SEPTEMBER2016

M.Sc. IN CIVIL ENGINEERING

TSIRY TSILAVO RAJAONARY

UNIVERSITY OF GAZİANTEP

GRADUATE SCHOOL OF

NATURAL & APPLIED SCIENCES

DESIGN AND OPTIMIZATION OF A HYDROKINETIC TURBINE DUCT WITH CFD

M.Sc. THESIS

IN

CIVIL ENGINEERING

BY

TSIRY TSILAVO RAJAONARY

SEPTEMBER 2016

Design and Optimization of a Hydrokinetic Turbine Duct With CFD

M.Sc. Thesis

In Civil Engineering

University of Gaziantep

Supervisor

Assoc. Prof. Dr. M. İshak YÜCE

by

Tsiry Tsilavo RAJAONARY

September 2016

© 2016 [Tsiry Tsilavo RAJAONARY]

REPUBLIC OF TURKEY
UNIVERSITY OF GAZİANTEP
GRADUATE SCHOOL OF NATURAL & APPLIED SCIENCES
CIVIL ENGINEERING DEPARTMENT

Name of the thesis : Design and Optimization of a Hydrokinetic Turbine Duct with CFD
Name of the student : Tsiry Tsilavo RAJAONARY
Exam date : 19.09.2016

Approval of the Graduate School of Natural and Applied Sciences

Prof. Dr. A. Necmeddin YAZICI
Director

I certify that this thesis satisfies all the requirements as a thesis for the degree of Master of Science.

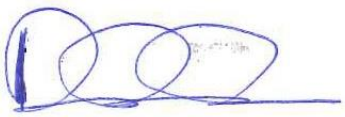
Prof. Dr. Abdulkadir ÇEVİK
Head of Department

This is to certify that we have read this thesis and that in our consensus opinion it is fully adequate, in scope and quality, as a thesis for the degree of Master of Science.

Assoc. Prof. Dr. M. İshak YÜCE
Supervisor

Examining Committee Members	Signature
Prof. Dr. Hanifi ÇANAKÇI	.....
Assoc. Prof. Dr. M. İshak YÜCE	.....
Assist. Prof. Dr. Abdullah MURATOĞLU	.....

I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all materials and results that are not original to this work.



Tsiry Tsilavo RAJAONARY

ABSTRACT

DESIGN AND OPTIMIZATION OF A HYDROKINETIC TURBINE DUCT WITH CFD

Tsiry Tsilavo RAJAONARY
MSc. In Civil Engineering
Supervisor: Assoc. Prof. Dr. M. İshak YÜCE
September 2016
85 pages

Duct is a special mechanism used to control to flow through turbines. Studies on ducts emerged at the beginning of the 20th century and they were focused on wind turbines. Though hydrokinetic technology is now starting to gain its place in the market, only few attempts have been made to implement duct in their designs. Most of the efforts done on studying ducts follow the traditional method of trial and error which is very time consuming and may lack in accuracy. In the present study new attempt has been made to optimize the output power of an hydrokinetic turbine using a duct mechanism. Though works on duct optimization are available, this study takes an unusual approach in the field of hydrokinetics. Adjoint method combined with mesh morphing tools present in some CFD software were used to optimize an airfoil based cross section duct. All simulations and optimizations have been conducted using ANSYS FLUENT software. At the end of the study, a 40% increase in power output was observed when the hydrokinetic turbine was put inside the designed duct. This result matched closely to the result from theoretical calculations.

Key Words: CFD, Hydrokinetic Turbine, Ducted Turbine, Adjoint Method, FLUENT

ÖZET

CFD İLE HİDROKİNETİK TÜRBİN KANAL TASARIMI VE OPTİMİZASYONU

Tsiry Tsilavo RAJAONARY
Yüksek Lisans Tezi, İnşaat Mühendislik Bölümü
Tez Yöneticisi: Dç. Dr. M. İshak YÜCE
Eylül 2016
85 sayfa

Kanal, türbinlerdeki akışı kontrol etmek için kullanılan özel bir mekanizmadır. Kanallar ile ilgili çalışmalar, 20. yüzyılın başında ortaya çıktı ve genellikle rüzgar türbinleri üzerinde duruldu. Hidrokinetik teknoloji artık pazardaki yerini kazanmaya başlıyor olsa da, sadece birkaç tasarımda kanal uygulaması yapılmıştır. Kanallar üzerine yapılan çalışmaların çoğu uzun zaman gerektiren ve tutarsız sonuçlar veren geleneksel deneme yanılma yöntemi ile yapılmıştır. Bu çalışmada hidrodinamik türbin çıkış gücünü optimize etmek için kanal mekanizması kullanan yeni bir yöntem denenmiştir. Kanal optimizasyonu çalışmaları mevcut olmasına rağmen, bu çalışma hidrokinetik alanında alışılmadık bir yaklaşım getiriyor. Kanaldaki Hava-folyosu kesit alanını optimize etmek için Bazı CFD yazılımlarındaki örgü geçişin araçları ile birlikte eşlenik yöntemi kullanılmıştır. Tüm simülasyonlar ve optimizasyonlar ANSYS FLUENT yazılımı kullanılarak yapılmıştır. Çalışmanın sonunda, hidrokinetik türbin dizaynı yapılan kanalın içerisine konduğunda, güçte 40% lik bir artış sağlandığı gözlemlendi. Bu sonuç teorik olarak yapılan hesaplamalara çok yakın bir değerdir.

Anahtar Kelimeler: CFD, Hidrokinetik Türbin, Kanali Türbin, Eşlenik Yöntemi, FLUENT

This work is dedicated to my family especially my parents and also to the people who strive every day in making lives around them a little bit better.

ACKNOWLEDGEMENTS

I would like to thank my supervisor Dr. Mehmet İshak YÜCE for his support, counsel and his patience during the study. He is more than just a supervisor, and for that I very thankful. Also, I would like to thank Dr. Abdullah MURATOĞLU for his help in providing key documents and information.

I would like to thank my parents for their constant support throughout every aspect of my life, without them none of this could have existed.

TABLE OF CONTENTS

ABSTRACT V
ÖZET VI
ACKNOWLEDGMENTS VIII
TABLE OF CONTENTS IX
LIST OF TABLES XI
LIST OF FIGURES XII
CHAPTER 1 1
INTRODUCTION 1
1.1 World Energy Status 2
1.2 Hydrokinetic Technology 5
1.3 Ducted Hydrokinetic Turbines 7
1.4 Motivation and objectives 9
1.5 Thesis organization 11
CHAPTER 2 12
FUNDAMENTAL BACKGROUNDS 12
2.1 Fundamentals of Hydrodynamics 13
2.1.1 Characteristics of Moving Fluids 13
2.1.2 Basic Principles of Moving Fluids 15
2.2 Horizontal Axis Hydrokinetic turbines (HAHT) 17
2.2.1 Energy Extraction Processes 19
2.2.2Properties of HAHTs 23
2.3 Duct Principles 27
2.4 Engineering Optimization 31
2.4.1 Optimization Methods 32
2.4.2 Shape Optimization 34
2.5 Computational Fluid Dynamics (CFD) 35
CHAPTER 3 38
LITERATURE REVIEW 38
3.1 Hydrokinetic Turbines 38
3.2 Turbine Duct 42
3.3 The CFD as Optimization Tool 47

CHAPTER 4 49
ANALYSIS AND DESIGN OF DUCT 49
4.1 ANSYS FLUENT 49
4.1.1 Simple flow simulation 50
4.1.2 Advanced Techniques 54
4.2 Turbine Information 55
4.3 Pre-Validation 56
4.3.1 Meshing Techniques 56
4.3.2 Simulation in 3D 58
4.4 Duct Design 63
4.4.1 Drag Analysis 63
4.4.2 Lift Analysis 67
4.4.3Pressure Drop Analysis 69
4.5 Ducted Turbine Simulation 71
CHAPTER 5 77
RESULTS AND CONCLUSION 77
5.1 Discussion of Results 77
5.2 Conclusions and Future Works 79
REFERENCES 80

LIST OF TABLES

Page

Table 1.1. Project cost comparison of different renewable energy sources 4
Table 2.1 Comparison of horizontal and vertical axis hydrokinetic turbines.....19
Table 2.2 Turbine characteristics with definitions 24
Table 2.3 List of popular CFD software.....36
Table 3.1 Timeline of breakthroughs in turbine analysis.....40
Table 4.1 Hydrokinetic turbine information.....55
Table 4.2 Details on the meshing of the outer and inner parts.....60
Table 4.3 Bare turbine simulation details.....61
Table 4.4 Ducted turbine meshing details 73
Table 4.5 Ducted turbine simulation details.....75
Table 5.1 Rotating bare turbine simulation results (2.7 m/s stream velocity).....77
Table 5.2 Turbine characteristics.....78
Table 5.3 Final comparison between bare and ducted turbine.....79

LIST OF FIGURES

Page

Figure 1.1 Primary energy consumption 3
Figure 1.2 World energy consumption by source, 1990-2040 3
Figure 1.3 Global new investment in renewable energy by asset class 4
Figure 1.4 New investment in renewable energy by sector, 2015, $BN 4
Figure 1.5Pelamis wave energy converter 6
Figure 1.6Boat propeller 6
Figure 1.7Horizontal and vertical axis hydrokinetic turbine 6
Figure 1.8 Water wheel 6
Figure 1.9 Classification of hydrokinetic system 7
Figure 1.10 Hydrokinetic turbine classification 7
Figure 1.11 Ducted horizontal axis hydrokinetic turbine 8
Figure 1.12 Ducted vertical axis hydrokinetic turbine 8
Figure 2.1 Branches of fluid mechanics 13
Figure 2.2. Main components inside the nacelle of horizontal axis turbine 18
Figure 2.3 Cross section of a turbine blade 20
Figure 2.4 Control volume in the actuator disc model 20
Figure 2.5 Power coefficient as a function of TSR for a two-bladed rotor 25
Figure 2.6 The workings of an airfoil 26
Figure 2.7 Duct with airfoil cross section 28
Figure 2.8 Venturi effect 29
Figure 2.9 Lift-generated circulation 29
Figure 2.10 Reduction of tip vortices using duct 30
Figure 2.11 Illustration of gradient-based methods 33
Figure 2.12 CFD methodology 35
Figure 3.1 First wind turbine used to produce electric power 39
Figure 3.2 Inclined Garman Turbine 41
Figure 3.3 Ducted windmill with flow augmenter 43
Figure 3.4 The Vortec 7 45
Figure 3.5 CAD model and wind-tunnel model of the new generation of Vortec 7 .45
Figure 3.6 Schematic view of flow mechanism around a flanged diffuser 46
Figure 4.1 FLUENT – User friendly 50
Figure 4.2 Solver and model setup 51
Figure 4.3 Boundary conditions and solution methods task page 52
Figure 4.4 Solution initialization and solver iteration 53
Figure 4.5 CAD model of the hydrokinetic turbine 55
Figure 4.6 Result of mapped face meshing around NACA 0015 56
Figure 4.7 Result of triangular meshing with inflation layers around NACA 0015 .57
Figure 4.8 Pressure coefficient of NACA 0015 at 0° angle of attack 57
Figure 3.9 Outer domain details 59
Figure 4.10 Inner domain details 59
Figure 4.11 Meshing details on the outer domain 60

Figure 4.12 Meshing details on the inner domain 60
Figure 4.13 Pressure contour around turbine 61
Figure 4.14 Pressure contour on all flow domain 62
Figure 4.15 Velocity contour on all domain 63
Figure 4.16 Flow solving steps in adjoint solver 64
Figure 4.17 Adjoint sensitivity to body force x-component contours 64
Figure 4.18 Adjoint sensitivity to mass sources contours 65
Figure 4.19 Sensitivity of drag to change in shape 65
Figure 4.20 Drag coefficient variation throughout drag optimization 66
Figure 4.21 Adjoint sensitivity to mass sources contours 67
Figure 4.22 Sensitivity of lift to change in shape 68
Figure 4.23 Lift coefficient variation throughout lift optimization 68
Figure 4.24 Drag and lift optimized airfoil 69
Figure 4.25 Airfoil rotation to increase the drop in pressure 69
Figure 4.26 Parametric study of pressure drop 70
Figure 4.27 Variation of pressure with full body rotation of airfoil 71
Figure 4.28 CAD model of ducted turbine 71
Figure 4.29 2D representation of ducted turbine with dimensions 72
Figure 4.30 Detail of ducted turbine outer domain 72
Figure 4.31 Detail of ducted turbine inner domain 73
Figure 4.32 Mesh of the outer domain of the ducted turbine 74
Figure 4.33 Mesh of the inner domain of the ducted turbine 74
Figure 4.34 Pressure contour on the ducted turbine 75
Figure 4.35 Pressure contour on all flow domain 76
Figure 4.36 Velocity contour on all flow domain 76

CHAPTER 1
INTRODUCTION

For the past six millennia, human being has been experiencing rise and decline in its civilization. This rise and fall is something natural to planet Earth. A very fast rise started from the late 19th century and still continues until now. This climb started with the discovery of electricity and the usage of fuel to energize industries and other machines. This event intersected with the industrial revolution which gave this rise in civilization a huge boost. The quest of better life and other comforts made human being dependent to these sources of energy which now play a major and crucial role in their daily lives. The more the quality of life increases the less the rate of mortality, this in its turn leads to increase in population i.e. energy demand. Because fuel is abundant and its extraction and refinement is cheap, this source of energy is the best candidate to serve societies. Fuel is used to facilitate daily lives, including electricity generation. The availability of electrical energy made so many types of technologies to emerge. The electricity need increased very fast during the past few decades. Though this revolution improved civilization exponentially, it is believed also to be a source of downfall. Because the main source of electricity comes mainly from conversion of fossil fuel energy they are directly proportional. The problem arises because, the residuals of fuel and other carbon based energy sources are polluting the environment and atmosphere. To help understanding more about this situation data and other materials related to current world energy status are presented in Section 1.1.

The acknowledgment of the risk of this problem led people from different field of studies to find solutions. Engineers and scientists on their side tried to invent devices to exploit other source of energy especially the clean and renewable ones.

Dams, wind turbines and solar panels are the most famous tools used nowadays for converting renewable energy into electricity. Other mechanisms and devices are also starting to appear and this thesis will focus on one of them. This recently birthed technology is called hydrokinetic turbine. It started to appear only few decades ago and studies about it are still few. The main goal of the thesis is to improve the output energy of this technology using a special mechanism called duct. More information on hydrokinetic technology and ducted turbines are presented in Section 1.2 and Section 1.3 respectively. The motivations that lead to this thesis together with details of its objectives are described in Section 1.4. Finally, Section 1.5 presents the layout of the rest of the thesis.

1.1. World energy status

Nothing is lost and nothing is added in the universe, it is always in balance. Because energy is a subset of the universe, it also behaves the same way. Energy does not disappear, it only changes from one form to another. This change can be caused naturally or induced using special mechanisms. Though energy is present literally everywhere, its definition can still be confusing. A simple and comprehensible definition of it was taken from [1] which states that energy is the measure of the ability of a body or system to do work or produce change. An example of this can be illustrated in daily lives, a person can be able to move its body because of its internal energy received from food.

Population in the world, 7.4 billion (2016), is currently growing at a rate of around **1.13%** per year. The annual growth rate is currently declining and is projected to continue to decline in the coming years. It is estimated that it will become less than 1% by 2020 and less than 0.5% by 2050 [2]. This decline will slow down the growth rate but the population will still increase at least until 2050. The electrical energy consumption of the world in 2014 (with 7 billion people) was estimated to be 20.45 trillion kWh [3]. To put that into perspective, in 2014 we needed about 5000 Hover Dams, with an annual generation of 4.2 billion kWh [4], to meet that energy demand. This staggering energy need will keep increasing as long as the world population increases. Figure 1.1. gives a

better information about energy consumptions in the world in 2015 and Figure 1.2 presents a past and future projection of world energy consumption based on 2012. From these figures we can easily see that fossil fuels are still and will still be the dominant source of energy at least until 2040.

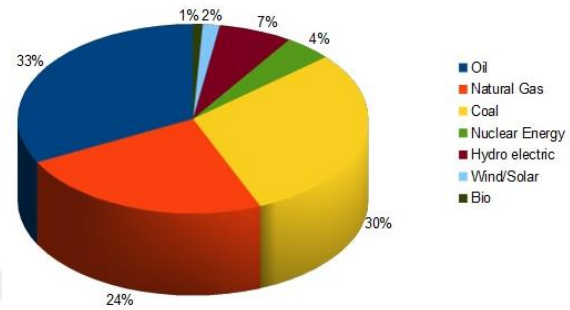


Figure 1.1 Primary energy consumption 2014 [5]

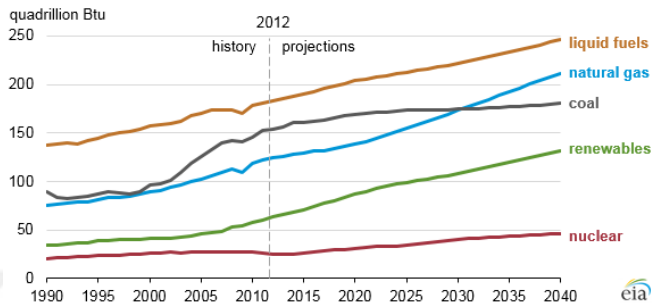


Figure 1.2 World energy consumption by source, 1990-2040 [6]

Beginning from the near end of the 20th century a significant trend on renewable energy was observed. Even though its contribution to the supply of world energy is still low compared to other sources it is rising. According to recent studies, renewable energy excluding large hydro have been estimated to make up 53.6% of the new power generating capacity installed in 2015 which prevented the emission of an estimated 1.5 gigatonnes of CO2 that year. [7]

The need to protect and to make the world a better place made people to think about alternative ways to extract energy. Investment on this subject have been growing for the past few years. In 2015, investment on renewable energy set a new record. . Investment in renewable energy on a global scale for the past decade is summarized in Figure 1.3. A more detailed information about investment in renewable energy in 2015 alone is presented in Figure 1.4.

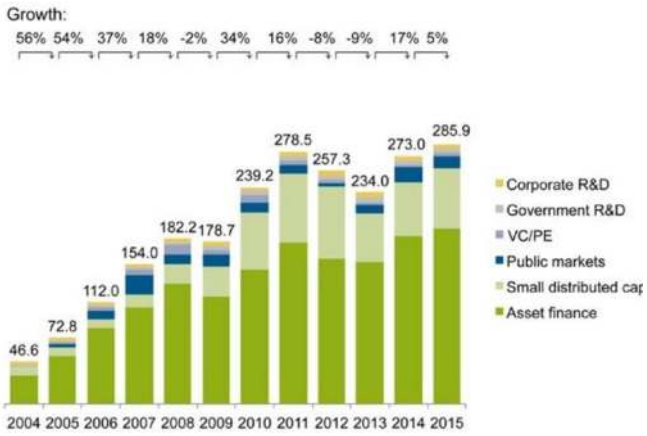


Figure 1.3 Global new investment in renewable energy by asset class, 2004-2015, $BN [7]

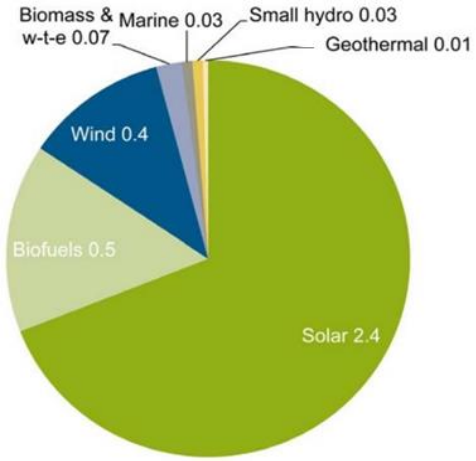


Figure 1.4 New investment in renewable energy by sector, 2015, $BN [7]

Among renewable resources, hydro, dominated by dam usages, still leads in energy output around the world. It leads for two main reasons. The first reason is its availability. All countries in the world have access to either ocean, river or both. To put that into numbers, 76.7% of the 206 countries in the world have access to ocean and 91.3% have river(s) within them. Second, it is cheap. If one compares the project cost of renewable power plants based on their power output, hydro is the cheapest. A simple comparison is illustrated in Table 1.1.

