AC 2012-3461: UNDERGRADUATE RESEARCH ON CONCEPTUAL DESIGN OF A WIND TUNNEL FOR INSTRUCTIONAL PURPOSES

Peter John Arslanian, NASA/Computer Sciences Corporation

Peter John Arslanian currently holds an engineering position at Computer Sciences Corporation. He works as a Ground Safety Engineer supporting Sounding Rocket and ANTARES launch vehicles at NASA, Wallops Island, Va. He also acts as an Electrical Engineer supporting testing and validation for NASA's Low Density Supersonic Decelerator vehicle. Arslanian has received an Undergraduate Degree with Honors in Engineering with an Aerospace Specialization from the University of Maryland, Eastern Shore (UMES) in May 2011. Prior to receiving his undergraduate degree, he worked as an Action Sport Design Engineer for Hydroglas Composites in San Clemente, Calif., from 1994 to 2006, designing personnel watercraft hulls. Arslanian served in the U.S. Navy from 1989 to 1993 as Lead Electronics Technician for the Automatic Carrier Landing System aboard the U.S.S. Independence CV-62, stationed in Yokosuka, Japan. During his enlistment, Arslanian was honored with two South West Asia Service Medals.

Dr. Payam Matin, University of Maryland, Eastern Shore

Payam Matin is currently an Assistant Professor in the Department of Engineering and Aviation Sciences at the University of Maryland Eastern Shore (UMES). Matin has received his Ph.D. in mechanical engineering from Oakland University, Rochester, Mich., in May 2005. He has taught a number of courses in the areas of mechanical engineering and aerospace at UMES. Matin's research has been mostly in the areas of computational mechanics and experimental mechanics. Matin has published more than 20 peer-reviewed journal and conference papers. Matin worked in auto-industry for Chrysler Corporation from 2005 to 2007.

Undergraduate research on Conceptual design of a wind tunnel for Instructional purposes

Abstract

Senior students in the engineering programs are challenged to thoroughly apply their learned engineering knowledge and research skills toward design and implementation of a challenging senior design project. A wind tunnel is often used in mechanical or aerospace engineering programs as a laboratory instrument to gather experimental data for investigation of fluid flow behavior. The authors have conducted research to implement a comprehensive design of a small size inexpensive wind tunnel for instructional purposes {overall length: 1.8105m, maximum diameter (contraction nozzle): 0.375m, working section dimensions: 0.25m in length X 0.125m in diameter}. The objectives of this research project are to engage an undergraduate engineering student: 1) to design a well-structured wind tunnel model by means of fluid mechanics fundamentals and simulation software, and 2) to develop wind tunnel experiments such as flow visualization, lift and drag measurement around different geometries including NACA airfoils. The wind tunnel designed is an open-loop circuit contained of three basic sections: the contraction nozzle, the working or experiment section, and the diffuser nozzle. Fluid thrust is delivered by an axial fan attached to the end of the diffuser nozzle. The Student was able to obtain the geometric properties of the contraction nozzle and diffuser nozzle by fluid mechanics theories governing a constant pressure decrease, or increase respectively. The working section is a duct of constant area that maintains a uniform fluid velocity. The student built the solid models of the contraction nozzle, working section and diffuser nozzle for flow simulation. The performance of the wind tunnel designed has been verified through CFD-based simulation. The data collected from the simulation results indicate that a uniform laminar flow is maintained in the working section as desired. Different testing models such as sphere and infinite wing have been included in the simulation to characterize the performance of the wind tunnel during testing. The simulation results are promising. Extensive engineering knowledge acquired throughout the course of undergraduate study is applied by the student. Significant understanding of shearing forces developed in the boundary layer has been gained throughout this undergraduate research.

Introduction

Wind tunnels, beginning from the rude but arguably famous Wright Brothers device circa 1903 to the great research facilities funded by NASA, have uncovered the dynamics existing between fluid and solid objects. The Wright Brothers recognized that by blowing air past a model of their aircraft in a device that could mimic conditions favorable to flight, they could ultimately deliver the answer sought after by man for millennia, the ability to fly. The Wright Brothers prevailed, and the history of the wind tunnel as an integral component to aerodynamic research was documented.

What the Wright Brothers failed to recognize, was that the complexity of flight and those to mimic the conditions in a device are equally complicated. It is the perplexing realization that the integral geometry to produce the desired fluid velocity in the wind tunnel, introduces flow regimes that are not present in a natural laminar flow. Such integral component responsible is the contraction nozzle of the wind tunnel.

