Addis Ababa University

Addis Ababa Institute of Technology

School of Mechanical and Industrial Engineering

Parametric Study of a Diffuser for HAWT Power Augmentation

A Thesis Submitted to the School of Graduate Studies of Addis Ababa University in Partial Fulfillment of the Requirement for the Degree of Master in Science in Thermal Engineering

By

Nathnael Bekele

Thesis Advisor

Dr-Ing. Wondwossen Bogale

June 2018

Parametric Study of a Diffuser for HAWT Power Augmentation

By

Nathnael Bekele

Approved by Board of Examiners

Dr-Ing. Wondwossen Bogale	____________________	____________________
Advisor	Signature	Date
Dr. Abdulkadir Aman	____________________	____________________
Internal Examiner	Signature	Date
Dr. Yilma Tadesse	____________________	____________________
External Examiner	Signature	Date
Dr. Yilma Tadesse	____________________	____________________
Dean, SMiE	Signature	Date

Candidate's Declaration

I, hereby, declare that the work which is presented in this thesis, entitled ***"Parametric Study of a Diffuser for HAWT Power Augmentation"*** in partial fulfillment of the requirements for the award of the degree of Master of Science in Thermal Engineering, is an authentic record of my own work carried out from November 2016 to June 2018 under the supervision of Dr-Ing. Wondwossen Bogale, School of Mechanical and Industrial Engineering, Addis Ababa Institute of Technology, Addis Ababa, Ethiopia. The matter embodied in this thesis has not been submitted by me or any other person for the award of any degree or diploma. All relevant resources of information used have been duly acknowledged.

Name	Signature	Date
Nathnael Bekele Candidate	__________________	________________

This is to certify that the statement made by the candidate is correct to the best of my knowledge. This thesis has been submitted for examination with my approval.

Name	Signature	Date
Dr-Ing. Wondwossen Bogale Thesis Advisor	____________________	_________________

Abstract

The paper is a compilation of a work involving the study of the parametric relationship of geometric parameters of a diffuser for power augmentation for horizontal axis wind turbines (HAWTs). Various works of researchers are studied and hypotheses and possible optimization techniques are put together to depict the picture of power augmentation. The bottom line shows that the power output and performance of a wind turbine can be improved through the use of diffusers. The effect of the use of these devices is to increase the mass flow rate at the rotor plane. This significantly augments the power developed by the wind turbine due to the cubic relationship of wind speed and power. This increase in speed, represented by the acceleration factor or velocity ratio, $\frac{U}{U_\infty}$, is dependent on the geometry of the concentrator device, specifically the diffuser. The basic diffuser geometry parameters were identified and their relationships were formulated based on CFD results. A commercial software, ANSYS-Fluent, is used for simulation where the results are analyzed with MS-Excel and Matlab. The results show good agreement with previous works.

Due to the presence of a diffuser in the flow field, the velocity ratio along the central axis of the diffuser shows an increase as the flow approaches the diffuser inlet. The velocity peaks at a location immediately after the diffuser inlet. The velocity, then, levels off and further decreases as the flow continues to the diffuser outlet and exits it. This suggests a possible location of a wind turbine at the vicinity of the inlet.

The length of the diffuser and the flange height were the major parameters considered in this study. Their relationship with velocity ratio was analyzed and the results show that both have a direct relationship with the velocity ratio. The no-flange configuration showed a 32.5% increase in the on-axis wind speed in reference to the bare wind turbine configuration. The flanged configurations showed a 45% and 51% increase on the on-axis wind speed with $\frac{H}{D} = 0.1 \ and \ 0.2$, respectively. The maximum power augmentation ranges from two-to-three times the power produced by a comparable bare wind turbine. The mathematical relations obtained for the major parameters and the velocity ratio can be used in the performance prediction and optimization of a diffuser. Finally, possible directions of further research and work are recommended considering that this work shows a good agreement with previous works and predictions.

Keywords: HAWT, DAWT

Acknowledgement

First and foremost, I express my deepest gratitude to God for His unreserved help and wisdom all through the work.

I would like to thank my advisor, Dr-Ing. Wondwossen Bogale, for the selection of the research topic, his constant follow up and eagerness to have quality work.

Also my gratitude goes to Prof. David Wood for his earnest comments and guidance during the work.

To my wife for continuous support and for believing in me. Her faith in me boosts my energy.

Motivation

Our nation is undertaking various developmental activities. The energy sector remains one of the core areas in driving and sustaining our development. However, I believe we have not strived our best to address out energy needs. All the efforts made in renewable energy sector are but a spoon scoop from the ocean. I believe small-scale renewable energy technologies will substantially augment our energy production. In line with this, I have focused on renewable energies ever since my undergraduate studies.

My team and I developed and manufactured a small wind turbine during our internship. Another student and I followed on that work, designed a better wind turbine blade, and built it from fiberglass-resin composite material. Mass production and optimization were considered during the work. Adding one giant step to this, my MSc research is currently on a diffuser for power augmentation of horizontal axis wind turbines upon recommendation of my advisor. My work on off-grid power generation will not stop here. I have developed a project on wind turbine blade optimization and mass production techniques recently and am working on creating linkages with various researchers in the field. I hope to see small wind turbines made in Ethiopia planted here and there.

Abbreviations

SMiE – School of Mechanical and Industrial Engineering

HAWTs – Horizontal Axis Wind Turbines

DAWTs – Diffuser Augmented Wind Turbines

CFD – Computational Fluid Dynamics

Re – Reynolds number

NS – Navier-Stokes

List of Tables

List of Figures

Table of Contents

Chapter 1 Introduction

1.1 Background

The world has been striving to meet its energy needs. Its development and advancement drive its needs to an unquenchable state. As the natural resources are exploited for energy, new concerns about the environment arise. Since the conventional energy reserve of the earth is limited, there is need for the development of new energy sources and energy production methods. Here, renewable energy sources come into view. Though these are not challenge-free in themselves, they are promising ventures to cope up with the world's needs without depleting the environment. Amongst the challenges of renewable energies is the requirement of large initial investment and, thus, a higher cost of energy.

As new treaties and protocols are signed, developing countries will be and are being stressed while working towards their development. The environmental impact of conventional energy generation methods is exaggerated in these countries. As these nations strive to improve the living standards of their nationals, their energy demand increases. Every ounce of investment made on any of their endeavors should be effective to have thriving economies. However, the challenge of reducing the cost of energy by making the most of the available energy resource is the joint challenge of both developing and developed countries.

In view of this, one of the cleanest renewable energy sources is wind. It is amongst the most promising renewable energy sources. As almost all energy sources, it ultimately comes from the sun. Uneven heating of the earth's surface and atmosphere creates a pressure gradient and, thus, generates flow from higher pressure areas to lower pressure areas. As long as the sun remains, wind remains. Moreover, wind is affected by terrain and landscape.

As far back as the development of wind sailing ships, wind energy has been in use in transportation. And since the development of the Danish wind mills, it has been in use as a source of mechanical power. They have been in use in agriculture for irrigation and grain milling for a long time. The area of interest of wind energy application for this study is its application for electricity generation.

Wind turbines are used to convert the wind energy into mechanical or electrical power. The power extraction sections of wind turbines, known as rotors or blades, convert the kinetic energy in the wind to mechanical power. The rotors are connected to the electric generator directly or through a gearbox. The output from the generators is connected to the electrical control, distribution or storage section. Depending on the site and required application, they can be used in large-scale integrated in wind farms, or as small as a micro-scale, standalone wind turbine powering a household. Rural electrification is one of their newly found applications. Various wind turbines have been developed at different capacities for these applications. Much has been done and many researches have been conducted to improve their performance.

It has been observed that the power available in the wind has a cubic relationship with the wind speed. Thus, any acceleration of the wind right before the wind rotor will have a considerably large impact on the generated power. Researches and developments have led to the use of flow concentration devices used to locally accelerate the wind. These devices are one of the options for improving the performance of wind turbines. Nozzles, cylindrical ducts and diffusers or a combination of these can be used. One of the advantages of these devices is the reduction of tip losses. Nozzles and diffusers increase the mass flow rate into the rotor plane. However, the optimum design of these devices should be identified for an economically viable design. This calls for identifying the major parameters and their relationships with performance.

Diffuser Augmented Wind Turbines (DAWTs) have been researched for quite a long period of time. DAWTs 'inhale' the oncoming wind by creating low pressure immediately behind the diffuser. The focus on DAWTs has been important because they have considerable amount of power augmentation. This allows them to function in low wind speed regions. They are not also significantly affected by fluctuations in wind speeds and the wind direction. In addition, many researchers claim that the power coefficient can be improved. Other advantages include reduced tip losses and noise, and better safety.

1.2 Problem Statement

The role of energy in economic development cannot be overemphasized. Despite the class of a nation, energy is required to drive its activities. However, limitation in science and technological development has put developing countries at a disadvantage. In line with this, studies and researches on energy production techniques is important.

Wind energy systems are one of the promising technologies in responsibly powering our world. Many have delved into launching large-scale wind farms. For a cost-effective and less costly venture into harnessing wind energy, small-scale wind turbine systems can be used for rural electrification. Though small wind systems are less costly, the cost of energy (COE) is still high. If more energy can be produced for the same developed system and cost, the COE can be reduced. The major energy extraction component of a wind turbine is the rotor. The energy extracted depends on the rotor area and aerodynamic performance of the rotor blades. However, the major variable is the wind speed at the rotor plane. Thus, the larger wind speed going through the rotor area, the larger the power production will be.

Nevertheless, the systems developed in this regard have not been much successful. In addition, they have failed to reach commercialization. However, the use of devices to increase the mass flow rate through the rotor blades has been proven feasible through many researches.

This study will be focused on conducting a comprehensive research on performance improvement devices. It involves identifying the important parameters in diffuser designs and their relationship with velocity increase and, thus, augmentation in power. It focuses on the parametric relationships of a diffuser for a micro-scale, horizontal axis wind turbine. Its results range from being an input for commercial product development endeavors to an input for other researches.

1.3 Objectives

1.3.1 Main objectives

- To conduct a parametric study of a diffuser for a horizontal axis wind turbine

1.3.2 Specific objectives

- To identify the important parameters of the diffuser
- To study the relationship of the parameters with diffuser performance
- To suggest an optimization technique

1.4 Scope of the study

The study of a diffuser for power augmentation involves many factors. From previous researches, it has been identified that there are various designs and, thus, different parameters that affect its power augmentation. In line with this, a simple geometry, easy-to-manufacture diffuser is selected and the various parameters involved are introduced and studied.

This study includes a review on previous efforts to augment the power of wind turbines with the use of flow concentrators. Claims of various researchers are critically observed and analyzed. However, the study is narrowed down to the diffuser-type flow concentrator.

The parameters that affect diffuser performance are identified and studied. The significant parameters are selected and CFD simulations are conducted. Empirical relations of these parameters are obtained from the CFD results. Finally, an experiment is conducted to validate the results of the CFD simulation.

Chapter 2 Literature Review

2.1 Background

As environmental concerns escalate, the world is faced with the challenge of developing better ways of powering its endeavors. Renewable energy sources, like solar, wind and water resources, seem to be the ways out of this pressure. As promising as these sources are, their cost of energy is high. Thus, our energy production has been limited to the use of well-established methods and energy conversion techniques. However, as human needs increase, different ventures are taken to improve the existing technologies to reduce the cost of energy.

Wind turbines are one of the most promising technologies in this regard. There is tremendous power in the wind, as seen in hurricanes and tornados. Through the centuries, there have been various endeavors to harness the energy in wind. Wind has been used for the propulsion of ships, grinding grain and pumping water for irrigation. One of the first attempts to use wind for electricity production came in the beginning of the twentieth century. However, with the development of the diesel engine and steam turbines, its use of electricity production dwindled until its rise again with the rise of environmental concerns and depletion of the earth's fossil fuel reserve (Martin O. L. Hansen 2008).

2.2 Wind Resource and Energy Conversion

Wind is the horizontal movement of air due to pressure gradient. This pressure gradient arises due to the uneven heating of the earth's surface by solar radiation. It can be correctly put that wind ultimately comes from the sun. Since it involves movement, wind has kinetic energy, which is capable of doing work. The energy per unit time in the wind is represented as follows;

$$P = \frac{1}{2}\dot{m}V^2 \qquad (2\text{-}1)$$

where $\dot{m}$ is the mass flow rate of air through the area of interest and V is the flow velocity normal to the area under consideration.