Table 1.1 Project cost comparison of different renewable energy sources

Type	Power Plant	Capacity (MW)	Cost (MN $)	Cost (MN $/MW)
Hydro	Kariba Dam (Zambia)	1319	480	0.36
Wind	Mojave Wind Farm (USA)	1320	1200	0.91
Solar	Topaz Solar Farm (USA)	500	2500	5

Dams are responsible of about 15% of world's electricity generation [8]. Sadly, dam projects can be very costly and that puts developing and especially underdeveloped countries in difficulties in making them. Still, hydropower is a good source of clean energy because it is renewable. Dams are built to convert river potential energy to electrical energy. But apart from potential energy, rivers have another useful energy which is kinetic energy. To harness that form of energy, Hydrokinetic turbines are used. Compared to dams, hydrokinetic turbines installation and maintenances are extremely cheap and with good engineering design and planning, decent energy can be obtained.

1.2. Hydrokinetic technology

When one talks about technology, most people assume that it is all about fast cars and electronic gadgets. Though it is not wrong, it is only partially true. To define it in one single sentence, technology is the application of scientific knowledge for practical purposes. This implies that every usable invention created fully or partially using general principles and scientific backgrounds are all technology. Hydrokinetic technology sums up all devices that interact and work with moving liquid. Hydrokinetic technology has many subcategories. They differ from each other by their functions and workings. The oldest hydrokinetic technology is the water wheel, Figure 1.8. The earliest evidence of a water-driven wheel is probably the Perachora wheel (3rd century BC), in Greece [9]. This device uses flowing water, from river or from man-made canal, as a power source for tools such as mill. By observing turbines used in dams, it is easy to conclude that these recent turbines were inspired by water wheel. In dams the water is stored to obtain higher energy. This energy is then condensed and released using special pipes to rotate the turbines. Other examples of hydrokinetic technology that extract moving liquid powers are wave energy converters, Figure 1.5, and hydrokinetic turbines, Figure 1.7. These devices are very versatile because they can be installed wherever there are exploitable flow of water or ocean. Because water movements occur naturally all over the planet, this characteristic significantly increases the number of potential installation sites and applications possible with hydrokinetic technology [10]. The devices

mentioned earlier uses moving fluid as energy source, on the other hand, device like propeller, Figure 1.6, do the opposite. They use mechanical energy to move liquid which propels the main device.

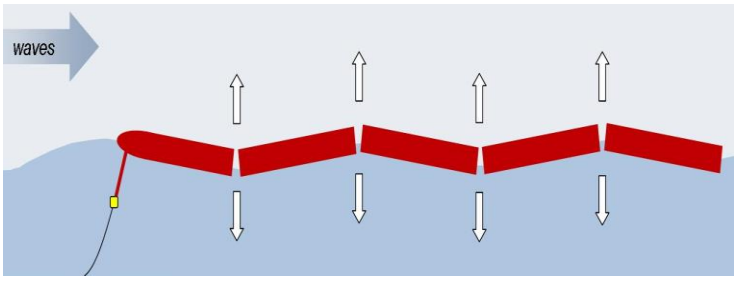


Figure 1.5 Pelamis wave energy converter [11]

Figure 1.6 Boat propeller [12]

Figure 1.7 Horizontal (left) [13]and vertical (right) [14] axis hydrokinetic turbine

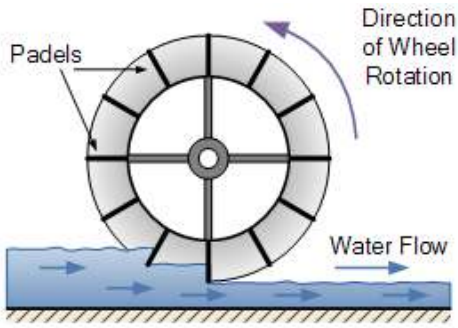


Figure 1.8 Water wheel [15]

This thesis focuses on the energy generating hydrokinetic technology. These devices convert kinetic energy of moving fluid into electrical energy, without building dams, at zero or near zero head. Conversion systems used by hydrokinetic devices is summarized in Figure 1.9. Most instream hydrokinetic devices use turbine(s) as main converting mechanism. Simple classification of hydrokinetic turbines is shown in Figure 1.10.

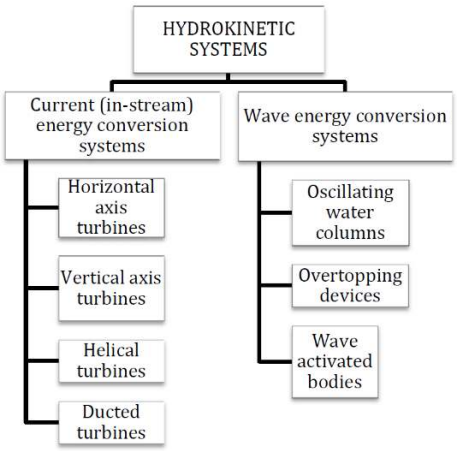


Figure 1.9 Classification of hydrokinetic system [16]

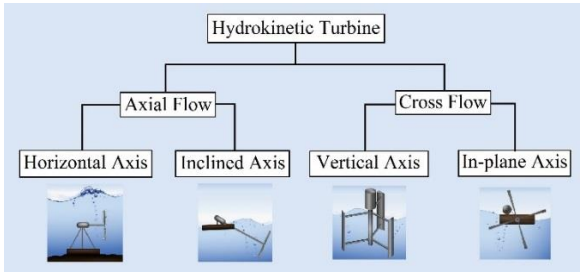


Figure 1.10 Hydrokinetic turbine classification

Almost all hydrokinetic turbines are inspired from wind turbines. Because they both work by converting fluid kinetic energy, aerodynamic principles can be adapted in hydrodynamic analysis and design of hydrokinetic turbines. One of the most well-known theory when it comes to wind turbine is the Betz limit. It states that wind turbines can only capture about 59% of available wind power. Because of their similarities, hydrokinetic turbines are also affected by this. However, if a mechanism is used to increase the airflow or water flow through the rotor disc, the Betz limit can be exceeded [17]. The usage of duct around turbine is an example of that mechanism.

1.3. Ducted Hydrokinetic Turbines

Compared to wind turbines, hydrokinetic turbines have lower power output. The main reason of this contrast is in the turbines sizes. Because wind turbine operates above ground its size can be increased easily. The largest recorded wind turbine has a 165 m diameter and installed at 220 m above the ground. On the other hand, hydrokinetic turbines especially the ones running with river flow are limited by the river and sea beds.

Power output of turbines depends mostly of its size and the velocity of the fluid. By increasing the size of the turbine or the velocity of the fluid, the power output can be boosted. Though the size augmentation of hydrokinetic turbine is not economical and sometimes not feasible, the velocity of liquid passing through it can be increased instread using special mechanisms. Turbine duct or augmentation channels can be used to accomplish that. These mechanisms induce a sub-atmospheric pressure within a constrained area and thereby increase the flow velocity. If a turbine is placed in such a channel, the velocity around the rotor would be higher than that of a free rotor [18]. Ducts can be installed in both horizontal axis, Figure 1.11, and vertical axis, Figure 1.12, hydrokinetic turbines.

Figure 1.11 Ducted horizontal axis hydrokinetic turbine [19]

Figure 1.12 Ducted vertical axis hydrokinetic turbine [20]

Hydrokinetic turbine technology is young but ducting technology is not. Studies and researches on ducts started many decades before the appearance of hydrokinetic turbines. Though it was not a big hit during the last century, statistics show that many hydrokinetic turbines are using duct(s). According to Khan et al. [21] 33% of reported axial flow turbines and 48% of cross flow turbines have duct mechanisms. There are a number of companies at the prototype stage in their development of ducted tidal turbines [22]. Examples of that include OpenHydro (Ireland), in France there is Alstom and Canada Clean Current.

Though augmentation mechanisms are encouraging, factors like design and material costs, workability, mooring structures should be taken into account while choosing between bare and ducted turbines. More information on ducts are presented with details in Section 2.3 and Section 3.2.

1.4. Motivations and objectives

It is common to say that everything starts from imagination. This is true, but most imaginations and creations starts their journey from real events, situations or problems. One can all agree that we don't need imagination to see and understand the current situation in the world. The word "developed", "developing" and "underdeveloped" are used to categorize countries in this planet. But these words are misleading because they are just generalizations. Nowadays, contrasts can be seen even between the so called developed countries. And the poorest countries in the world are so poor that the word "underdeveloped" is not low enough to categorize them. Some of them do not come even closer to the economy of the richest countries one century ago. But still, the world is developing, things are invented and life is beeing modernized at a very fast pace. Though most of these events take stage in developed and developing countries, they arrive in third world countries almost instantaneously because of globalization. Information are also transmitted in seconds thanks to internet. Countries became intertwined with each other and poor countries are the victims because they are chasing after the rich ones without having the necessary resources. Changes need to be made in order to change this situation if not the more these poor countries run toward the rich ones the more they will fall back. This situation is the major motivation that fueled of this thesis.

Researches and projects on energy extraction and related technologies are mostly done by big companies from rich countries. When the design and analysis of these technologies are done they are mostly priced for rich countries. The poor countries cannot afford them. With the fast growing world, energy is the key to development. To help poor countries have access to energy, efforts have been made to extract energy using cheap technology. And this thesis is one

of those efforts. The results of this study may help people around the world get access to cheap, reliable and sustainable energy. This energy can be harnessed anywhere as long as there is a river having the required parameters. Not only this device will help reduce the damage on the ecosystem, it will develop societies, especially the ones in poor countries, economically.

Quest of cheap and reliable energy has always been one of the biggest challenge of engineers. The growing population, the fast developing technology and the increase of standard of living make the challenge even bigger. The desire to achieve something where others have failed is one of the golden goal of people in general. The desire of searching solution to the challenge mentioned earlier also motivated the writing of this thesis. Compared to other researches in different fields of science and engineering, only few studies have been done on hydrokinetic turbines. Because of that, this study will strengthen the information and experiments that have been done on this subject.

Among these natural resources, water is the most used and dominant. Most of the clean energy obtain from water is harnessed using dams. The bigger the dam, the bigger the energy obtained. But dams have huge impact on ecosystem. Though many measures have been taken to solve this problem most of them are only temporary and some are even ineffective. In the hydro-world, hydrokinetic turbines are promising, they are cheap and the damages they cause to the environment is slim compared to dams. But they lack the key ingredient, which is the energy output. Still, if the main focus of the study is for poor countries, and devices that can generate energy can make a big difference.

The main goal of this thesis is to increase the power output of hydrokinetic turbines using duct. To reach that goal, the effect of duct on hydrokinetic devices was analyzed and new duct was designed and optimized to increase the flow of liquid passing through the turbine. Apart from that this thesis presents a more practical and faster way of optimizing fluid related devices using adjoint based design tool incorporated in the famous computational fluid dynamics (CFD) software ANSYS FLUENT. This optimization technique avoids using trial and error method which involve using different designs. Computers boost the level of understanding by showing details that can be difficult to understand and to see in real life. This thesis shows how to take advantages of that capacity. To

sum up, the objective of this thesis is to use the power of computer software to maximize power production of hydrokinetic turbines, while minimizing the efforts in design and analysis.

1.5. Thesis organization

This thesis was intended to present complete information about the main subject without any interruptions. To do that, the information were organized in such a way that readers will feel continuity between every chapters and sections. Also to give readers ideas about each chapters, at the beginning of a chapter all sections present on that chapter are summarized. To start, fundamentals backgrounds of the studies are presented in Chapter 2. This will give ereaders a sense of control to the subject. Then, literature review which summarizes past researches and breakthroughs related to the study and techniques used in this thesis are presented in Chapter 3. This in its turn, will give readers a starting point in understanding the real problem(s) and to give them even more control on the subject. Chapter 4 explains every steps of analysis, design, simulation and optimization that took place during the study. This chapter will welcome the readers in the problem solving part of the thesis. The illustrations presented in this chapter help a lot in achieving that. Discussion of the results obtained from this chapter together with conclusions are presented Chapter 5. This final chapter helps readers be more confident about what they have acquired from previous chapters, this is where they will receive the fruit of the study. Also, the conclusions can be used as a future guide.

CHAPTER 2
FUNDAMENTAL BACKGROUNDS

Together with innovative ideas, fundamental principles constitute the core of any scientific studies or any studies for that matter. These principles and ideas help scientists and engineers throughout their researches. They are used to define and to analyze problems which help achieve the goal of the study. And sometimes they can be modified to create something new. An example of this can be seen in Shives' studies [22] where the principles of BEM, used to model bare turbines, were used to create a new BEM version which takes into account the presence of duct around turbines. As stated in Section 1.4 the objective of this thesis is to design an optimal shape for a duct to increase power output of hydrokinetic turbines. This objective will be achieved by using the help of recent advancement in technology, optimization tools and optimization techniques. But before diving deep into the main subject, it is necessary to review and to understand the bases. This chapter presents the main ideas and principles used to achieve the objective of this thesis. Every principles, methods and techniques discussed in this chapter have important roles in upcoming chapters.This chapter is divided into five sections. Section 2.1 reviews the fundamentals of hydrodynamics. Section 2.2 explains the workings and elements of horizontal axis hydrokinetic turbines. Section 2.3 provides information about how duct can improve the power output of hydrokinetic turbines. Section 2.4 describes optimization methods used in engineering applications. Finally, Section 2.5 introduces CFD and its major contribution in fluid dynamic analysis and optimization.

2.1. Fundamentals of hydrodynamics

From Figure 2.1 we can see that all fluid studies are sub-branch of fluid mechanics. Fluid mechanics is divided into two studies, fluid statics and dynamics. When it comes to water related study, fluid statics and dynamics are called hydrostatics and hydrodynamics respectively. Hydrodynamics is defined as

thescientificstudyofthemotionoffluids,especiallyincompressibleliquids,underthe influenceofinternalandexternalforces. Hydrodynamics is also divided into two studies, kinetics and kinematics. Kinetics is the study of forces that cause motion (ex. torque, gravity, friction, etc.) and can be classified into two groups; linear and angular motion, kinematics on the other hand, is the study of describing movement (ex. displacement, time, velocity, etc.) [23].

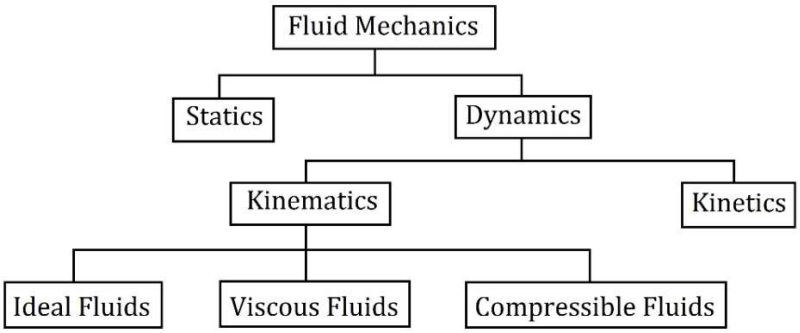


Figure 2.1 Branches of fluid mechanics

2.1.1. Characteristics of moving fluids

The first observable characteristics of a flowing fluid is its flow velocity. Newton's first law of motion says that an unbalanced force affects the velocity of an object. The basic forces that make fluids move are gravity, pressure differences, and surface stresses. The mass i.e. the density of the fluid also plays a role in the motion of the fluid. The higher the density the bigger the force needed to move the object. Density is defined as the mass per unit volume of a substance. Velocity (v), density (ρ) together with the cross sectional area (A) of

an object defines an important characteristics of fluid dynamics which is the mass flow rate ($\dot{m}$) (Equation 2.1).

$$\dot{m} = \rho v A \tag{2.1}$$

In order to describe most of the events occurring in fluid dynamics, it is important understand the concept of viscosity. Viscosity is defined as a measure of the resistance of a fluid, which is being deformed by either shear stress or tensile stress. In more common words, viscosity is the "internal friction" of a fluid, it is the friction between two layers of a fluid when the two layers move relative to each other. Sir Isaac Newton theorized that the shear stress (τ) between layers is proportional to the velocity gradient(∂u) in the direction perpendicular to the layers (y) (Equation 2.2). Here μ is a proportional constant and represents the viscosity of the fluid.

$$\tau = \mu \frac{\partial u}{\partial y} \tag{2.2}$$

Because flow passing through a body, ducted turbine, is the real focus of this thesis, it is important to mention eddy and vortex (eddies and vortices in plural). Though eddy and vortex are two close terms they are not exact. An eddy is defined as the swirling of a fluid and the reverse current created when the fluid flows past an obstacle. The moving fluid creates a space devoid of downstream-flowing fluid on the downstream side of the object then fluid behind the obstacle flows into the void creating a swirl of fluid on each edge of the obstacle, followed by a short reverse flow of fluid behind the obstacle flowing upstream, toward the back of the obstacle. On the other hand, vortex do not create void but only an area of lower pressure, but still the fluid rotates because of a backflow. [24]

Two types of flow, laminar and turbulent, was observed by Professor Osborne Reynolds in 1883. Because laminar flow is a neat flow no eddies are observed. Turbulent flow on the other hand is a disorder flow and occurs with significant eddy formation. Laminar flow occurs when viscous forces are dominant and

turbulent flow occurs when it is negligible. Transitional flow type is located between these two. The measure of the relative importance of viscous forces can be understood from Reynolds number. This number is given by Equation 2.3, v is the velocity of the fluid, ρ its density, μ its dynamic viscosity and l the characteristic length. [25]

$$\boldsymbol{Re = \frac{vl\rho}{\mu}} \tag{2.3}$$

When flow characteristics do not change anywhere with respect to time, the flow is said to be steady otherwaise it is called unsteady. In reality, all flows are unsteady if looked at over a long enough time period, but much can be learned from a study of steady flows. [25]

2.1.2. Basic principles of moving fluids

The main task in fluid dynamics is to find the velocity field describing the flow in a given domain [26]. To accomplish that the basic equations of fluid flow, conservation of mass and momentum, should be used. If the flow is compressing the fluid, conservation of energy must be considered.