Modern wind tunnel design progressed sharply thereafter, and researchers set conditions on the quality of the flow present in the experiment section. The usual condition was that the velocity of the flow regime at the end of the contraction nozzle must be fairly uniform^[13,3]. Hsue-Shen Tsien stated that the curvature of the contraction nozzle is responsible for “regions of adverse pressure gradient” that may lead to the boundary layer separating from the wall of the contraction^[13,3]. The boundary layer separation introduces turbulence to the flow, and the presence of the adverse pressure gradient is evidence of a non-uniform velocity. Tsien based his design by applying the fundamental theorem of fluid mechanics, the continuity equation, to derive a stream function that would predict the contraction curvature.

Beginning in the 1960s with the advent of powerful computing machines and the computational methods necessary to process mathematical operations, more variables were considered during contraction nozzle design. The new variables governing design, in addition to flow uniformity, are the overall length and the contraction ratio (CR). This ratio is defined as the ratio of cross sectional areas of the entrance to the exit of the contraction nozzle. In reference to Tsien’s work, Morel proposed that once the CR and length variables are selected, the contraction contour is established by matching two cubic arcs at an inflection point^[3,10]. Mikhail progressed upon Morel’s work with the proposition that the overall length of the contraction can be controlled by optimizing the design. For once, the optimum length is achieved for a desired CR only then is the dynamic load and boundary layer growth at their minimalist values^[3,8,11].

During the later part of the 20th century, only a handful of universities possessed the funding necessary to build and operate viable wind tunnels that they could in turn utilize to reinforce the curriculum in their mechanical and aerospace programs. Fortunately during this same time, the power of computing machines and the computer languages necessary to program the foundational mathematics started increasing exponentially. Conversely to this trend, the cost to develop such systems was decreasing in an equally opposite trend. The conditions were met for an economical study of fluid flow prediction to evolve into the field known as computational fluid dynamics (CFD). With the help of CFD software and its complimentary CAD graphic interface, one can accurately design and evaluate the flow regimes of a highly capable wind tunnel device.

The objective of this work is to engage an undergraduate engineering student to utilize the fundamentals of fluid mechanics along with CFD tools to design a small size low-cost wind tunnel for instructional purposes.

Nomenclature

AR = area ratio

CR = contraction ratio

$P_{static1}$ = static pressure prior to the contraction nozzle

$P_{static2}$ = static pressure at the entrance to contraction nozzle

$P_{static3}$ = static pressure in the working section

$P_{static4}$ = static pressure at the exit of the diffuser nozzle

$P_{ambient}$ = ambient pressure

P_{stag1} = stagnation pressure

U_1 = velocity prior to the contraction nozzle

U_2 = velocity at the entrance to the contraction nozzle

U_3 = velocity in the working section

U_4 = velocity at the exit of the diffuser nozzle
 A_2 = cross sectional area at the entrance to contraction nozzle
 A_3 = cross sectional area of the working section
 A_4 = cross sectional area of the exit of the diffuser nozzle
 R_2 = radius of the entrance to the contraction nozzle
 R_3 = radius of the working section
 R_4 = radius of the exit of the diffuser nozzle
 L = length of the diffuser
 θ = equivalent cone angle of the diffuser nozzle
 θ_{CF} = cross flow angle
 θ_{UF} = up flow angle
VFR = volume flow rate
 ΔP_r = required pressure recovery of the fluid pump
 U = x-component of velocity
 V = y-component of velocity
 W = z-component of velocity
 x = x- coordinate
 y = y-coordinate
 z = z-coordinate
 k = stream function's constant of integral
 ρ = fluid density
 μ = fluid viscosity
 g = gravitational acceleration
 h_l = head loss
 K = loss coefficient
 ΔP = pressure loss
 K_{ws} = loss coefficient of the working section
 K_{nt} = loss coefficient of the contraction nozzle
 K_d = loss coefficient of the diffuser nozzle
 K_f = loss coefficient of the diffuser nozzle due to friction
 K_{ex} = loss coefficient of the diffuser nozzle due to expansion
 f = friction factor
 f_{ws} = friction factor of the working section
 f_{cn} = friction factor at the entrance to the contraction nozzle
 f_{avg} = average friction factor of the contraction nozzle
 D = hydraulic diameter
 D_{ws} = hydraulic diameter of the working section
 L_{ws} = length of the working section
 L_n = length of the contraction nozzle
 ε = roughness
 A_c = cross sectional area
 Per = wetted perimeter
 Re = Reynolds number
 V_{avg} = average velocity
 BR = blockage ratio

