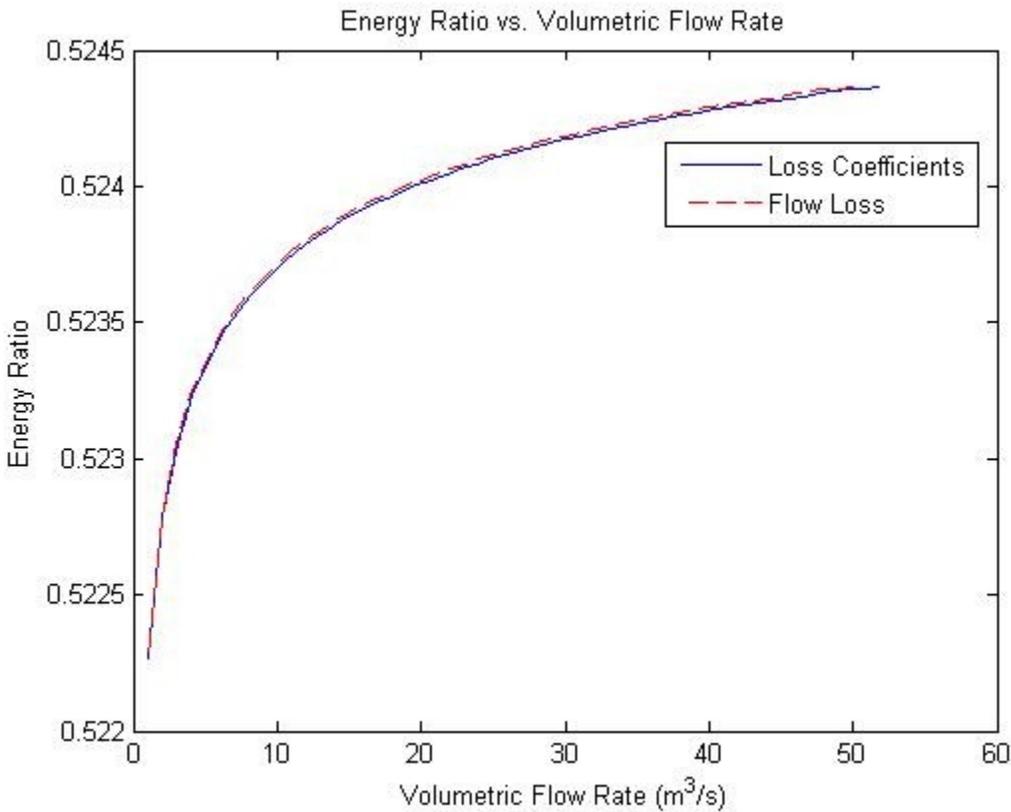


Figure 10- Energy Ratio vs. Volumetric Flowrate

The plot shows the energy ratio plotted against the volumetric flow rate. Of the two plots, one includes the energy ratio with no losses within the calculations. The other are the sum of the losses. Because they closely resembled each other, I could proceed with using the losses to dimension the tunnel.

Head Losses

In using the loss coefficient equations as mentioned earlier, plots of the pressure drops within different sections of the tunnel with respect to the volumetric flow rate were made. The changing variable within this analysis was the airspeed in the tunnel. Note that losses due to the motor were not considered in this analysis because the losses from the motor depend on the motor selection, which has not been determined yet.

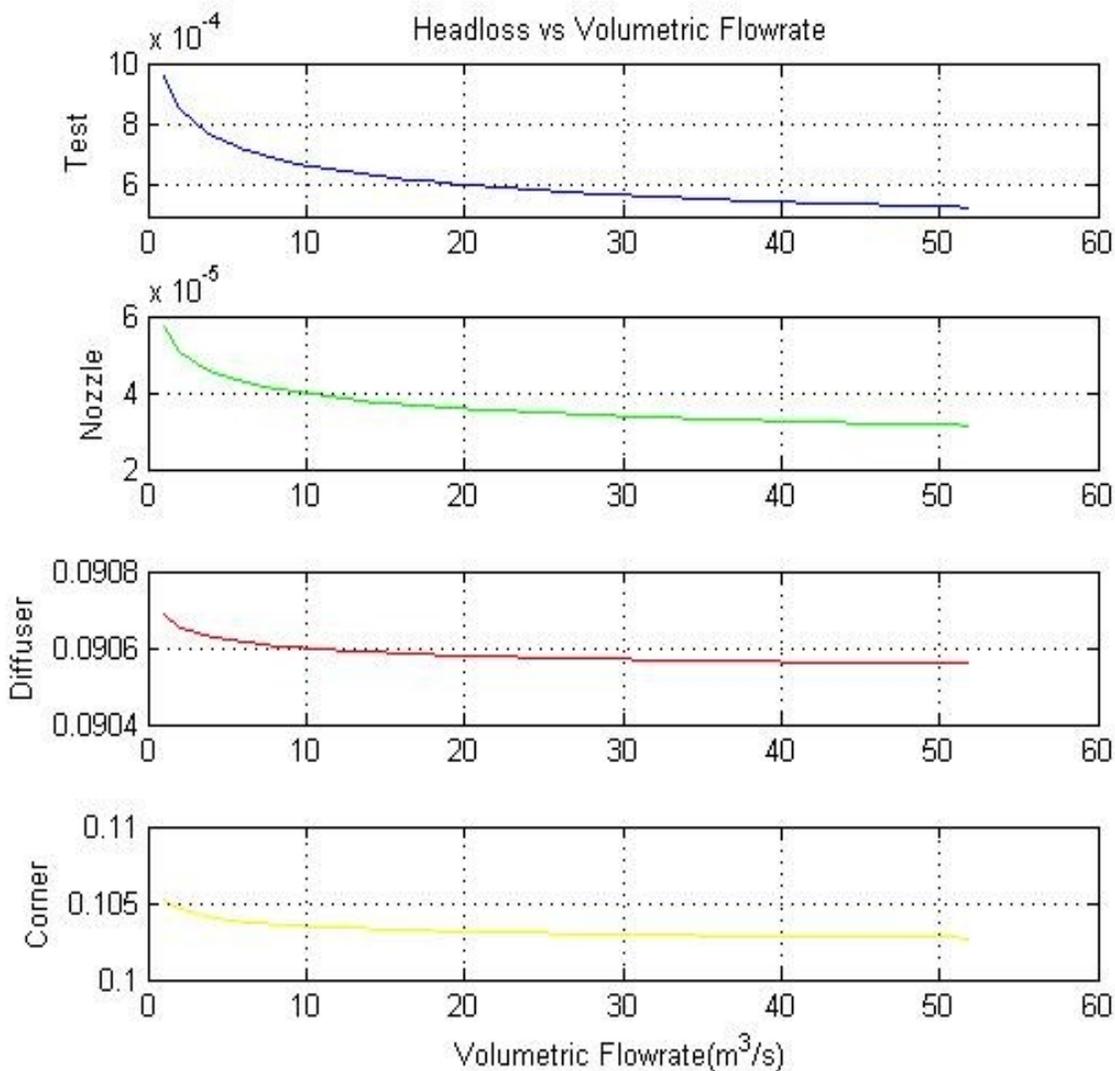


Figure 11- Pressure losses due to friction

From the tables above, one can see that the pressure drop due to losses tends to be the greatest around the diffuser section and the constant area section. In the calculations for this analysis, the constant area section consists of the losses from 2 full left turns, a protection screen and a flow straightener screen. In comparison to the test section, which can also be considered a constant area section, the losses are a lot less. The nozzle tends to have the least amount of losses mainly because the loss coefficient for this section tends to be used as an approximation.

For the analysis of the different coefficients of the tunnel, I will begin with the diffuser section since it is the longest portion of the tunnel and plays a big role in configuration. In minimizing the loss coefficient of this section, I can obtain the length of the entire diffusion section, as well as the angle at which to diffuse the airflow. The following graph shows the head loss coefficients with respect to the different lengths of the diffuser section of the tunnel. With this information, I can keep in mind the length that I want the tunnel to be. I will also have a

range to choose from. Notice how, in the graph that the trend tends to decrease rapidly, then slow down its curvature. But after a certain length, the trend actually reverses again. The bottom of this curve would be the most ideal length to use for the tunnel. Along with that length will come an angle to use. The equations used for these calculations are as follows:

$$K_{ex} = .09623 - .004152 * \theta \quad \text{For Theta between 0 and 1.5}$$

$$K_{ex} = .1222 - .04590 * \theta + .02203 * (\theta^2) + .003269 * (\theta^3) - .0006145 * (\theta^4) - .00002800 * (\theta^5) + .00002337 * (\theta^6) \quad \text{For } \theta \text{ between 1.5 and 5}$$

$$K_{ex} = -.01322 + .05866 * (\theta) \quad \text{For } \theta \text{ greater than 5}$$

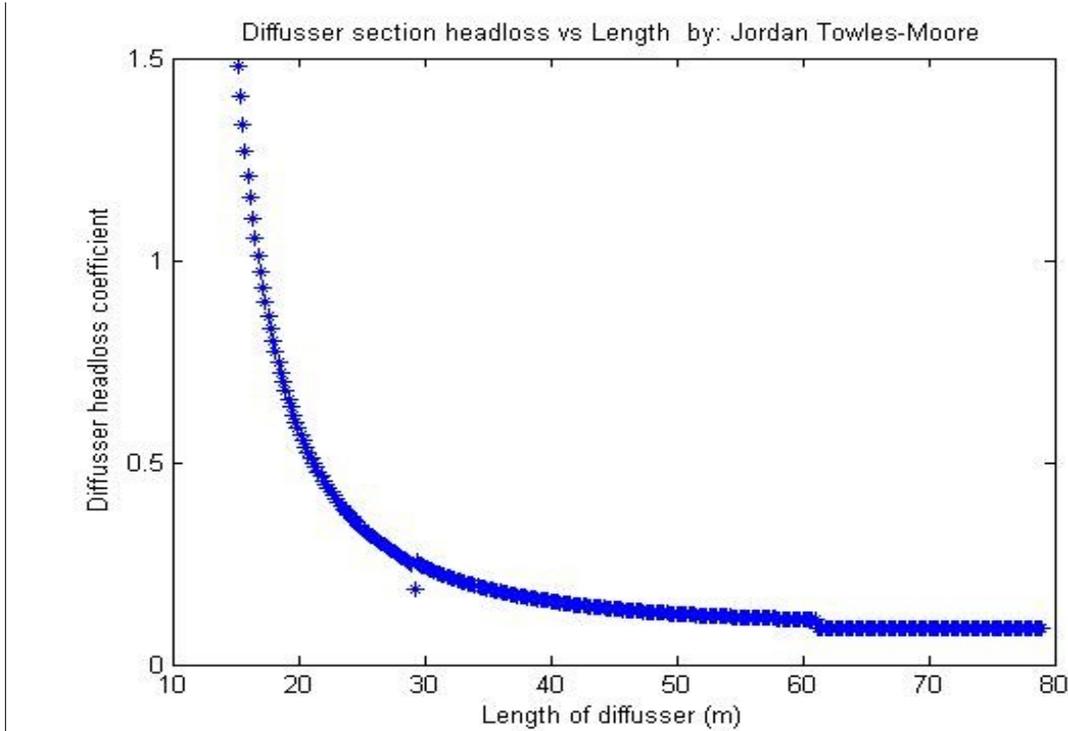


Figure 12- Diffuser head losses vs length

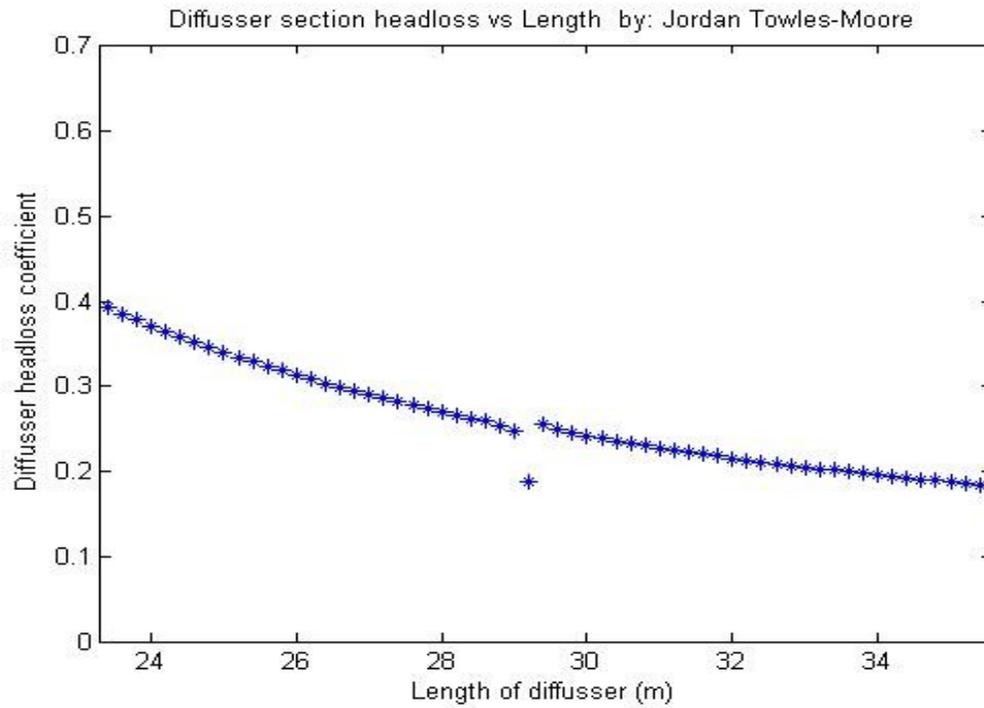


Figure 13- Diffuser head losses vs length (2)