For a flow area of A and flow velocity V for an incompressible flow, the power can be rewritten as:

$$P = \frac{1}{2}\rho A V^3 \qquad (2\text{-}2)$$

where ρ is the density of air

From the above relation, the power highly depends on the wind speed. Thus, the wind resource of a location is very important in siting the location of the wind turbine for maximum performance. Another important variable is the flow area. For a circular area, it is dependent on the square of the diameter. Hence, taking all other parameters as constant, the following relations can correctly depict the relationship of the power and the rotor diameter (d) and the wind speed (V);

$$P = f(d^2, V^3) \qquad (2\text{-}3)$$

However, no matter the wind resource available in a specific location, the power production is limited by the cut-in and cut-out wind speed. The cut-in wind speed represents the wind speed where the turbine starts generating power while the cut-out speed denotes the wind speed above which the turbine deliberately shuts down to prevent damage to the generator and other components.

2.3 Wind Turbines

Different wind turbine configurations have been in use. The most common classifications include horizontal and vertical axis. A wind turbine whose axis of rotation is parallel to the ground and perpendicular to the ground is classified as horizontal axis wind turbine (HAWT) and vertical axis wind turbine (VAWT), respectively. Wind turbines can be used for both grid-connected and off-grid power generation. The science of power extraction has been under study for decades. In recent years, improving the performance of wind turbines has been the focus of various researches.

Wind turbines extract energy from wind through wind and turbine blade interaction. In lift-type turbines, the profile of the blades allows for the generation of lift. The tangential component of the lift, then, generates the torque. Hence, the kinetic energy in the wind is converted into mechanical energy (torque and rotation).

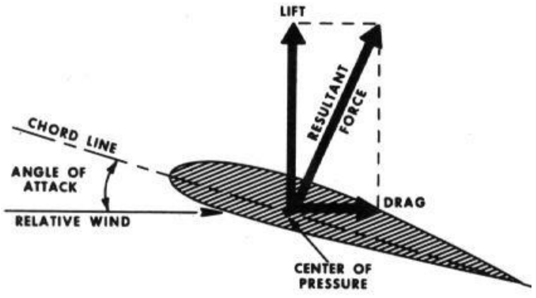


Figure 2-1: Forces on an airfoil section ("Forces on an Airfoil" n.d.)

Technically, all the power in the wind cannot be harnessed. If the whole power is extracted the air immediately behind the rotor plane would be stationary, restricting continued flow of air. Only a certain fraction of the power in the wind can be converted to mechanical power. The power coefficient defines this portion. It represents the factor of mechanical energy extracted by the rotor blades from the available wind energy. This power coefficient can be improved by aerodynamic optimization. New airfoils have been developed for use in wind turbines for optimum power extraction application.

From a 1-D, momentum analysis on a wind turbine, the theoretical maximum limit for power extraction is discovered. This limit was discovered by Betz, a German scientist, to be 16/27 (59.3%). The limit provided to the power coefficient is due to the expansion of the wind right before the rotor blades. This leads to a lesser drop in pressure across the rotor blades limiting the power extracted to the Betz limit. The efficiency of the wind turbine is further reduced by the inefficiencies of the generator and coupling systems and losses in the electrical system. Thus, the amount of energy extracted is further reduced.

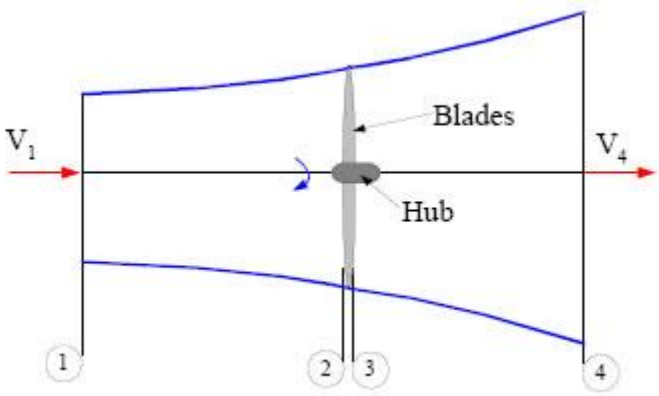


Figure 2-2: Stream tube in 1D-momentum theory ("Momentum Theory" n.d.)

Ever since the discovery of Betz limit, many researchers, including Betz himself, have been working on breaking the limit. One alternative to improve the performance of wind turbines is to reduce the expansion before the rotor plane or re-concentrate it. This can be achieved by using concentrators. Generally, wind turbines with flow concentration devices are called shrouded wind turbines.

2.4 Flow Concentration Devices

Various structures can have different effects on the flow of a fluid. As fluid goes through hollow structures, it interacts with the structure wall and its parameters are affected by the geometry of the structure. From the no slip condition in basic fluid mechanics, the fluid layer in immediate contact with the wall will be stationary. Due to this, there will be a boundary layer extending from the wall surface to the free stream level. Amongst others, the roughness of the wall surface affects the boundary layer thickness (Yunus A. Cengel and John M. Cimbala 2006). Another important aspect is the geometry of the structure. The structure can have different inlet and outlet flow area. The variation in area affects the characteristics of the flow. This is known as the inlet-outlet area ratio. In line with this, different researchers have endeavored to develop and test various hollow structures to obtain the desired output, augmenting the turbine power. These techniques have not only been tried on wind turbines but have been successfully used in water current turbines.

(Yuji Ohya and Takashi Karasudani 2010) compared different hollow structures; a nozzle-, cylindrical- and diffuser-type shrouds. The results showed that the wind tends to avoid the nozzle-type concentrator. The cylindrical-type acted as a normal duct. The diffuser-type model seemed to 'inhale' the wind. Thus, it was observed that a hollow-type diffuser is best suited for the purpose of local acceleration of the wind. (Buyung Kosasih and Andrea Tondelli 2012) investigated with a nozzle-diffuser structure. The results showed that it had 1.7% increase in performance that a diffuser-only structure. However, the nozzle is short and can be considered as an inlet lip. (Kamyar Mansour and Peyman Meskinkhoda 2014) also observed that adding an inlet to the diffuser had advantages.

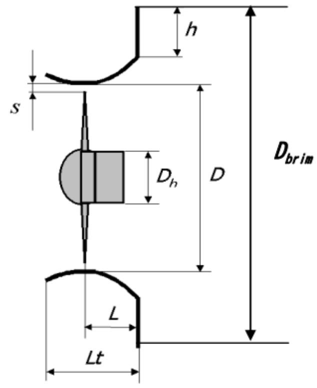


Figure 2-3: Diffuser with wind turbine

(Yuji Ohya and Takashi Karasudani 2010) conducted experimental investigations to identify the basic parameters of the diffuser that affects its performance. Furthermore, it was observed that a curved sectional shape of a diffuser had better performance than a straight shaped diffuser.

Academia and companies studied and tried to develop diffuser augmented wind turbines (DAWTs), as they are called. However, none succeeded in developing a commercially viable product. Thus, further studies were done to come up with better designs.

Figure 2-4: A Shrouded wind turbine (Yuji Ohya et al. 2008)

2.5 Diffuser Augmented Wind Turbines

(M. M. Ehsan et al. 2012) investigated the use of a diffuser to extract low wind speeds. The diffuser increased the inlet wind speed by a factor of 4. (Ken-ichi Abe and Yuji Ohya 2004) observed that the performance of a diffuser strongly depends on the existence of a flow separation inside the diffuser. Moreover, the loading coefficient has a strong relationship with the generation of separation inside the diffuser. A relatively small loading coefficient that avoids a separation and maintains a high pressure-recovery coefficient gives a high performance. (S. A. H. Jafari and B. Kosasih 2014) observed that the augmentation is strongly dependent on the shape and geometry of the diffuser. Thus, the length and expansion angle are the important parameters. The most influential factor is the sub-atmospheric back pressure. This factor is affected by the diffuser area ratio according to their work.

(Aly M. El-Zahaby et al. 2016) carried out CFD analysis on a flanged diffuser and observed that the addition of a flange had a significant increase in the wind velocity. The results also show that the flange height-to-throat diameter ratio (H_c/D_a) above 0.1 had no significant effect.

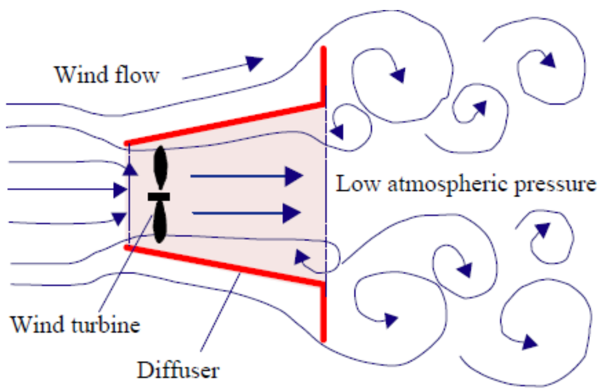


Figure 2-5: Flow around a Flanged diffuser(Toshio Matsushuma, Shinya Takagi, and Seiichi Muroyama 2005)

The (Yuji Ohya and Takashi Karasudani 2010) research with a flanged diffuser shows that the wind speed going through the rotors was “1.6–2.4 times that of the approaching wind speed”. The flange generates a low pressure by creating vortices right behind the diffuser. This creates a high-pressure-to-low-pressure gradient which enhances the mass flow rate into the diffuser. Thus, power increases of 4-5 times were observed. From the observation that as the length of the diffuser increased the velocity ratio increased. However, the wind turbine and diffuser combination will be a heavy structure, which is undesirable due to wind loads and cost. Thus, the design was further modified and a compact “brimmed diffuser” was developed. The researchers were able to achieve two- to three-folds of a ‘comparable’ bare wind turbine power output. This research shows that compact diffusers can be developed. Thus, the wind loads and cost on the structure can be reduced. In addition, Ohya’s research considered different geometries of the diffuser where the wind turbine configuration was changed into a downwind type. This leads to the other advantage of a diffuser. The diffuser controls the yaw motion. Furthermore, it significantly reduced the noise and provided safety from broken blades (Yuji Ohya et al. 2008). (Ken-ichi Abe and Yuji Ohya 2004) considered the flow fields around flanged diffusers. The research further considered the opening angle and loading coefficients and parameter relationships.

Figure 2-6: A Wind Lens Turbine (Yuji Ohya and Takashi Karasudani 2010b)

(F. Bet and H. Grassmann 2002) used a wing structure (an airfoil ring) around the wind turbine. The wing structure had two airfoil rings. The advantage of using an airfoil profile is to reduce flow separation inside the diffuser. The use of two airfoil rings will make use of the air flowing near but not through the blades. The mixing of these two air streams will create turbulence at the exit of the diffuser. This maintains the low pressure behind the diffuser and thus, maintaining the acceleration of the air. CFD analysis was made and demonstrated that the power increased by a factor of 2.

(S. A. Hosein Jafari and Buyung Kosasih 2014) conducted a CFD analysis and concluded that the diffuser gives significant results for higher tip-speed-ratios. On another work, (Buyung Kosasih and Andrea Tondelli 2012) observed that an increase in diffuser length shifted the performance curve to higher tip speed ratios. In addition, the flange height improved the power coefficient as it increased.

(Aly M. El-Zahaby et al. 2016) tried to obtain the optimum flange angle by conducting CFD analysis with ANSYS. The flange angles considered ranged from -25^{o} to 25^{o} and the results showed that a flange angle of 25^{o} had the best performance.

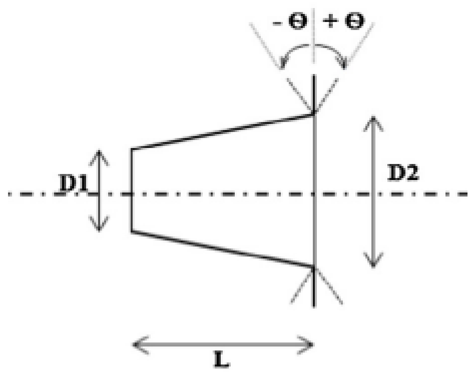


Figure 2-7

Figure 2-8: Flange angle (Aly M. El-Zahaby et al. 2016a)

(Toshio Matsushuma, Shinya Takagi, and Seiichi Muroyama 2005) identified the diffuser parameters and conducted a simulation by varying one of the parameters while fixing the rest. These parameters were the diffuser main body length, entrance diameter, expansion angle and flange length. The simulation results show that as the diffuser length increased the wind speed increased. The maximum wind speed was observed at the entrance of the diffuser. However, there is a limit to the amount of increase in the diffuser length. The maximum wind speed was obtained while the expansion angle was 6 degrees. For the values of the diffuser flange greater than 0.1 meters, the wind speed ratio remained constant for a one-meter diameter diffuser. These show similar results to that of (Aly M. El-Zahaby et al. 2016). A 1.7 folds wind speed increase and 5 folds power augmentation was observed. It should be noted that the simulation was conducted with the diffuser only (without the presence of a wind turbine).