Problem Statement

The purpose of this study is to engage an undergraduate engineering student to design a wind tunnel laboratory that will aid aerospace / mechanical curriculums. The wind tunnel shall be designed so that an undergraduate engineering student majoring in an aerospace or mechanical discipline can conduct experiments that enforce the fluid flow theories he / she will learn during the undergraduate study. The design needs are as follows:

- The maximum fluid velocity in the working section of the wind tunnel is 25m/s (Sub-Sonic Flow).
- The flow quality allows for basic observation of fluid flow phenomena.
- Lab experiments that aid aerodynamics or fluid mechanics courses are designed and developed. These labs may include fluid flow development, boundary layer visualization, laminar/ turbulent flow visualization, flow around a cylinder, sphere or a wing of infinite length, and etc.

The design constraints are as follows:

- The design fits in the existing lab space; the length is not to exceed 3 meters.
- Cost to manufacture a prototype of the successful design is one-tenth in comparison to a commercially available device roughly priced \$30,000.
- Flow Quality Standards enforced in the working section:
 - Dynamic Pressure difference $< 1\%$ from the mean.
 - Cross Flow/ Up Flow Angles are held to 0.1° with a max of 0.2° [2,7,11].
- No modifications to the existing lab space structure.
- Power available is limited to 115 VAC at 20 Amps.

Initial Design

In an open circuit design, the fluid is taken from an ambient state, far away from the entrance to the contraction, and is accelerated to the desired velocity by the contraction nozzle. The fluid velocity is then maintained the length of the working section. The flow is then returned to near ambient pressure in the diffuser section. Finally, the fluid is exhausted into the open environment.

As shown in Figure 1, the major component subassemblies of the wind tunnel are:

- *Compressor*, referred to as the Contraction nozzle - a section that accelerates the fluid to the desired velocity.
- *Test Section*, referred to as the Working area - a section with constant cross sectional geometry to conduct experiments.
- *Diffuser*, referred to as the Diffuser nozzle – a section that returns the flow to near ambient pressure.

Figure 1 Basic Components of a Wind Tunnel

Two different configurations are considered during the initial research. A blow down configuration as seen in Figure 2, and a suck down configuration as seen in Figure 3. The Suck down configuration is chosen over the blown down by virtue of the length savings incurred without the added, larger centrifugal pump, and the associated diffuser section (Area 1). It is worth noting in each of these figures that the optimal length is displayed. This length is determined in the first phase of the design approach.

Figure 2 Blow Down Configuration

Figure 3 Suck Down Configuration

Design Approach

The design is to progress in the following 3 phases; with a decision at the conclusion of each phase to either reiterate or proceed:

Phase 1: Design to meet the length requirement.

Phase 2: Design for the required pump considering the constraints with the assumption of ideal flow.

Phase 3: Design based on flow quality constraints with the assumption of viscous flow.

Phase 1:

The purpose of the first phase is to meet the length requirement.

A constant cross sectional area is assigned to the working section. The length (and also diameter) of the working section are assigned based on the design needs as shown in Figure 4. The length of the contraction nozzle is set as shown in Figure 4. The diffuser nozzle length is dependent on two variables. The first variable is the diameter of the working section and the second is the area ratio (AR) of the diffuser nozzle set by the designer. The typical AR is around 3 in conjunction with an equivalent cone angle of 3° [2,7,11]. Therefore the length for the diffuser nozzle is found by following the formulations presented below referencing Figure 5.

The AR is set at 3:

$$AR = \frac{A_4}{A_3} = 3 \quad (1)$$

A_3 , the area of the octagonal cross section at the working section is known since its radius is set by the designer:

$$A_3 = 8 \tan \frac{\pi}{8} R_3^2 \quad (2)$$

Figure 4 Phase 1 of Design

Figure 5 Diffuser Length Calculation

Combining (1) and (2), the area of the diffuser nozzle at the exit A_4 is determined. Since this cross section is circular, its radius is then calculated according to:

$$R_4 = \sqrt{\frac{A_4}{\pi}} \quad (3)$$

Setting a standard equivalent cone angle for the diffuser nozzle, the length of the diffuser is calculated based on Figure 5 as:

$$L = \frac{R_4 - R_3}{\tan\theta} \quad (4)$$

The first phase in the design process is concluded by summing the lengths and verifying that the overall length meets the constraints. If the overall length calculated meets the requirements, the design proceeds to phase 2. If not, the design reiterates with a reduced radius in the working section until the constraints are met.