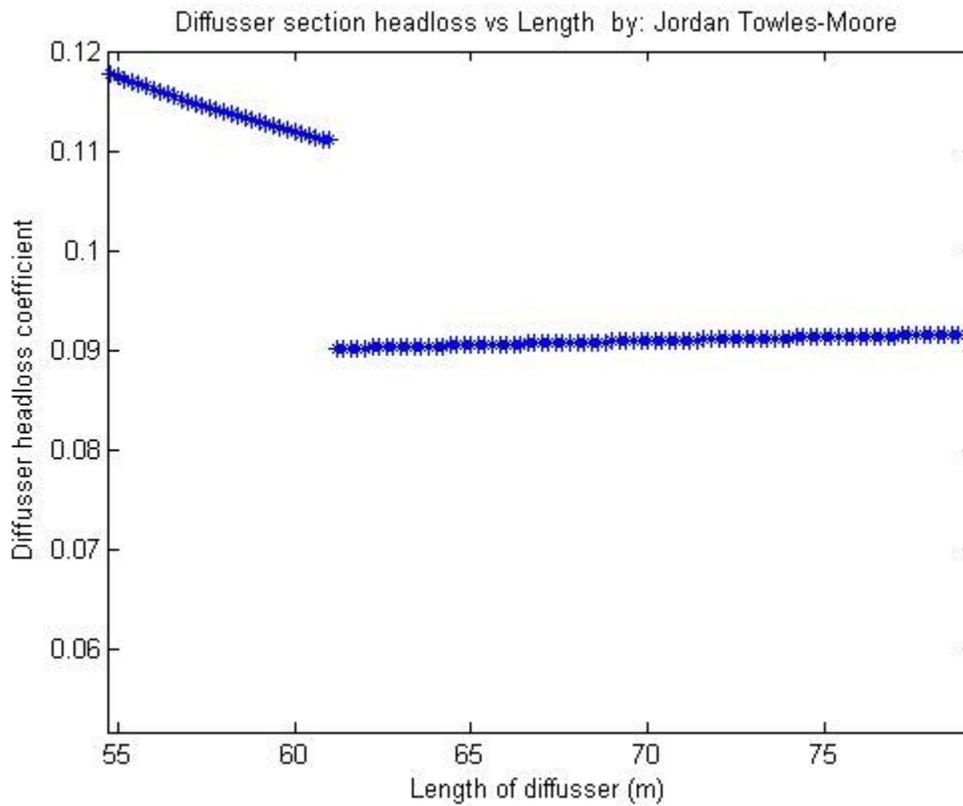


Figure 14- Diffuser head losses vs length (3)

The two previous graphs are close ups of the first graph. They show slight discontinuities in the curve. This is because the curve is made from calculations created from previous tests. Where the breaks are represent the changes in the angle of the diffuser section. They represent three different equations.

After examining the graph, a good length that the tunnel could be was about 65 meters in length. As stated before, the curvature of the head loss tends to eventually curve upward. It is around that area that the length was chosen. To complement the length, the angle of the diffusion section was also acquired using the following equation

$$14. \theta_e = \arctan\left(\frac{R_2 - R_1}{L}\right)$$

From this, the angle of the diffusion section came out to be 5.89 degrees. The diffuser section head losses were calculated through equations organized by Barlow. From the plots, the choice of a diffuser section of about 65 meters was selected for the tunnel. From this information, a preliminary design of the diffuser section could be made.

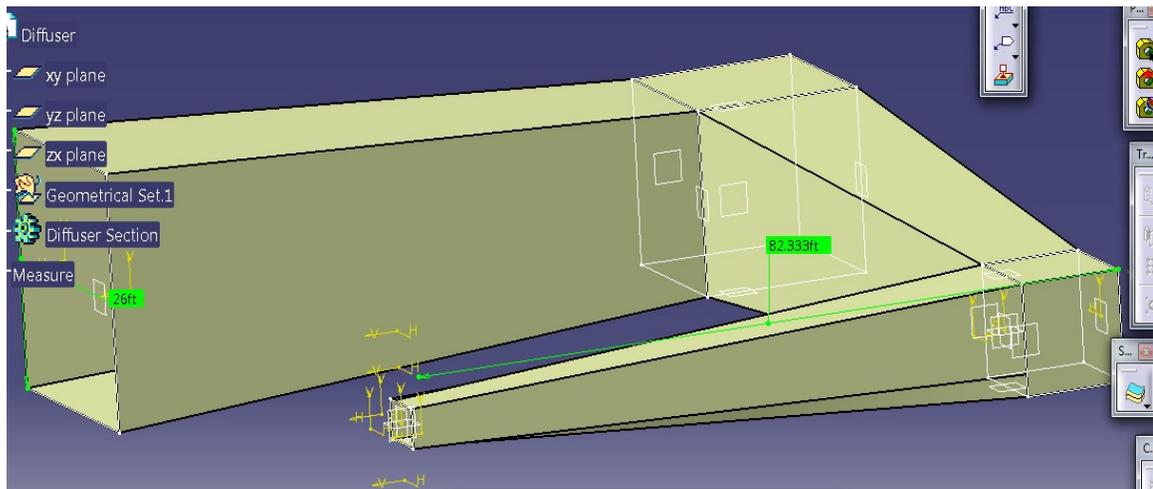


Figure 15- Preliminary design of diffuser section

With this completed, the next step can be made in the design process. The tunnel needs to turn in order to complete the loop, so the corners will be considered next. For simplicity, I will stick with the design of square tunnels and find the most efficient way to turn the flow without hurting-allowing the energy required to recirculate to be lost. But since that energy needs to be known, I will consider the flow power provided by the fan first.

Turning Vanes

With respect to previous wind tunnel analysis, the biggest turn in a closed circuit wind tunnel is the first two turns following the test section (Barlow, 1965). In order to minimize losses in these sections, turning vanes will be place at all four corners of the tunnel. For simplicity, the style of vane that will be used will model a curved plate with an extended trailing edge. This is chosen for two reasons. First, the cost to make airfoils for the tunnel will bring the cost of construction up. The second reason is that when looking at recorded data from previous testing, the amount of loss that is recovered from an airfoil instead of a curved plate is about 4%. From

this information, it would not be desirable to pursue an airfoil turning vane at this time (Barlow, 1965).

To apply the use of a turning vane, CFD analysis was done on the flow of air around the vanes at the max performance speed of the tunnel. The idea here is to get a visualization of how the flow is affected turning a corner with and without vanes. Below are the results of this analysis.

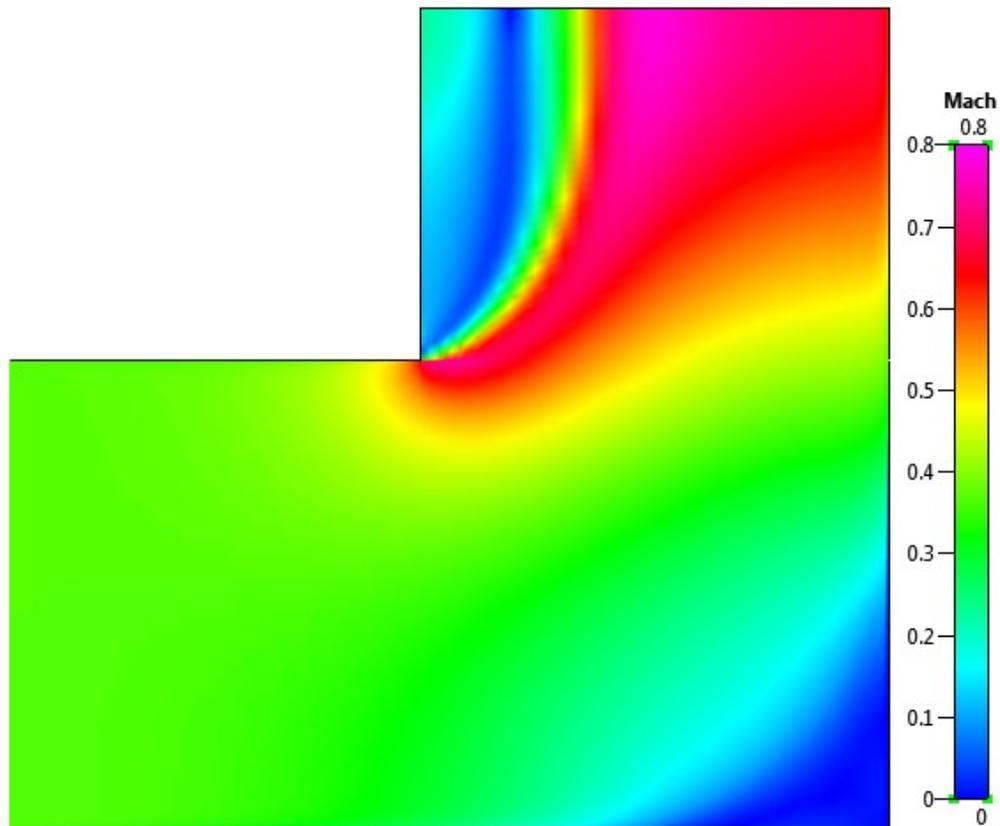


Figure 16- CFD image of corner with no vanes

Above, one can see the flow around a corner of a section of the tunnel with no vanes. Notice how much the flow is disturbed in order to change directions. This is where those losses occur. As mentioned before, this is one of the largest losses for a wind tunnel. The faster the airflow, the more aggressive this loss becomes. For a system with vanes, the loss is greatly reduced.

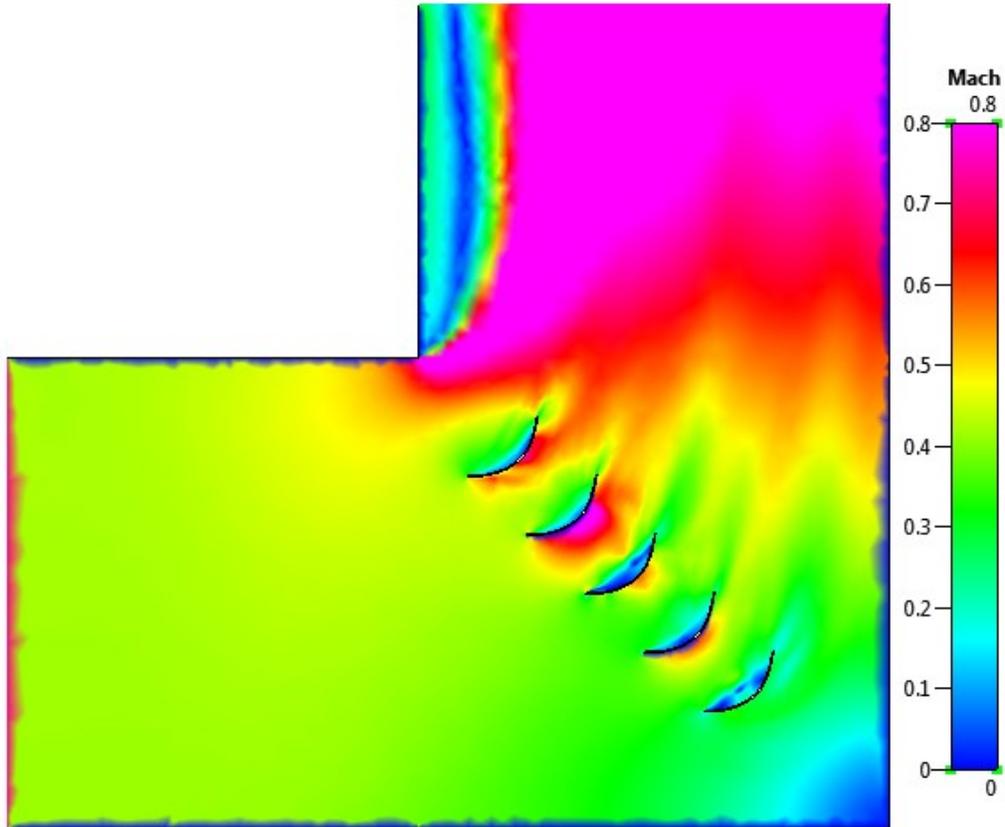


Figure 17- CFD of a corner with vanes

From the above images, flat plate vanes were inserted and tested. They greatly assist with the turning of the flow. Although, as can be seen, there are some areas that still will cause a lack of ideal flow. For these, larger vanes can be inserted and placed properly in order to redeem the flow to a more desired characteristic. Below is the most desired flow for this environment.