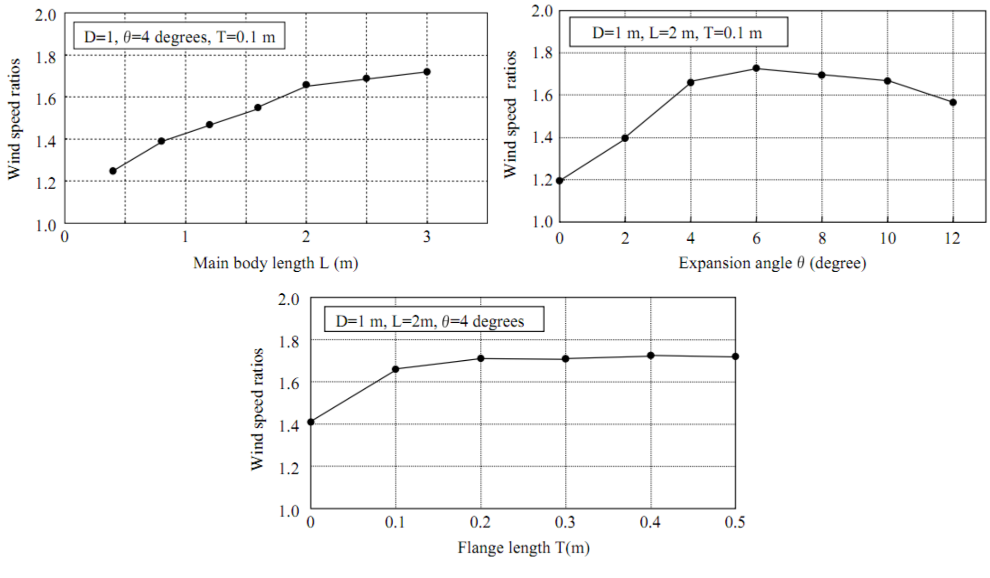


Figure 2-9: Results from (Toshio Matsushuma, Shinya Takagi, and Seiichi Muroyama 2005)

The above figures show that as the length of the diffuser increased the velocity ratio increased, expansion angle values greater than 6 degrees had an adverse effect and flange lengths after 0.2 meters had similar results.

A field test was conducted two turbines (with and without a diffuser). A five-blade propeller wind turbine was used. An increase in energy production was observed; a maximum 1.75 times power output and 1.16 times power output for the entire day were observed. For a fixed wind turbine direction, a maximum of 2.44 folds energy increase and a 1.65 folds wind energy production for an entire day was obtained. However, the effect of solidity was not considered. The performance will be appreciable if a better yawing mechanism can be developed (Toshio Matsushuma, Shinya Takagi, and Seiichi Muroyama 2005).

The numerical analysis from (Srikanth K. S. and Tushar 2016a) also showed that flange height to width (flange length) ratios up to 0.6 had better performance. (Farouk Owis et al. 2015) performed numerical investigation to study the influence of the different diffuser parameters with performance. The results showed that expansion angles in the range of 0 to 12 degrees were suitable for the acceleration of flow. Regarding the diffuser length, a range of diffuser length-to-inlet diameter ratios were studied and a ratio of 1.25 was found suitable from a range of 0.5 to 1.25. Flange height-to-inlet diameter ratio of 0.75 was also found suitable.

2.6 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a "field of study devoted to solution of the equations of fluid flow through the use of computers" (Yunus A. Cengel and John M. Cimbala 2006). It is one of the methods of numerical solutions. Analytical solutions have been observed to be tedious and requiring simplification because the solutions involve solving differential equations which are difficult to solve. However, the use of computers has simplified the task. This makes CFD very important. It involves replacing the partial differential equations with discretized algebraic equations that approximate the partial differential equations. These equations are then solved to obtain fluid field values at the discrete points in space and/or time (Bruce R. Munson et al. 2013).

Generally, CFD involves three essential sections. Pre-processing, Solution and Post-processing. Its major steps include defining the computational domain, setting boundary conditions and initial conditions. CFD has a wide variety of applications in the industry. Automotive, industrial, HVAC, naval, civil, chemical and biological industries are the few amongst others. It is choicest because it is a cost-effective way of simulating actual scenarios. It is clear, though experiments are accurate than CFD, they are expensive. Based on these advantages, the use of CFD shortens the design cycle and reduces the design cost. Large parametric tests can be "conducted on new designs in a shorter time." It also "enhances visualization of complex flow phenomena." (Bruce R. Munson et al. 2013)

There are various commercial CFD codes in use. Each one has its own method of solving the fluid flow equations. However, there are important steps that are common to all. An important step in the use of CFD is the grid or mesh generation, which follows the computational domain definition. Here, "the domain is divided into many small elements called cells" (Yunus A. Cengel and John M. Cimbala 2006). The governing equations are "solved on the computational mesh" (F. Moukalled, L. Mangani, and M. Darwish 2016). The computational mesh could have structured or unstructured grids. A structured grid has "planar cells with four edges (2D) or volumetric cells with six faces (3D)" while an unstructured grid consists of "cells of various shapes" (Yunus A. Cengel and John M. Cimbala 2006). Note that mesh and grid can be used interchangeably.

Grid generation is a critical step because a quality grid gives more reliable results. In addition, the results will converge more rapidly. To check the quality of the mesh, grid independency tests needs to be conducted. "The standard method to test for grid independency is to increase the resolution and repeat the simulation" and check if the results have an appreciable difference (Yunus A. Cengel and John M. Cimbala 2006). Another important step is specifying boundary conditions. Boundary conditions are essential because they make the general equation unique. Cautions should also be taken since wrong boundary conditions lead to wrong solutions (F. Moukalled, L. Mangani, and M. Darwish 2016). Typical boundary conditions include wall, inflow and outflow. Wall boundary condition sets the no-slip condition, which dictates that the fluid layer in contact to that boundary is stationary. Due to this, a boundary layer is expected in the immediate fluid layers. Inflow and outflow boundary conditions can be specified with the use of pressure and velocity. Hence, the total pressure at the inlet and outlet or the inflow or outflow velocity can be specified. However, both conditions cannot be set with pressure or velocity because the governing equations have coupled the pressure and the velocity (Yunus A. Cengel and John M. Cimbala 2006).

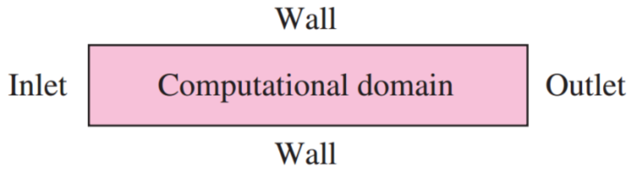


Figure 2-10: A simple computational domain with boundary conditions (Yunus A. Cengel and John M. Cimbala 2006)

ANSYS is a multi-physics platform to solve engineering problems. These include electrostatic, structural, thermal, fluid flow and more analyses. Such analysis tools have significant importance in the preliminary design of products.

Amongst the fluid flow platforms, ANSYS Fluent stands out. It provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent, and steady-state and unsteady flow. It combines a broad range of mathematical models for transport phenomenon with the ability to model complex geometries ("ANSYS Fluent Theory Guide" 2013). It has wide applications in ranging from modeling air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Fluent covers a broad reach, including special models with capabilities to model in-cylinder combustion, aero-acoustics, turbomachinery and multiphase systems ("ANSYS Fluent," 2017).

Fluent addresses fluid flow problems be solving the mass and momentum conservation equations. Robust and accurate turbulence models are available in ANSYS Fluent. The available turbulence models in Fluent include Spalart-Allmaras, Standard k-e, RNG k-e, Realizable k-e, standard k-w and SST k-w. (ANSYS Training) "Turbulence models are developed for particular geometries and flow conditions and may be in accurate or unrealistic for others" (Frank M. White 2001). Clearly, there are no universally accepted models for all problems. "The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation." ("4.2 Choosing a Turbulence Model" 2009) There is need to identify the available options for modelling turbulent flows.

Experimental methods in this parametric study are largely tedious and expensive. Thus, the use of CFD in this study is of utmost importance. However, the computational domain and solutions methods should be carefully selected and implemented, else this will create a lot of error and misconception.

2.7 Simulation of flow around diffusers and DAWTs with CFD

For the case of the diffuser analysis, the flow behavior includes both laminar and turbulent flow. The turbulent flow is desired in that it creates the low-pressure region immediately behind the diffuser. This low-pressure region creates a draft in to the diffuser due to the pressure gradient inside the diffuser. However, flow separation can also occur in the diffuser. Thus, turbulence models are important in the simulation of flow with flow separation (Kamyar Mansour and Mohsen Yahyazade 2011). This way the analysis will be able to capture the turbulent flow around the diffuser. However, a well-designed diffuser should be able to allow the maximum flow augmentation without flow separation inside the diffuser.

(Kamyar Mansour and Mohsen Yahyazade 2011) observed that boundary conditions are important for low-Re flow in wind turbines. Generally, boundary conditions are essential in solving ordinary and partial differential equations. Different boundary conditions give different results for the same general equation. Thus, wrong boundary conditions will give incorrect results. Therefore, the identification and assignment of boundary conditions should be carefully done (F. Moukalled, L. Mangani, and M. Darwish 2016).

(Srikanth K. S. and Tushar 2016a) conducted a two-dimensional CFD analysis of a Wind Lens. Gambit was used as a preprocessor with a mesh scheme of simple quadrilateral meshes and fine meshes around the diffuser. Dense grids were used around the diffuser surface to resolve the flow features. Fluent 6.3 was used as a CFD solver with Spalart-Allmaras turbulence model and second order upwind scheme. However, the computational method was not validated. In addition, grid independence study was not conducted.

(Kamyar Mansour and Peyman Meskinkhoda 2014) conducted a 2D axisymmetric analysis and compared the results with a previous experimental result. (M. O. L. Hansen, N. N. Sorensen, and R. G. J. Flay 2000) conducted a simple 1D analysis using the actuator disc approach. The research was to compare the approach with CFD. The CFD was a 2D analysis with 16,640 cells. The grid had 5 times the diffuser length upstream, 10 times downstream and radially. Free stream velocity was specified at the inlet. K-ω turbulence model with SST limiter was used.

(Aly M. El-Zahaby et al. 2016) observed that the FLUENT package had results that were in good agreement with experimental results. Here, a 2D analysis was conducted. 27,900 nodes with 2mmX2mm grid size were used and verified with experimental results. The mesh was also tested for independence.

(Kamyar Mansour and Mohsen Yahyazade 2011) Compares the different models. Spalart-Allmaras, Standard k-e and RNG k-e were compared. Observed that Spalart-Allmaras was suitable for turbulence closure for low wind speeds and RNG k-e was suitable for higher wind speeds. Spalart-Allmaras has much better convergence and low computing cost. Standard k-e cannot accurately predict flow separation.

Chapter 3 Methodology

3.1 Modelling

Various models were generated and analyzed with ANSYS Fluent. The different models incorporated different values for the parameters. Thus, the parametric relationships were obtained. Analytical equations were derived from the obtained results. As discussed in the literature review, the following are the major parameters;

- Diffuser area ratio (β) or opening angle (α)
- Diffuser length or width (L)
- Flange height (H)
- Flange angle (θ)

Diffuser area ratio (β) is the ratio of the diffuser outlet area with the diffuser inlet area. It can also be represented by the opening angle, which can be calculated using the diffuser inlet and outlet radius (or diameter) with the diffuser length.

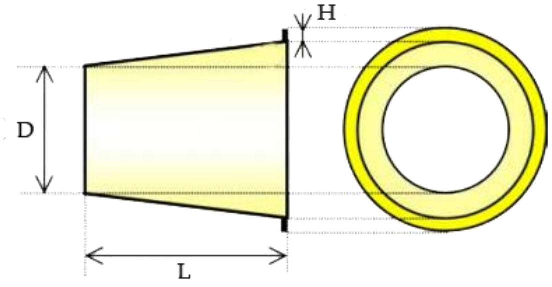


Figure 3-1: Diffuser Parameters

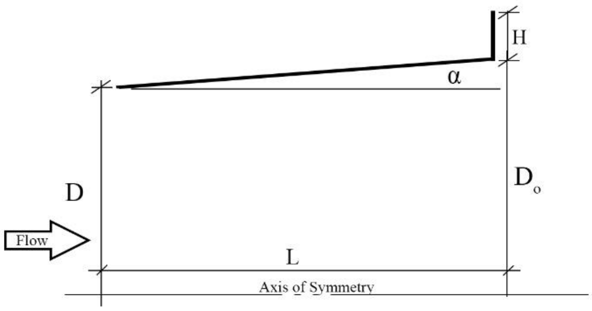


Figure 3-2: Major Parameters (Sectional view)

The area ratio and the opening angle can be mathematically presented as follows;

$$\text{Area ratio, } \beta = \frac{Area_{outlet}}{Area_{inlet}} \quad (3\text{-}1)$$

$$\alpha = \text{atan}\left(\frac{x}{L}\right), \quad (3\text{-}2)$$

$$\text{where } x = \frac{D_o - D}{2} \quad (3\text{-}3)$$

The diffuser length (L) is the central axis distance between the diffuser inlet and outlet. It is normalized by the diffuser inlet diameter (D) for convenience. The remaining are also normalized by the diffuser inlet diameter (D). The area ratio or the opening angle can be used to define the outlet diameter. Consequently, there is no need to define two diameters. Therefore, the diffuser length-to-inlet diameter ratio will be $\frac{L}{D}$ and the flange height to inlet diameter ratio will be $\frac{H}{D}$.