Before moving on to the next phase, the radius of the entrance to the contraction nozzle is also determined with a similar process. It is worth noting that the contraction nozzle retains an octagonal geometry the entire length. Therefore:

$$A_2 = 8 \tan \frac{\pi}{8} R_2^2 \quad (5)$$

The contraction nozzle CR is set by the designer, with CR ratios optimum between the values of 7-12^[2,7,11]. The CR is set at 9, therefore:

$$CR = \frac{A_2}{A_3} \quad (6)$$

Once again A_3 , the area of the octagonal cross section at the working section, is known, so the entrance area to the contraction nozzle and consequently the associated radius R_2 are calculated:

$$R_2 = \sqrt{\frac{A_2}{8 \tan \frac{\pi}{8}}} \quad (7)$$

Phase 2:

The purpose of the second phase is to evaluate the fluid flow assuming a two-dimensional, steady, incompressible and inviscid flow, where the frictional forces are negligible, in order to determine if the required pump can meet the cost constraint.

Figure 6 is used during this phase of design. The expected velocity in the working section is set at $U_3 = 25\text{m/s}$, as a result the volume flow rate is known on its entrance and exit; additionally the working section shares coincident surfaces with the contraction nozzle and diffuser nozzle. Assuming that the fluid velocity remains constant as it proceeds through the working section, the expected velocity at any cross section in the wind tunnel can be solved for based on the continuity equation. The cross sectional areas for the contraction nozzle entrance and the diffuser nozzle exit are known by the previous design phase. Therefore, the unknown velocities at the entrance to the contraction nozzle and at the exit of the diffuser nozzle are solved for.

$$U_2 = \frac{U_3 A_3}{A_2} \quad (8)$$

$$U_4 = \frac{U_3 A_3}{A_4} \quad (9)$$

These velocities are then used to predict the pressure at the respective control surface.

Figure 6 Inviscid Fluid Flow Model

With the stated assumption that the flow is steady, incompressible, inviscid and the frictional forces are negligible, Bernoulli's equation is valid. The dashed centerline in Figure 6 is chosen as the streamline to relate the flow between any two points. Prior to the entrance of the contraction nozzle, the ambient velocity U_1 is zero, and the pressure is the ambient pressure. Thus this point is taken as the stagnation point.

$$P_{static1} = P_{ambient} = 101.325kPa = P_{stag1} \quad (10)$$

Knowing the stagnation pressure, Bernoulli's equation is implemented to calculate the expected pressure at the entrance to the contraction nozzle, in the working section, and at the very exit of the diffuser nozzle, respectively, as follows:

$$P_{static2} = P_{stag1} - \rho \frac{U_2^2}{2} \quad (11)$$

$$P_{static3} = P_{stag1} - \rho \frac{U_3^2}{2} \quad (12)$$

$$P_{static4} = P_{stag1} - \rho \frac{U_4^2}{2} \quad (13)$$

It should be noted that the Bernoulli's equation is only valid prior to the fluid pump placed at the exit of the diffuser nozzle. The pressure prior to the fluid pump is the same as pressure at the exit of the diffuser nozzle. The pressure after the pump is the ambient pressure. Since the pressures are known on both sides of the pump, the required pressure recovery provided by the pump is calculated. The fluid pump should also provide the volume flow rate (VFR) required by the wind tunnel, and match with the diameter of the diffuser nozzle. Based on these factors, the fluid pump has been selected with the following specifications:

$$\text{VFR} = 683.345 \text{ Cubic feet per minutes}$$

$$\Delta P_r = 0.16703 \text{ Column inches of water}$$

$$\text{Diameter} = 8.52 \text{ in}$$

The estimated cost for such pump is reasonable. It should be noted that these specifications correspond to ideal flow where there is no pressure loss. These specifications are corrected in coming sections to take the effect of losses into consideration.

Phase 3:

The purpose of the third phase is to design the contraction contour, evaluate the flow as viscid, and evaluate if the flow meets the defined constraints with CFD simulation.

Phase 3-1: Contraction Contour Design

The purpose of this section is to design the contraction contour.