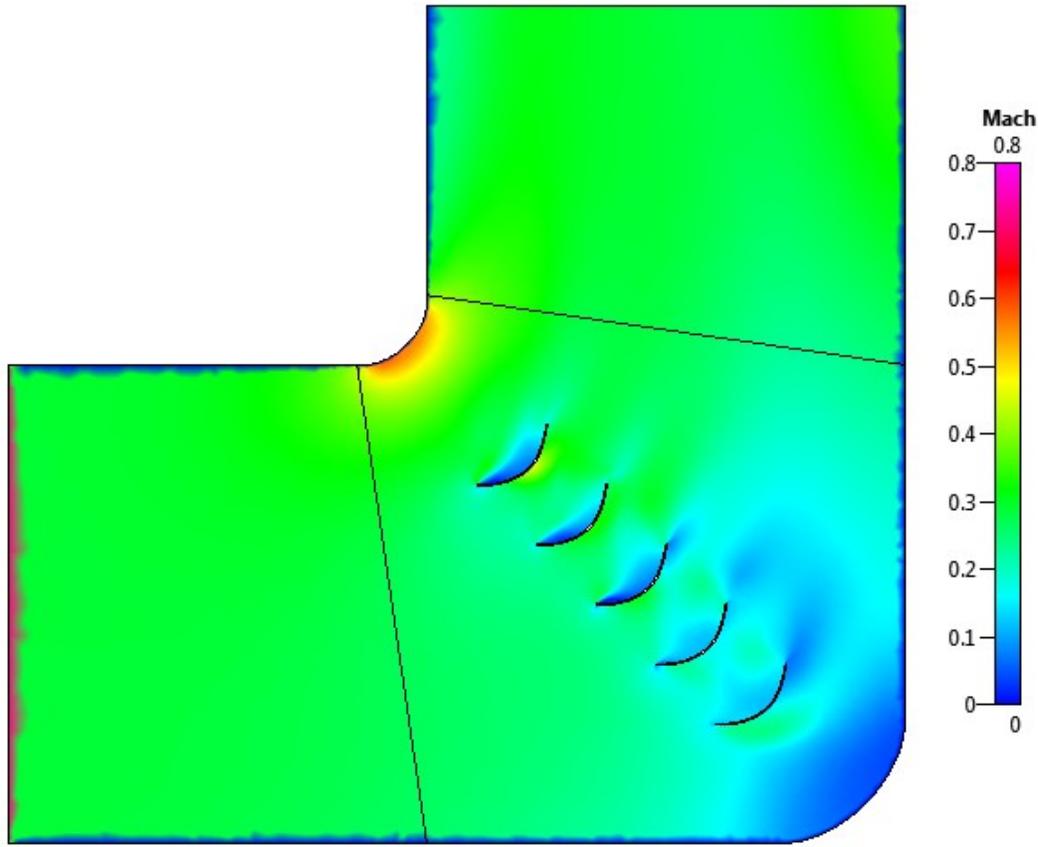


Figure 18- CFD of corner with optimized conditions

In this flow, the flow around the back corner is stopped. Note that because of this, the flow is compressed a little, because the total area of the fluid moving through this section will be slightly reduced. This will cause an acceleration of the flow instead of a deceleration, which is actually desired. But, the losses will still be minimized from a more turbulent flow. Results of this can be seen in the table below. It shows the total average forces in the x and y directions of the total flow in the area around where the vanes are located. It also shows the total average moment and area of these sections as well.

Aerodynamic Forces Around Vanes					
	F _x (N)	F _y (N)	M (N-m)	A (m ²)	Force Vector
Cornered Vane	196060	2.04E+05	-2.83E+04	5.1352	282703.011 1
No Vane	4.97E+05	5.03E+05	-7.43E+04	2.4384	707165.801 3
Desired Flow	200690	198200	-31179	5.2736	282063.319 3

Notice how there is only a slight difference in the cornered vane and the desired flow. This is what was expected, but the difference can be clearly seen in the flow visualization above.

In the cornered vane turn, the corner generates a cutoff on the flow, where it accelerated around the turn. For transonic flows, this is dangerous because it can easily increase the flow to Mach, which is very undesirable.

Also notice how the flow tends to speed up turning around the corner. This told me that the flow; when not properly turned and sufficiently diverged, could actually move faster than the flow in the test section even if the flow area is being diverged. This can provide a big problem when being run at max speed. But the desired flow did not provide these characteristics.

For the vane design, the goal is to mainly redirect the flow 90 degrees. Because another goal is to minimize costs, the design of an airfoil as a vane was not considered for this design. Instead, the idea of a curved plate vane design was implemented. As stated before, the differences in efficiency between the two is very small and not worth the expense.

~~Also notice how the flow tends to speed up turning around the corner. This told me that the flow; when not properly turned, could actually move faster than the flow in the test section even if the flow is being diffused. This can provide a big problem when being run at max speed. But the desired flow did not provide these characteristics.~~

~~For the vane design, the goal is to mainly redirect the flow 90 degrees. Because another goal is to minimize costs, the design of an airfoil as a vane was not considered for this design. Instead, the idea of a curved plate vane design was implemented. As stated before, the differences in efficiency between the two is very small and not worth the expense.~~

Screens

Screens for the tunnel are only important in the aspect of protecting the fan from debris. For a wind tunnel of this magnitude, there will need to be a decent sized screen in the diffuser section to guard against large objects being sucked in. The coefficient for energy loss due to the screen is observed in the calculations for the total losses in the diffuser section.

The turbulence screen serves a different purpose for the tunnel. It will be positioned right before the nozzle to rid the flow of any turbulent, uncontrolled airflow. For this tunnel, given its size, an alternative screen design is mentioned. Barlow states that instead of using one screen with small honeycomb cross sections, one should use multiple screens with larger cross sectional areas to provide a better pressure drop and a more promising flow (Barlow, 1965).

Boundary Layer Analysis

When looking at the test section, a boundary layer analysis will give a good interpretation as to the test capability of the tunnel. For this project, the test section will be approximately 20 feet long. The area section dimensions are as stated before, 6 ft. wide by 4 ft. tall. In order to analyze this boundary layer, the following assumptions are made:

- The length of the test section is as four solid plates connected in a rectangle.
- The boundary layer will be calculated for the max speed tolerated of 255 m/s.
- The analysis will be made using a modified Runge-Kutta Solution.

To do this analysis, I chose to follow a Runge-Kutta solution of a boundary layer analysis of Blasius' equation(as can be seen below) in order to accurately show the boundary layer at different stations within the test section of the tunnel. The analysis was taken from a Fortran execution of Runge-Kutta and modified by myself to run in Matlab.

15. $f''' + ff'' = 0$ Blasius' Equation

Which, when simplified becomes

$$16. \quad 2f''' + ff'' = 0$$

Given that the equation is a third order, non-linear, ordinary differential equation, it can be solved using the Runge-Kutta solution. This solution basically takes small increments of the boundary in the y direction going away from the wall. Given the equation, there are two boundary conditions that are known, but one is unknown. In this method the third boundary layer at the wall is assumed and is integrated with the other two for a specific number of increments. Then, if the number matches the edge of the boundary layer after the integration, the equation is then satisfied. If it does not match, then the assumed boundary layer condition is increased. The process repeats until the equation is satisfied. To do this, I created a code in Matlab that will observe the boundary layer of the fluid using this method of increments. The code can be seen in the appendix. Results of the analysis can be seen below.

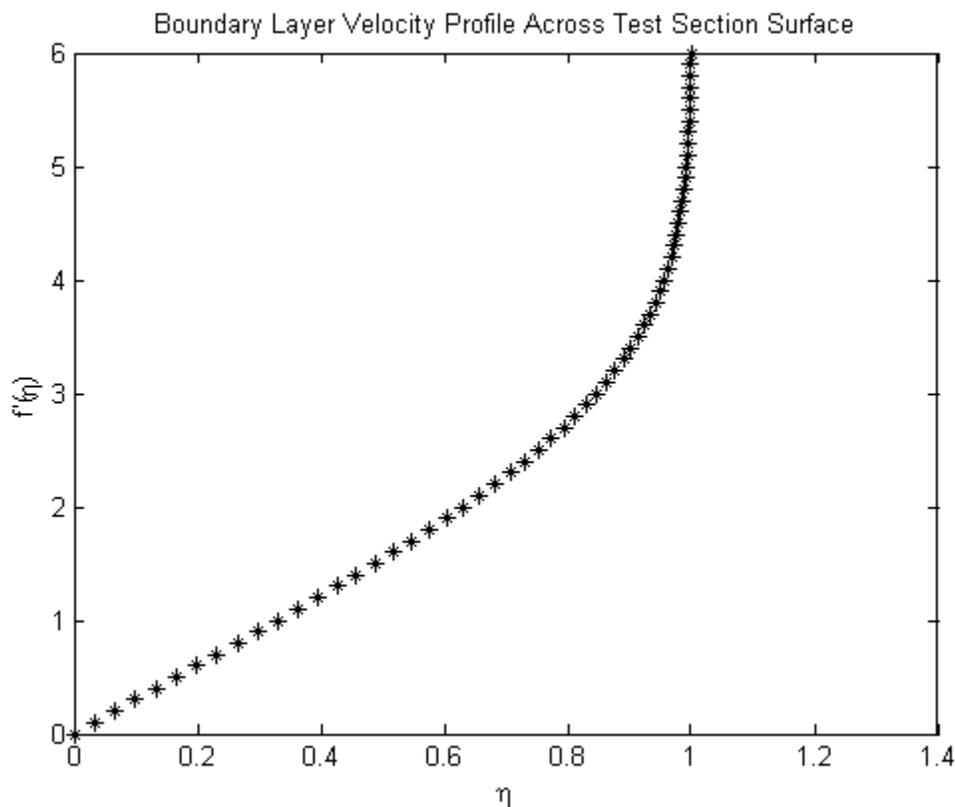


Figure 19- Velocity profile across the wall of the test section.

The above image is the velocity profile for the wall of the test section. The profile was formed using the modified version of Blasius' solution. Also using this solution, a profile was made for different locations, as can be seen below.

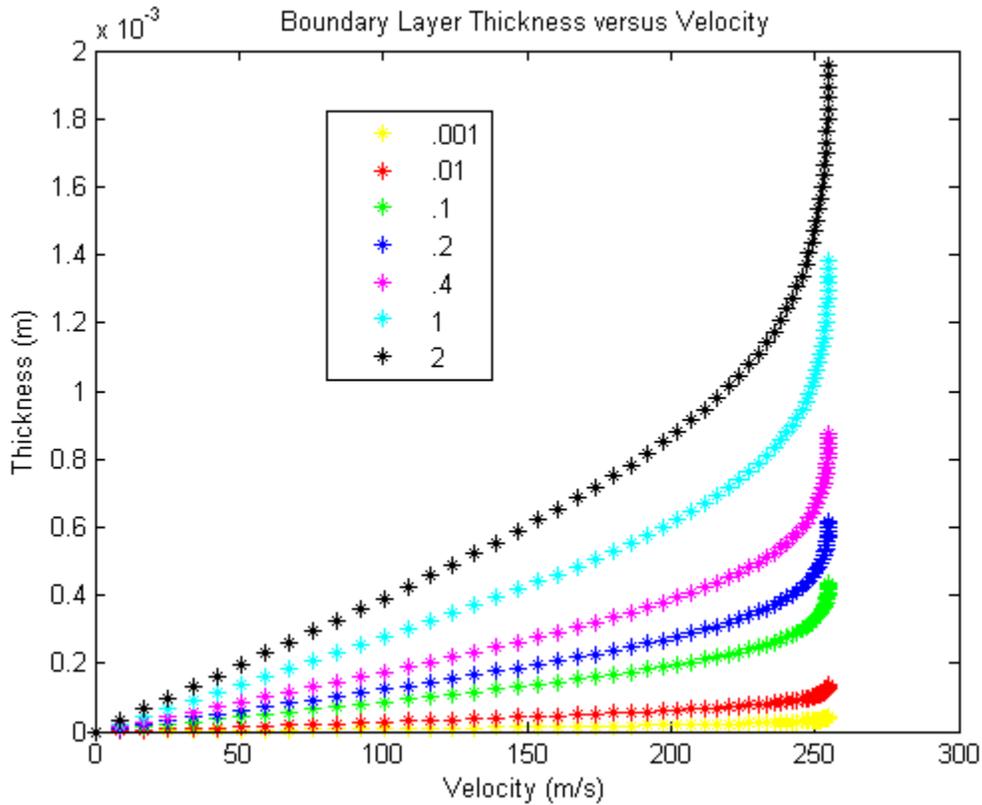


Figure 20- Velocity profiles for different stations along test section

Here, the boundary layer is plotted for different locations going across the test section, as represented in the legend. Notice that the boundary layer increases as the flow continues across the test section. From the graph, the boundary layer plots are observed at that of the top speed that the tunnel will be designed to run. From estimating from the graph, the boundary layer would sit somewhere around .0014 meters. This is noted from the area on the farthest away plot of the boundary layer, (the black line). Notice how at some point, the max velocity is reached. This would denote when the boundary layer is no longer present and the flow is in free stream.

Chapter 3- Test Section Flow Quality

Blockage/Boundary Correction

Another concept to look at when examining the test section of the tunnel is the amount of blockage that is present in the test section. When an object is placed inside the tunnel, it makes the area of the section smaller; thus according to Bernoulli, speeds up the flow. This increase in velocity has come to be known as solid blockage. In order to find the limitations for the amount of blockage that can exist in the test section, I will take a mathematical approach, using a simple model such as a right circular cylinder. When considering Rae's analysis of the cylinder, It will consist of a simulated doublet. Its strength is given below.

17. \square^{\square}

Where (a) is the radius of the cylinder.