However, only the diffuser length and flange height were used in the simulation. The opening angle was excluded from the analysis because its effects were negligible for small angles (0^o - 6^o). The flange angle was not also included in the analysis because the researcher believes that the parameter should be studied after a complete parametric analysis. This is validated in that only one researcher (Aly M. El-Zahaby et al. 2016) considered the flange angle.

The flange height has been included in the major parameters, because various researchers have obtained results that flanges improved the diffuser performance by generating turbulence immediately behind the diffuser (Aly M. El-Zahaby et al. 2016; Farouk Owis et al. 2015; Kamyar Mansour and Peyman Meskinkhoda 2014; Ken-ichi Abe and Yuji Ohya 2004; S. A. H. Jafari and B. Kosasih 2014; Srikanth K. S. and Tushar 2016a; Toshio Matsushuma, Shinya Takagi, and Seiichi Muroyama 2005; Yuji Ohya and Takashi Karasudani 2010). Since flow separation occurs in the diffuser due to adverse pressure gradient, the low pressure created at the diffuser outlet helps the flow to remain attached to the diffuser wall.

For simplicity, the dimensionless forms of these parameters are used. This allows for few changes in the simulation. Both parameters are normalized by the diffuser inlet diameter. The dependent parameter in this study is the acceleration factor (or the velocity ratio). This is the local velocity normalized by the free stream velocity, $\left(\frac{U}{U_\infty}\right)$ is referred to velocity ratio hereafter.

3.2 Validation

Though a curved diffuser interior surface is found to be advantageous as observed in the research conducted by (Yuji Ohya and Takashi Karasudani 2010), this model does not include such a feature for ease of manufacturing. In addition, the interior surface is not in the scope of this study.

Based on this, the remaining parameters can be set. Note that there should be a clearance from the diffuser. This is taken to be 0.01m based on (Yuji Ohya and Takashi Karasudani 2010). Hence, the inlet diameter will be D = 0.2m.

Table 3-1: Parameters for the validation model

No	Parameters	Value	Dimensions
1	Area ratio	(Opening angle: 4°)	-
2	L/D	1.5	L = 0.3m
3	H/D	0.5	H = 0.1m

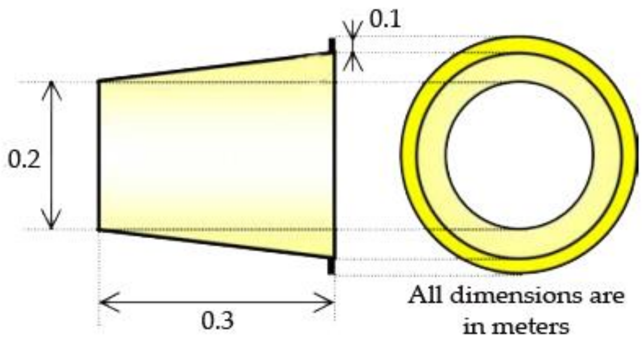


Figure 3-3: Model Diffuser

3.3 CFD Modeling and Simulation

Two-dimensional analysis was used to reduce computation resource and time. This method is also used by various researchers (Aly M. El-Zahaby et al. 2016; Farouk Owis et al. 2015; Srikanth K. S. and Tushar 2016) and found to be useful and acceptably accurate. Furthermore, the Spalart-Allmaras turbulence model is used for the CFD solver (Srikanth K. S. and Tushar 2016). In addition, from (Kamyar Mansour and Mohsen Yahyazade 2011)'s work, Spalart-Allmaras is a good model for turbulence. Second order upwind scheme was set.

3.4 Mesh Independence Test

Elsewhere in this paper, it has been mentioned that the quality of the mesh is very important. A finer mesh will give a more accurate result. Furthermore, the results will converge more rapidly. However, a finer mesh also means longer computational time and higher cost. There is need to balance between accuracy and cost (and/or time). “The standard method to test for grid independency is to increase the resolution and repeat the simulation” and check if the results have an appreciable difference (Yunus A. Cengel and John M. Cimbala 2006). In line with this, a mesh independence test was conducted based on a validation model. The following results were obtained and the final mesh was selected.

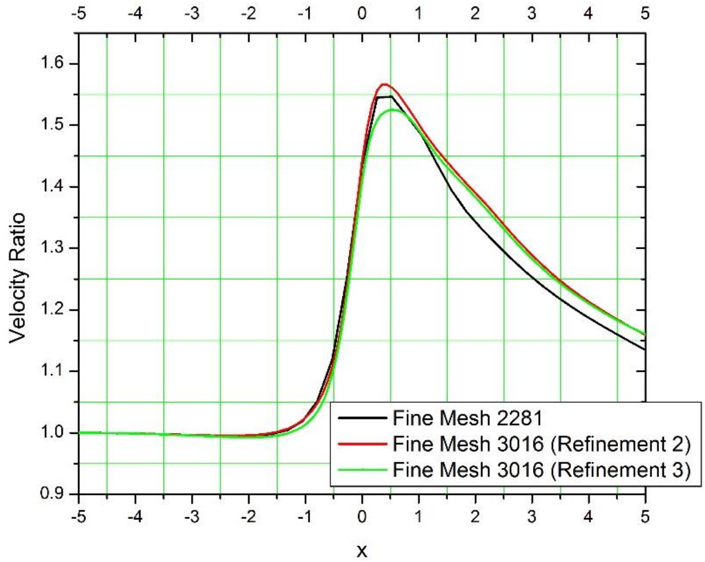


Figure 3-4: Grid Independence Test

The validation was done based on the research by (Farouk Owis et al. 2015). Their work was validated based on the experiment by (Ken-ichi Abe and Yuji Ohya 2004).

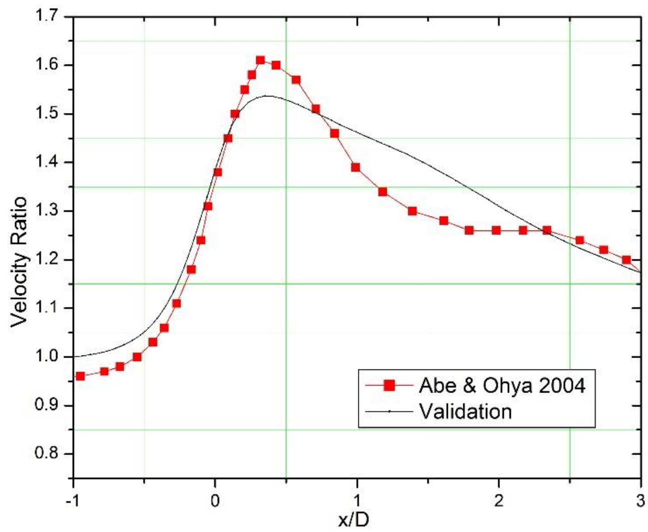


Figure 3-5: On-axis Velocity Ratio Distribution

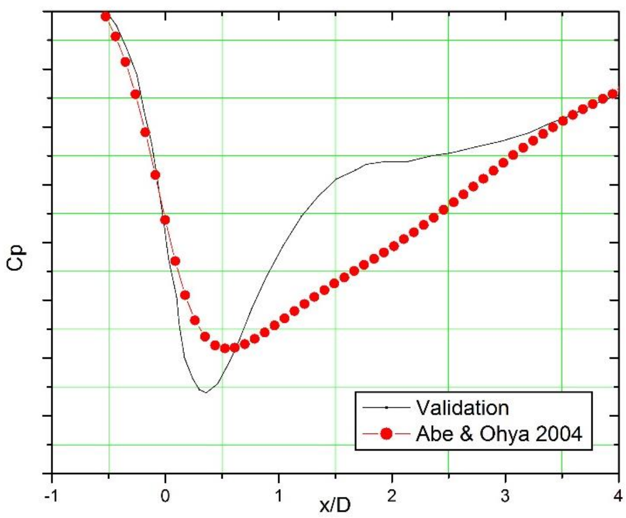


Figure 3-6: On-axis Pressure Coefficient Distribution

The model underestimates the velocity increase and over-predicts the lowering of the pressure. Since the work majorly involves the velocity distribution, the model can be used. The general trend of the velocity and pressure distribution are observed to follow the trend of the experimental work. The velocity distribution obtained has a maximum error percentage of 8%.

The two-dimensional and axisymmetric model of the diffuser and the computational domain was modelled in SOLIDWORKS 2017 and exported as an IGES model. The computational model was developed based on Owis' work (Farouk Owis et al. 2015).

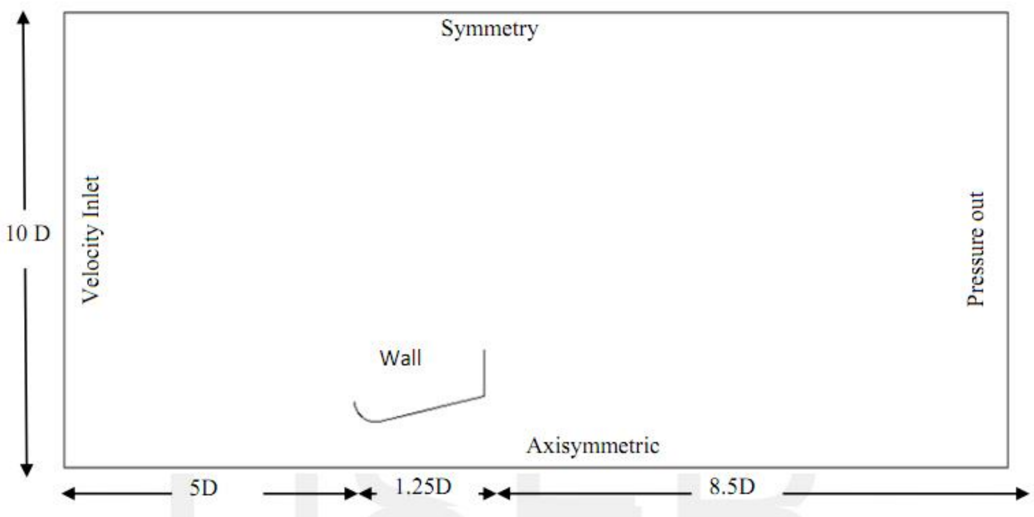


Figure 3-7: Computational Domain (Farouk Owis et al. 2015)

The inlet lip has an advantageous effect as observed by Mansour and Ohya (Kamyar Mansour and Peyman Meskinkhoda 2014; Yuji Ohya and Takashi Karasudani 2010). However, the length of the inlet lip is important in actually defining its effect. For long lengths, it will have a disadvantage in that it will act as a nozzle and the flow avoids it (Yuji Ohya and Takashi Karasudani 2010). The inlet lip is not included due to this.

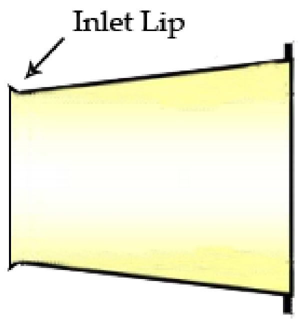


Figure 3-8: Diffuser with Inlet Lip

Since the analysis is wind flow, the inlet boundary condition can be modelled as a uniform flow with the outlet boundary condition as a pressure outlet. A symmetry boundary condition will be used for the lower boundary along the axis of symmetry and the upper boundary. The diffuser surface are considered as walls and, thus, the no-slip condition holds. This agrees with various researchers work (Aly M. El-Zahaby et al. 2016; Farouk Owis et al. 2015).