As shown in Figure 7, the major criterion for the design of the contraction contour is that the velocity of the fluid flow at the exit of the contraction nozzle (or at the entrance of the working section) should be uniform. Uniform velocity is defined and checked by the flow quality constraints bulleted in the introduction. Smooth transition from the contraction contour with its varying cross sectional area, to the working section with its constant cross sectional area, facilitates uniform flow in the working section. This smooth variation translates to zero slope at the exit of the contraction nozzle, where the contraction contour coincides with the working section. Similarly, the slope at the entrance of the contraction nozzle should be zero as well. In the literature ^[3,5,6,8,9], the contraction contours are formed by matching two cubic arcs at an inflection point. Careful consideration is taken in constructing such curves, since an excess of wall curvature results in regions of adverse pressure gradients developed near the contraction's entrance and exit. These adverse pressure gradients produce a thickening boundary layer that can lead to the layer's separation. The result is degraded flow uniformity at the exit of the nozzle.

Figure 7 Contraction Contour approach by Morel

To avoid the complexity associated with the previous works cited earlier, the authors have developed a simple innovative approach to design the contraction contour. The main steps of the proposed method are as follows:

- Derive a stream function that predicts the contraction contour.
- Assign a suitable constant of integration in order to plot the stream line representing the wall.
- Choose (x, y) coordinates of a point as the inflection point along the stream line.
- Design the cubic arcs by computational interpolation.

First, a stream function needs to be derived. The continuity equation in differential form when the flow is assumed to be two-dimensional, steady, and incompressible is given as:

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0 \quad (14)$$

The x-component of velocity U is assumed to increase in a linear trend as a function of x only, from the entrance to the exit of the contraction nozzle.

$$U = U_1 + \left[\frac{U_2 - U_1}{L} \right] x \quad (15)$$

At any arbitrary point, the velocity vector is always tangent to the stream lines.

$$\frac{dy}{dx} = \frac{V}{U} \quad (16)$$

Combining equations (14), (15), and (16), the equations of the stream lines are derived by solving for y in terms of x as ^[4]:

$$y = \frac{k}{U_1 + \left(\frac{U_2 - U_1}{L} \right) x} \quad (17)$$

The constant of integration is set to plot a streamline that approximates the wall contour of the contraction nozzle as shown in Figure 8. As it is seen, the transition at the exit and particularly at the entrance is not smooth. Thus, this curve needs to be modified. To this end, an inflection point is selected on the curve arbitrarily around the midpoint of the contraction nozzle length. Two cubic arcs need to be constructed. The first cubic is from the entrance to the inflection point and the second cubic is from the inflection point to the exit.

Figure 8 Contraction Streamline

Figure 9 Secondary Inflection Point

Figure 10 Contraction Contour Family

In order to produce the desired curvature for each cubic, a second inflection point is necessitated for each section. As shown in Figure 9, such point could be chosen by rotating the radial value of the contraction nozzle with any one of the 30°, 45° or 60° angles. The positions of the secondary inflection point, primary inflection point, and entrance point along with zero slope condition will produce a series of contours known as the first cubic to be evaluated. The same can be said for the second cubic. Such contours are constructed using MATLAB software utilizing the spline function as shown in Figure 10. Only the contour with a secondary inflection point defined by the 45° radial coordinate and a primary inflection point at 0.2m (shown in Figure 10) is used in the design and validation efforts.

Phase 3-2: Loss Calculations

The purpose of this section is to evaluate the flow as viscid to calculate the successive pressure losses that eventually need to be balanced by the fluid pump as a realistic pressure recovery.

As shown in Figure 11, the wind tunnel is made of three main sections. Each section features different geometries, and different velocity conditions. The pressure loss is calculated in each section independently. The sum of each section's pressure loss represents the total pressure loss. Such pressure loss is then subtracted from the pressure at the exit of the diffuser, establishing the actual pressure recovery required by the fluid pump. In general, the head loss is given based on following formulas:

$$h_l = K \frac{V^2}{2g} = \frac{\Delta P}{\rho g} \quad (18)$$

Thus, the pressure loss ΔP may be calculated using:

$$\Delta P = K \frac{V^2}{2} \rho \quad (19)$$

where K is the loss coefficient, V is the average fluid velocity, and ρ is the fluid density. In general, the loss coefficient K is a function of friction factor and the geometry of the section.