According to the principle of mass conservation, mass (and volume) in a control volume should always be constant for incompressible flow. In other words, volume flow rate (Q), which is the product of area (A) and velocity (v), should always be the same at every cross section of the control volume (Equation 2.4).

$$\boldsymbol{Q = Av = A_1v_1 = A_2v_2 = \cdots = A_iv_i} \tag{2.4}$$

The conservation of momentum principle deals with momentum exchanges or acceleration. Newton's law of motion relating mass and acceleration for a solid body can be applied to an isolated body of fluid within a control volume. For a steady flow, the momentum theorem is shown in Equation 2.5 where $\rho$is the fluid's density, $Q$is its volume flow rate and $v$is its velocity.

$$\sum \boldsymbol{F} = (\boldsymbol{\rho Q v})_{\boldsymbol{1}} - (\boldsymbol{\rho Q v})_{\boldsymbol{2}} \tag{2.5}$$

The principle of conservation of energy requires that the total energy of a flow at some downstream location must be equal to the energy at some upstream location minus any energy lost or abstracted (e.g., due to friction or power extracted by a turbine), and plus any energy added (e.g., by a pump) between the two points.

The principles mentioned above are the fundamentals, when it comes to complex studies the form of these equations need to be changed to take into account various phenomena. The most notable forms of these equations are the Bernoulli's equation and the Navier-Stokes equation.

The Bernoulli equation is defined for an incompressible, inviscid fluid on a steady and non-turbulent flow. For such a fluid, the summation of the hydrostatic pressure, the kinetic energy per unit volume and the potential energy per unit volume is a constant. This can be applied to any arbitrary line of flow in the fluid. However, fluids in real don't obey this equation since they are compressible and viscous. Equation 2.6 portrays the Bernoulli equation, here P represents the pressure, ρ the fluid's density, v the fluid's velocity, g the acceleration due to gravity and h the elevation head of the center of the cross section with respect to a datum.

$$\boldsymbol{P_1 + \frac{1}{2}\rho v_1^2 + \rho g h_1 = P_2 + \frac{1}{2}\rho v_2^2 + \rho g h_2} \tag{2.6}$$

The Navier-Stokes equations are a family of equations that fundamentally describe how fluid flows through its environment. But despite Navier-Stokes' importance and wide use across fields, parts of it remain a mathematical mystery. Understanding the unknowns of Navier-Stokes equations is one of the biggest questions in mathematics, which explains why The Clay Mathematics Institute continues offering $1 million to anyone who can solve them. The Navier-Stokes equations are long and complex. To understand it's basics, an equation defining a simple situation is necessary (Equation 2.7). This equation is suitable for simple incompressible constant coefficient of viscosity problem.

$$\boldsymbol{\rho}\left(\frac{\partial \boldsymbol{v}}{\partial \boldsymbol{t}} + \boldsymbol{v}.\boldsymbol{\nabla}\boldsymbol{v}\right) = -\boldsymbol{\nabla}\boldsymbol{p} + \boldsymbol{\mu}\boldsymbol{\nabla}^{\boldsymbol{2}}\boldsymbol{v} + \boldsymbol{f} \tag{2.7}$$

Consider the right hand side of the equation, it shows the various forces acting on the fluid body. In the above equation, the forces are (from left to right): the pressure force which drives the motion, the viscous forces which causes dissipation of momentum and the body forces such as weight or buoyancy.
And to the left hand side (from left to right): the local acceleration and the convective acceleration or space rate of change of velocity, it can also be considered as how the momentum is carried by the flow along the space.

2.2. Horizontal axis hydrokinetic turbines (HAHT)

Turbines having their rotational axis parallel or inclined to the direction of flowing water are classified as horizontal axis turbines or axial flow turbines. These turbines usually feature rotors that resemble aircraft propellers, so they are also called propeller turbines [27]. The main parts of an HAHT are the rotor blades and the nacelle. Nacelle is the enclosed part of a turbine which houses the generating components. All the energy conversion processes take place inside the nacelle. The design of a nacelle depends mostly on its location. For most hydrokinetic turbines, the nacelle is submerged inside water so they must be corrosion resistant and most importantly they must be water resistant. Main components inside the nacelle are the driveshaft, gear box and the generator (Figure 2.2). The drive shaft transmits torque and rotation produced from the

rotor blades to the gearbox. Simultaneously, the gearbox amplifies the rotational speed of the rotor blades using different size of gears. This augmented rotation is then used in the generator to produce electricity. The gearbox plays a significant role in energy production, without the amplification processes the frequency of rotation of the rotor blades is very low to produce any useful electricity.

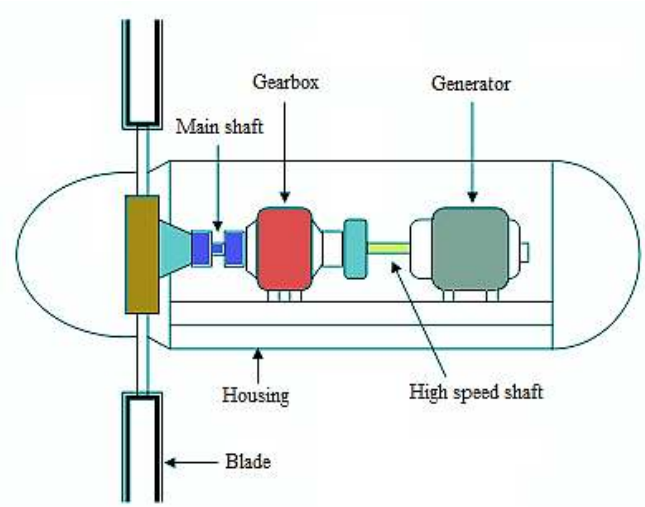


Figure 2.2 Main components inside the nacelle of horizontal axis turbine

Even though horizontal axis hydrokinetic turbines are, in general, more efficient compared to cross flow or vertical axis types, the later shows promising results. In regard to this, it is necessary to make a broad comparison between them. Table 2.1 shows some major pros and cons of the two hydrokinetic turbine classes.

Table 2.1 Comparison of horizontal and vertical axis hydrokinetic turbines

	Horizontal axis	**Vertical axis**
Efficiency	High	Low
Rotational speed	High	Low
Flow direction	Sensitive	Insensitive
Yawn mechanism	Yes	No
Self-start	Yes	No
Area of installation	Large	Small
Torque ripple	Low	Significant
Failure rate	High	Low

2.2.1. Energy extraction processes

Rotor blades are the centerpiece in the energy extraction process of a turbine. These blades convert the kinetic energy of the water into shaft power, which is called torque. This is achieved by slowing down the water as it passes over the blades. The forces which decelerate the water are equal and opposite to the thrust type lifting forces which rotates the blades. The curved shape of the blades also generates lift force perpendicular to the flow direction. The most efficient turbine is the one that can generate the most lift from a given thrust. A representation of a cross section of a turbine blade is shown in Figure 2.3 where, $U_b$is the velocity at blade, U_{rel} is the relative velocity, $r\omega$ is the speed caused by the rotation of the rotor, F_N is the normal force, F_T is the tangential force, F_L is the lift force, F_D is the drag force, φ is the angle of relative wind blow, α is angle of attack and Θ is section pitch angle.

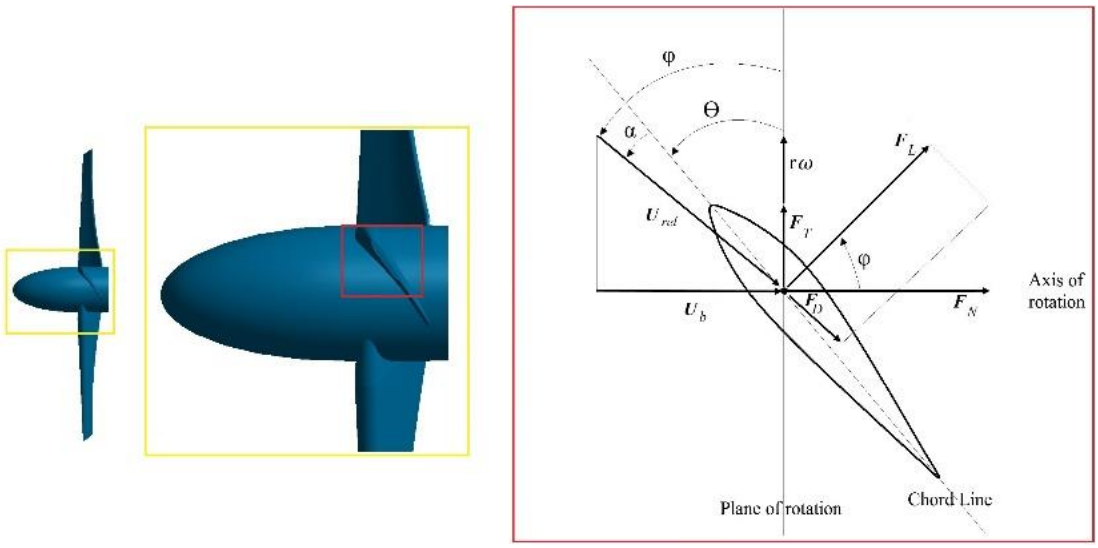


Figure 5.3 Cross section of a turbine blade

A range of models are available for analyzing the performance of turbines. They differ from each other by their complexity and their approaches. A simple model known as actuator disc model can be used to calculate the power output of an ideal turbine rotor and the thrust on the rotor. In this model, the turbine is replaced by a circular disc through which the water stream flows (Figure 2.4).

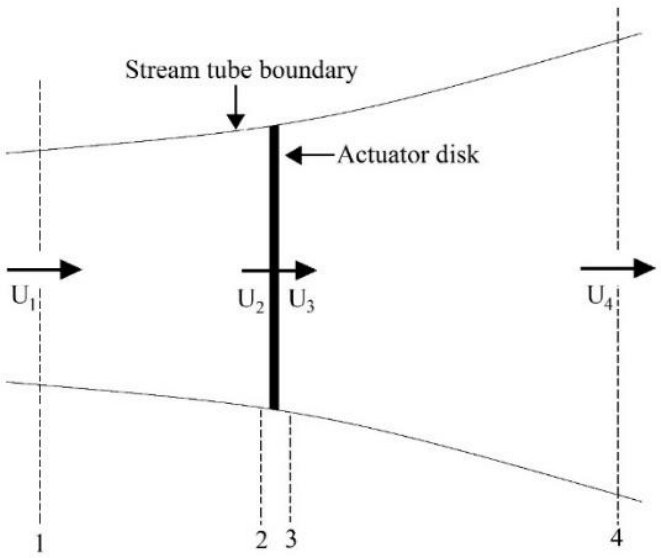


Figure 2.4 Control volume in the actuator disc model

Application of conservation of linear momentum to the control volume yields Equation 2.8 where the thrust T is equal to the opposite of the net force on the contents of the control volume. The Bernoulli function can be used in the two control volumes on either side of the actuator disk (Equations 2.9 and Equation 2.10), since in this model it is assumed that no work is done on either side of the turbine rotor. In these equations, ρ is the water density, A is cross sectional area, U is the velocity of water and p is the pressure.

$$\boldsymbol{T = U_1(\rho AU)_1 - U_4(\rho AU)_4} \tag{2.8}$$

$$\boldsymbol{p_1 + \frac{1}{2}\rho U_1^2 = p_2 + \frac{1}{2}\rho U_2^2} \tag{2.9}$$

$$\boldsymbol{p_3 + \frac{1}{2}\rho U_3^2 = p_4 + \frac{1}{2}\rho U_4^2} \tag{2.10}$$

Rearranging and manipulating the above equations with the help of the following assumptions:

(i) far upstream and far downstream pressures are equal (p_1=p_4),

(ii) velocity across the disk remains the same (U_2=U_3),

(iii) $\dot{m} = (\rho AU)_1 = (\rho AU)_2 = (\rho AU)_4$, for steady state flow where and $\dot{m}$ is the mass flow rate,

(iv) nonrotating wake,

(v) no frictional drag,

Equation 2.11 is obtained.

$$\boldsymbol{U_2 = \frac{U_1 - U_4}{2}} \tag{2.11}$$

In literature the axial induction factor$a$ (Equation 2.12) is often introduced to simplify the analysis. The more complex version of this analysis uses another term which is called angular induction factor, (Equation 2.17). The power in the rotor can be expressed as the product of the trust and velocity (Equation 2.13). Replacing terms in this equation, the power P and power coefficient C_p can be expressed as functions of a (Equations 2.14 and Equation 2.15).

$$\boldsymbol{a = \frac{U_1 - U_2}{U_1}} \tag{2.12}$$

$$\boldsymbol{P = TU_2 = \frac{1}{2}\rho A_2 (U_1^2 - U_1^2) U_2} \tag{2.13}$$

$$\boldsymbol{P = \frac{1}{2}\rho A_2 U_1 4a(1-a)^2} \tag{2.14}$$

$$\boldsymbol{C_p = \frac{Rotor\ power}{Available\ power} = \frac{P}{\frac{1}{2}\rho U_1^3 A_2} = 4a(1-a)^2} \tag{2.15}$$

The maximum C_p is determined by taking the derivative of the power coefficient with respect to a, and setting it equal to zero, yielding $a = 1/3$.

$$\boldsymbol{C_{p,max} = \frac{16}{27} = 0.593}$$

The maximum power coefficient developed by an ideal actuator disk in an open flow condition is around 59.3%, which is known as the Betz limit. The power coefficient limit and its behavior with the ratio of the downstream and the upstream velocities are usually taken as a baseline for the design of wind and hydrokinetic turbines. A more detailed explanation of the actuator disc model can be found in [28]. This method is just a simplification, in reality the rotor blades are rotating. This rotation generates angular momentum which needs to be included in the hydrodynamic analysis of the turbine. The flow behind the rotor or wake rotates in the opposite direction to the rotor, in reaction to the

torque exerted by the flow on the rotor. Thus, a more ideal HAHT analysis should take wake rotation into account. When blades rotation and wake rotation are taken into account, another more realistic equation of C_p emerges (Equation 2.16). In this equation a is the axial induction factor (Equation 2.12) and a' is the angular induction factor, it is expressed as the ratio between the rotational movement of the fluid, w_f, and the blade angular velocity, w_b (Equation 2.17).

$$C_p = \frac{4a(1-a)^2}{(1+a')} \tag{2.16}$$

$$a' = \frac{w_f}{w_b} \tag{2.17}$$

2.2.2. Properties of HAHTs

All horizontal axis turbines work the same way be it wind-driven or hydro-driven. Because they all work with fluids, the methods used in analyzing and designing them can be very similar. Still, every turbine has unique characteristics. These characteristics include thrust coefficient, power coefficient, rated power, tip speed ration and so on. Other tangible features like size, blade number and blade profile, angle of attack, pitch and stall regulation, etc., also differentiate horizontal axis turbines. Table 2.2 provides a simple definition of the important characteristics of a turbine.

Table 2.2 Turbine characteristics with definitions

Characteristic	Definition
Power coefficient	ratio of power produced by turbine divided by the total available power
Angle of attack	the angle between the line of the chord of an airfoil and the relative air or water flow
Stall regulated	when a turbine is stall regulated, its blades are designed so that the rotational speed decreases with increasing flow speed above a certain value
Pitch regulated	pitch-regulated turbines have an active control system that can vary the pitch angle of the turbine blades to manipulate the rotational speed according to the speed of flow
Yawn mechanism	mechanism used to turn the turbine rotor against the flow for better efficiency
Rated power	highest power output of a turbine
Cut in speed	point at which the turbine starts generating electricity from turning
Cut out speed	denotes how fast the turbine can go before wind speeds get so fast that it risks damage from further operation

Most of the characteristics mentioned earlier depends on the blades shape and features. Since the rotor blades are the most important component of a turbine system, designing them should be in the center of hydrokinetic investigations. The energy extraction process presented in Section 2.2.1 was explained just by using the momentum theory but it does not consider the effects of blades geometry. For this reason, blade element theory needs to be added for a more accurate analysis. Blade element theory involves dividing up the blade into a sufficient number of elements and calculating the flow at each one. Overall performances of the blades are then determined by numerical integration along the blade span [28]. The combination of the blade element and the momentum theory yields the well-known Blade Element Momentum theory (BEM), which is widely used in turbine and propeller analyses and designs.

The blade shape profile and the number of turbine blades affect an important characteristics of a rotating turbine which is its tip-speed-ratio (TSR). Similar to wind turbines, the power coefficient (C_p) of a hydrokinetic turbine depends on its tip-speed-ratio. TSR is the ratio of the speed of the rotor tip to the water flow

velocity (Equation 2.18). If the turbine blades rotate slowly, it allows too much water to pass through undisturbed, which prevents it to extract as much energy as it potentially could. On the other hand, if it rotates too fast, the turbine may act as a large flat rotating disc, which creates a large amount of drag.

$$\lambda = \frac{\Omega R}{v} \tag{2.18}$$

In order to get an optimum design for a turbine, there must be enough delay between two blades passing through the same location so that new water can enter and the next blade can harness the power from that new water, not the used, turbulent water. Apart from its effect on TSR, number of turbine blades also affect the overall performance of the turbine. Figure 2.5 depicts the Betz, ideal constant, and actual wind turbine power coefficient as a function of the TSR.

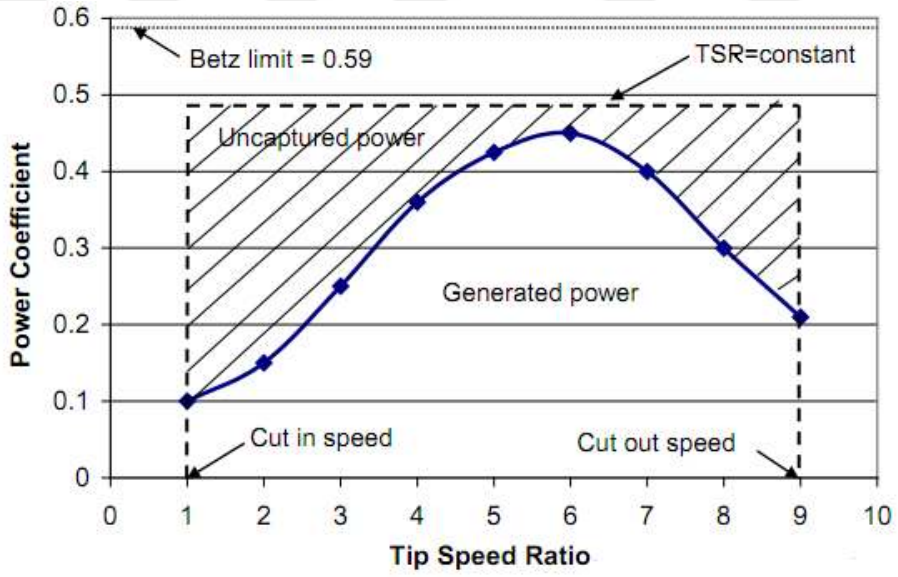


Figure 2.5 Power coefficient as a function of TSR for a two-bladed rotor [29]

From here, it is obvious that the TSR of a turbine play a crucial role in power output. As seen earlier, this ratio depends on the rotational speed of the blades. To understand more about how blades rotate, the concept of airfoil should be looked closer. Figure 2.6 will be used to explain how it works.