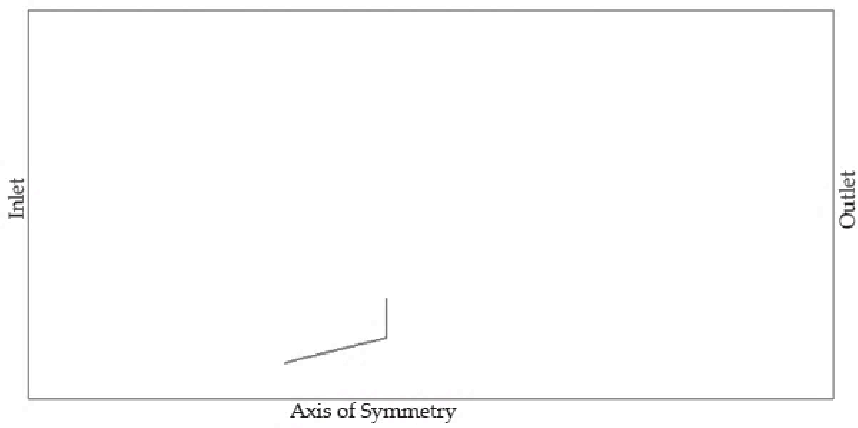


Figure 3-9: Computational Domain

Table 3-2: Summary of Boundary Conditions

Boundary Condition	**Type**
Inlet	Inlet Velocity: 5m/s
Outlet	Pressure Outlet
Diffuser Wall	Wall: No-slip

3.5 Experimental setup

The CFD results have to be validated through experiments. Due to the unavailability of a suitable wind tunnel and measurement instruments, an existing airflow bench was modified for a 2D experimental setup. The diffuser was modelled with specific geometric parameters and made to match the simulation Re number.

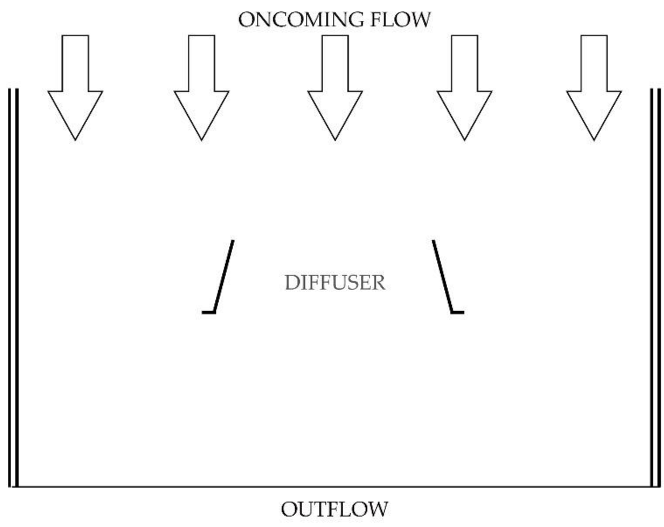


Figure 3-10: Arrangement in airflow bench

The airflow bench used is an old product of Tecquipment made for education purpose. The bench has different setup for various fluid mechanics related experiments like drag coefficient determination, flow visualization, etc. The flow visualization module was modified for this experiment. The 2D model was designed to fit this module. The model was 3D printed and was attached to a customized board for attachment. Photos are attached in Appendix.

Chapter 4 Results and Discussion

The simulation included runs with different values of the parameters of interest. The diffuser was modeled in SOLIDWORKS 2015. The following geometric parameters were used.

Table 4-1: Parameters Used

No	Parameters	Value	Remark
1	Opening angle	4^o	Not studied
2	Thickness	5mm	Fixed
3	L/D	0.1 - 2	Varied
4	H/D	0 - 0.2	Varied

The length-to-diameter ratio was varied in the range 0.1 to 2 with an increment of 0.1. The range was chosen based on reasoning as cost and weight of diffuser structure if the size exceeds 2 times the rotor diameter. This was conducted for case with no flange and $\frac{H}{D}$ values of 0.1 and 0.2.

Though the on-axis velocity distribution does not represent the velocity field inside the diffuser, it is used as the dependent variable because it has a unique value for each configuration.

4.1 No Flange

For a no flange configuration, the velocity distribution along the central axis was obtained to be as follows.

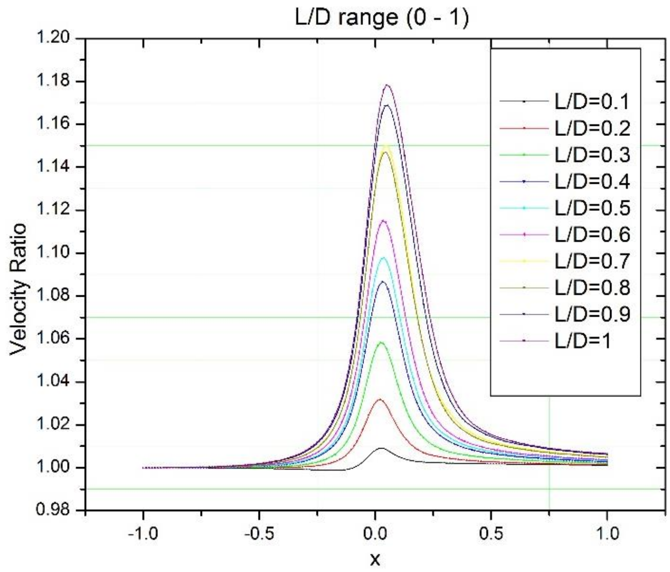


Figure 4-1: On-axis velocity distribution for diffuser with no flange configuration and L/D range between 0 and 1

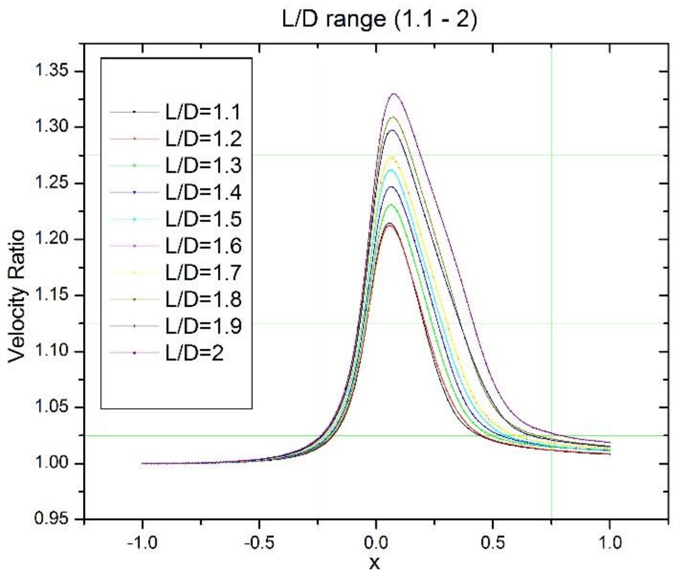


Figure 4-2: On-axis velocity distribution for diffuser with no flange configuration and L/D range between 1.1 and 2

In the $\frac{L}{D}$ range of 0.1 to 2, there was a direct relationship between the maximum velocity ratio and $\frac{L}{D}$. As the value of $\frac{L}{D}$ increased, the velocity ratio increased. Unless another limit is specified, the velocity will increase with increasing $\frac{L}{D}$.

In order to develop the relationship between the velocity ratio and $\frac{L}{D}$, the maximum velocity ratio along the central axis is selected. This is because that value is unique in comparison to other values. Therefore, the maximum velocity ratio is plotted with $\frac{L}{D}$ below.

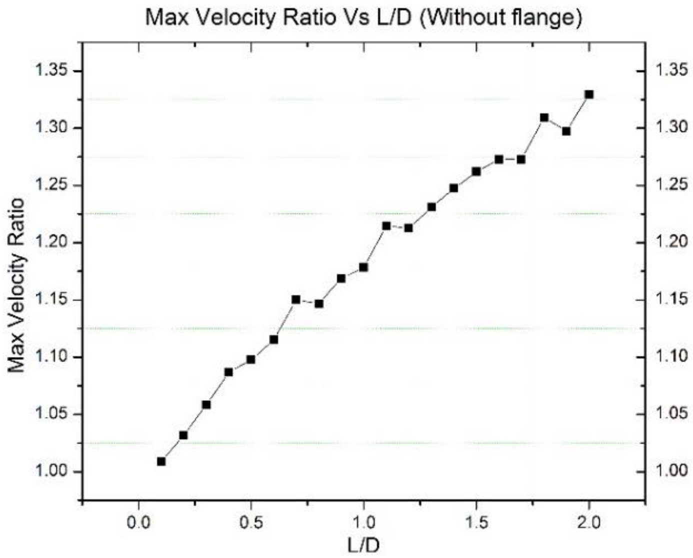


Figure 4-3: Max Velocity Ratio Vs L/D (Without Flange)

For a no-flange configuration, a maximum on-axis velocity ratio of about 1.325 was achieved for the maximum length-to-diameter ratio for this study. This velocity corresponds to a velocity of 6.625m/s. However, the maximum velocity along the central axis does not represent the maximum velocity inside the diffuser. It is seen in the following figures that the velocity also varies along the radial position from the central axis. The velocity increases with increasing radial position until it becomes maximum and diminished due to no-slip condition at the diffuser wall. The maximum velocity for $\frac{L}{D} = 2$, is about 6.82 m/s near the diffuser wall.

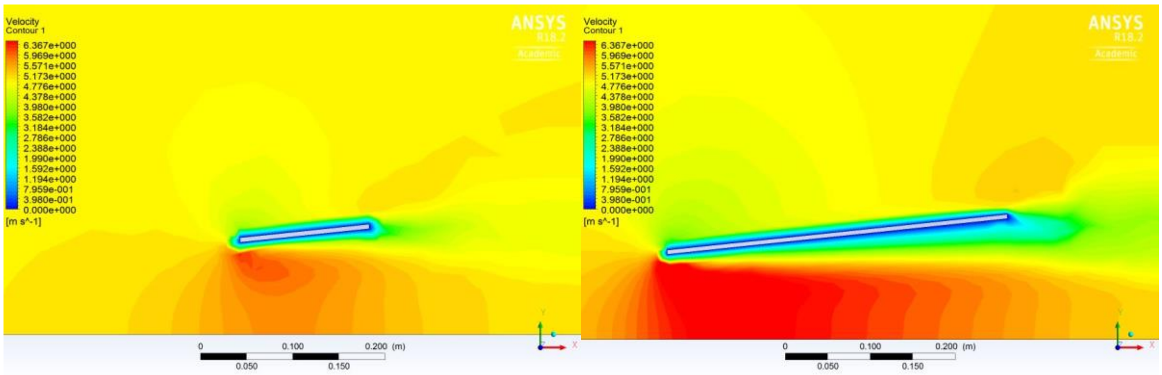


Figure 4-4: Comparison of L/D with 0.7 (left) and 2 (right)

The velocity increase is the highest at the diffuser inlet region. As the length-to-diameter ratio increases the ‘increased velocity’ region expands to the central region of the diffuser. This is shown in the figure above (Figure 7-4). The ‘increased velocity’ region decreases in size as the flow progresses to the outlet.

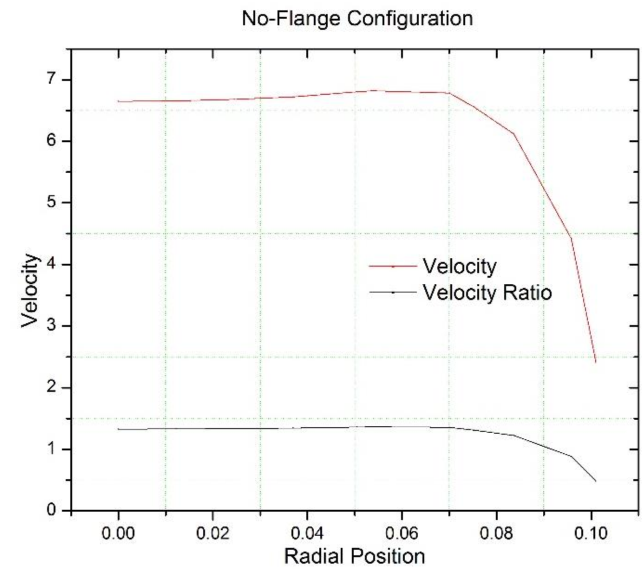


Figure 4-5: Radial variation of velocity and velocity ratios (H/D = 0)

4.2 With Flange ($\frac{H}{D}$ = 0.1)

For a diffuser with flange configuration, the flange height-to-diameter ratio was evaluated for values of 0.1 and 0.2. The following central axis velocity distribution was obtained for a flange height-to-diameter ratio of 0.1.