Section Losses + Pressure recovery = Axial Fan Requirements

Figure 11 Viscous Model Losses

The loss coefficient of the working section is calculated as follows ^[2,7,11]:

$$K_{ws} = f_{ws} \frac{L_{ws}}{D_{ws}} \quad (20)$$

where f_{ws} , L_{ws} and D_{ws} are the friction factor, the length and the hydraulic diameter of the working section, respectively. The friction factor may be calculated using the Colebrook equation ^[4] as:

$$\frac{1}{\sqrt{f}} = -2 \log_{10} \left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re \sqrt{f}} \right) \quad (21)$$

With the assumption of zero roughness ($\varepsilon = 0$) in the interior of the working section, the Colebrook equation simplifies to:

$$\frac{1}{\sqrt{f}} = 2 \log_{10}(Re \sqrt{f}) - 0.8 \quad (22)$$

where Re is Reynolds number, and is given as:

$$Re = \frac{\rho V_{avg} D}{\mu} \quad (23)$$

where ρ and μ are the density and viscosity of the flow, respectively. V_{avg} is the fluid average velocity in the working section, which is set at 25m/s by design. D is the hydraulic diameter, which is defined as:

$$D = \frac{4A_C}{Per} \quad (24)$$

where A_C is the cross sectional area, and Per is the wetted perimeter. Since the geometry of the working section is known, its hydraulic diameter is calculated using equation (24). Then the associated Reynolds number Re is calculated based on equation (23). With the Reynolds number known, equation (22) is numerically solved for the friction factor in the working section. Once friction factor f_{ws} is known, equation (20) is used to calculate the loss coefficient of the working section. Subsequently, the pressure loss in the working section is calculated using equation (19).

The loss coefficient in the contraction nozzle can be estimated based on Wattendorf's Approximation as ^[2,7,11]:

$$K_{nt} = 0.32f_{avg} \frac{L_n}{D_{ws}} \quad (25)$$

where f_{avg} , L_n and D_{ws} are the average friction factor, the length of the contraction nozzle and the hydraulic diameter of the working section, respectively. As shown in equation (26), the average friction factor f_{avg} is estimated based on the previously-determined friction factor of the working section f_{ws} , and the friction factor at the entrance of the contraction nozzle f_{cn} as:

$$f_{avg} = \frac{f_{ws} + f_{cn}}{2} \quad (26)$$

The friction factor at the entrance of the contraction nozzle f_{cn} is determined by solving equation (22) with the assumption of zero roughness ($\varepsilon = 0$) in the interior of the contraction nozzle. For this purpose, the Reynolds number is evaluated at the entrance of the contraction nozzle using equation (23). It should be noted that the hydraulic diameter is also evaluated at the entrance using equation (24) for the Reynolds evaluation. Once f_{cn} is calculated, f_{avg} is calculated, and then the loss coefficient of the nozzle K_{nt} is estimated. Next, the pressure loss associated with contraction nozzle needs to be estimated. For the contraction nozzle, equation (19) is still valid if the average velocity between the entrance and exit is substituted.

The loss in the diffuser nozzle is not only as a result of friction but also as a result of expansion as well. Thus for the diffuser, two loss coefficients are defined. One loss coefficient is associated with friction with the assumption of zero roughness ($\varepsilon = 0$) in the interior of the diffuser nozzle, and the other is associated with expansion^[2,7,11]:

$$K_d = K_f + K_{ex} \quad (27)$$

where K_f and K_{ex} are given as:

$$K_f = \left(1 - \frac{1}{A_R^2}\right) \frac{f_{ws}}{8\sin\theta} \quad (28)$$

$$K_{ex} = K_e(\theta) \left(\frac{A_R - 1}{A_R}\right)^2 \quad (29)$$

where A_R and θ are the area ratio and equivalent cone angle of the diffuser nozzle, respectively. Known values have been assigned to these variables during previous phases of the design ($A_R = 3$ and $\theta = 3^\circ$). For $1.5^\circ < \theta < 5^\circ$, $K_e(\theta)$ may be estimated as^[2,7,11]:

$$K_e(\theta) = 0.1709 - 0.1170\theta + 0.03260\theta^2 + 0.001078\theta^3 - 0.00090760\theta^4 - 0.00001331\theta^5 + 0.00001345\theta^6 \quad (30)$$

With approximating $K_e(\theta)$ based on equation (30), K_{ex} and K_f are estimated using equation (29) and (28) respectively. Then, the loss coefficient of the diffuser is calculated using equation (27). Subsequently, the pressure loss is calculated in the diffuser nozzle based on equation (19).

Since the pressure loss in all three sections has been estimated, the actual pressures at different sections of the wind tunnel are revised as shown in Figure 12.