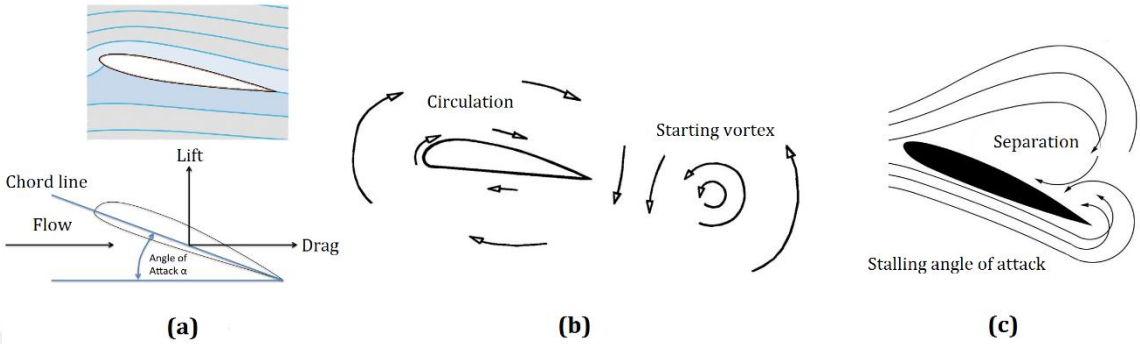


Figure 2.6 The workings of an airfoil

The special shape of the airfoil (Figure 2.6 a) provides lift by creating a situation where the pressure above the wing is lower than the pressure below the wing. To catch up with the lower part of the fluid under the airfoil, the upper part has to accelerate because it travels a longer distance. According to Bernoulli's equation, because the velocity of the fluid below the wing is lower than the velocity of the fluid above the wing, the pressure below the wing is higher than the pressure above the wing. Since the pressure below the wing is higher than the pressure above the wing, there is a net force upwards. To create this pressure difference, the surface of the wing must satisfy one or both of the following conditions. The wing surface must be cambered (curved) and/or inclined relative to the flow direction. There are many different types of airfoils, some are more appropriate in a certain situation than others. [30]

Viscosity is essential in generating lift. The effects of viscosity lead to the formation of the starting vortex, which, in turn is responsible for producing the proper conditions for lift. Because the angular momentum should be satisfied, an equivalent motion should exist to oppose the vortex movement that starts to rotare counter-clockwise. A circulation around the wing takes form (Figure 2.6 b).

Lift increases together with the increase in angle of attack. However, stalling occurs when this angle become very large. During this situation a decrease in lift is observed. The phenomena responsible for stalling is flow separation (Figure 2.6 c). Flow separation occurs when the fluid flow does not follow the surface of the airfoil.

2.3. Duct principles

There is no debate as to whether using ducts can improve the power production of turbines of a given rotor diameter in an unbounded flow, this is well established [22]. This increased power is obtained from:

- a higher axial velocity through the turbine by controlled contraction and/or diffusion and by other duct features
- a reduction in tip loss.

A duct is then said to be optimum if it can induce a decent amount of velocity to the flow and if it can reduce the losses at the tip of the blades. Before entering into the details, it is important to mention that the duct that will be analyzed and designed in this thesis has an airfoil based cross section (Figure 2.7). This shape was chosen because researches on airfoil based duct have been made in the past. Those researches provides vital aid to the thesis because results obtained from the thesis can be compared easily with the earlier results. Also, because of its shape which provides key features required for an optimum duct, this type was chosen.

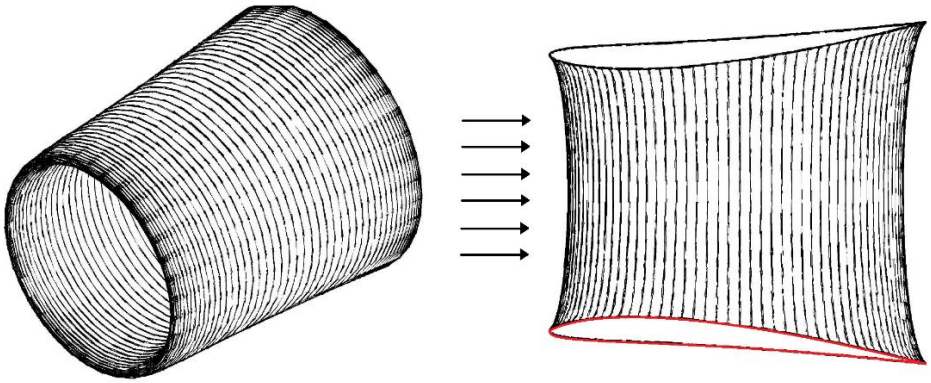


Figure 2.7 Duct with airfoil cross section

To get the idea behind how the velocity inside a duct can be increased, the venturi effect and the principles related to it should be understood. Venturi effect is the reduction in fluid pressure that results when a fluid flow through a constricted area. According to the principle of mass continuity, a fluid's velocity must increase when it passes through a constriction and consequently its static pressure must decrease in accord with the principle of conservation of mechanical energy (Figure 2.8). If one observes the duct in Figure 2.7, it can be observed that from its far left up until about 1/3 of its length there is a contraction. Let's assume a water flow from the left of that duct. Because of the contraction, if a turbine is put at that location (about 1/3 length from left) it will rotate faster because the flow rate at that location is higher compared to the flow rate in bare condition. Apart from the contraction upstream, the expansion on the downstream side of the turbine will also help increase the flow rate through the throat of the duct. This happens because the forced expansion reduces the pressure on the downstream side of the turbine.

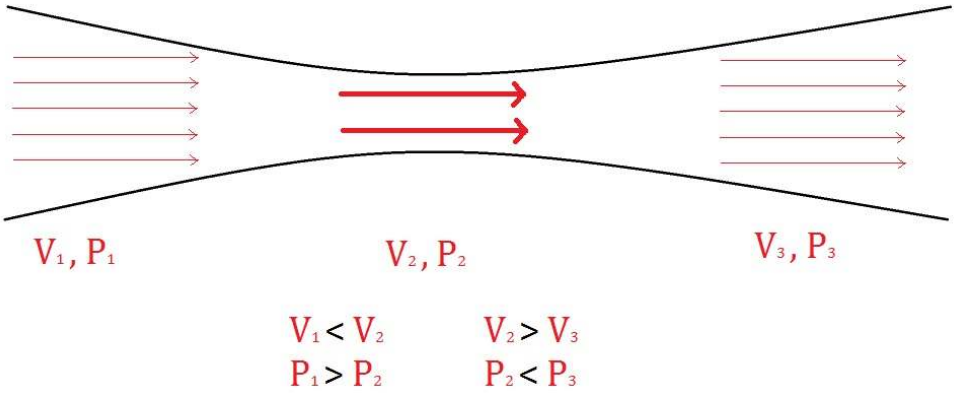


Figure 2.8 Venturi effect

If the cross-section of the diffuser is shaped as an airfoil, the generated lift will give the circulation of a ring vortex [17] (Figure 2.9). The more the lift, the more the water will be sucked through the duct. This additional flow will also contribute in the additional power output of the turbine because it increases the stream velocity at the turbine location. Still, everything has limit, in this case the separation of the boundary layer of the duct wall limits the amount of lift.

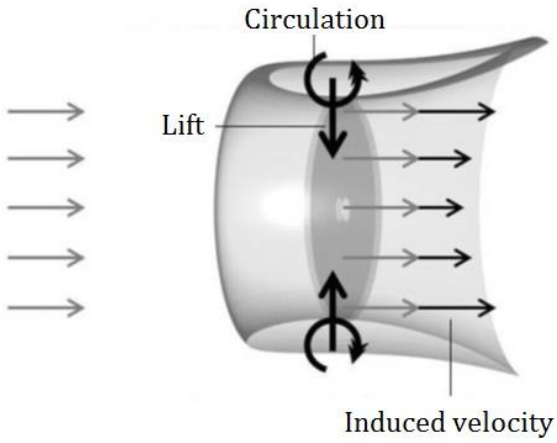


Figure 2.9 Lift-generated circulation [22]

The tip vortex of a wind turbine rotor blade originates as a result of a complex distribution of vorticity along the blade tip thickness [31]. To get a basic idea about it, a straightforward explanation is necessary. When turbine rotates, the centrifugal force pushes the water to the edge of the blades. Because of the pressure difference between the faces of the blades, the water at the tip tries to

flow from high pressure part to the low part. The water at the edge trying to flow from low to high pressure will create vortices at the tip of the blades. This flow reduces the pressure difference at the faces of the blades which leads to the reduction of lift. Because of that tip vortices equal wasted energy. In case of wind turbines, the energy that goes into creating these vortices is transformed into noise and heat. When a duct is put around the turbine, leaving a small gap between them, the border of the duct blocks this motion so the vortices become almost non-existent depending on the amount of gap between the duct and the blade (Figure 2.10). In this case, the lift becomes stronger because the vortices do not cancel out the low pressure on the other side of the blade, so more power is produced. Apart for increase in lift, a decrease in drag can be anticipated. Because of the swirling nature of a vortex, it increases the drag force on the device. Because the duct can almost eliminate vortices, it can reduce the amount of drag on the turbine. According to Spera [32], six percent reduction in drag can be gained by the elimination of blade tip vortices.

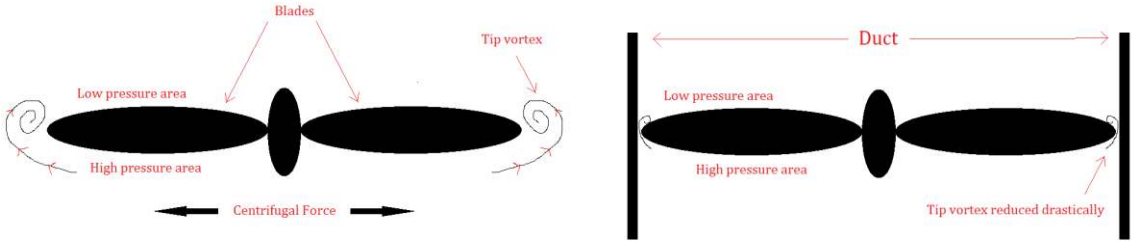


Figure 2.10 Reduction of tip vortices using duct

There are other minor benefits in putting duct around turbine. The duct may attenuate some of the turbulence present in the ambient flow reducing fatigue loading on the blades. It may also help to better align off-axis flows with the rotor plane, which could eliminate the need for a yawing mechanism. Many types of duct have been created for different kinds of situations and purposes. Information on those will be provides in Section 3.2.

2.4. Engineering optimization

Engineering optimization is a part of a series of steps called Engineering Design Process that engineers follow to solve engineering problems. This design process consists of five steps, problem specification, gathering of information, generation of multiple solutions, analysis and solution selection and solution testing. Questions are asked, in the first step, to identify the design problem. Appropriate solutions are then presented. During the second step, information related to that problem are collected. Information should consist of materials such as available solutions related to the problem. Records of attempts on solving the problem should also be collected. When sufficient information are collected, then step three is carried out. In this step, the information are interpreted and engineers brainstorms to find solutions. These solutions are then analyzed in step four. At the end of this step the best solution is selected and later it will be tested in step five. When step five is over, an informed decision can be made. If the results are good enough, the design can be finalized. If not, then the design is must be improved. This design improvement is called optimization.

To define an optimization problem, the following questions should be answered:

- What performance targets must be met? (constraints)
- If all constraints are satisfied, what should be minimized/maximized? (objective)
- What characteristics are relevant to my problem? (responses)
- What should I operate on to achieve my target? (variables)

In the simplest case, an optimization problem consists of maximizing or minimizing a real function by systematically choosing input values from within an allowed set and computing the value of the function [33]. Before starting a review on optimization techniques, the standard form of the general optimization problem should be presented. The standard form for a single-objective, non-linear, constrained optimization problem is provided in Equation 2.19.

$$\begin{aligned} &\textbf{Minimize} && : && f(x) && && (2.19)\\ &\textbf{subject to} && : && g_j(x) \le 0 && j=1,m\\ &&&&& h_k(x) = 0 && k=1,p\\ &&&&& x_{iL} \le x_i \le x_{iU} && i=1,n \end{aligned}$$

In this equation, $f(x)$represents the objective function, $g_j(x)$ and $h_k(x)$ represent an inequality and equality constraint functions respectively. The $x$vector represents the design variables that are modified to obtain the optimum. The searchable design space is defined by the upper and lower bounds,$x_{iL}$and x_{iU}, of the design variables, referred to as the side constraints. [34]

2.4.1 Optimization methods

In engineering applications optimization problems can be handle with different methods. In general, these methods fall into two categories, heuristic methods and gradient-based methods. Heuristic methods are commonly known as global methods and gradient-based methods as local methods.

Global optimization algorithms typically make use of a set of design points, generally referred to as a population, to find the optimum design. Because of the randomness used in these methods, they are also referred as stochastic methods. These methods are typically inspired by some phenomena from nature and have the advantage of being extremely robust, having an increased chance of finding a global or near global optimum, being easy to implement and being well suited for discrete optimization problems. The big drawbacks associated with these algorithms are high computational cost, poor constraint-handling abilities, problem-specific parameter tuning and limited problem size. [34]

Currently, two of the most popular evolutionary algorithms are the more established GeneticAlgorithm [35], which was inspired by Darwin's principle of survival of the fittest, and Particle Swarm Optimization [36], which is based on a simplified social model. Because the optimization technique used in this thesis is gradient based, no further information about heuristic methods will be presented.

A gradient is the observable change in the magnitude of a property (e.g pressure, temperature) happening between two different locations or moments. Gradient-based optimization techniques use gradient information to find the optimal solution of Equation 2.19. Unlike the global methods, where a set of design points are used to find the optimum, local methods update single design point from one iteration to the next using gradient information. As indicated by the name, local methods can only locate a local optimum, theyhave difficulty solving discrete optimization problems, they are complex algorithms that are difficult to implement efficiently, and they may be susceptible to numerical noise [34].

To reach the optimum, two steps are generally used in gradient-based algorithms. This process can be understood using Figure 2.11. The boy in this figure has to reach the highest point on the hill. Because this is an analogy to a gradient-based algorithm, the boy can reach the hill by taking two steps at a time, one in the x-direction and one in the y-direction. From these steps, the boy can obtain information about his location and can estimate a direction that would take him uphill.

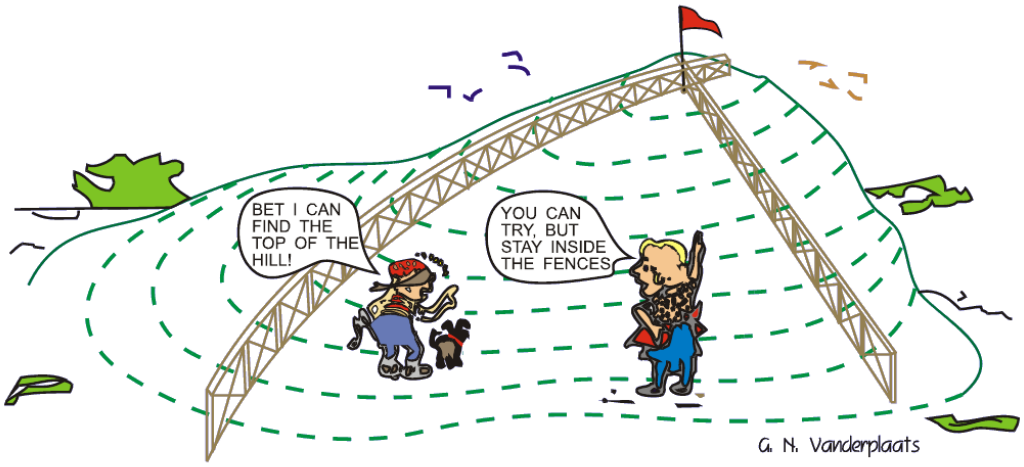


Figure 2.11 Illustration of gradient-based methods [37]

2.4.2 Shape optimization

Because gradient-based methods can only locate local minimum, it is best used to optimize shapes that are close to optimum. An example can be the maximization of the lift to drag ratio of an airfoil. When it comes to shape optimization, a common method used in gradient-based optimization is the adjoint approach. Adjoint method uses adjoint models to achieve its goals. Adjoint models are powerful tools for many studies that require an estimate of sensitivity of model output with respect to input. Actual fields of sensitivity are produced directly and efficiently, which can then be used in a variety of applications. [38]

Given the solution x of a discretized Partial Differential Equation (PDE) or some other set of M equations parameterized by P variables p (design parameters, a.k.a. control variables or decision parameters), one often wishes to compute some function $g(x,p)$ based on the parameters and the solution. For example, if the PDE is for a fluid, one might wish to know the flow rate somewhere. Often, however,more than just the value of $g$are wanted, its gradient $\frac{dg}{dp}$ may also be wanted. Adjoint methods give an efficient way to evaluate $\frac{dg}{dp}$, with a cost independent of P and usually comparable to the cost of solving for x once. The gradient of g with respect to p is extremely useful. It gives a measure of the sensitivity of the answer to the parameters p. Or, one may want to perform an optimization of g, picking the $p$that produce some desired result; in this case the gradient indicates a useful search direction. For large-scale optimization problems, the number P of design parameters can be hundreds, thousands, or more—this is common in shape or topology optimization, in which p controls the placement and shape of arbitrary blobs of different materials constituting a given design. Sometimes, this process is called inverse design: finding the problem that yields a given solution instead of the other way around. [39]

2.5. Computational fluid dynamics (CFD)

Computational fluid dynamics (CFD) is a tool used to analyze and solve fluid mechanics problems. To solve the problems, CFD uses numerical analyses and algorithms and most importantly the power of computers. Besides, conducting all calculations and analyses, computers help one to understand fluid flow phenomena by showing visual contents. CFD makes it possible to simulate real life experiments which saves valuable time and money. Though the results obtained from CFD simulations can be very accurate, they must be compared with experimental results or theoretical results.

The processes that a typical fluid mechanics analysis follows are similar in all CFD software. First the flow boundaries and the body/bodies present within these boundaries are drawn using CAD software. CFD software solve fluid mechanics problems by approximating equations in every element inside the boundaries. To do that, the space must be partitioned into elements, this step is called meshing. The number of elements is directly proportional to the accuracy and quality of the simulation but unfortunately it is also directly proportional to the computational time. Meshing is so crucial in a CFD simulation so much effort should be spent in generating a good quality mesh. After the meshing is done, the last step in this process can be initiated. In this step, information about the flow, materials, boundary conditions and other details related to the simulation are inputted. The simulation is done and said to be successful when it converges. The results from this step can then be visualized and manipulated to give details and information about the simulation. Most CFD software also provide tools to create animation to give even more understanding about the situation.

A summary of CFD methodology is presented in Figure 2.12.

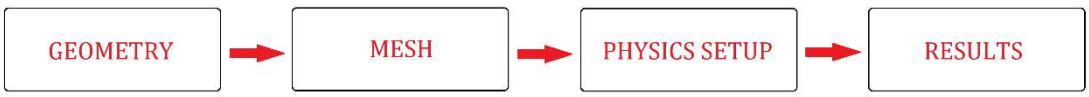


Figure 2.12 CFD methodology

Almost all non-commercial CFD software are open source, which means that the original codes are available and can be modified. Software like OpenFoam and SU² are written in C++ which makes them easy to be improved. Though these software are cheap to get, most of them need third party software for other tasks like meshing or visualization. Generally, these software are not compatible with each other which can lead to inaccurate results or even crashed simulation. The commercial ones do not have such serious problems. They are all equipped with CAD tool, meshing tool and other tools.