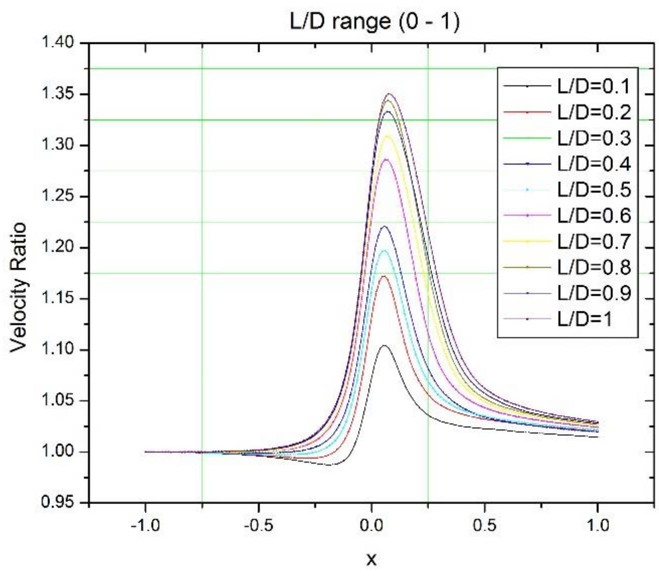


Figure 4-6: On-axis velocity distribution for diffuser with flange configuration and L/D range between 0.1 and 1

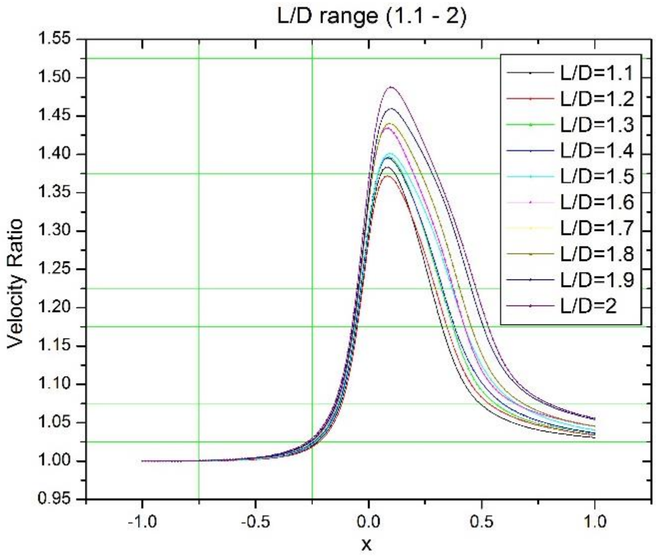


Figure 4-7: On-axis velocity distribution for diffuser with flange configuration and L/D range between 1.1 and 2

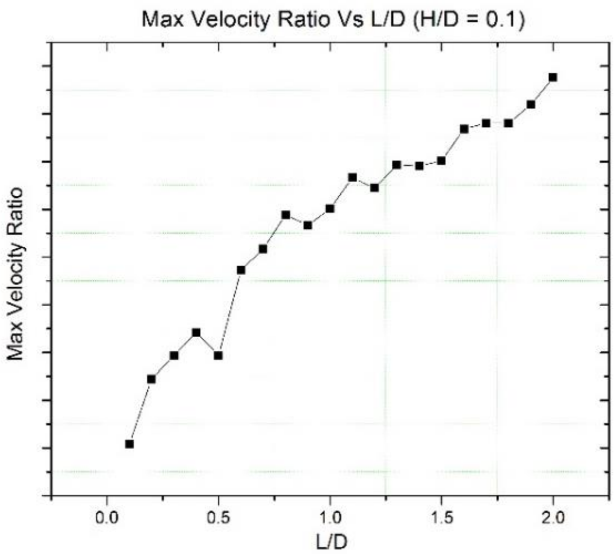


Figure 4-8: Maximum On-axis Velocity Ratio Vs L/D (H/D=0.1)

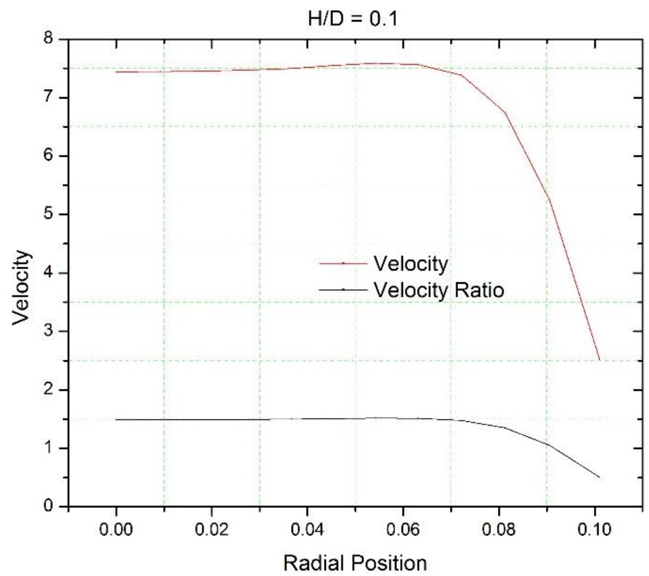


Figure 4-9: Radial variation of velocity and velocity ratios (H/D = 0.1)

For a with- flange configuration of $\frac{H}{D} = 0.1$, a maximum on-axis velocity ratio of about 1.45 was achieved for the L/D ratio of 2, the maximum for this study. This velocity corresponds to a velocity of 7.25 m/s. However, as discussed earlier, the actual maximum velocity inside the diffuser is found further away from the central axis along the radial position and has a value of 7.59 m/s near the diffuser wall.

4.3 With Flange ($\frac{H}{D}$ = 0.2)

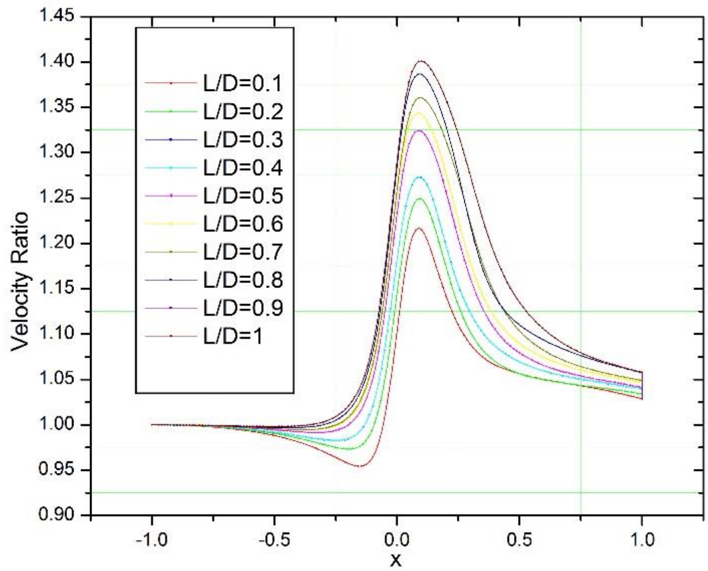


Figure 4-10: On-axis velocity distribution for diffuser with flange configuration and L/D range between 0.1 and 1

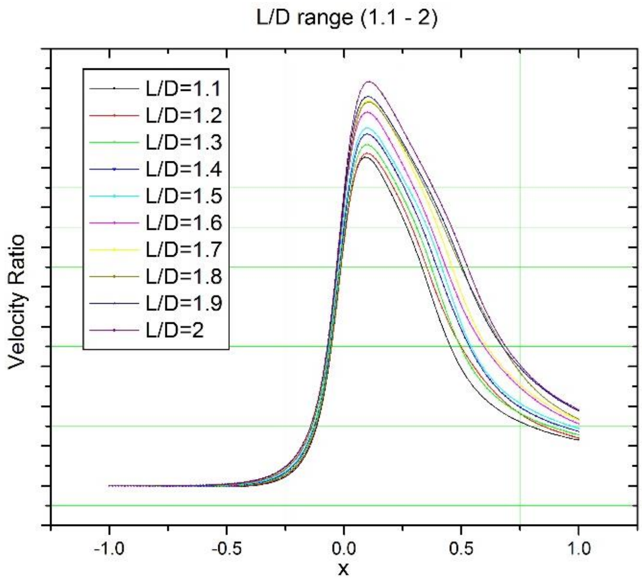


Figure 4-11: On-axis velocity distribution for diffuser with flange configuration and L/D range between 1.1 and 2

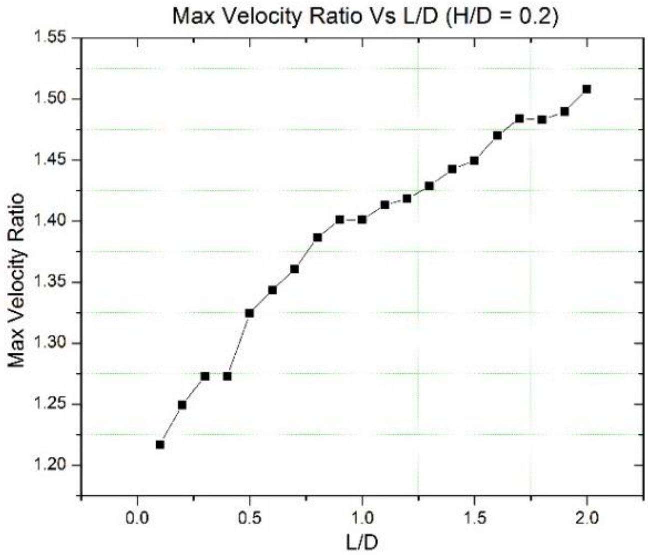


Figure 4-12: Max Velocity Ratio Vs L/D (H/D=0.2)

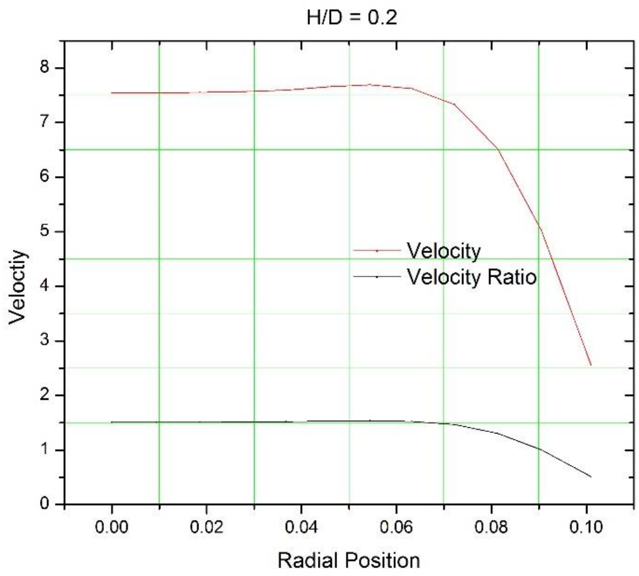


Figure 4-13: Radial variation of velocity and velocity ratios (H/D = 0.2)

For a with- flange configuration of $\frac{H}{D} = 0.2$, a maximum on-axis velocity ratio of about 1.51 was achieved for the L/D ratio of 2, the maximum for this study. This velocity corresponds to a velocity of 7.55 m/s. However, as discussed earlier, the actual maximum velocity inside the diffuser is found further away from the central axis along the radial position and has a value of 7.69 m/s near the diffuser wall.

The maximum on-axis velocity increased as a flange was added and the flange height increased. By the addition of a diffuser to the flow field, the on-axis wind speed increased from 5 m/s to 6.625 m/s for $\frac{L}{D} = 2$. This is a 32.5% increase. In summary, the increments are as follows;

Table 4-2: Summary of Maximum Velocities for the Different Configurations

Configuration	$\boldsymbol{\frac{H}{D}}$	$\boldsymbol{\frac{L}{D}}$	**Maximum on-axis Velocity (m/s)**	**Maximum velocity (m/s)**	**% Increase in reference to**		**Potential Power Increase**
					5 m/s	**No-flange configuration**	
No flange	0	2	6.625	6.82	32.5	-	2.32 times
With flange	0.1	2	7.25	7.59	45	9.43	3.05 times
With flange	0.2	2	7.55	7.69	51	13.96	3.44 times

The above tables shows that general trend where the effect of increasing the flange size reduces with increasing flange size. The magnitude of increment reduces with increasing flange size.

In all the three cases above, the maximum velocity is found at the immediate inlet position. This makes the position of that location a suitable location for the turbine. The location of the maximum velocity are at locations at distances of 0.068968, 0.09569 and 0.109052 meters from the diffuser inlet along the central axis.

4.4 Empirical Relations

The following figures show the maximum on-axis velocity ratios plotted with $\frac{L}{D}$ for different flange sizes.