Then, the axial velocity of the doublet is given by,

$$18. \quad \frac{\Delta V}{V_u} = a^2/h^2$$

Here, V_u is equal to the uncorrected velocity of the free stream. Because the velocity produced by a doublet is inversely proportional to the square of the distance from the doublet, the doubly infinite doublet series is given as

$$19. \quad \sum_{n=-\infty}^{\infty} \left(\frac{a^2}{z - z_n} \right) \sum_{m=-\infty}^{\infty} \left(\frac{a^2}{z - z_m} \right)$$

So, it ends up looking like the following

$$20. \quad \left(\frac{a^2}{z} \right) \left(\frac{a^2}{z} \right)$$

Now, for the blockage of an airfoil with given thickness (t), a good representation of the approximate cylinder would be one of a diameter $t(\lambda_2)^{\frac{1}{2}}$.

With this, the solid blockage for any 2-D body can be found using the simplified doublet summation. When combining the cylinder with the approximated one for an airfoil, we get

$$21. \quad \left(\frac{a^2}{z} \right) \left(\frac{a^2}{z} \right) \left(\frac{a^2}{z} \right)$$

The variable of λ_2 is estimated from a graph of the fineness ratio found in the image below from Glauert.

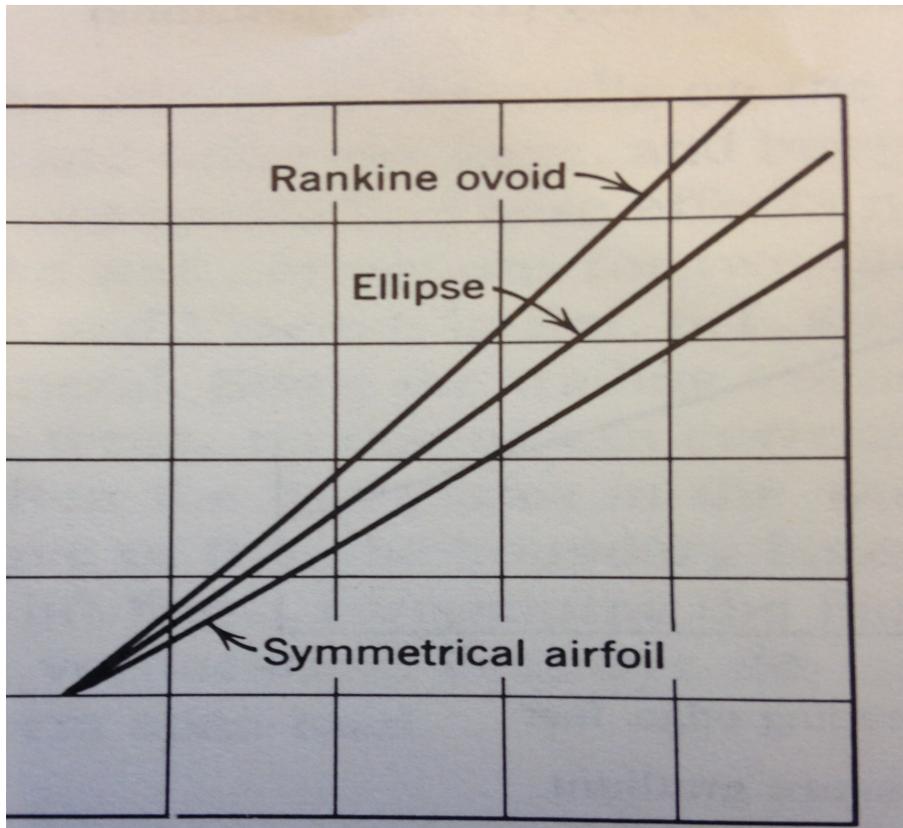


Figure 21- Reference for variable Lambda (Rae, 1984)

In this image, the variable λ depends on the ratio of the length of the airfoil and the thickness. Using this reference, one can use the equation for the solid blockage velocity increment to place a limit to how large an object can be when running the tunnel. In the case of this tunnel, the cap will need to be placed whereas the velocity will tend to exceed transonic speeds.

To do this, I considered the free stream velocity to be around 255 meters per second. The Mach velocity will be about 340 meters per second. For the velocity around the object to reach this Mach velocity, the increment will have increase by 1.33 percent. I will basically increase the thickness of a right cylinder until it reaches the limits of velocity. For this tunnel that will be the approximate limit of how large an object can be inside the tunnel when running at max speed. I will not consider the approximation for an airfoil because of the constant variable lambda, which will simultaneously change with every change in thickness.

The results of this analysis came out to give a max cylinder diameter of .154 meters. This means that for a cylinder, if it exceeds this thickness at max speed in the tunnel, there will be Mach flow running in the test section of the tunnel. For this project, it is very undesirable. This is, of course for max speed, so for slower speeds, the object can be much larger. The analysis was done in Matlab and can be seen in the appendix. Below is a plot of the basic airfoil, designed in Matlab.

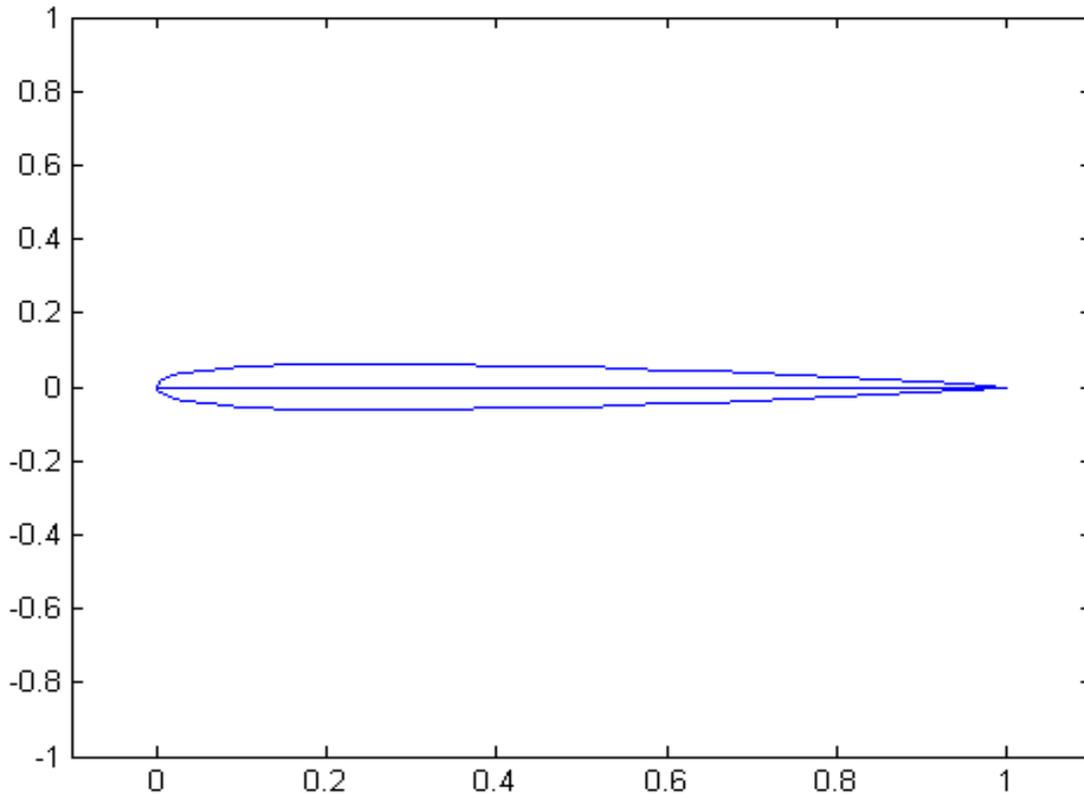


Figure 22- NACA 0012 Airfoil (Sankar, 1997)

Another method that was used for solid blockage was Thom's method. With this method, the following equation was used:

$$22. \quad C_{D1} = \frac{C_{D0} \left(\frac{A_0}{A} \right)^2}{1 - K_1 \left(\frac{A_0}{A} \right)}$$

Where K_1 is equal to approximately .74 for wings that span the tunnels width, and C_{D0} is the tunnels test section area. With this method, 3-D flow is taken into account. Also, an approximation of the volume of an airfoil was used. A NACA 0012 airfoil was simulated for this analysis. From the results, the solid blockage correction comes out to be about 4.27 percent. The error between the two methods falls under the use of 2-D and 3-D specifications. Also, when an airfoil is made, it allows for far better flow. (Rae, 1984)

As for correcting the boundaries of the tunnel, we will have to combine the information from the blockage with the information received for the boundary layer of the walls. For this, I will keep in consideration the NACA 0012 airfoil.

For an airfoil this size, I wanted to look at the positioning of the airfoil in the tunnel whereas the flow will not be disturbed by the boundary layer. Given the information for the max thickness of the object and the information for the approximate blockage corrections, It is found that an object should be within approximately $\frac{1}{4}$ the ratio of the area of the front of the object to the total area of the test section. For this tunnel, it comes out to be approximately $\frac{1}{4}$. But for a tunnel running at the speed that I would like to achieve, it must be less than 1.33.

From the boundary layer analysis, the boundary sits about .0014 meter out from all wall surfaces. This makes the wall that we want go from 1.2192 meters to 1.2164 meters. From there, I reanalyzed the blockage, which came about to be approximately the same. This is as expected though. The boundary layer ends up being so small that it can generally be neglected at these speeds. But at the same time, the blockage that occurs cannot be neglected.

In providing limitations for objects placed in this tunnel, I have come down to the fact that the object cannot be larger than ~6 inches in thickness when operating at full speed.

Chapter 4 - Fan Design

For a tunnel of this size, a very efficient motor will have to be provided in order to allow the velocities required of this project. Pope describes that the more efficient fans provide a lift to drag ratio of approximately 50. It is also implied that for higher rpm's of a fan motor, the faster velocity of the wind running through the tunnel is a better location for the mounting of the fan. When comparing that to the past tests of nacelle drag as the velocity increases, the more efficient place to put the fan is downstream of the second turn of a circulating wind tunnel coming out of the test section.

As for the L/D for the fan, 50 is a very high number that is used in reference to an infinite aspect ratio. In order to obtain this, I will need to simulate it by having an end plate on both sides of the fan blade. On a more realistic level, the tips of the blades coming out from the root will need to be very close to the wall of the tunnel. More ideally, I will need a motor supply that is more centralized inside the tunnel. The best description of this would be as a wind turbine nacelle-type motor.

Ultimately, one would want to keep the diffusion cross sectional area constant even through the fan power section. If the motor supply is in the middle of the tunnel, then the nacelle would be very large due to power demands. Because of this, the cross sectional area that the nacelle will take up will need to be gained back by enlarging the total cross sectional area of the tunnel in that section. At the same time, the larger the nacelle is, the longer it will need to be to allow a quality flow to go through. In a sense, one would need to use area ruling in order to allow a good streamlined flow.

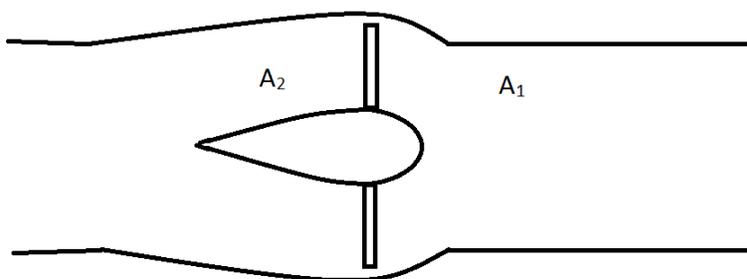


Figure 23 - Area ruling for nacelle section of the wind tunnel

As seen in the image above, the cross sectional area should stay approximately the same throughout the length of the of the fan supply. This will increase the efficiency of the motor and its ability to generate the max flow required. (Rae, 1984)

Fan Straightener

Another thing to consider when looking at the flow design of the fan section is the amount of twist that shows up in the flow when coming off the blades. This causes an increase in axial

velocity, and in turn will increase the absolute velocity. The idea here is to remove this excess velocity as opposed to turning it in the correct direction. This will be accomplished through straightening vanes.

Following the fan straightener theory provided by Pope, I started with the power output of fan.

$$23. \frac{P_{out}}{P_{in}}$$

Where A_b is the area of the fan nacelle at the root of the blade, A_f is the cross sectional area of the fan, and η_t is the total efficiency of the straightener.

From that, we get

$$24. \frac{P_{out}}{P_{in}} = \frac{A_b}{A_f} \eta_t$$

Where A_t is the test section area and V_t is the test section velocity.

When we apply the definition of the energy ratio along with the law of continuity, we will get the following equation for k:

$$25. k = \frac{A_t V_t^2}{A_b V_b^2}$$

Then the fan efficiency is simply derived starting from the basic relationship of power out over the power in.

$$26. \frac{P_{out}}{P_{in}}$$

Where Q is the torque from the fan and n is the number of revolutions per second.

It turns out being more convenient to rewrite the efficiency in a manner that can reference the radius of the annulus, so the efficiency is rewritten as

$$27. \frac{Q}{\rho u^3 r^3}$$

And defining dQ as $2\pi r * dr * \rho u * \omega r^2$, we get

$$28. \frac{Q}{\rho u^3 r^3}$$

Defining the local advance ratio as $j = u\pi / \Omega r$, the flow rotation as $e = \frac{\omega r}{u}$

Whereas $\eta_t = kj / 2\pi e$

Then we can define a head loss due to the straightener as

$$29. \frac{\Delta P}{\rho u^2}$$

Where e is taken from the equation of η_t .