To take analyses and designs a step further, some CFD software are equipped with optimization tools (Table 2.3).

Table 2.3 List of popular CFD software

	CDF SOFTWARE	Optimization tool
Commercial	ANSYS Fluent/CFX	Yes
	CD-Adapco	Yes
	Solidworks Flow Simulation	Yes
	Autodesk CFD	Yes
Non-Commercial	OpenFoam	Yes
	SU2	Yes
	OVERFLOW	n/a

Traditionally, engineers try to achieve an optimum design using a time consuming trial and error process based on engineering experience and product knowledge. The well-crafted usage of computer power by CFD software help reduce that time drastically. ANSYS FLUENT can be a good example of that. The adjoint solver in ANSYS FLUENT accomplishes the remarkable feat of calculating the derivative of a single engineering observation with respect to a very large number of input parameters simultaneously via a single computation [40]. Example of an observable can be the airfoil lift or drag, the pressure drop can also be set as an observable.

This ability together with the mesh morphing feature, also present in ANSYS FLUENT, makes optimization even faster. More information on adjoint solver and mesh morpher are presented in Section 3.3 and Section 4.2.

CHAPTER 3
LITERATURE REVIEW

John Muir said, “Don’t try and reinvent the wheel – just work on making it better than anyone else.”. The main idea behind this is that valuable time and efforts shouldn’t be wasted on creating something that is already created. The right thing to do is make existing inventions more beautiful and/or more efficient. But still, to accomplish that, it is crucial to understand how the object in question works, how it was made. It is also important to have information about how other people tried to improve it, what techniques they used. The study and analysis of those information is called literature review. This review is vital for any kind of studies because it gives researchers a control to the subject.

This chapter reviews important works related to the main subject of this thesis which is the design and optimization of an horizontal axis hydrokinetic turbine duct using CFD. This chapter is divided into three sections. In Section 3.1, works done on hydrokinetic turbines are reviewed. Section 3.2 reviews the evolution of turbine duct since its first recorded applications in turbines. Section 3.3 reviews optimization studies that used CFD as optimization tool.

3.1. Hydrokinetic turbines

Energy from sun and wind have been the most common types of clean energy that everybody knew, but there is another abundant renewable resource covering 75% of the planet, that is water [15]. The energy from water is considered renewable due to the hydrologic cycle. Kinetic energy of flowing water can be converted into electricity by the usage of hydrokinetic devices.

Though various types of hydrokinetic devices were presented in Chapter 1, only researches and breakthroughs related to horizontal axis hydrokinetic turbines (HAHT) will be reviewed in this section.

The working principles of hydrokinetic turbines are quite similar to wind turbines'. If one goes back in time and observe the evolution of these devices, it can be concluded that hydrokinetic turbines are inspired from wind turbines. Because of that, to make a good review on hydrokinetic turbines past works on wind turbines should also be mentioned.

While it is not exactly known when wind first began to be used as a source of power, it is likely that some form of windmill was used in Japan and China 3000 years ago [41]. Early mills provided significant mechanical energy which was used in a variety of industries from flourmills and water pumps to lumber mills and the processing of various foodstuffs, spices and grains [42]. In 1892 the first wind turbine used to produce electric power (Figure 3.1) was built in Denmark by Poul la Cour. He was a pioneer in the field of aerodynamics, with his wind tunnel experiments, he developed the theory for the ideal windmill blade profile and discovered that the airflow behind the blade was just as important as the wind's forward pressure [43].

Figure 3.1 First wind turbine used to produce electric power [43]

The world's first megawatt wind turbine is built and connected to the power grid in Castleton, Vermont (US) in 1941. The turbine has 75-foot blades and weighs 240 tons [44]. With the fast developing technology, more and more powerful wind turbines were created. Enercon E126, a German made wind turbine, is an example of that, it is capable of generating 7.58 Megawatts by itself [45].

Though technology has improved exponentially for the past century, the methods used in analyzing and designing wind turbines did not change much. Most breakthroughs related to wind turbine design, analysis and modelling happened between the late 19th and the early 20th century. Table 3.1 provides a timeline of those breakthroughs.

Table 3.1 Timeline of breakthroughs in turbine analysis

1865 (Rankine W.J.M.)	**Momentum theory**: mathematical model of an ideal actuator disk
1878 (Froude W.)	**Blade element momentum theory**: mathematical process used to determine the behavior of propellers
1889 (Rankine–Froude)	**Actuator disk model**: approximation of blade element momentum theory
1921 (Betz A.)	**Approximate correction** of "Rankine–Froude actuator-disk"
1926 (Glauert H.)	**Refinement** of blade element momentum theory

One of the first recorded hydrokinetic turbine is a riverine turbine that was developed and field tested by Peter Garman. In 1978 this work resulted in the so-called Garman Turbine (Figure 3.2) which was specifically meant for water pumping and irrigation. Other designs also appeared during the early stage of hydrokinetic technology's naissance. An example of that is the Tyson Turbine which is an Australian design [46].

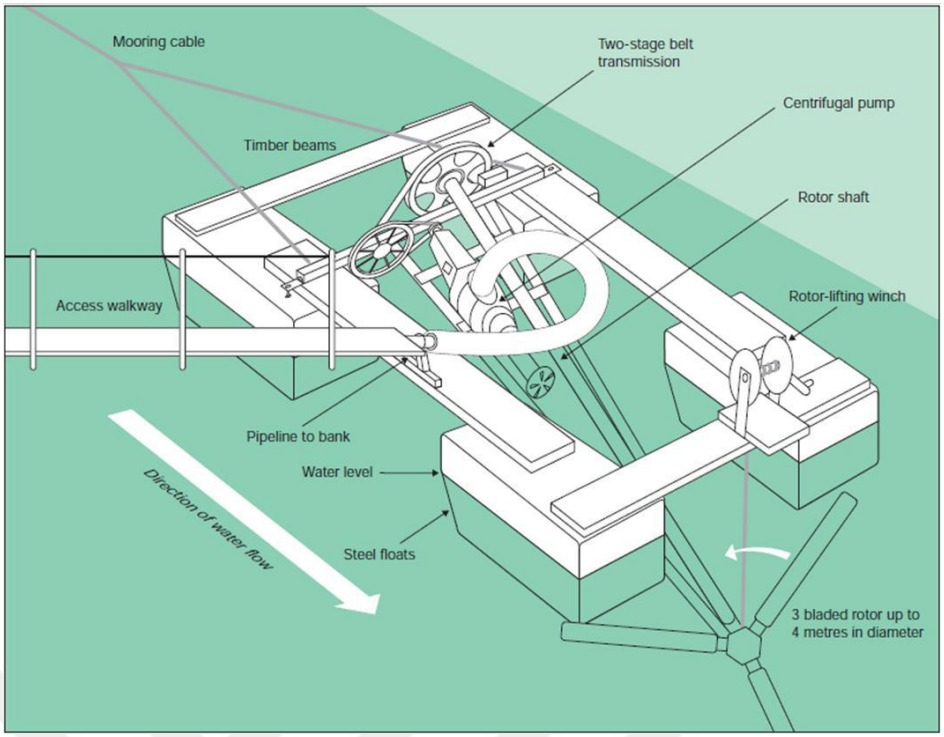


Figure 3.2 Inclined Garman Turbine [47]

One of the most popular underwater HAHT is the SeaGen Turbinedeveloped by Marine Current Turbines Company. It has twin horizontal axis propeller type of rotors and each rotor has 18 m diameter. There is also the Verdant Power Turbineproduced by Verdant Power Company. It is a three-bladed turbine with 5 m diameter and has rated power about 35 kW with high efficiency relative to the other CEC systems over large range of water velocities.

The other popular horizontal axis turbines are; Tidal Stream Multiple, TidEl system, Andritz Hydro Hammerfest, Tidal Generation turbines, also, RiverStar, TidalStar, OceanStar and CurrentStar turbines of Bourne Energy, Scotrenewables and Open Hydro turbines. [48]

Many researches have been made to understand the workings of hydrokinetic turbines. Chica et al. [49] explained the steps involved in the design and numerical simulation of a small horizontal axis hydrokinetic turbine rotors. André et al. [50] presented a general methodology for the efficient design of horizontal axis hydrokinetic turbines with variable rotation. Laurens et al. [51] exploited the known methods used in designing marine propeller to design a marine current turbine. Kolekar et al. [52] used hydrodynamic analyzing tools to optimize hydrokinetic turbines. Ismail and Batalha [53] made a comparative

study on river hydrokinetic turbine blade profiles. These works are just few examples of the works done on HAHTs. Since its first appearance, not much have been changed in term of method and techniques in designing HAHT, only technical and functional improvements were observed. Most of the past achievements on this subject are about things like, cavitation study [54], optimization [52,55], environment impact [56]. The most tangible change that occurred during the past two decades is the usage of computer to simulate these devices. This situation gave studies on turbines a new perspective. Together with momentum theory, the blade element momentum theory, numerical simulation plays now an important role in validating turbine designs and analysis. Since then almost all studies on turbines incorporate computer simulation as a main tool or as a validating tool for the studies. Still, all these efforts have been done to make turbines more robust, more reliable and most importantly more efficient. A different approach used to make turbines more efficient is the usage of duct or diffuser. The review on this topic is presented in the following section.

3.2. Turbine duct

Attempt on ducting turbines to extract more power started in the early 20th century. Most of those early works focused on how to increase the power output of windmills. Though, it has been almost a century since the starting of duct studies, theoretical studies and duct modelling is still far from maturity. One of the reason why this field is progressing slowly is the fact that only few studies have been done on it. Because of this situation, apart from achievements and breakthroughs related to duct designs, modelling techniques used by researchers will also be emphasized in this section.

One of the first pioneer in ducted turbine studies is Sanuki [57]. During his experiments, he used different number of blades together with different settings for the windmill. This windmill was installed inside a cone having 1.3 and 1.1 diameter respectively greater than the windmill diameter. The experiment of Sanuki show that with relatively crude ducting and no effective slipstream diffusion the increase in power output is about 86% for a two bladed windmill

and somewhat less for windmills having larger numbers of blades. This work is important because for the first time experimental results about ducts are given.

In 1953, Isawaki [58] came up with a cylindrical shape diffuser claiming 30% increase from the power output of the unshrouded windmill. His experimental and theoretical work show that the reduction in tip loss due to the presence of the diffuser contributed greatly to that increase.

The first in depth study of ducted turbines was done in the context of wind power by Lilley and Rainbird [59]. They used the simple one-dimensional momentum theory and potential flow methods to compare the performance of the ducted windmill and that of the unshrouded windmill. They concluded from their work that the power output is proportional to the rate of volume flow through the windmill disc for a given disc loading and the performance factor depends on the value of the pressure coefficient at exit i.e. the power output of a ducted windmill will be increased significantly if the diffuser outlet is placer in a lee of an obstacle or by providing a flow augmenter (Figure 3.3). According to their calculation, a ducted windmill that has an internal loss coefficient of 0.15, exit pressure coefficient of -0.15 and a diffuser expansion ratio of 3.5 gives an output power 65% greater that of the ideal unshrouded windmill. Results from their studies also suggested that duct can reduce gust loads on the blade because an uniform flow across the windmill is created by the contraction cone ahead.

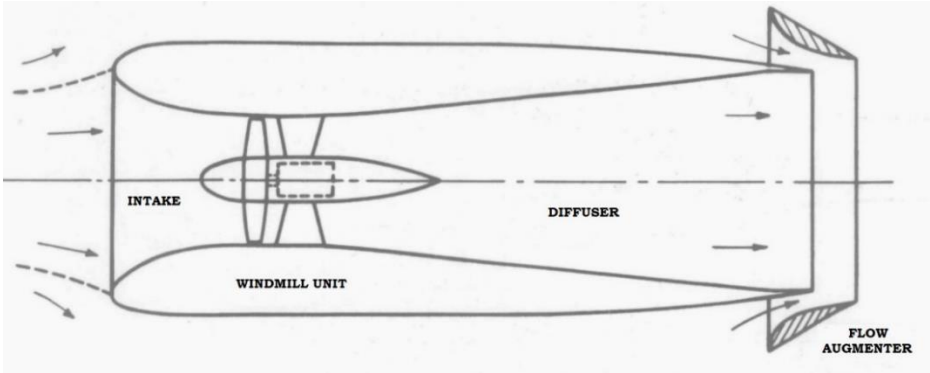


Figure 3.3 Ducted windmill with flow augmenter [59]

Between the late 1970's and early 1980's, extensive work on ducted turbines had been performed by Foreman and his co-workers at the Grumman Aerospace Corporation. This effort resulted with the publication of several papers. To achieve their goals, the first approach they took uses the energetic external wind to prevent separation of the diffuser's internal boundary layer working on DAWT by placing slots to control the boundary layer separation. Another method uses high lift airfoil contours for the diffuser wall shape [60]. At the earlier stage of the study, diffuser model tested have indicated almost a doubling of wind power extraction capability compared to conventional turbines. Apart from successful attempts in reducing the surface area requirements of an efficient diffuser, many parameters that affect the performance of the diffuser system have also been examined in small-scale wind tunnel tests with a family of compact diffusers, using screens and center bodies to simulate the presence of a turbine [61]. Results from [62] demonstrated power augmentation for a given turbine diameter and wind speed by a factor of about 3.5 compared with an ideal conventional system. They claimed that refinements are expected to raise this factor to the range between 6 and 8.

The success of the works done by Forman and his co-workers drew the attention of big corporations. In 1997, New Zealand based company Vortex made Vortec 7, a 17 m tall diffuser augmented wind turbine with a rotor blade diameter of 7.3 m, based on Grumman's design. The initial testing of this device showed a mismatch between the power generated and the power predicted by the researchers. To solve that problem, CFD software called PHOENICS was used [63,64] to understand more about the problem and to make fast optimization. Modifications done on the original Vortec 7 are illustrated in Figure 3.4.

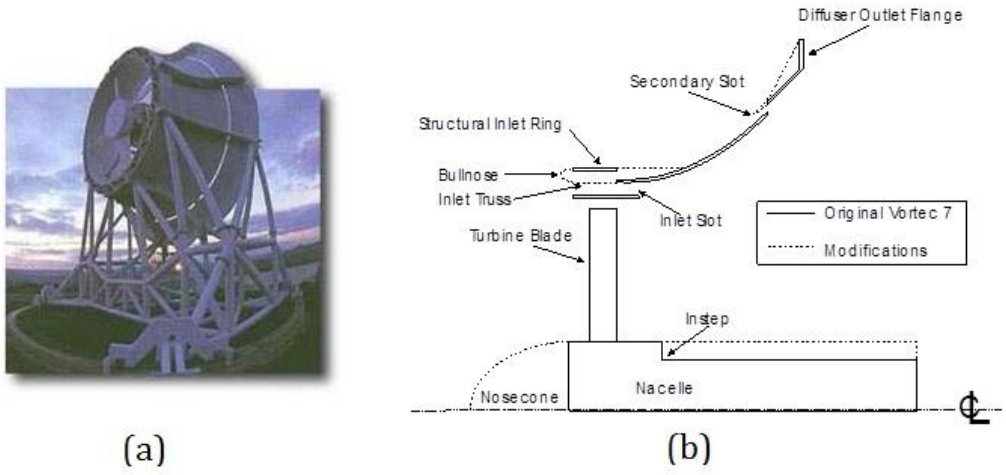


Figure 3.4 Vortec 7 (a) As built: flow is from left to right. (b) Geometric features [64]

Together with CFD simulation and analysis, considerations of the effect of the inlet slot, diffuser outlet flange, nacelle shape and size have been examined using the existing area ratio and diffuser exit angle lead to the development of a new design (Figure 3.5). Increases in power coefficient by a factor of approximately four compared with the original design have been predicted. Much larger inlet speed-ups were predicted with the optimal power coefficient around 1.9 occurring at a local disc loading between 0.3 and 0.4.

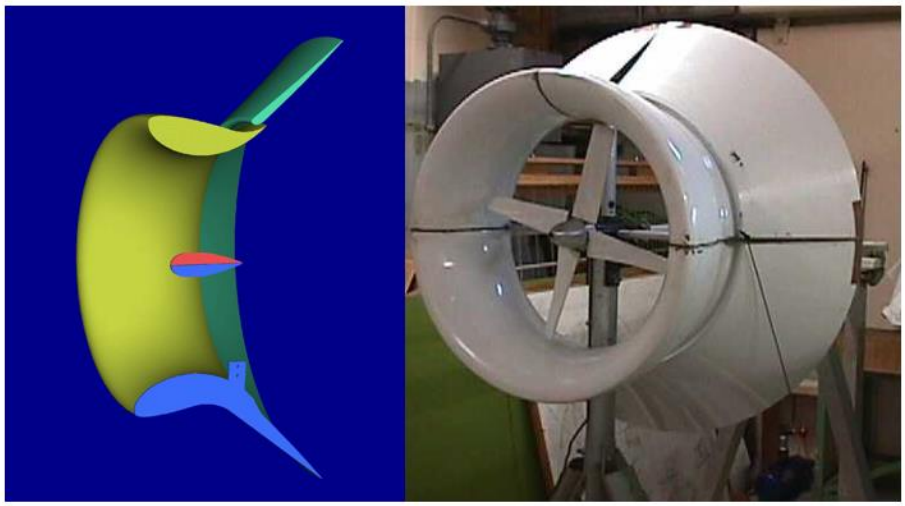


Figure 3.5 CAD model and wind-tunnel model of the new generation of Vortec 7 [65]

In his work, Hansen [17] performeda simple 1D analysis of a ducted turbine. He used a deformed NACA0015 airfoil for the cross section of the duct. For the CFD part of his work, he simulated the flow using 2D geometry and introduced a volume for across the blade plane in the diffuser to model the rotor as an actuator disc. This method was also used during the study of Vortec 7. The result of this work showed proportionality between the power increase and the induced relative mass flow increase in mass flow through the rotor.

As seen from the information above, to accelerate the flow velocity on the turbine, diffusers are used. The expansion and/or contraction of the diffuser is responsible of the flow augmentation. Another method to increase the flow velocity is the attachment of a large flange at the exit of the diffuser (Figure 3.6). This flange generates large size of separation behind it, where a very low-pressure region appears to draw more wind compared to a diffuser with no flange [66]. This system is very promising because, according to the results, the flanged diffuser increases the power coefficient of the turbine beyond the Betz limit.