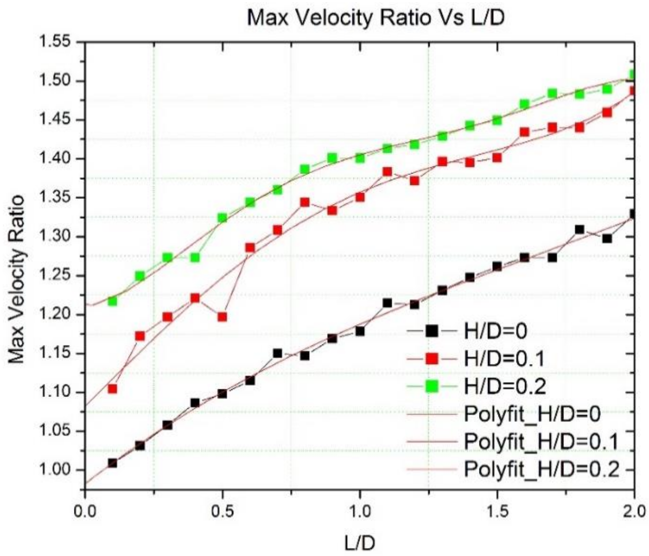


Figure 4-14: Curve fitting and data points [Superimposed]

A polynomial curve fitting technique was used to obtain a relation for the max velocity ratio and the length-to-diameter ratio.

Table 4-3: Curve fitting for maximum velocity ratio and length-to-diameter ratio

H/D	Fitting Equation	r^2
0	$0.01742x^3 - 0.08631x^2 + 0.27344x + 0.98294$	0.99393
0.1	$0.04988x^4 - 0.14872x^3 + 0.02384x^2 + 0.35004x + 1.08215$	0.97821
0.2	$-0.10326x^5 + 0.5579x^4 - 1.05174x^3 + 0.75034x^2 + 0.04171x + 1.2026$	0.99325

$$\frac{U}{U_\infty} = 0.01742\left(\frac{L}{D}\right)^3 - 0.08631\left(\frac{L}{D}\right)^2 + 0.27344\left(\frac{L}{D}\right) + 0.98294 \quad for\,\frac{H}{D} = 0 \tag{4-1}$$

$$\frac{U}{U_\infty} = 0.04988\left(\frac{L}{D}\right)^4 - 0.14872\left(\frac{L}{D}\right)^3 + 0.02384\left(\frac{L}{D}\right)^2 + 0.35004\left(\frac{L}{D}\right) + 1.08215 \quad for\,\frac{H}{D} = 0.1 \tag{4-2}$$

$$\frac{U}{U_\infty} = -0.10326\left(\frac{L}{D}\right)^5 + 0.5579\left(\frac{L}{D}\right)^4 - 1.05174\left(\frac{L}{D}\right)^3 + 0.75034\left(\frac{L}{D}\right)^2 + 0.04171\left(\frac{L}{D}\right) + 1.2026 \; for\,\frac{H}{D} = 0.2 \tag{4-3}$$

The above relations can be used for the optimization of a diffuser for a wind turbine. Required increases in velocity can be supplied and the different parameters of the diffuser can be obtained. The outputs of these relations can be used in the prediction of power production, size of diffuser and cost. This can reduce time and cost for simulations and analysis by provided an initial guess to obtain the required output.

4.5 Experimental Work

The results show a similar trend with the simulation work. The deviation of the results is as expected in that the blockage ratio of the setup is high. Due to this, there is a maximum error of 8% between the experimental and CFD work.

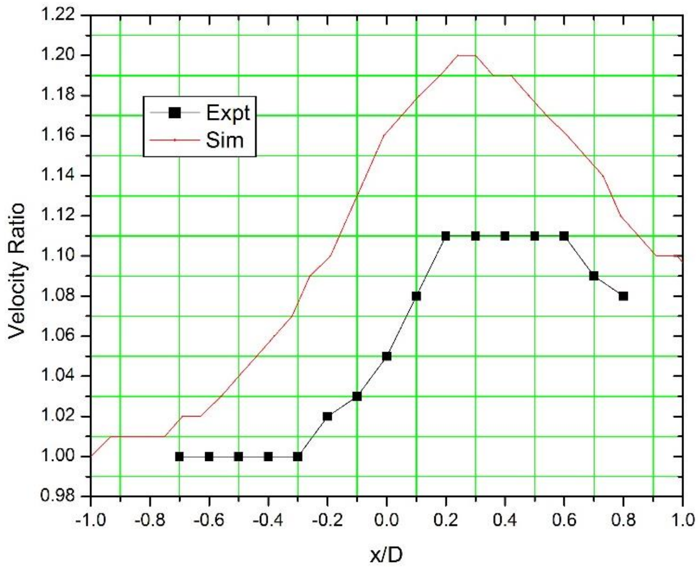


Figure 4-15: Experimental work

Chapter 5 Conclusion

The major parameters for a diffuser were identified and studied. The diffuser was modeled as a two-dimensional model and imported in to ANSYS Fluent for simulation. The diffuser parameters were varied and different models were generated. The velocity distribution was reported and analyzed.

5.1 Major Findings

With the use of a diffuser around a wind turbine, the velocity field inside the diffuser can be increased. The increase has a significant relationship to the diffuser geometry. For both cases ($\frac{L}{D}$ *and* $\frac{H}{D}$), a direct relationship with the maximum on-axis velocity ratio was observed. The new velocity field is a function of the radial and on-axis position. Due to this, though the maximum on-axis velocity ratio was studied, that does not represent the maximum velocity ratio in the diffuser. The maximum velocity is obtained near the diffuser inlet at the far end from the central axis.

5.2 Empirical Relationships

Empirical formulae were developed for the relationship between the on-axis maximum velocity ratio and $\frac{L}{D}$ *and* $\frac{H}{D}$. These formulae can be used for the design and optimization of diffusers for further works. In addition, these formulae can be used for the prediction of the power augmentation on a wind turbine with BEM method or other methods.

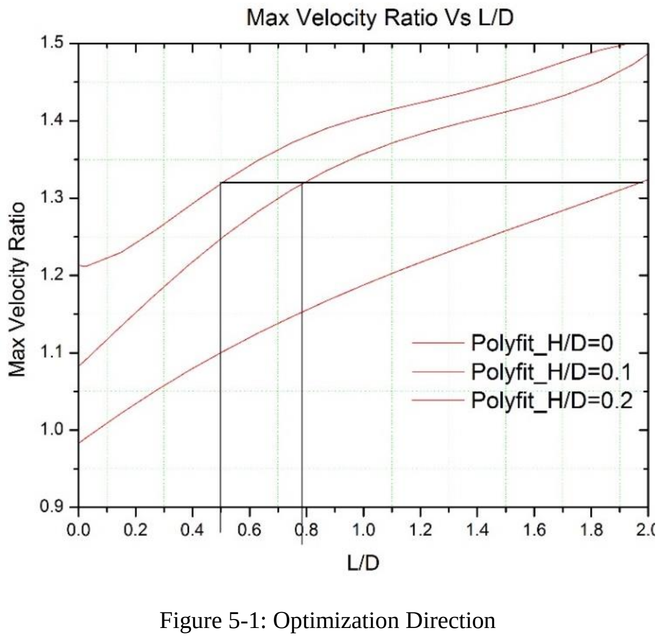


Figure 5-1: Optimization Direction

The above figure shows that the maximum achievable velocity ratio by the no flange configuration, which is about 1.3, can be achieved with a shorter flanged diffuser. Length-to-diameter ratios of 0.8 and 0.5 can give the same results for $\frac{H}{D}$ values of 0.1 and 0.2, respectively. Therefore, the empirical relations can be used to develop a compact diffuser with lower cost and lower wind load. This can enhance the commercialization of DAWTs and reduce the structural load and, thus, the tower size.

In conclusion, the major findings in this work include:

- The addition of a flange at the diffuser exit is advantageous. It creates a low pressure region immediately after the diffuser allowing the drafting of more air into the diffuser.
- The velocity distribution along the possible turbine location is predicted to enhance the power augmentation by lowering the cut-in wind speed by enhance the starting characteristics of the turbine. In addition, the power produced is augmented due to the increase in the velocity of the wind at the power extraction region of the turbine.
- Significant power augmentation can be achieved by optimization with $\frac{L}{D}\, and\, \frac{H}{D}$.

Chapter 6 Future Development

As promising as the power augmentation with diffusers is, the study is yet to be exhausted. The study is confined to a single diffuser geometry and few of the parameters involved. The researcher believes that there is much more to be done in this area than what is done in this work. Thus, the following are suggested for further works;

- Simulations that include the wind turbine need to be conducted. In this case, an actuator disc can be used to represent the wind turbine or a three dimensional analysis can be conducted.
- Experimental works need to be conducted to validate the simulation results. Well-suited experimental setups need to be developed. In addition, field tests can be conducted to further affirm the results of the experiments.
- Since the concept of a diffuser is 'inhaling' more wind in to the rotor area, further studies should not be limited to the diffuser geometry studied. Other geometries that can create the low pressure after the diffuser should be developed and tested for greater velocity ratios.
- The reported velocity increases should be used to predict the power augmentation due to the presence of a diffuser. This can be through the development of an integration to an existing wind turbine analysis tool.
- The cost increment due to the addition of a diffuser should be estimated. A good picture of the advantage of using diffusers can be obtained by relating the Cost of Energy of DAWTs and comparable HAWTS.

References

"4.2 Choosing a Turbulence Model." 2009. ANSYS Fluent 12.0 Theory Guide -4.2 Choosing a Turbulence Model. 2009. http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node44.htm.

Aly M. El-Zahaby, A. E. Kabeel, S. S, Elsayed, and M. F. Obiaa. 2016. "CFD Analysis of Flow Fields for Shrouded Wind Turbine's Diffuser Model with Different Flange Angles." *Alexandria Engineering Journal*.

"ANSYS Fluent Theory Guide." 2013. ANSYS, Inc.

Bruce R. Munson, Theodore H. Okiishi, Wade W. Huebsch, and Alric P. Rothmayer. 2013. *Fundamentals of Fluid Mechanics*. John Wiley & Sons, Inc.

Buyung Kosasih, and Andrea Tondelli. 2012. "Experimental Study of Shrouded Micro-Wind Turbine." *Elsevier*.

F. Bet, and H. Grassmann. 2002. "Upgrading Conventional Wind Turbines." *Elsevier*.

F. Moukalled, L. Mangani, and M. Darwish. 2016. *The Finite Volume Method in Computational Fluid Dynamics*. Springer.

Farouk Owis, M. T. S. Badawy, K. A. Fawaz, and Amr Elfeky. 2015. "Numerical Investigation of Loaded and Unloaded Diffuser Equipped with a Flange." *International Journal of Scientific and Engineering Research*.

"Forces on an Airfoil." n.d. Accessed June 18, 2018. http://avstop.com/ac/flighttrainghandbook/forcesonanairfoil.html.

Frank M. White. 2001. *Fluid Mechanics*. Fourth.

Kamyar Mansour, and Mohsen Yahyazade. 2011. "Effects of Turbulence Model in Computational Fluid Dynamics of Horizontal Axis Wind Turbine Aerodynamics." *WSEAS TRANSACTIONS on APPLIED and THEORETICAL MECHANICS* 6 (3).

Kamyar Mansour, and Peyman Meskinkhoda. 2014. "Computational Analysis of Flow Fields around Flanged Diffusers." *Journal of Wind Engineering and Industrial Aerodynamics*.

Ken-ichi Abe, and Yuji Ohya. 2004. "An Investigation of Flow Fields around Flanged Diffusers Using CFD." *Elsevier/Journal of Wind Engineering and Industrial Aerodynamics*.

M. M. Ehsan, Enaiyat Ghani Ovy, H.A. Chowdhury, and S. M. Ferdous. 2012. "A Proposal of Implementation of Ducted Wind Turbine Integrated with Solar System for Reliable Power Generation in Bangladesh." *International Journal of Renewable Energy Research* 2 (3).

M. O. L. Hansen, N. N. Sorensen, and R. G. J. Flay. 2000. "Effect of Placing a Diffuser around a Wind Turbine." *Wind Energy*.

Martin O. L. Hansen. 2008. *Aerodynamics of Wind Turbines*. Second.

"Momentum Theory." n.d. Accessed June 18, 2018. http://www.esru.strath.ac.uk/EandE/Web_sites/09-10/MCT/html/Technical/bemoverview.html.

S. A. H. Jafari, and B. Kosasih. 2014. "Flow Analysis of Shrouded Small Wind Turbine with a Simple Frustum Diffuser with Computational Fluid Dynamics Simulations." *Journal of Wind Engineering and Industrial Aerodynamics*.

S. A. Hosein Jafari, and Buyung Kosasih. 2014. "Analysis of the Power of Augmentation Mechanisms of Diffuser Shrouded Micro Wind Turbine with Computational Fluid Dynamics Simulations." *Wind and Structures: An International Journal*.

Srikanth K. S., and Tushar. 2016. "Numerical Analysis of Wind Lens." *International Journal of Innovative Research in Science, Engineering and Technology*.

Toshio Matsushuma, Shinya Takagi, and Seiichi Muroyama. 2005. "Characteristics of a Highly Efficient Propeller Type Small Wind Turbine with a Diffuser." *Renewable Energy*.