Figure 12 Viscid Model

Based on the total pressure loss, the pressure recovery required by the fluid pump is calculated. Accordingly, the actual cost-effective fluid pump is selected with following specifications:

$$\begin{aligned} \text{VFR} &= 683.345 \text{ Cubic feet per minutes} \\ \Delta P &= 0.38828 \text{ Column inches of water} \\ \text{Diameter} &= 8.52 \text{ in} \end{aligned}$$

Phase 3-3: CFD Simulation

The purpose of this section is to evaluate the flow with CFD simulation

The solid model of the wind tunnel designed is built using SolidWorks 2010. As shown in Figure 13, the final assembly of the wind tunnel is made of three solid model parts: the contraction nozzle, working section, and diffuser nozzle. The fluid flow simulation is performed with SolidWorks Flow Simulation 2010. The following preprocessing steps are taken to set up the model before the simulation starts:

- The expected ambient pressure value at the entrance to the contraction nozzle is assigned.
- The expected static pressure at the very end of the diffuser nozzle is assigned.
- Wall roughness is assigned zero.
- Fluid properties are assigned.

Figure 13 Solid Model Assembly of Wind Tunnel

After the model sets up, the simulation can run.

Flow Quality

The flow quality produced by the design is investigated in a vertical plane normal to the centerline. The vertical plane is evaluated by utilizing a software tool to produce a cut plot, a thin slice of the flow regime. This cut plot reveals the static pressure present in the selected plane. In Figure 14, the static pressure cut plot reveals three distinct regions of near uniform pressure. Such uniform pressure is desired, leading to a nearly uniform fluid velocity in the working section. Based on the static pressure values, the dynamic pressures are also calculated. The

variation of each regions dynamic pressure to the mean is found to be less 1%, satisfying the defined constraint.

Figure 14 Pressure Cut Plot

The main direction of the flow is considered along the wind tunnel axis, y direction. Taking a vertical plane, the fluid velocity along z is compared against the y-direction velocity. The comparison is conducted by calculating the cross flow angle. The lower this angle is, the lower cross flow exists, and the higher quality of flow is achieved. The same concept is applied to up flow. Figure 15 shows how cross flow and up flow are evaluated based on cross flow angle and up flow angle, respectively.

Figure 15 Cross Flow and Up Flow

$$\tan\theta_{CF} = \frac{|U|}{|V|} \quad (31)$$

$$\tan\theta_{UF} = \frac{|W|}{|V|} \quad (32)$$

Figure 16 and Figure 17 show the fluid velocity along x and z directions on a vertical and horizontal plane, respectively. As it can be seen, these two components of fluid velocity are nearly zero. That is a good indication of low cross flow and up flow. Figure 18 and Figure 19 are the close up of the fluid velocity along x and z directions in the working section of the wind tunnel. The fluid velocity along the main direction of the flow (y direction) is also included in these figures. The cross flow angle, and up flow angle are calculated based on the values observed in these figures. The angles estimated are fairly small meeting the associated design constraints. These two figures indicate negligible cross flow and up flow, which is desirable.

Figure 16 x-Component of Velocity

Figure 17 z-Component of Velocity

Figure 18 Cross Flow Illustrated

Figure 19 Up Flow Illustrated

Simulated Lab Experience

Two examples of laboratory experiments are simulated to gauge the performance of the wind tunnel designed with a model placed in the working section. The two experiments are the flow around a cylinder and the flow around an infinite airfoil. Before inserting a model in the working section, the blockage ratio of the object is determined. As a rule of thumb, the ratio should be less than 7.5%^[4]. The blockage ratio (BR) is calculated as follows:

$$BR = \frac{100}{A_{\text{working section}}} [A_{\text{model}}(\text{model scale})^2] \quad (33)$$

Flow around a Sphere

Based on a blockage ratio of 6.6%, a model sphere is placed in the wind tunnel designed. The flow observed around the sphere matches well with the theory^[1]. As shown in Figure 20, the high velocity regions are formed on the top and bottom of the sphere. The low velocity region is depicted on the left hand side of the sphere, where the stagnation point is formed. The velocity field also illustrates the boundary layer separation. The angle at the separation point is measured and compared against the theory. In such experiment, the effect of Reynolds number can be studied as well.