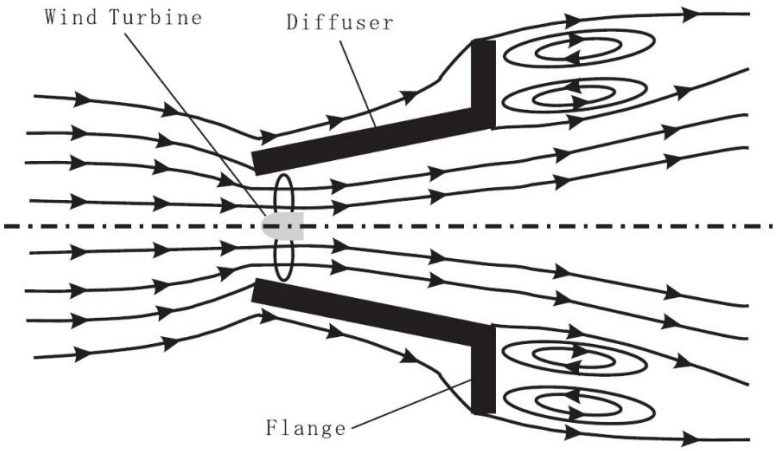


Figure 3.6 Schematic view of flow mechanism around a flanged diffuser [66]

3.3. CFD as optimization tool

CFD analyses are used to understand the physical phenomena in the flow field and to investigate its sensitivity to various parameters [67]. Results from CFD simulations are analyzed to improve selected characteristics in the fluid flow. From this brief explanation, one can conclude that CFD can be a tool used to optimize.

With the help of adjoint solvers and mesh morphing technologies, CFD software are now capable of bringing optimization to the next level.

A mesh morpher is a tool capable to perform mesh modifications, in order to achieve arbitrary shape changes and related volume smoothing, without changing the mesh topology [68]. Optimization using mesh morphing can be very advantageous because the work flow is accelerated and automated even for very large meshes.

Mesh morpher tools are either based on Radial Basis Functions (RBF), or based on Bernstein polynomials. Based on observed results, between these two, RBF based mesh morpher is leading in terms of performance and efficiency.

One of the most popular mesh morpher tool is the RBF-Morph which is an integrated system for morphing and shape optimization tailored for the CFD solver ANSYS FLUENT. RBF-Morph is a unique mesh morphing technology that combines a very accurate control of the geometrical parameters with an extremely fast mesh deformation, fully integrated in the solving process [69].

Most of the CFD software available nowadays have mesh morpher tool. Though RBF-Morph was tailored for ANSYS FLUENT, this later has mesh morpher tool incorporated within it which based on Bernstein polynomials. Another example is Altair, whose software has mesh morphing capability.

An adjoint solver allows specific information about a fluid system to be computed that is very difficult to gather otherwise. Becasue the adjoint solutions are not very useful in their raw form, they must be post-processed appropriately. In a single calculation adjoint solvers can compute the derivative of an engineering quantity with respect to all of the inputs for the system.

Because of this ability, desire changes can be selected easily. Once this desired change to the geometry of the system has been selected, mesh morphing tools

will be used to provide a convenient and powerful means of changing the geometry and the computational mesh.

CHAPTER 4
ANALYSIS AND DESIGN OF DUCT

The purpose of the previous chapters is to act as a support for the main study of the thesis which will be presented in this chapter. Now that the fundamental backgrounds and the literature have been reviewed it is time to apply them to the study. In this chapter every steps and techniques used during the design and analysis are explained, and the obtained results are presented. This chapter is divided into five section. Because most of the works in this chapter uses the power of ANSYS FLUENT, Section 4.1 helps reader to familiarize with this software. Section 4.2 provides information about the hydrokinetic turbine used throughout the study. Validation of simulation techniquesused throughout this chapter are presented in Section 4.3. Design, analysis and optimization of the duct are explained in details in Section 4.4. The final simulation of the hydrokinetic turbine together with the designed duct is presented in Section 4.5.

4.1. ANSYS FLUENT

Before works on the turbine and the duct start, it is important to understand the tools and techniques that will be used. To achieve the goal of the thesis, a CFD software called ANSYS FLUENT is used primarily. Though good CFD software are available, ANSYS FLUENT was chosen because of its popularity and the fact that it is a reliable software.

Because of the limited space and resources, only key steps and techniques used throughout this chapter will be detailed in a simple manner. To avoid overwhelming readers, illustrations will be used for explanation.

Also for the sake of presentation from now on instead of writing “ANSYS FLUENT” only the word “FLUENT” will be used.

4.1.1 Simple flow simulation

As mentioned in Section 2.5, CFD simulation consists of four steps. First the geometry is prepared using CAD software, then it is meshed, then the details of the problem are provided to the CFD software at the end results are analyzed. Before continuing, it is important to know that FLUENT is very user friendly. The standard steps to performer a simulation are listed, from top to bottom, in a logical fashion. Also, in cases where details about a step are needed, help is present in every pages. (Figure 4.1)

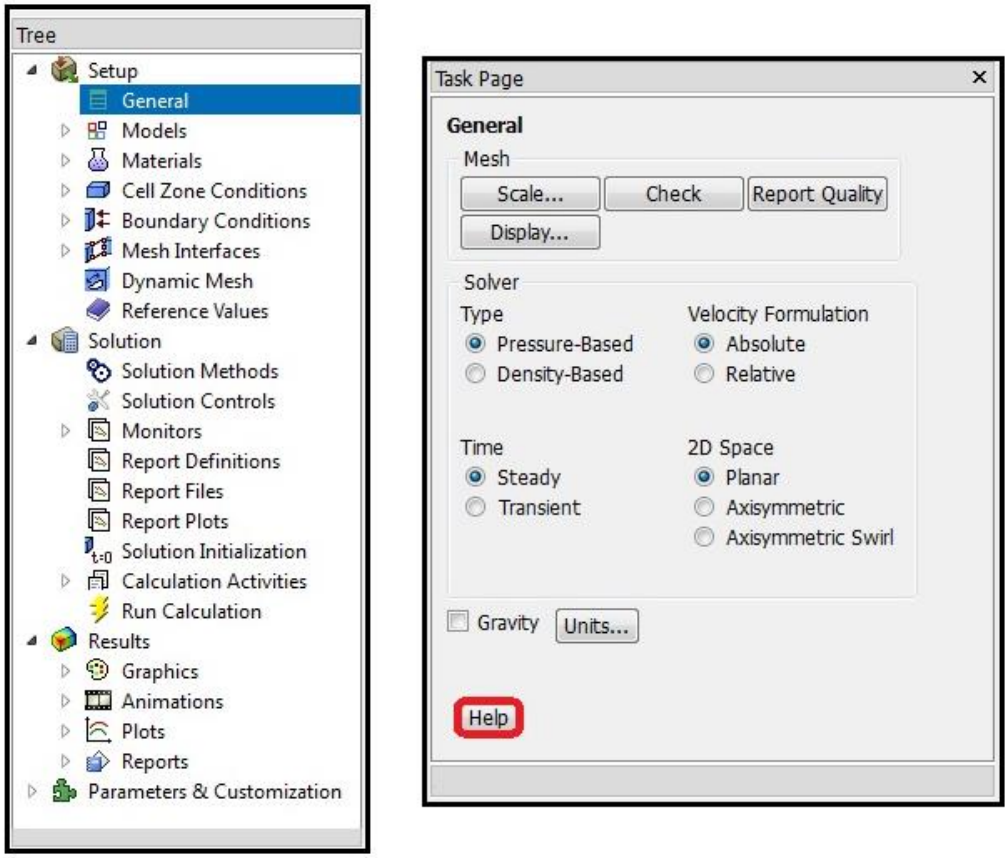


Figure 4.1 FLUENT – User friendly

To have an idea about how FLUENT solve fluid mechanics problems, let's assume a simple simulation. Also let's assume a meshed geometry was already created. The first two steps in performing a simulation is to choose solvers and models (Figure 4.2). Solvers can be Pressure-Based or Density-Based and Steady or Transient. These two differs from their way of solving problems. In density-based, the density field is obtained form continuity equation and the equation of state is used to determine the pressure field. On the other hand, in the pressure-based approach, the pressure field is extracted by solving a pressure or pressure correction equation which is obtained by manipulating continuity and momentum equations. The difference between steady and transient state simulation is that steady state simulation ignores many of the cross terms and higher order terms dealing with time. These terms all go to zero in steady state so they don't affect the steady state result. The transient simulation includes all these terms. Usually this means the steady state model has an easier convergence as there are less terms to model and some transient non-linearities are removed. Models are chosen based on the nature of the problem.

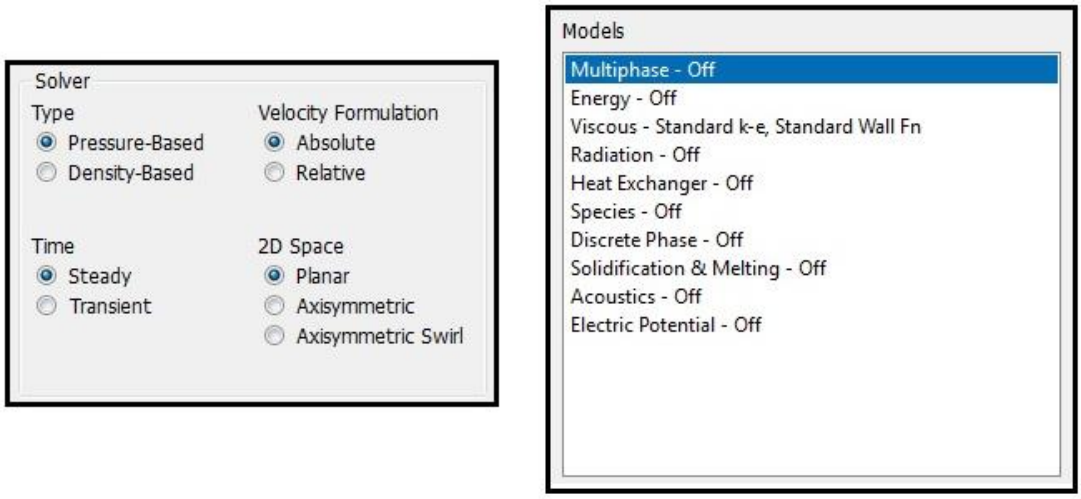


Figure 4.2 Solver and model setup

The next step in the simulation process is the material determination. This step allows users to set properties for any fluid or solid (or mixture, if applicable) materials in the simulation. Common materials are available in the fluent database. If multiple zones containing different materials are present in the simulation, then the materials on those zones should be defined from the cell zone condition page.

The next step is the specification of the boundary conditions. The Boundary conditions task page allow users to set the type of a boundaries and display other dialog boxes to set the boundary condition parameters for each boundary. After the boundary conditions are defined, various parameters associated with the solution method to be used in the calculation are specified. (Figure 4.3)

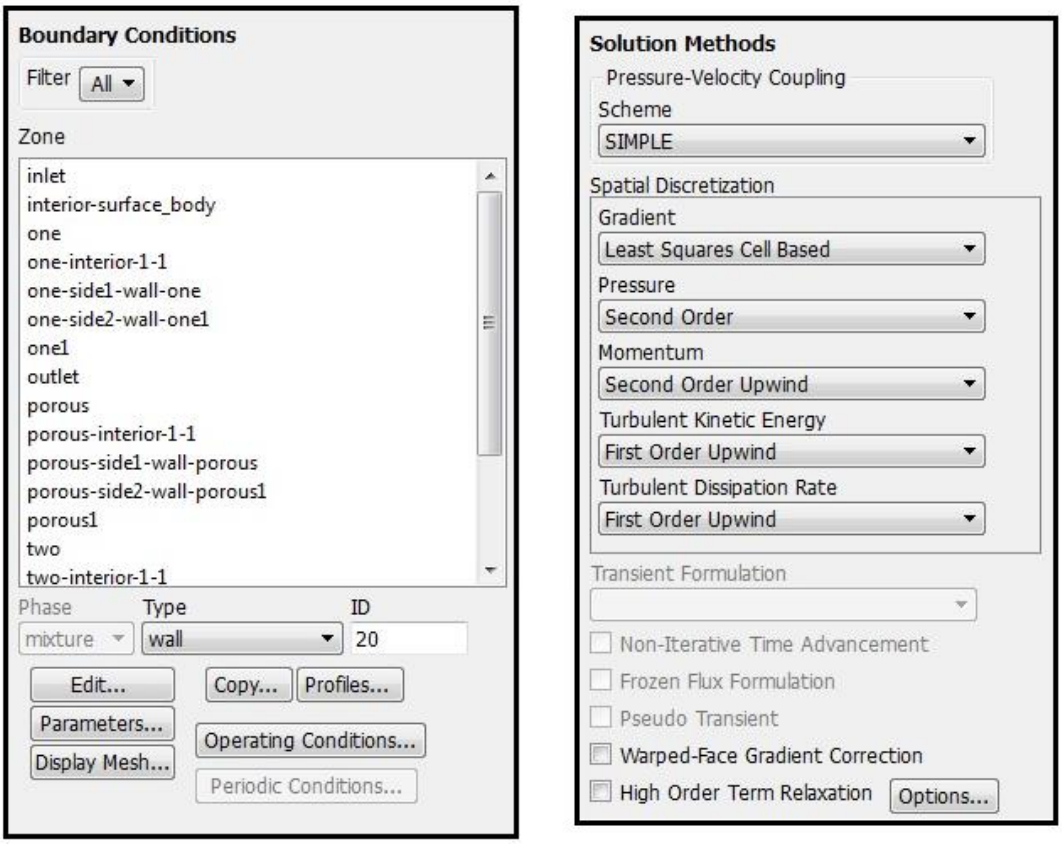


Figure 4.3 Boundary conditions and solution methods task page

The last steps in the simulation process is to start the solver iteration. But before that the values of flow variables must be defined the flow field to these values must be initialized. And also, before starting the solver, the number of iterations must be specified. Figure 4.4 illustrates these steps. The fluent solver works iteratively, so before the first iteration of the solver, a value must be given for every quantity in every cell. This is called initialization. The more realistic the values are, the faster the solution converges.

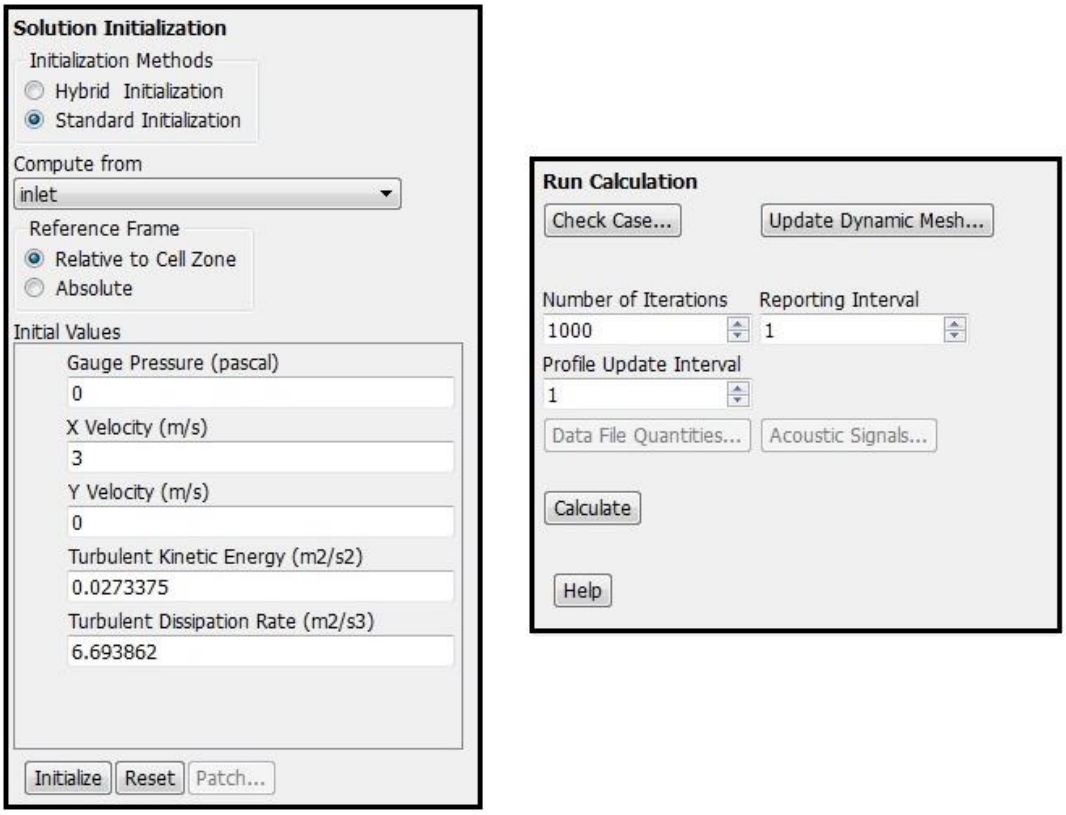


Figure 4.4 Solution initialization and solver iteration

When calculations converge, results can then be used and manipulated to understand the simulation. FLUENT provides post-processing tools to give users control on the results of the simulation. Contours, graphs, reports and many more tools can help users understand more about their simulation.

4.1.2 Advanced techniques

The reason why CFD was created in the first place is to help engineers solve complex problems. Though the steps explained in the previous section is enough to perform a simulation, most of the time those steps are not enough to better simulate certain flows. In the present work, more advanced techniques have been used to get more accurate results and to better simulate the flow as close as in real life.

Besides getting accurate results some techniques were used to accelerate the study, to save time. One example of that is the usage of symmetry while performing a simulation. Another example is the 2D modelling of 3D situations.

Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow have mirror symmetry. Though it may sound an easy task, the usage of symmetry needs to be understood and performed with delicacy. Symmetry reduces computational time because the number of mesh elements are reduced.

To obtain better results and to make the simulation as accurate as possible, a technique called Moving Reference Frame (MRF) was used. MRF is a relatively simple, robust, and efficient steady-state, CFD modeling technique to simulate rotating machinery. An MRF assumes that an assigned volume has a constant speed of rotation and the non-wall boundaries are surfaces of revolution (e.g., cylindrical, spherical, conical). [70]

One of the most important techniques used in this work is the usage of adjoint solver and mesh morpher. Details about these tools were presented in Section 2.6. From that section it was clear that shape optimization using mesh morpher and adjoint solver is very flexible. The ability of adjoint solver to calculate the derivatives of an observation, such as drag or lift, with respect to a very large number of input parameters make it a better candidate when it comes to optimization. Together with mesh morpher, the two form a very robust and flexible tool in engineering designs.

4.2. Turbine information

The hydrokinetic turbine used in the present work is the result of the study done by Muratoğlu[48]. The turbine has been named as TIGRIS-27H based on being a horizontal axis turbine having 27 kW rated power. It has been attributed to The River Tigris, which is one of the two biggest rivers of Mesopotamia. TIGRIS-27H turbine is a stall regulated hydrokinetic energy converter having an average efficiency of 0.43. An optimum rotational velocity of 45 rpm has been proposed for high productivity and minimum impact on the river habitat. [48] Table 4.1 summarize the important information about the turbine that are necessary for this work. A 3D representation of the turbine is shown in Figure 4.5.