From this, we can state that the fan efficiency can be known from

$$30. \frac{P_{out}}{P_{in}}$$

Next, we define the elemental thrust and multiply it by the elemental area as the pressure rises.

31.

Changing it to coefficient form, we get

$$32. \frac{C_{T,el} \rho \omega^2 r^2 dr}{2 \pi r^2 \rho \omega^2 r^2}$$

The total pressure rise required for the fan and the flow straightener together is

$$33. \frac{P_{fan}}{P_{flow}}$$

Which makes the elemental thrust coefficient

$$34. \frac{C_{T,el}}{2 \pi r^2} (\rho \omega^2 r^2)$$

Which we can then write the elemental torque as

$$35. \rho \omega^2 r^3 \left(\frac{C_{T,el}}{2 \pi r^2} \right) (2 \pi r^2 dr)$$

Where x_o is the ratio of the radius at the root, and J is the advance ratio at the fan blade tip.

This equation provides the torque input that would be necessary to realize the presence of the pressure change, hence giving us the power required. Ultimately, we want to get an equation of the efficiency that relates the advance ratio, e , and L/D .

With that, we move on to the pressure and power.

$$36. \frac{P_{fan}}{P_{flow}}$$

$$\frac{P_{fan}}{P_{flow}}$$

Making the fan efficiency

$$37. \frac{P_{fan}}{P_{flow}}$$

Which, when applied in coefficient form becomes

$$38. \frac{C_{T,el} \rho \omega^2 r^2 dr}{2 \pi r^2 \rho \omega^2 r^2}$$

With some substitution,

$$39. \frac{C_t}{C_d} \left(\frac{C_t}{C_d} \right)$$

Now, to get L/D involved in the equation, we express the elemental thrust in such a way as to relate to normal coefficients for wings.

$$40. \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d}$$

Where ϕ is the neglected twist, N = the number of blades on the fan, and C_t is the thrust coefficient. We also define the torque force coefficient as

Reducing the above equation, we get

$$41. \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d}$$

The corresponding torque is

$$42. \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d}$$

Substituting in the previous form of the efficiency coefficient, we get

$$43. \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d}$$

And using some trigonometry from ϕ

$$44. \tan \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d}$$

And finally, inserting L/D, we get

$$45. \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d} \frac{C_t}{C_d}$$

When designing the fan for this tunnel, one would seek to get the most efficiency out of the fan. Using this equation, which incorporates all aspects of the circulating tunnel, one can use the solutions to achieve the best possible efficiency for the fan; and in doing, so come across the best design for the fan blades as well as the fan straightener (Rae, 1984).

Flow Power

In the test section, an estimate of the power needed to allow for the max airflow will need to be made. For this, the power was calculated for an ideal fan for the tunnel using the following equations.

$$46. \quad p_c = \frac{1}{A_c} \iint_{A_c} p \, dA \quad \text{Static Pressure}$$

$$47. \quad p_t = \frac{1}{2} \dot{m}_t V_t^2 = \frac{1}{2} \rho_t A_t V_t^3 \quad \text{Jet Power}$$

$$48. \quad E_R = \frac{P_t}{P_c} \quad \text{Energy Ratio for flow losses}$$

Where P_t is the power of the test section, and P_c is the loss from the flow around the tunnel. Using this equation, I looked at the flow loss to see how much power I would need in order to operate the tunnel at the necessary speed. The following plot shows my results.

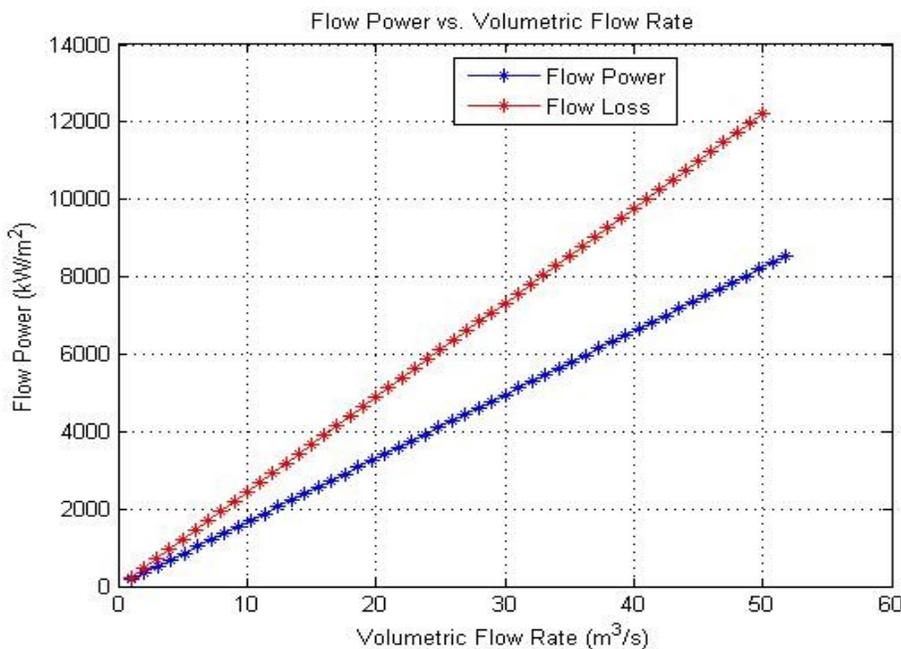


Figure 24- Flow power graph for power requirement

It is my understanding that the flow loss represents the flow that would need to be made in order to achieve the goal of flow rate for this tunnel. The differences between the two are the fact that the flow loss accounts for the total losses within the tunnel and how the energy required to maintain this power changes. Basically, this shows that due to the largest losses in the system, (the diffuser and the corners), the amount of energy required gets a lot larger as the volumetric flow rate increases. It is almost like an exponential growth. In other words, to get a flow rate of 53 m³/s, One would need about 9000 kW/m², but due to all the losses in the system, One would actually need 12,000 kW/m².

|

Turning Vanes

With respect to previous wind tunnel analysis, the biggest turn in a closed circuit wind tunnel is the first two turns following the test section (Barlow, 1965). In order to minimize losses in these sections, turning vanes will be placed at all four corners of the tunnel. For simplicity, the style of vane that will be used will model a curved plate with an extended trailing edge. This is chosen for two reasons. First, the cost to make airfoils for the tunnel will bring the cost of construction up. The second reason is that when looking at recorded data from previous testing, the amount of loss that is recovered from an airfoil instead of a curved plate is about 4%. From this information, it would not be desirable to pursue an airfoil turning vane at this time (Barlow, 1965).

To apply the use of a turning vane, CFD analysis was done on the flow of air around the vanes at the max performance speed of the tunnel. The idea here is to get a visualization of how the flow is affected turning a corner with and without vanes. Below are the results of this analysis.

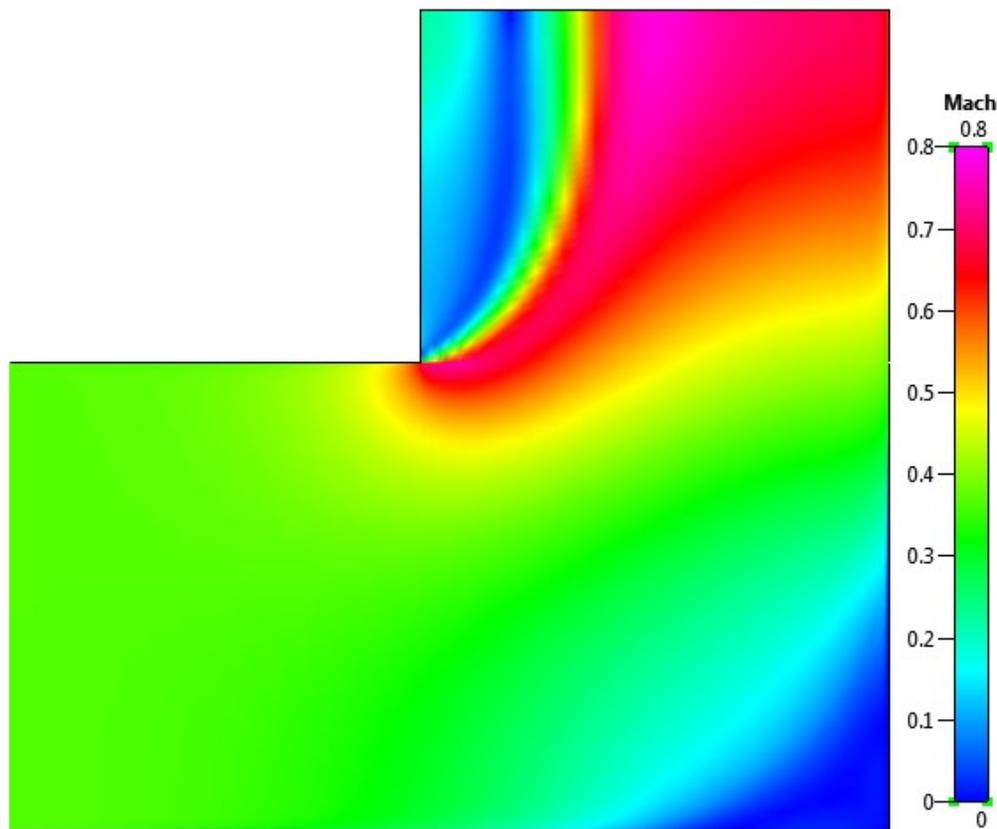


Figure 25—CFD image of corner with no vanes

Above, one can see the flow around a corner of a section of the tunnel with no vanes. Notice how much the flow is disturbed in order to change directions. This is where those losses occur. As mentioned before, this is one of the largest losses for a wind tunnel. The faster the airflow, the more aggressive this loss becomes. For a system with vanes, the loss is greatly reduced.

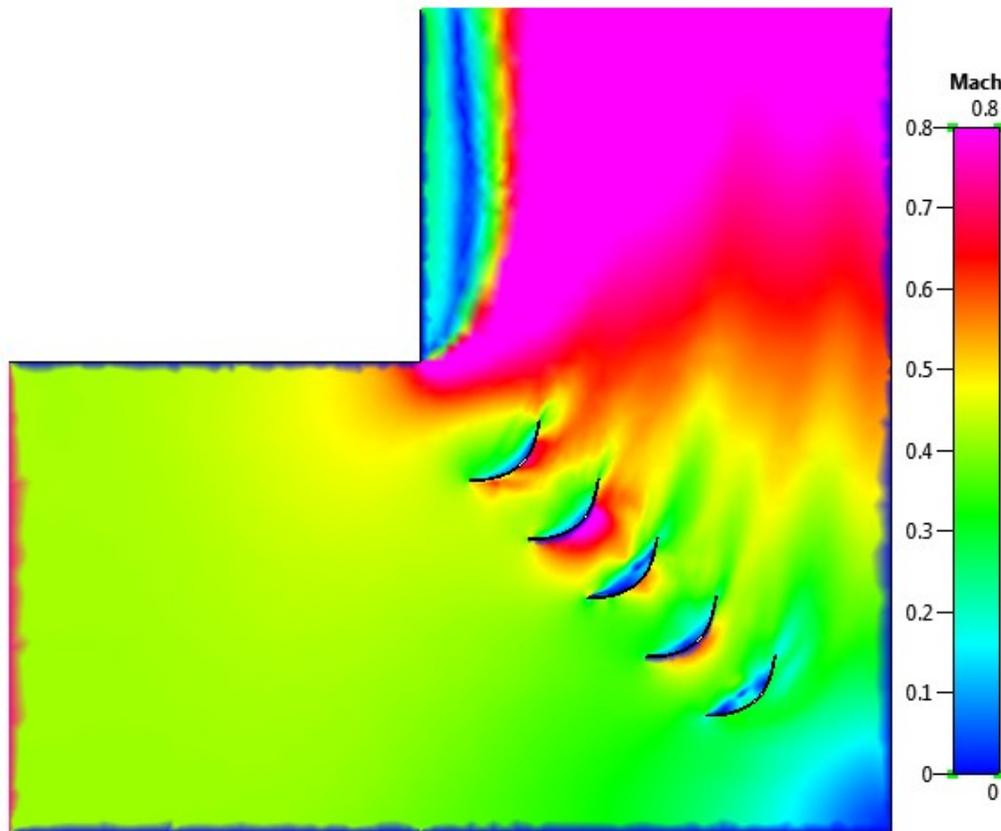


Figure 26—CFD of a corner with vanes

From the above images, flat plate vanes were inserted and tested. They greatly assist with the turning of the flow. Although, as can be seen, there are some areas that still will cause a lack of ideal flow. For these, larger vanes can be inserted and placed properly in order to redeem the flow to a more desired characteristic. Below is the most desired flow for this environment.