Yuji Ohya, and Takashi Karasudani. 2010. "A Shrouded Wind Turbine Generating High Output Power with Wind-Lens Technology." *Energies*.

Yuji Ohya, Takashi Karasudani, Akira Sakurai, Ken-ichi Abe, and Masahiro Inoue. 2008. "Development of a Shrouded Wind Turbine with a Flanged Diffuser." *Journal of Wind Engineering and Industrial Aerodynamics*.

Yunus A. Cengel, and John M. Cimbala. 2006. *Fluid Mechanics: Fundamentals and Applications*. 1st ed.

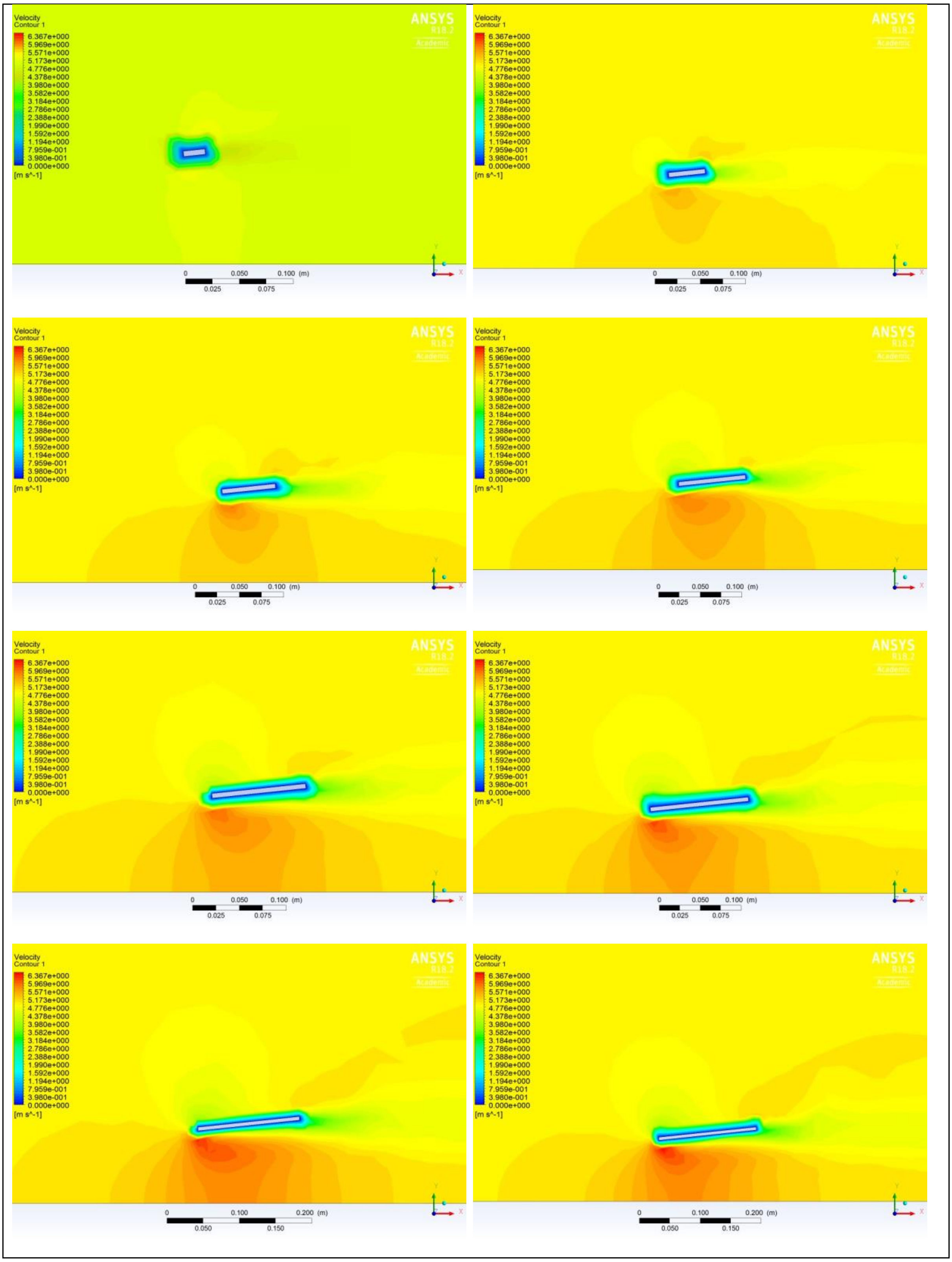
Appendices

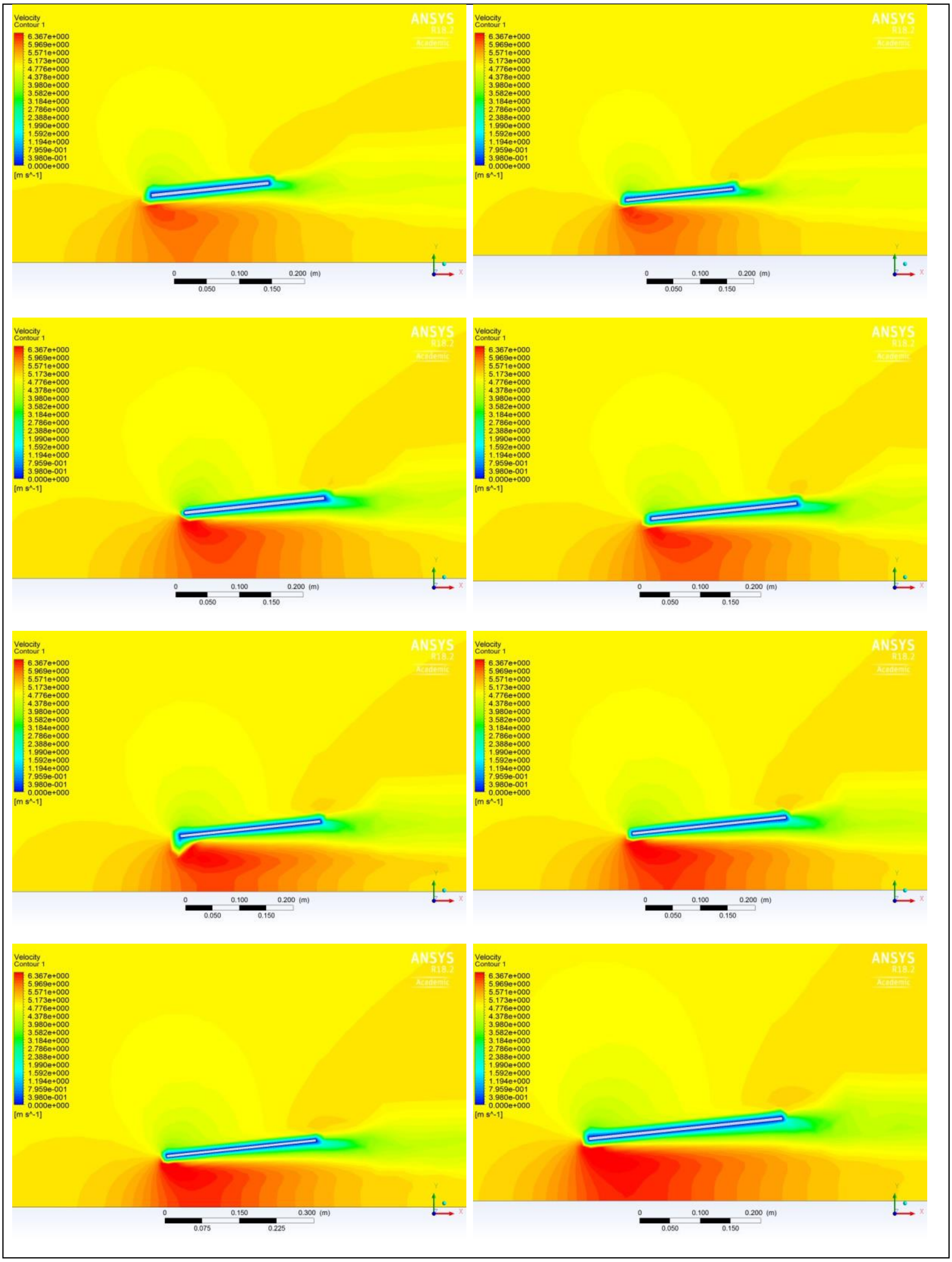
Airflow Bench

Figure 1: Airflow Bench

Figure 2: Flow Arrangement

Figure 3: Test Specimen Attached to Board


Velocity
Contour 1
6.367e+000
5.969e+000
5.571e+000
5.173e+000
4.776e+000
4.378e+000
3.980e+000
3.582e+000
3.184e+000
2.786e+000
2.388e+000
1.990e+000
1.592e+000
1.194e+000
7.959e-001
3.980e-001
0.000e+000
[m s^-1]
ANSYS
R18.2
Academic
0
0.050
0.100 (m)
0.025
0.075
0.100
0.200 (m)
0.050
0.150


Velocity
Contour 1
6.367e+000
5.969e+000
5.571e+000
5.173e+000
4.776e+000
4.378e+000
3.980e+000
3.582e+000
3.184e+000
2.786e+000
2.388e+000
1.990e+000
1.592e+000
1.194e+000
7.959e-001
3.980e-001
0.000e+000
[m s^-1]
ANSYS
R18.2
Academic
0 0.100 0.200 (m)
0.050 0.150
0 0.150 0.300 (m)
0.075 0.225

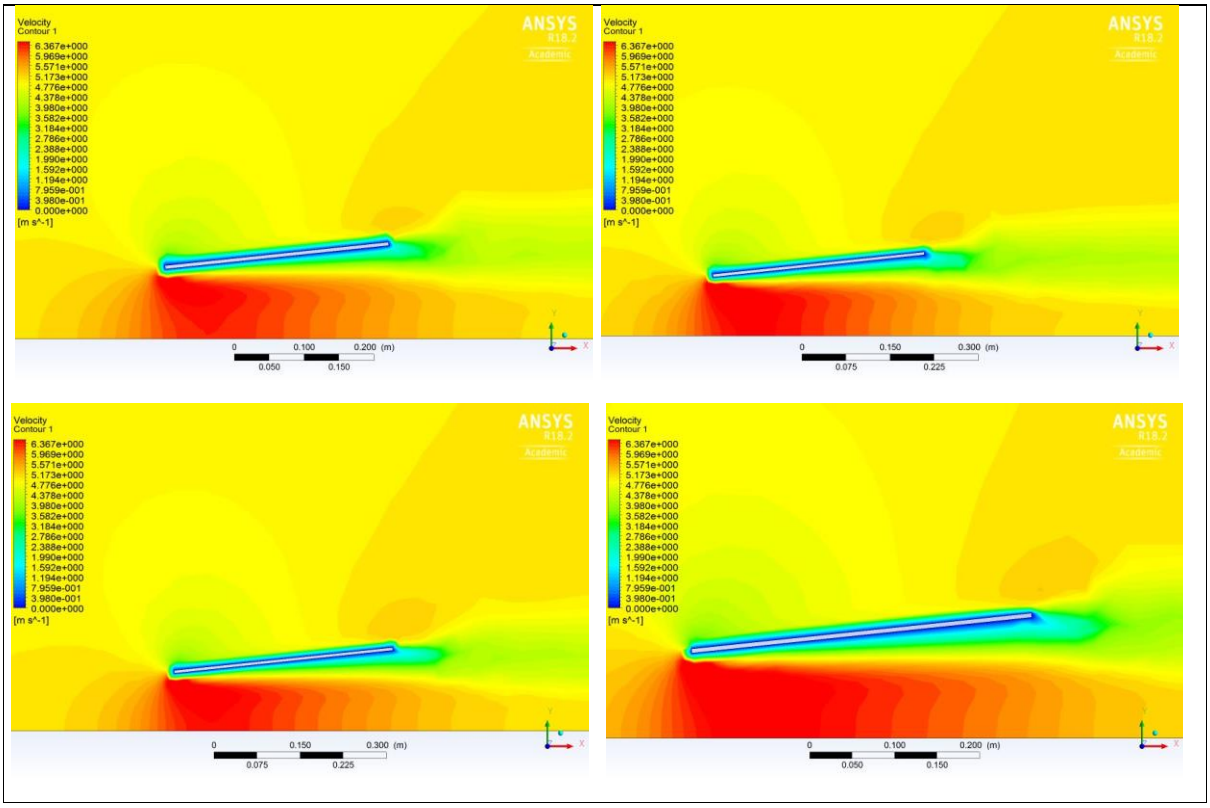


Figure 4: Velocity Distribution for a Diffuser with no flange at different L/D values