Table 4.1 Hydrokinetic turbine information

Radius from axis (m)		1.5
Nacelle radius (m)		0.4
Distance from hub tip to nacelle bottom (m)		1.2
Optimum rotational velocity (rpm)		45
Average power coefficient		0.43
Simulation results from a 2.7 m/s stream velocity (theoretical results)	**Power (kW)**	28.99 (27.48)
	Torque (kN.m)	6.17 (5.83)
	Thrust (kN)	17.48 (17.05)

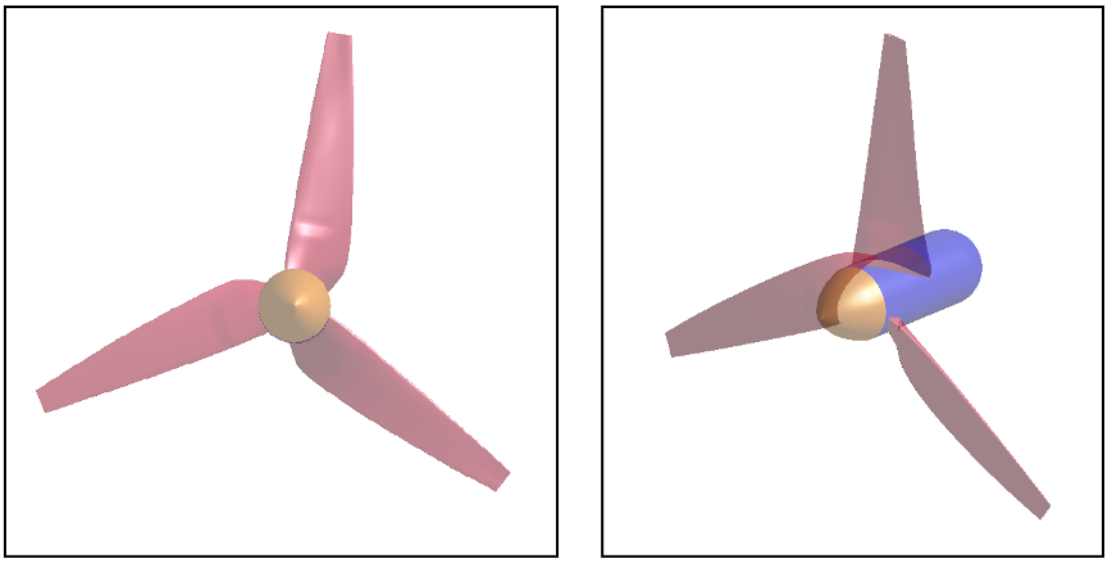


Figure 4.5 CAD model of the hydrokinetic turbine

4.3 Pre-validation

4.3.1 Meshing techniques

One of the most important step in CFD simulation is the meshing part. It has a key role in the quality of the simulation, unfortunately it is overlooked sometimes. Different techniques are used by engineers in generating mesh based on the nature of the problems and the geometry used in the simulation. Before starting the study, it is necessary to validate the meshing technique used. To do that a NACA 0015 airfoil was simulated using two different meshing techniques, then the pressure coefficient obtained from them are compared with the pressure coefficient of the same airfoil obtained from JavaFoil (Figure 4.8). JavaFoil is a relatively simple program, which uses several traditional methods for airfoil analysis. The following two methods build the backbone of the program: The potential flow analysis is done with a higher order panel method (linear varying vorticity distribution) [71].The obtained meshes are illustrated in Figure 4.6 and Figure 4.7.

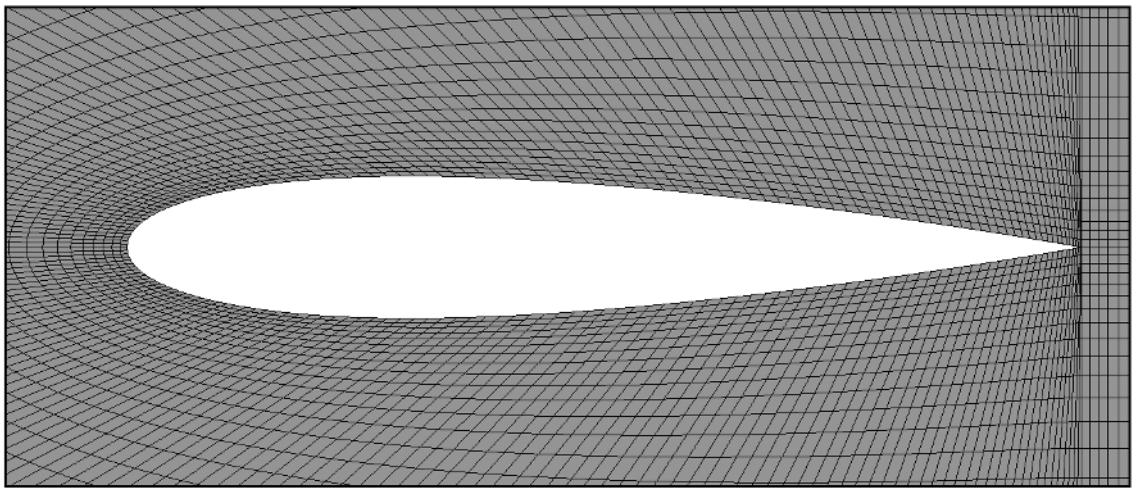


Figure 4.6 Result of mapped face meshing around NACA 0015

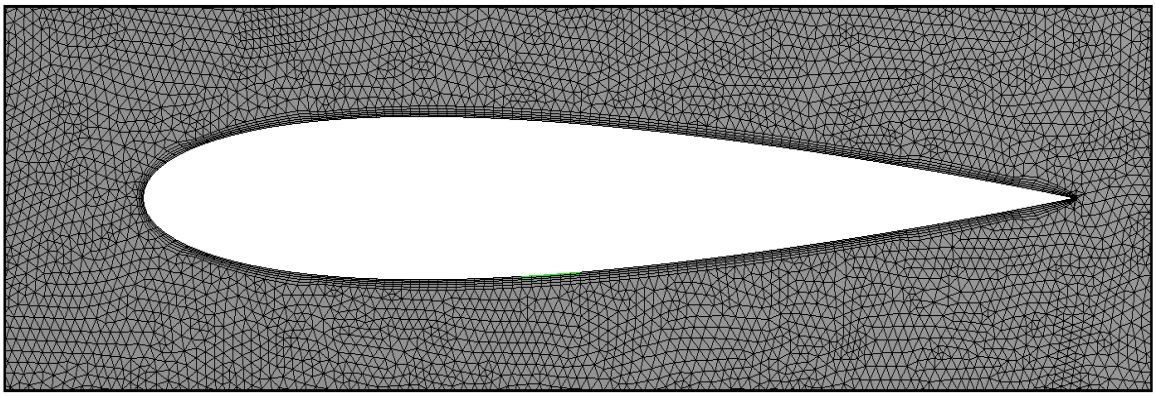


Figure 4.7 Result of triangular meshing with inflation layers around NACA 0015

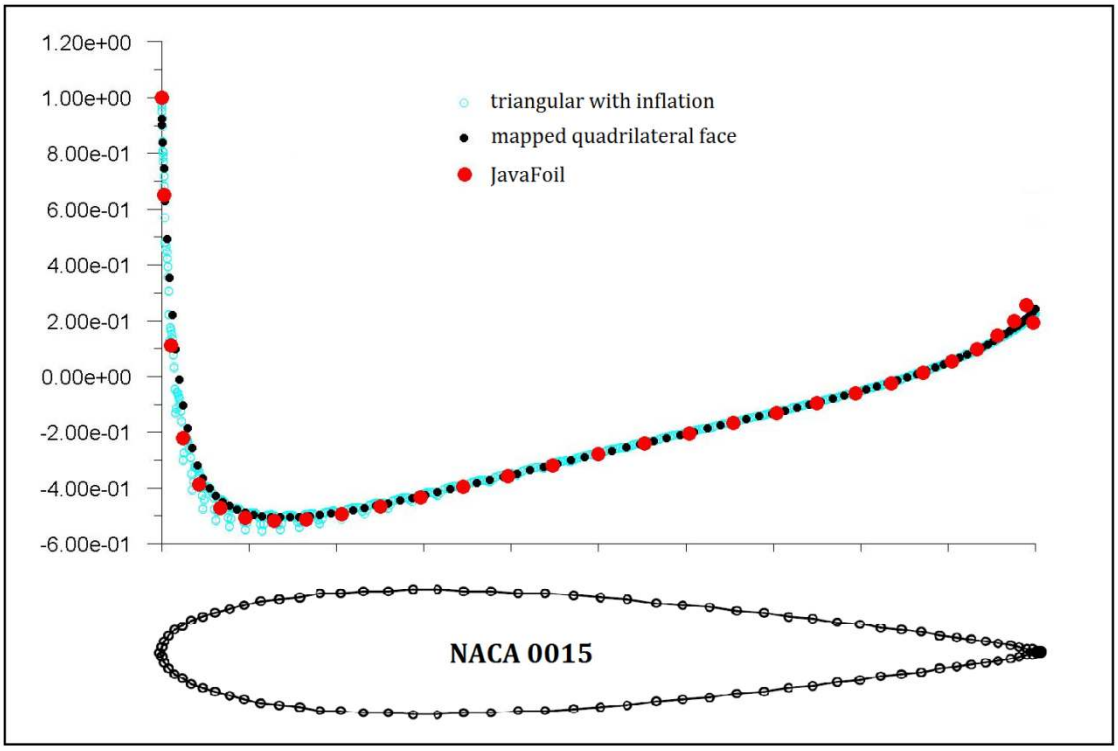


Figure 4.8 Pressure coefficient of NACA 0015 at 0° angle of attack

4.3.2 Simulation in 3D

At the end of the design and optimization processes, the obtained duct will be simulated alongside the hydrokinetic turbine. To get a realistic results, a 2D simulation is not an option because it omits many criterias that are crucial to the flow, so a 3D simulation should be performed. Also, because results from 3D simulation of the bare turbine are available, it is necessary to simulate the ducted turbine in 3D to better compare the power output of the bare and the ducted turbine.

Two simulations of same object cannot be perfectly similar even if all the conditions are the same, the reasons for this may be one of the followings:

- Different meshing software
- Different number of mesh elements
- Different simulation software
- Different versions of the same software

The results of the 3D simulation of the bare turbine that were received prior to this study were obtained from ANSYS CFX software. Because a different software, ANSYS FLUENT, will be used throughout this study, it is essential to simulate the bare turbine first using this later. The results from the two simulations are then compared to validate the simulation technique used in FLUENT.

In the present simulation, the turbine will have a rotation so two geometries should be prepared, one stationary and one rotating. The sationary part is the outer domain and the rotating part is the inner domain. Details on those two domains are illustrated in Figure 4.9 and Figure 4.10. The meshing and the simulation details are presented in Table 4.2 and Table 4.3 respectively.

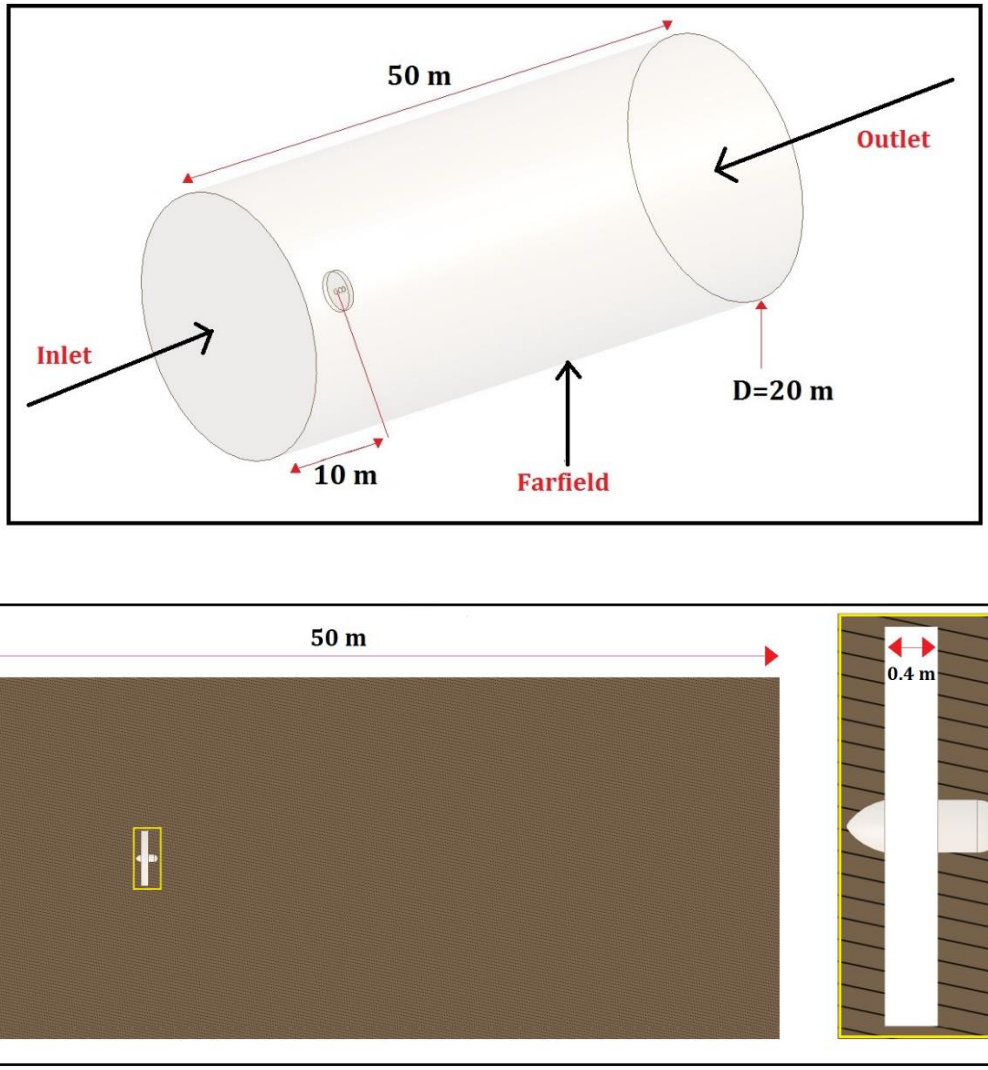


Figure 6.9 Outer domain details

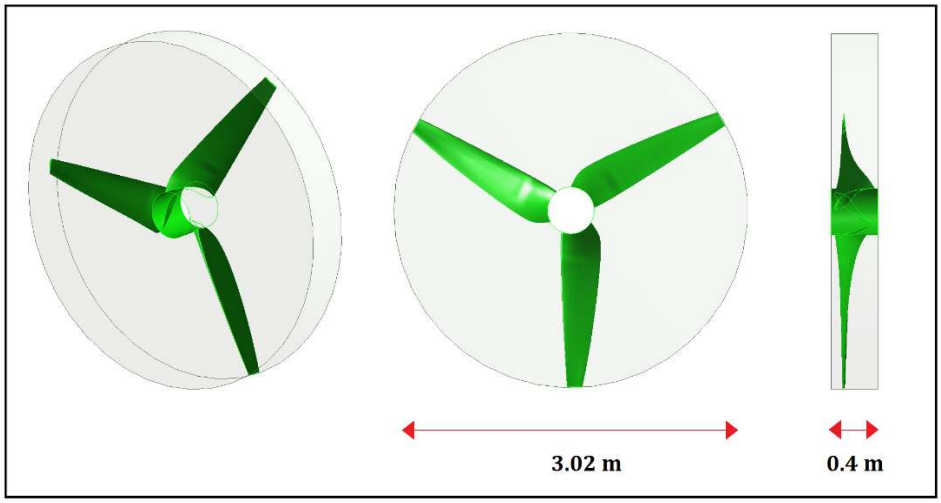


Figure 4.10 Inner domain details

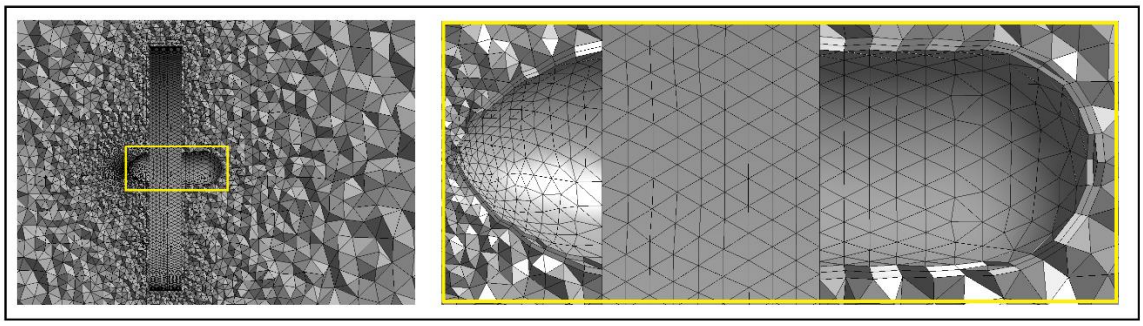


Figure 4.11 Meshing details on the outer domain

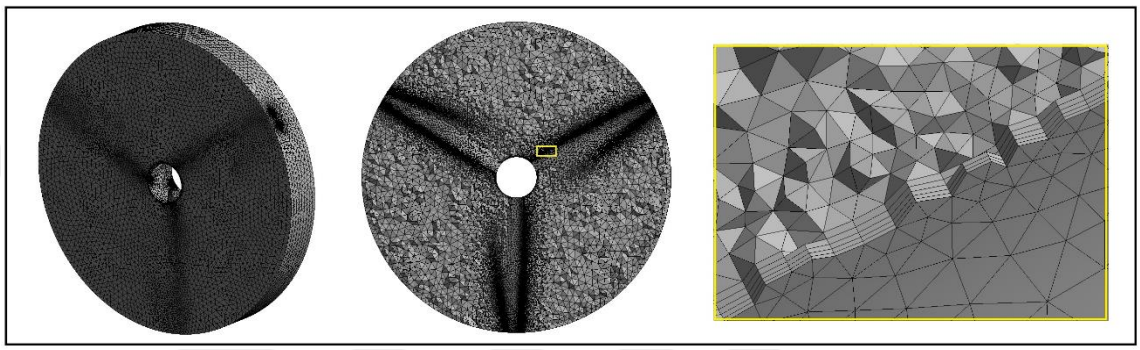


Figure 4.12 Meshing details on the inner domain

Table 4.2 Details on the meshing of the outer and inner parts

		Outer Part	**Inner Part**
Size function		Curvature	Curvature
Relevance Center		Fine	Fine
Smoothing		Fine	Fine
Curvature Normal Angle		18.0°	18.0°
Min Size		Default (7.76 e-3 m)	Default (6.26 e-4 m)
Max Face Size		Default (0.77)	Default (6.26 e-2 m)
Max Tet Size		Default (1.55 m)	Default (0.13 m)
Growth Rate		1.1	1.2
Inflation	**Transition Ratio**	0.272	0.272
	Maximum Layers	2	5
	Growth Rate	1.2	1.2
Statistics	**Nodes**	158137	1532295
	Elements	894317	6289847

Table 4.3 Bare turbine simulation details

Options		Double Precision
Processing options		Parallel Solver
		8 Processes
Solver		Steady Pressure-Based
Model		k-ε, Standard Wall Fn
Cell Zone Conditions (Inner)		Frame motion, 45 rpm (-x)
Boundary Conditions	**Blades**	Wall
	Farfield	Free Slip Wall
	Inlet	2.7 m/s
	Outlet	148452 Pa
Solution Methods	**Scheme**	SIMPLE
	Gradient	Least Squares Cell Based
	Pressure	Second Order
	Momentum	Second Order Upwind
Solution Initialization		Hybrid Methods
Number of Iterations		800