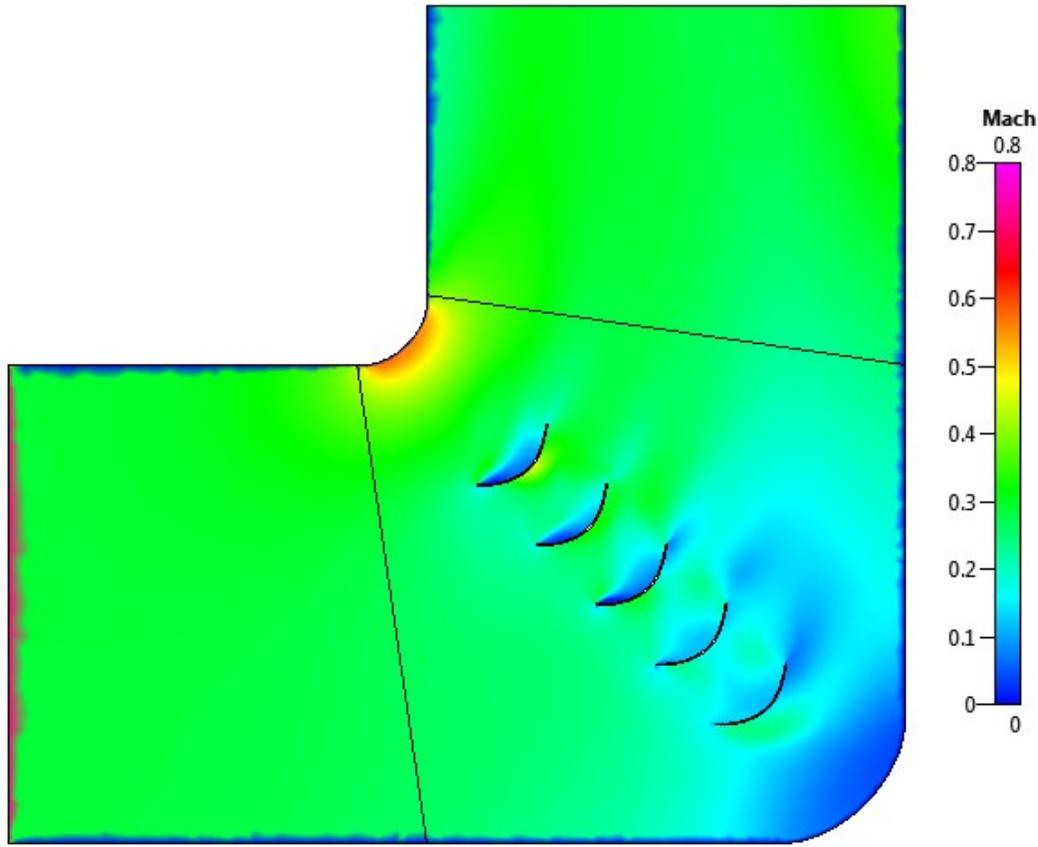


Figure 27 CFD of corner with optimized conditions

In this flow, the flow around the back corner is stopped. Note that because of this, the flow is compressed a little, because the total area of the fluid moving through this section will be slightly reduced. This will cause an acceleration of the flow instead of a deceleration, which is actually desired. But, the losses will still be minimized from a more turbulent flow. Results of this can be seen in the table below. It shows the total average forces in the x and y directions of the total flow in the area around where the vanes are located. It also shows the total average moment and area of these sections as well.

Aerodynamic Forces Around Vanes					
	F_x (N)	F_y (N)	M (N-m)	A (m ²)	Force-Vector
Cornered-Vane	196060	2.04E+05	-2.83E+04	5.1352	282703.011
No-Vane	4.97E+05	5.03E+05	-7.43E+04	2.4384	707165.8013
Desired Flow	200690	198200	-31179	5.2736	282063.3193

~~Notice how there is only a slight difference in the cornered vane and the desired flow. This is what was expected, but the difference can be clearly seen in the flow visualization above. In the cornered vane turn, the corner generates a cutoff on the flow, where it accelerated around the turn. For transonic flows, this is dangerous because it can easily increase the flow to Mach, which is very undesirable.~~

~~Also notice how the flow tends to speed up turning around the corner. This told me that the flow, when not properly turned, could actually move faster than the flow in the test section even if the flow is being diffused. This can provide a big problem when being run at max speed. But the desired flow did not provide these characteristics.~~

~~For the vane design, the goal is to mainly redirect the flow 90 degrees. Because another goal is to minimize costs, the design of an airfoil as a vane was not considered for this design. Instead, the idea of a curved plate vane design was implemented. As stated before, the differences in efficiency between the two is very small and not worth the expense.~~

Screens

~~Screens for the tunnel are only important in the aspect of protecting the fan from debris. For a wind tunnel of this magnitude, there will need to be a decent sized screen in the diffuser section to guard against large objects being sucked in. The coefficient for energy loss due to the screen is observed in the calculations for the total losses in the diffuser section.~~

~~The turbulence screen serves a different purpose for the tunnel. It will be positioned right before the nozzle to rid the flow of any turbulent, uncontrolled airflow. For this tunnel, given its size, an alternative screen design is mentioned. Barlow states that instead of using one screen with small honeycomb cross sections, one should use multiple screens with larger cross sectional areas to provide a better pressure drop and a more promising flow (Barlow, 1965).~~

Fan Power

When considering power to provide an airflow of around 660 ft/sec, one must look at the amount of horsepower per unit area to use. Barlow gives a graph that gives good estimates on how much horsepower is required. The equation used in calculating this is of the following form:

$$49. \quad P_t = \frac{1}{2} \rho_t A_t V_t^3$$

Where ρ_t is the density of air, A_t is the area at the test section, and V_t is the velocity at that point. For this project, all of the variables are known. So, estimated from the graph provided by Barlow, The amount of horsepower required to move air through this tunnel is about 500 hp/ft². Since the test area of this tunnel is designed to be 6 feet by 4 feet, a motor with 12,000 hp is required. This motor will probably be the largest investment for this tunnel (Barlow, 1965).

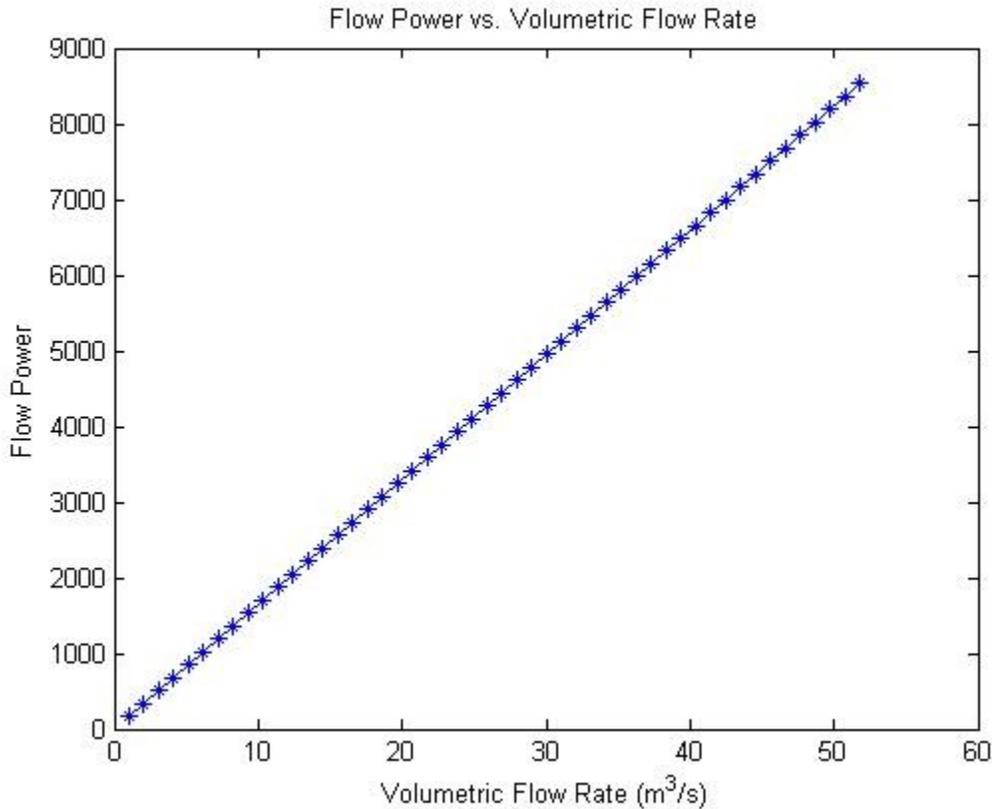


Figure 28--Power graph for power requirement

Balance

For this project, one of the requirements is for the tunnel to provide basic lift and drag forces on any object placed in it. For this, a single component or 3 component balanced will need to be purchased for the tunnel. Integrating it into the tunnel will require optimizing once the tunnel is built. In the case of this project, the tunnel will be built based on the fan housing, so the balance will need to be optimized for the test section that will already be built.

Chapter 35- Design Break-Down

Design Break-down

As for the structure of the tunnel, a systems engineering approach will be taken in determining the basic design for this project. Throughout this process, the information obtained in the previous chapter will be taken in to account while determining the structure of the tunnel.

From a provider's perspective, two things that will hinder the production of this tunnel are cost and complexity. To account for this, the main focus in designing the tunnel will fall to designing a high speed subsonic wind tunnel that will allow for laminar flow through a test section that is both simple and inexpensive.

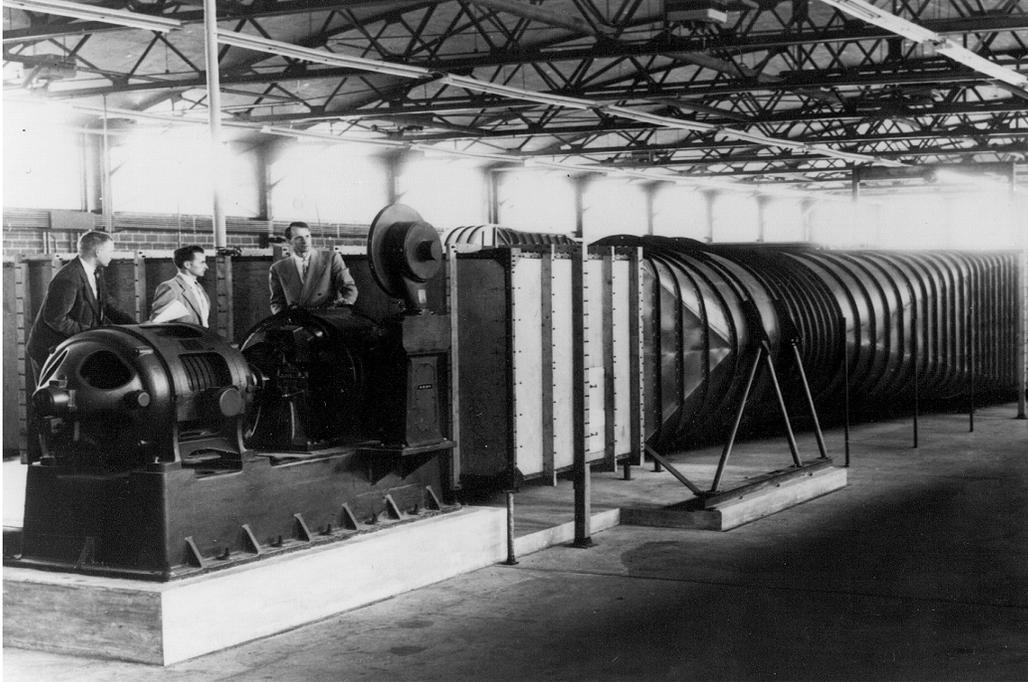


Figure 29- George Palmer's original wind tunnel design(Purdue University)

Figure 30- George Palmer's original closed-circuit-wind-tunnel-design (Purdue University)

From an aerodynamicist's point of view, steady controllable flow is the primary objective. So, the inner wall materials will need to be very slick, but at the same time, very strong as to not move when the pressure against the walls increases. As seen in chapter 2, there is an exponential pressure increase as the wind tunnel approaches its ideal max speed. To compensate for that, the structure must be very sturdy. For this project, weight is a factor that shows no concern, so an allotment of material can be used for the walls of the tunnel. To keep the flow smooth on the surface of the inner walls, a surface sealing coat can be placed on a material if it is not ideally smooth enough to minimize friction. The outer wall does not really matter and can be made in any way. The difficult part is connecting the different panels together. As seen in chapter 1, in the preliminary design, the different pieces are essentially sections that are joined together. Where the problem lies is vibration from the motor and air leaks from construction error. From a design perspective, this may be one of the more difficult parts of design, because there will be many sections for a tunnel of this magnitude, and just one misalignment can cause unsteady flow through the test section. More realistically, as long as the walls from the flow straightener through the test section have smooth transitions, the test section should get a good laminar flow. But if other parts are not, the losses will build up and cause poor performance.

From a design perspective, a consistent style of connecting the sections together will be a great asset to the construction of the full tunnel. Ideally, each section should be built separately and then connected. To account for vibration control, each section can be bolted together tightly

with locking nuts so that the full tunnel can be considered as 1 rigid body. If this is the case, then the tunnel can also sit on rubber pads to dampen the vibration. To account for the air leak problem, the sections can be sealed and sanded at each adjoining part from the inside. This would be more cost effective than placing padding in between each section.

When considering the design of the test section, it would be cost effective to build the nozzle so that the top and bottom are constructed the same, as well as the sides. In order to do this, the test section will need to be centered within the cross-sectional area of the tunnel. From a design aspect, this leads to the shape design of the tunnel. A circular shape would be more ideal for the tunnel, but a rectangular shape for the different sections of the tunnel can also yield good flow through the test section, and at the same time is very cost effective.