Figure 20 Velocity Around a Sphere

Flow around an airfoil

Based on a blockage ratio of 1.2%, a solid model of a NACA series 2412 airfoil is built. The airfoil coordinates were initially generated with a MATLAB program developed by Phillips^[12] and then imported into SolidWorks. Figure 21 illustrates the pressure distribution developed around the airfoil as the angle of attack is changed in increments of two degrees from zero to twelve degrees. The pressure distribution observed is reasonable. The low pressure area is formed on the top while the high pressure area is formed on the bottom of the airfoil. The lift value is returned by the simulation software as the force in the z direction. The lift value is then plotted against the angle of attack and compared to the theoretical results generated by the MATLAB program. As depicted in Figure 22, the lift observed is in a good agreement with theoretical prediction with the corrected velocity.

Figure 21 Pressure Around Airfoil

Figure 22 Lift versus Angle of Attack

As the angle of attack increases, the blockage ratio increases and the mean fluid velocity decreases in the working section. That leads to a drop in the amount of the lift around 8 degrees of angle of attack.

Student Learning Outcomes

- 1- The project exposed a student to design process of a real world fluid system with realistic design requirements and design constraints.
- 2- The student developed a logical 3-phase design approach to design the main components of the wind tunnel.
- 3- The student learned how to apply the fundamentals of fluid mechanics to design for the main components such as contraction nozzle, working section and diffuser.
- 4- The student learned how to build a solid model of the system, and how to run a flow simulation to verify the design.
- 5- The student gained hands-on experience working with different modern math and engineering software such as MATLAB, SolidWorks, SolidWork Flow and etc.

Conclusions

Fluid mechanics fundamentals along with the state-of-art CFD simulations have been utilized to design a small size wind tunnel for instructional purposes. The CFD investigations have validated the design since a relatively uniform dynamic pressure has been obtained in the working section of the wind tunnel, confirming a nearly uniform fluid velocity. The flow quality is shown to be acceptable since minor cross flow and up flow angles have been observed. Flow around a sphere and an infinite airfoil have been simulated as instructional experiments. The performance of the wind tunnel under such experiments is adequate. The cost estimated is well below the commercial systems available in the market. Valuable levels of knowledge have been gained through this undergraduate research in the areas of fluid mechanics, CFD simulations, computational methods, solid modeling and design.

References

1. Anderson, John D Jr. Fundamentals of Aerodynamics 5Th Ed. McGraw-Hill. New York, NY. 2011.
2. Barlow, Jewel B. Rae Jr, William H. Pope Alan. Low Speed Wind Tunnel Design 3rd Edition John Wiley and Sons Inc. New York, NY. 1999.
3. Borger, G. G. The Optimization of Wind Tunnel Contractions for the Subsonic Range. NASA TTF-16899.
4. Cengel, Yunus A. Cimbala, John M. Fluid Mechanics Fundamentals and applications. McGraw Hill. New York, NY. 2006.
5. Chmielewski, G. E. Boundary Layer Considerations in the Design of Aerodynamic Contractions. Journal of Aircraft, Vol. 11, Aug. 1974, pp. 435-438.
6. Jones, D. J., South, J. C., and Klunker, E. B. On the Numerical Solution of Elliptic Partial Differential Equations by the Method of Lines. Journal of Computational Physics, Vol. 9, June 1972, pp. 496-527
7. Lindgren, Bjorn. Johansson, Arne. V. Design and evaluation of a low-speed Wind-tunnel with expanding corners. Technical report, TRITA-MEK 2002:14. Dept. of Mechanics, KTH, SE-100 44 Stockholm, Sweden.
8. Mikhail, M. N. Optimum Design of Wind Tunnel Contractions. American Institute of Aeronautics and Astronautics Journal., Vol. 17, NO. 5, pg 471-477, May 1979. Article NO. 78-819
9. Mikhail, M.N. Optimum Design of Internal Flow Passages with Specific Reference to Wind Tunnel Contractions. Ph.D. Thesis, Carleton University, Ottawa, Canada, Oct. 1976.

10. Morel, T. Comprehensive Design of Axisymmetric Wind Tunnel Contractions. *Journal of Fluids Engineering.*, ASME, Vol. 97, pg 225-233, June 1975.
11. Owen, Kevin. F. Owen, Andrew. K. Measurement and assessment of wind tunnel flow quality. *Progress in Aerospace Sciences Journal.*, Vol. 44, pg 315-348, July 2008.
12. Phillips, Ben. Estimation of Lift by thin Airfoil theory. MATLAB program. March 2011, University of Maryland Eastern Shore.
13. Tsien, Hsue-Shen. On the Design of the Contraction Cone for a Wind Tunnel. *Journal of the Aeronautical Sciences*, February 1943.