From a structures point of view, the main focus of the tunnel would be the areas that must support weight. For this, the bottom of the wind tunnel is considered to be the most important part. Ideally, one would want all 4 walls that run through the tunnel to be made of the same material, but for this tunnel, the floor would need to be strong enough for at least a 200 lb. person to be able to walk around in. This would be a requirement for construction. As for the test section, it will need to be supported in mid height between the nozzle and the diffuser sections. The floor of that section will also need to be able to support around 500 + lbs., as mentioned in chapter 1.

Another concept to consider is the door to the test section as well as access points to get inside different parts of the tunnel. As for the test section, it would be both cost effective and overall efficient to make one wall the whole door. If this is done, then the other three walls will simply be reinforced. The picture below is an example of a test section enclosure where the right side of the section is the door. This would be better for larger objects that are placed in the tunnel for testing.



Figure 31- Example test section door for a 4x6 wind tunnel. (Purdue University)

Figure 32- Example test section door for a 4x6 wind tunnel. (Purdue University)

As for the full design of the tunnel, the picture below gives a basic design for the full scale, fully dimensioned inside area of the tunnel.

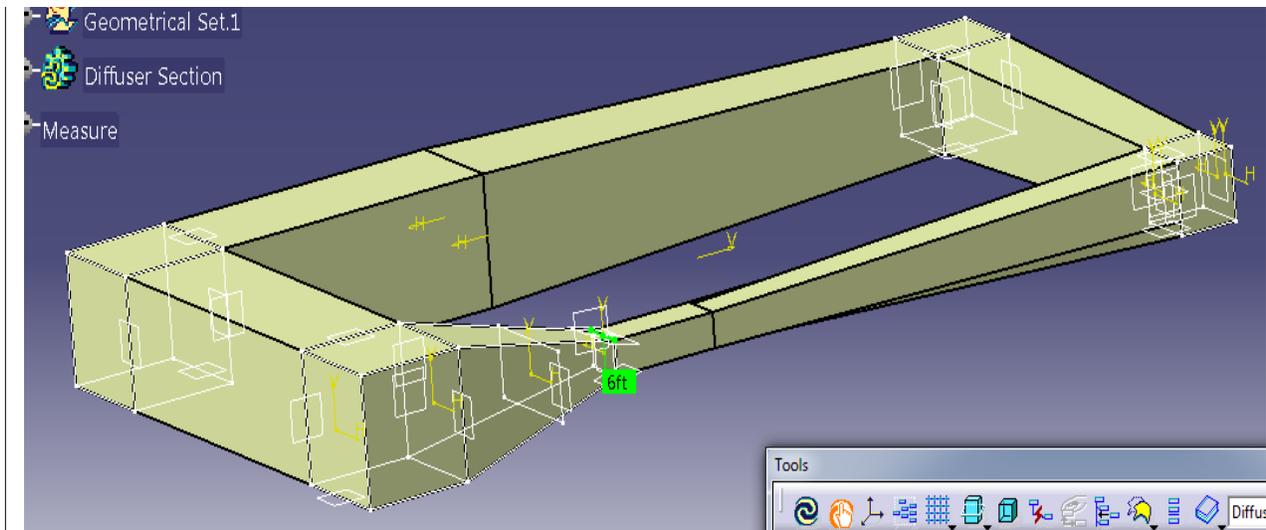


Figure 33- Full scale dimensioned design of wind tunnel

Figure 34- Full scale dimensioned design of wind tunnel

As can be seen, the main part of the tunnel is the test section. The rest of the tunnel is required in order to get the flow recirculated back around and ready to provide steady laminar flow once again. Of course, within the tunnel, one would find the turning vanes at all 4 corners, as well as a flow straightener right before the compression section. For simplicity and cost effectiveness, notice how at each turn, the flow pauses its angled diffusion. This simply allows less work to be put on constructing the angled turns when building the tunnel. The losses due to this are very minimal and will not affect the overall flow.

Balance

For this project, one of the requirements is for the tunnel to provide basic lift and drag forces on any object placed in it. For this, a single component or 3 component balanced will need to be purchased for the tunnel. Integrating it into the tunnel will require optimizing once the tunnel is built. In the case of this project, the tunnel will be built based on the fan housing, so the balance will need to be optimized for the test section that will already be built.

From a materials point of view, a large selection is available for this project. As mentioned before, what limits one of the design properties of this tunnel is cost. As for the complexity of the project, basic materials that are similar throughout the tunnel would be both cost effective and easier to construct, yet at the same time fulfill the requirements of the customer. These materials are broken down and analyzed by cost in the section below.

Cost Analysis

As one of the more important design requirements, the cost of the materials and components play the largest role for this project. The idea is to know exactly what is needed and the price to do so. Looking at the requirements for this tunnel, the most costly of items would fall under the walls, fan motor, and balance. Although these are most of the parts of the tunnel, these parts allow the requirements to be met, so they will be the foundation of construction.

The table below shows a basic summary of the required parts in order to build the wind tunnel. For some parts, a selection is available for choice. The idea is to minimize the cost of the parts without destroying the integrity of the design. Basically, this tunnel needs to be able to hold up to a good beating, seeing as it will not have an expiration date.

Table 2-Cost analysis for wind tunnel

Cost Analysis							
Materials	Link	Size	Cost(each)	Amount	Total cost	Choice	
Plastic Laminate top	http://w	96x36x1(5/8)	243.29	40	9752.27965	9752.27965	
Maple tops	http://w	120x36x1(3/4)	557.03	40	22328.54755		
Partideboard tops	http://w	96x30x1(3/4)	196.55	40	7878.70675		
Section Separators							
UHMW Polyethylene	http://w	1.5x1.5x.25 L8	23.36	160	3739.5856	3739.5856	
Structural Fiberglass	http://w	1.5x1.5x.25 L10	39.56	160	6332.9626		
Brackets	http://w	1x1x.5	2.26	320	723.3921	723.3921	
Screws							
Flathead Phillips Machine Screws	http://w	1/4"-20—#3 Drive (25 pk)	13.21	20	265.32285	265.32285	
Nylon insert heavy hex Locknut	http://w	1/4"-20—.5"w 3/8"h (10 pk)	6.64	45	299.3644	299.3644	
Honeycomb							
AL Honeycomb Structure	http://w	24x48x.5	164.02	36	5918.6617	5918.6617	
Turning Vanes							
Vanes	http://w	6ft	14.67	160	2348.44695	2348.44695	
Rails	http://w	6ft	15.21	16	244.65285	244.65285	
Special Components							
Cast Acrylic	http://w	96"x48"x.25	427.82	2	892.0047	892.0047	
Fan housing	Estimated		650000	1	705250	705250	
Pitot tube	http://w	3"	11.28	4	46.0788	46.0788	
Balance system	estimated		5000	1	5425	5425	
						734904.79	tol-6000.00
						Total	740904.79

From the table above, it is obvious that there are a few things that are unaccounted for. For example, there will need to be a support system to hold the test section up, as well as the diffuser section. This will be considered an optimizing cost and will be covered under an extra amount allotted for going over the estimated cost. For this project, an excess of 6,000.00 dollars will be added to the required cost to make up for anything required that is not covered.

Future Steps

From this point, the next step to this project is to obtain funds to complete this tunnel. The layout from this [thesisproject](#) analyses the main concepts within the design of a wind tunnel. Research within the area of motor supplies for operation will need to be developed for this design criteria. A test section analysis will also need to be done for this design in order to allow for the best flow. ~~Another~~The last topic that would need to be analyzed is the balance, which will allow for all the calculations for lift, drag and yaw. Most balances for tunnels of this size are customized specifically for the tunnel. Once all these areas are covered, a ~~full~~basic wind tunnel will be ready to be built based on these specifications.

Appendix A

Matlab Coding

Pressure (test section)

```
%% Pressure Analysis ThesisProject Jordan Towles-Moore
clc
clear all
%% Variables
Vmax=(0:1:255); % max velocity in m/s
rho=1.225; %density at standard conditions in kg/m^3
p1=1; %Initial pressure before being compressed (atm)
Aratio=6; %ratio size of the open to test section
a2=24;
%% Conversion
a2=a2*(.3048^2); % conversion to meters
a1=a2*Aratio; %area of open section in m^2
p1=(p1*101.3)*1000;% conversion to N/m^2
V1=(a2.*Vmax)./a1;
%% Calculations
p2=p1+(rho./2).*((Vmax.^2)-V1.^2);
p2=(p2./1000)/101.3; %Conversion back to atm
%% Plots
plot(Vmax,p2,'r')
xlabel('Velocity (m/s)');
ylabel('Pressure (atm)');
title('Velocity vs Pressure (Test Section) - By: Jordan Towles-Moore')
```

Pressure Distribution

```
%% Pressure Analysis ThesisProject Jordan Towles-Moore
clc
clear all
%% Variables
Vmax=255; % max velocity in m/s
rho=1.225; %density at standard conditions in kg/m^3
p1=1; %Initial pressure before being compressed (atm)
Aratio=6; %ratio size of the open to test section
a2=24;
%% Conversion
a2=a2*(.3048^2); % conversion to meters^2
a1=(a2*Aratio:-.02:a2); %area of open section in m^2
p1=(p1*101.3)*1000;% conversion to N/m^2
V1=(a2.*Vmax)./a1;
%% Calculations (Nozzle)
p2=p1+(rho./2).*((V1.^2)-(Vmax.^2));
x1=(0:1/(numel(p2)-1):1);
%% Calculations (Test Section)
p=p2(numel(p2));
x2=(1:1/(numel(p2)-1):3);
%% Calculations (Diffuser Section)
a3=(a2:.02:a2*Aratio); %area of open section in m^2
V3=(a2./a3).*Vmax;
```

```

p3=p-(rho./2).*( (Vmax.^2)-(V3.^2));
x3=(3:13/(numel(p3)-1):16);
p2=(p2./1000)/101.3; %Conversion back to atm
p=(p./1000)/101.3; %Conversion back to atm
p3=(p3./1000)/101.3; %Conversion back to atm
%% Plots
plot(x1,p2,'r')
hold on
plot(x2,p,'b');
plot(x3,p3,'g')
axis([0 16 .6 1.1])
xlabel('Distance (m)');
ylabel('Pressure (atm)');
title('Distance vs Pressure (Nozzle-Test-Diffusser) - By: Jordan Towles-
Moore')

```

Headlosses

```

%% Pressure Analysis   Head loss coefficients   Jordan Towles-Moore
clc
clear all
%% givens
rho=1.225; %kg/m^3
Aratio=6; %ratio size of the open to test section
%for Aratio=(4:2:8)
a2=24;%Test section area
a2=a2*(.3048^2); % conversion to meters
a1=a2*Aratio; %area of open section in m^2
x=1;
for V_inf=(5:5:251) % max velocity in m/s
q_inf=.5 *rho * (V_inf^2);
gamma=1.4;
mu=1.783e-5;%viscosity kg/m-s

%% inputs
H_l=1; %Total pressure drop
K_l=H_l./q_inf; % Local section loss
%E_l=A1.*V_inf.*H_l; %Time rate of energy loss
q_test=q_inf; %dynamic pressure in test section

%% Test section Reynolds Number
Area_test=a2; % m^2
Area_local=sqrt((Area_test^2)*q_inf/q_test);
D_h=a1*sqrt(Area_test/pi); % Hydraulic diameter at test section
l_test=3; %length of test section
D_t=D_h; %diameter of test section vertically
Mach_test=.7;
Mach_local=(Mach_test/(1+.2*(Mach_test^2))^3)*(Area_test/Area_local)*(1+.2*(Mach_test^2))^3;
R_et=rho*V_inf*D_t/mu;
Re=R_et*(l_test/D_t)*(Area_test/Area_local)*((1+((gamma-1)/2)*Mach_local^2)/(1+((gamma-
1)/2)*Mach_test^2))^0.76;
F=1;
for z=(1:1:4)
    f=(2*log(Re*sqrt(F))-0.8)^-2;
    F=f;
end
f;
%f=(K_l*D_h)/l_test;
%% Section losses
K_test=f*(l_test/D_h);%(H_l/q_inf)*(q_inf/q_test);
%Diffusser
Ar=a1/a2;
L=120*(.3048^2);
D1=16.97056275*(.3048^2);
Theta=atan(.5*(sqrt(Ar)-1)/(L/D1));

```

```

Theta=Theta*(180/pi);
K_f=(1-(1/(Aratio^2)))*(f/(8*sin(Theta))); %friction loss
if Theta<1.5
    K_ex=.09623-.004152*Theta;% expansion loss
elseif 1.5<=Theta<=5

K_ex=.1222-.04590*Theta+.02203*(Theta^2)+.003269*(Theta^3)-.0006145*(Theta^4)-.00002800*(Theta^5)
+.00002337*(Theta^6);
elseif Theta>5
    K_ex=-.01322+.05866*(Theta);
end
% Diffuser loss coefficients
K_diffuser=K_f+K_ex;
% Corner loss coefficients
K_corner=.10+(4.55/(log(Re)^2.58));
% Screen loss Coefficient
K_screen=.785*((Re/241)+1.0)^-4+1.01;
% Honeycomb loss coefficient
L_h=8.333e-4; % honeycomb thickness in flow direction
D_h=.04166*(.3048^2);% Hydraulic diameter
K_honeycomb=.22*(.3048^2);% Referenced from book for square honeycomb
% Nozzle Loss Coefficient
L_n=6*(.3048^2); % nozzle length - meters
D_ts=D1; %open section hydraulic diameter
K_nozzle=.32*f*(L_n/D_t);
%% equations
Q(x)=(6*(.3048^2))*(4*(.3048^2))*V_inf;
E_r(x)=1/(K_test+K_diffuser+K_corner+K_screen+K_corner+K_corner+K_honeycomb+K_nozzle);
dP_test(x)=.5*K_test*rho*(V_inf^2);
dP_diffuser(x)=.5*(K_screen+2*K_corner)*rho*(V_inf^2);
dP_straight(x)=.5*(2*K_corner+K_honeycomb)*rho*(V_inf^2);
dP_nozzle(x)=.5*K_nozzle*rho*(V_inf^2);
x=x+1;
end
figure(1)
subplot(2,1,1)
plot(Q,dP_test,'r')
hold on
plot(Q,dP_nozzle,'y')
xlabel('Volumetric Flowrate(m^3/s)')
ylabel('Total Pressure Drop(Pa)')
legend('Test','Nozzle');
title('Total Pressure Drop vs Volumetric Flowrate by: Jordan Towles-Moore')
subplot(2,1,2)
plot(Q,dP_diffuser,'g')
hold on
plot(Q,dP_straight,'b')
xlabel('Volumetric Flowrate(m^3/s)')
ylabel('Total Pressure Drop(Pa)')
legend('Diffusser','Constant Area');
title('Total Pressure Drop vs Volumetric Flowrate by: Jordan Towles-Moore')
figure(2)
plot(Q,E_r,'-*)
xlabel('Volumetric Flowrate')
ylabel('Energy Ratio')
title('Energy Ratio vs Volumetric Flowrate by: Jordan Towles-Moore')
hold off

```

Boundary Layer

```

%% AE 295 Project Thesis Jordan Towles-Moore
clc; clear all;
%% inputs
g3l=.01; % .1
g3r=.5; % .5
deltanul=(0:.1:6);%change in distance across plate (0:.2:10)
nsteps=20;% gives number of iterations

```

```

%% Iterations (Modified Runge-Kutta Solution)
for n=(1:1:nsteps)
    g30=(g31+g3r)/2;
    g_3=g30;
    g_1=0;
    g_2=0;
    g0=[g_1,g_2,g_3];
    [TIME,STATE]= ode45('blfunc',deltanu1,g0);%dif equation solver
    F1=STATE(:,1);
    F2=STATE(:,2); %U
    F3=STATE(:,3);
    g_1=F1(length(F1),1);
    g_2=F2(length(F2),1);
    g_3=F3(length(F3),1);
    if g_2>1
        g3r=g30;
    else
        g3l=g30;
    end
end
end

-
nu1=transpose(deltanu1);
U=F2; % dimensionless
V2=255; % freestrem velocity in m/s
u=U.*V2; % velocity profile in x direction
V=.5.*((nu1.*u)-F1); % dimensioned

-
%% Plots
figure(1)
plot(U,deltanu1,'*k'); hold on
%plot(deltanu1,V,'.b');
xlabel('\eta');
ylabel('f'('\eta'));
title('Boundary Layer Velocity Profile Across Test Section Surface');

-
kinvisc=.00001359;
%deltanu1=deltanu1*.508; % conversion to 20 inches in meters.

-
xaxis=(0.001);
y=deltanu1./sqrt(V2./(xaxis.*kinvisc));

-
figure(2)
plot(u,y,'*y'); hold on
xaxis=(.01);
y=deltanu1./sqrt(V2./(xaxis.*kinvisc));
plot(u,y,'*r');
xaxis=(.1);
y=deltanu1./sqrt(V2./(xaxis.*kinvisc));
plot(u,y,'*g');
xaxis=(.2);
y=deltanu1./sqrt(V2./(xaxis.*kinvisc));
plot(u,y,'*b');
xaxis=(.4);
y=deltanu1./sqrt(V2./(xaxis.*kinvisc));
plot(u,y,'*m');
xaxis=(1);

```

```

y=deltanu1./sqrt(V2./(xaxis.*kinvisc));
plot(u,y,'*c');
xaxis=(2);
y=deltanu1./sqrt(V2./(xaxis.*kinvisc));
plot(u,y,'*k'); hold off
xlabel('Velocity (m/s)');
ylabel('Thickness (m)');
legend('.001','.01','.1','.2','.4','1','2');
title('Boundary Layer Thickness versus Velocity');

```

Function Call

```

function xdot =blfunc(n,g)
xdot(1) = g(2);
xdot(2) = g(3);
xdot(3) = -.5*g(1)*g(3);
xdot = xdot';
return

```

Blockage/ Corrections

```

%% Blockage Analysis
% Airfoil being used NACA 0012

-
% Panel Code in MATLAB
%
% Coded by L. sankar, April 1997
% Modified by J. Towles-Moore, March 2014
%
% Open a File and read airfoil coordinates
%
clc; clear all; close all
fid = fopen('NACA0012.txt','r') %'panel.data.txt'
alpha = fscanf(fid,'%f',1)% Angle of attack
%
% read number of points on the upper side of airfoil
%
nu = fscanf(fid,'%d',1);
%
% read number of points on the lower side of airfoil
%
nl = fscanf(fid, '%d',1);
%
% read Flag that states if this airfoil is symmetric
% if isym > 0 then airfoil is assumed symmetric
%
isym = fscanf(fid,'%d',1);
%
% Read a scaling factor
% The airfoil y- ordinates will be multiplied by this factor
%
factor=fscanf(fid,'%f',1);
-

```

```

if(isym>0)
    nl = nu;
end
%
% Allocate storage for x and y
%
x = zeros(1,100);
y = zeros(1,100);
%
% Read the points on the upper surface
%
for i = nl:nl+nu-1
    a=fscanf(fid,'%f',1);
    b = fscanf(fid,'%f',1);
    x(i) = a;
    y(i) = b * factor;
end
if isym == 0
%
% If the airfoil is not symmetric, read lower side ordinates too..
%
    for i = 1:nl
        a=fscanf(fid, '%f',1);
        b = fscanf(fid, '%f', 1);
        x(nl+1-i) = a;
        y(nl+1-i) = b * factor;
    end
else
-
    for i =1:nl
        x(nl+1-i) = x(nl-1+i);
        y(nl+1-i) = - y(nl-1+i);
    end
end
fclose(fid);
%
% Plot the airfoil on window #1
%
plot(x,y);
axis([-1 1.1 -1 1]);
n=nu+nl-2;
A=zeros(n+1,n+1);
ds=zeros(1,n);
pi=4. * atan(1.0);
%
% Assemble the Influence Coefficient Matrix A
%
for i = 1:n
    t1= x(i+1)-x(i);
    t2 = y(i+1)-y(i);
    ds(i) = sqrt(t1*t1+t2*t2);
end
for j = 1:n
    a(j,n+1) = 1.0;
    for i = 1:n
        if i == j

```

```

    a(i,i) = ds(i)/(2.*pi) *(log(0.5*ds(i)) - 1.0);
else
    xm1 = 0.5 * (x(j)+x(j+1));
    ym1 = 0.5 * (y(j)+y(j+1));
    dx = (x(i+1)-x(i))/ds(i);
    dy = (y(i+1)-y(i))/ds(i);
    t1 = x(i) - xm1;
    t2 = y(i) - ym1;
    t3 = x(i+1) - xm1;
    t7 = y(i+1) - ym1;
    t4 = t1 * dx + t2 * dy;
    t5 = t3 * dx + t7 * dy;
    t6 = t2 * dx - t1 * dy;
    t1 = t5 * log(t5*t5+t6*t6) - t4 * log(t4*t4+t6*t6);
    t2 = atan2(t6,t4)-atan2(t6,t5);
    a(j,i) = (0.5 * t1-t5+t4+t6*t2)/(2.*pi);
end
end
a(n+1,1) = 1.0;
a(n+1,n) = 1.0;
end
%
% Assemble the Right hand Side of the Matrix system
%
rhs=zeros(n+1,1);
alpha = alpha * pi /180;
xmid=zeros(n,1);
for i = 1:n
    xmid(i,1) = 0.5 * (x(i) + x(i+1));
    ymid = 0.5 * (y(i) + y(i+1));
    rhs(i,1) = ymid * cos(alpha) - xmid(i) * sin(alpha);
end
gamma = zeros(n+1,1);
%
% Solve the syetm of equations
% In MATLAB this is easy!
%
gamma = a\rhs;
cp=zeros(n,1);
cp1=zeros(n,1);
%
% Open a file to write x vs. Cp and the Loads
%
% Change the file name below, to open a new file every time
%
fid=fopen('cp4.dat','w');
fprintf(fid,' X CP\n\n');
for i = 1:n
    cp(i,1) = 1. - gamma(i) * gamma(i);
    cp1(i,1) = - cp(i,1);
    xa = xmid(i,1);
    cpa = cp(i,1);
%
% Write x and Cp to the file
%
% The xa- coordinate is the center points of panel 'i'
% Cpa is the Cp value at that point

```

```

%
fprintf(fid,'%10.4f %10.4f\n',xa,cpa);
end
%
% Open a new figure and plot x vs. Cp
%
figure(2);
plot(xmid,cp1);

-
-
%
% Compute Lift and Drag Coefficients
%
cy = 0.0;
cx = 0.0;
cm = 0.0;
% We assume that the airfoil has unit chord
% we assume that the leading edge is at i = nl;
for i=1:n
dx = x(i+1) - x(i);
dy = y(i+1) - y(i);
% xarm is the moment arm , equals distance from
% the center of the panel to quarter-chord.
xarm = 0.5 * (x(i+1)+x(i))-x(nl)-0.25;
cy = cy - cp(i,1) * dx;
cx = cx + cp(i,1) * dy;
cm = cm - cp(i,1) * dx * xarm;
end
%
% Print Lift and Drag coefficients on the screen
%
cl = cy * cos(alpha) - cx * sin(alpha)
cd = cy * sin(alpha) + cx * cos(alpha)
cm
%
% Write lift and Drag coefficients to a file
% %
% fprintf(fid,' CL          CD  CM\n');
% fprintf(fid,'%10.4f %10.4f %10.4f\n', cl,cd,cm);
% fclose(fid);

-
%% Thom's Solid blockage correction
thick=.061*2;
Mach=340.29; %mach speed in m/s
c=.6096;% chord length in m, (~2 ft.)
s=1.8288; % span in m, (~6 ft)
K_1=.7*thick*c*s;
h=1.2192; % height of the tunnel in meters
w=1.8288; % width of the tunnel in meters
C=w*h; % Wind tunnel test section area (m)
Epsilon=(K_1)/C % solid blockage correction
sigma=((pi^2)/48)*((c/h)^2);
delta_alpha=((6*sigma)/(pi^2))*cl
V=255; %Freestream velocity (m/s)
aoa_corr=(57.3*sigma/(2*pi))*(cl+(4*cm))
% Must be kept less than 3.2 ft, or .97536 meters.

```

```
% Glauert's Solid Blockage Increments
h2=1.2164;
Eps=0;
radius=.061;
while Eps<.0133
    Eps=((pi.^2)./3)*((radius.^2)./(h2.^2));
    radius=radius+.001;
end
radius=radius-.002
Eps=((pi.^2)./3)*((radius.^2)./(h2.^2))
Eps=Eps *100; % conversion to percentage
```

Works Cited

- Aerolab. (2012). *Aerolab*. Retrieved March 03, 2013, from <http://www.aerolab.com/index.html>
- Anderson, J. D. (2005). *Fundamentals of Aerodynamics*. Maryland: McGraw-Hill.
- Anderson, J. D. (n.d.). *Modern Compressible Flow*. New York: McGraw-Hill.
- Baals, D. D. (2009, April 30). *Whirling Arms and the First Wind Tunnels*. Retrieved February 08, 2013, from <http://www.grc.nasa.gov/WWW/K-12/WindTunnel/history.html>
- Barlow, J. B. (1965). *Low-Speed Wind Tunnel Testing*. New York: John Wiley & Sons.
- Carlone, G. (2008, May). *Building a Wind Tunnel*. Retrieved March 02, 2013, from Tom Carlone: <http://www.tomcarlone.com/WindTunnelProject/WindTunnelReport.pdf>
- Kotwani, K. (2003). *Wind Tunnel Performance analysis*. Retrieved 04 23, 2013, from <http://www.casde.iitb.ac.in/store/training/2003/kailash-wt-perf-analysis.pdf>
- NASA Observatorium. (1995). *Parts of a wind tunnel*. Retrieved March 04, 2013, from Flights of Inspiration: <http://www.fi.edu/flight/first/tunnelparts/index.html>
- Rae, A. P. (1984). *Low Speed Wind Tunnel Testing*. Seattle: John Wiley and Sons.
- Sankar, L. (1997, April). Panel Code in Matlab.
- Vertical Wind Tunnel Corporation. (n.d.). Retrieved March 03, 2013, from Vertical Wind: <http://www.verticalwind.com/circuits.html>