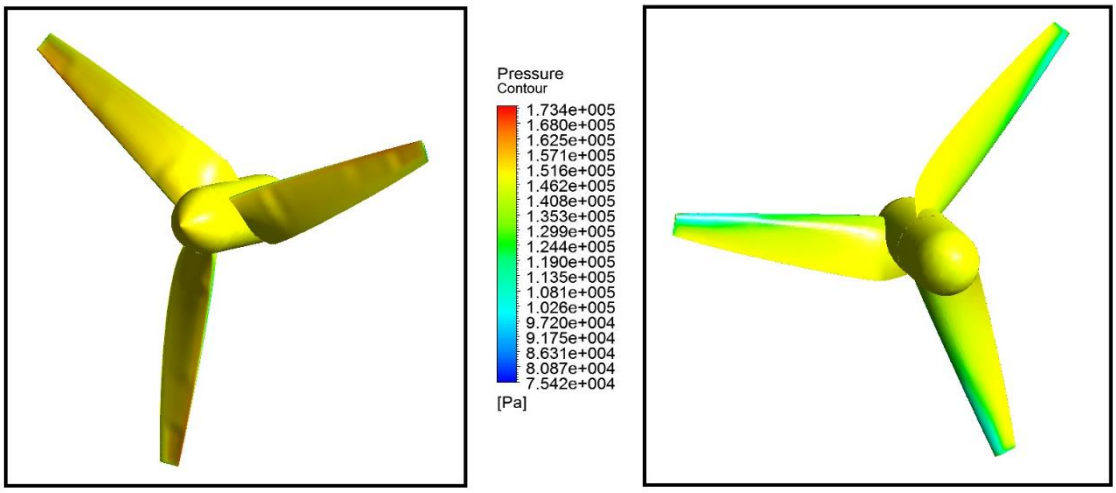


Figure 4.13 Pressure contour around turbine

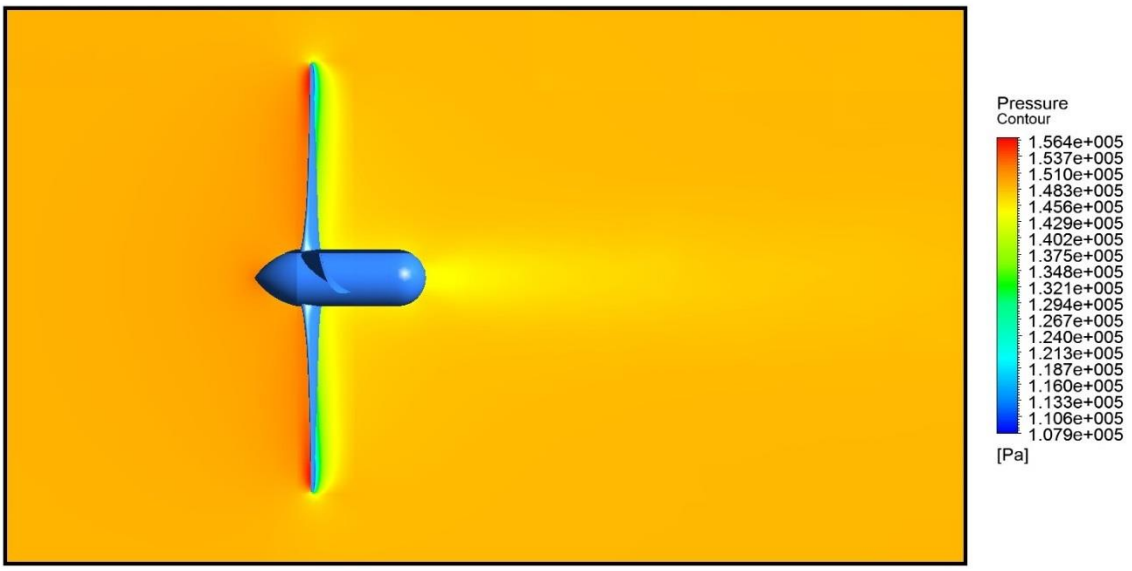


Figure 4.14 Pressure contour on all flow domain

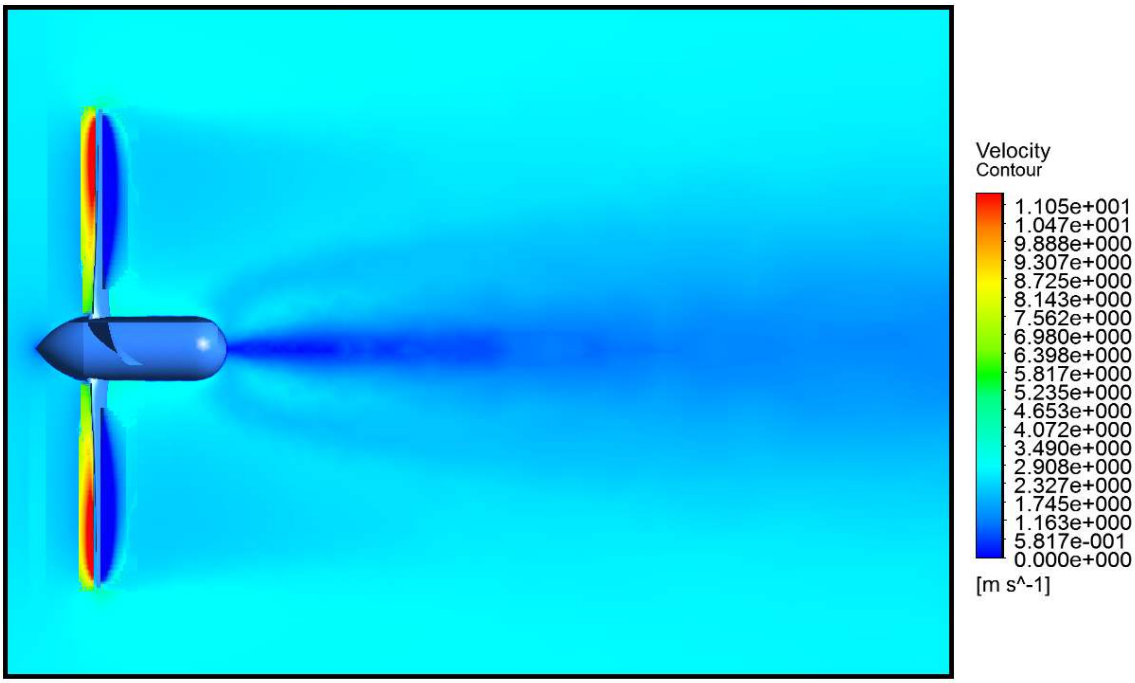


Figure 4.15 Velocity contour on all domain

4.4. Duct design

The design of the duct is divided into three works. These works represent three major factors in duct performances. The first work is on duct drag, the second is on lift and the last is on the pressure drop downstream of the turbine.
The airfoil used as a staring shape of the cross section of the duct is the NACA 0015. The starting does not make a big difference because the optimization tool will transform the shape be it NACA 0015, NACA 0012 or others, the important thing is that the design starts with an airfoil.

4.4.1 Drag analysis

Drag is a force acting opposite of the flow. The more the drag the more the force acting on the flow. Drag is generated by the difference in velocity between the solid object and the fluid. There must be motion between the object and the fluid. If there is no motion, there is no drag. It makes no difference whether the object moves through a static fluid or whether the fluid moves past a static solid object.
Drag slows the motion of fluid so it must be minimized. Attempt to minimize the drag on the duct is presented in this section.
When doing drag analysis and shape modification using adjoint solver, drag should be the observable. When the adjoint solver solve the flow, it will calculate how all parameters affect the drag on the object. Adjoint solver is a flow solver on its own, it has all the important steps that a flow analysis should have (Figure 4.16).

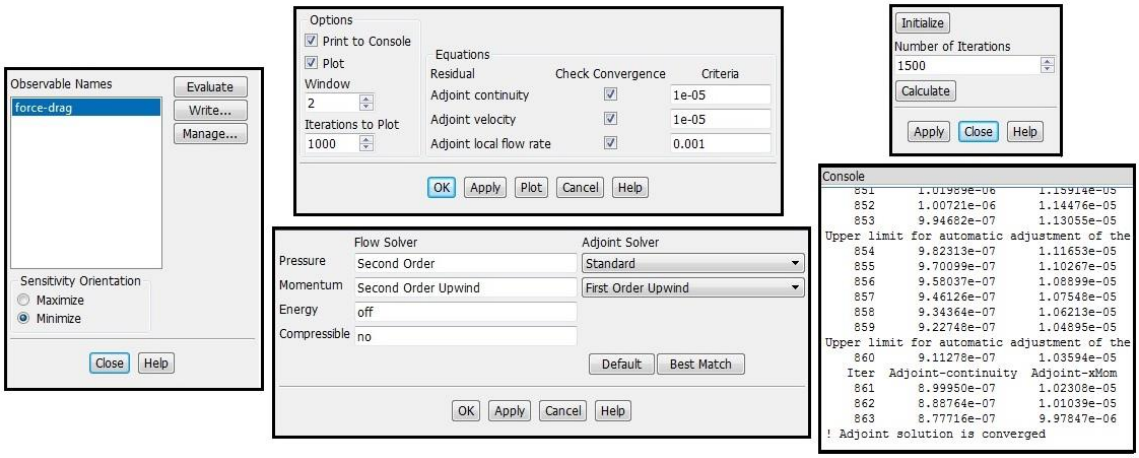


Figure 4.16 Flow solving steps in adjoint solver

After the adjoint solver converges, the analysis and design can start. First it is important to understand how certain flow conditions affect the observable in question which is the drag. This is important because it is helps users understand more about object to optimize. It also gives users an idea about what changes are to be expected when shape modifications will be applied on the object.

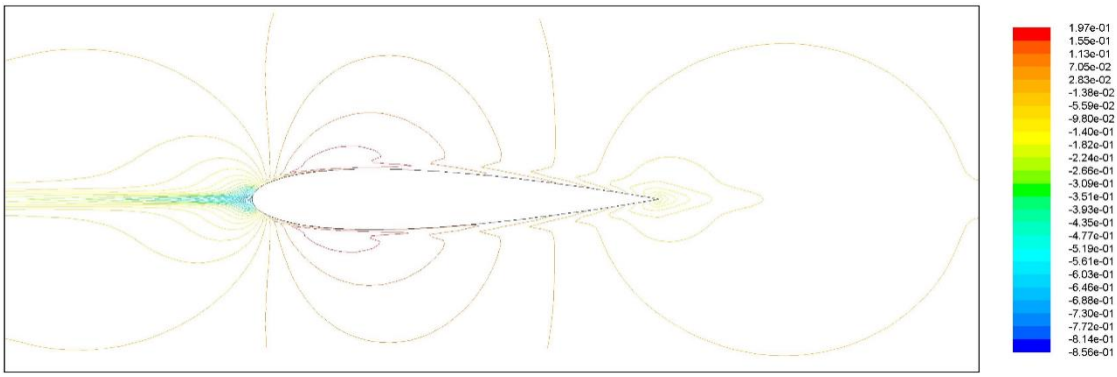


Figure 4.17 Adjoint sensitivity to body force x-component contours

Figure 4.17 shows how sensitive the drag on the airfoil is to the application of a body force in the direction in the flow. Note that the effect of applying a body force downstream of the cylinder is minimal, as expected. If a body force is applied directly upstream of the cylinder; the disturbed flow is incident on the cylinder and modifies the force that it experiences.

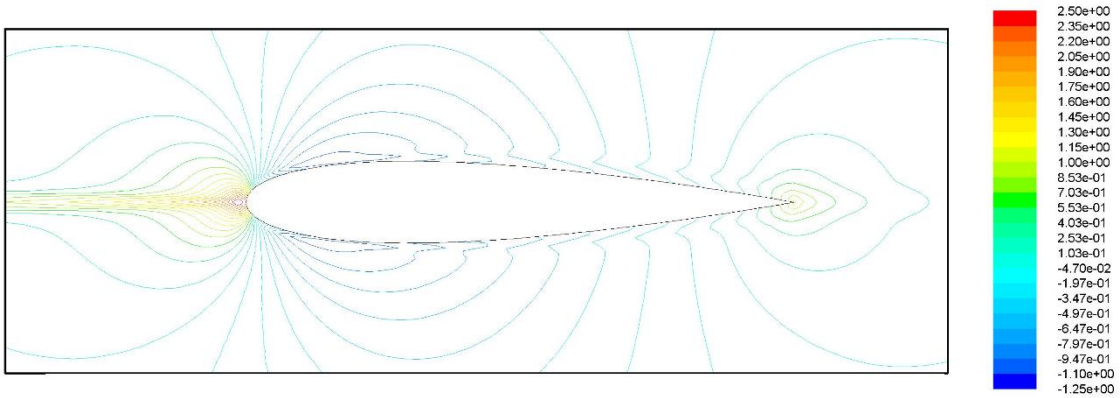


Figure 4.18 Adjoint sensitivity to mass sources contours

Figure 4.18 shows how the drag on the airfoil is affected by the addition of sources and sinks of mass. If material were to be systematically removed from the flow field upstream of the airfoil, then the drag will be reduced. This can be deduced

since the adjoint pressure is negative upstream of the cylinder.

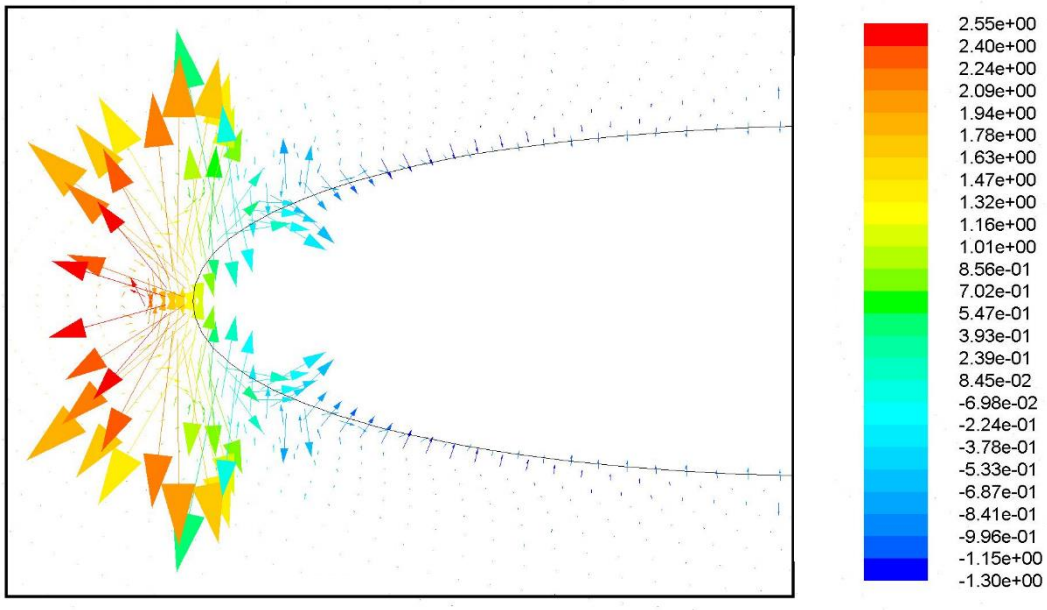


Figure 4.19 Sensitivity of drag to change in shape

Figure 4.19 shows how sensitive the drag on the airfoil is to changes in the surface shape. The drag is affected more significantly if the airfoil is deformed on the upstream rather than the downstream side. Maximum effect is achieved by narrowing the cylinder in the cross-stream direction.

When a general understanding of the sensitivity of the drag is acquired, desired changes can now be imputed and expected changes are presented. Then the mesh morpher will modify the mesh to so that the final shape will yield the desired change. This step can be repeated as much as needed until a reasonable and meaningful changes are observed. The result of the drag reduction after each iteration is illustrated in Figure 4.20. From this figure it can be seen that after each drag optimization iteration, the drag coefficient always decreases which proves that the optimization is going on the right path. Here the iteration was stopped because the obtained shape from the mesh morphing starts to diverge from the desired geometry.

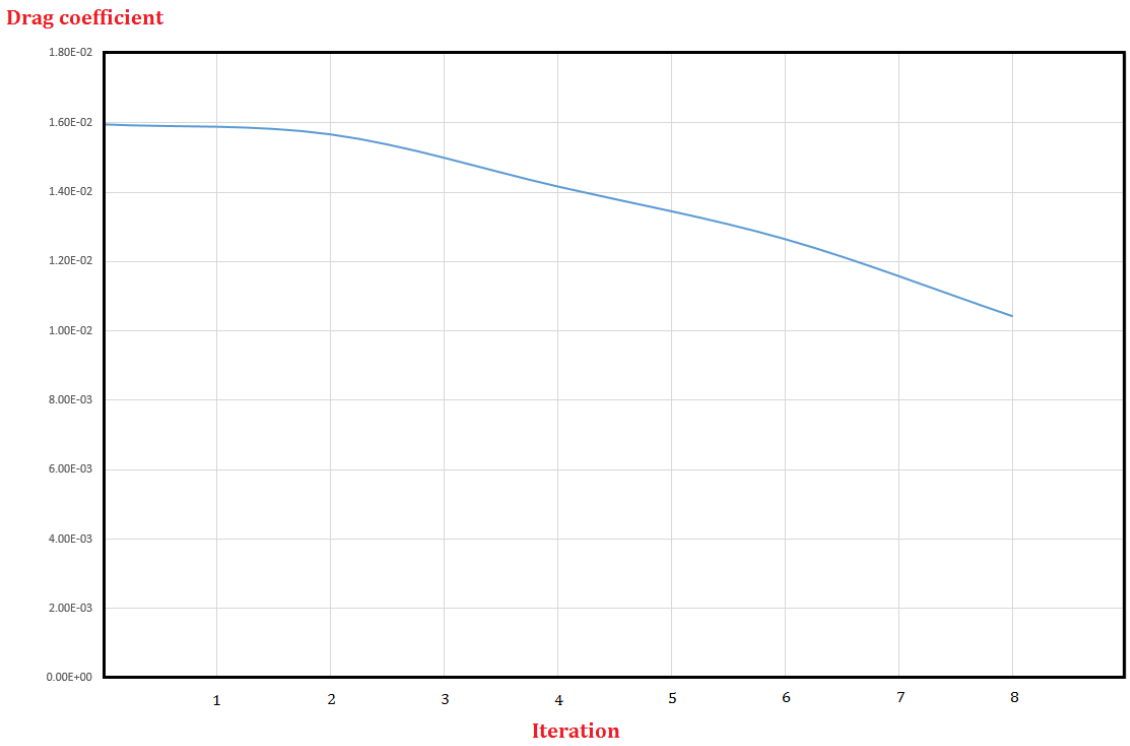


Figure 4.20 Drag coefficient variation throughout drag optimization

4.4.2 Lift analysis

From Section 2.3 it was stated that the generated lift on the airfoil will produce a ring vortexcirculation which will help move more water through the duct. This additional flow will also contribute in the additional power output of the turbine because it increases the stream velocity at the turbine location.

The same study will be done one the airfoil but this time the observable used in the adjoint solver will be the lift force. Again preliminary analysis will be made to understand more about the relationship between lift and any changes in the flow or in the shape. Figure 4.21 shows how the lift on the airfoil is affected by the addition of sources and sinks of mass.

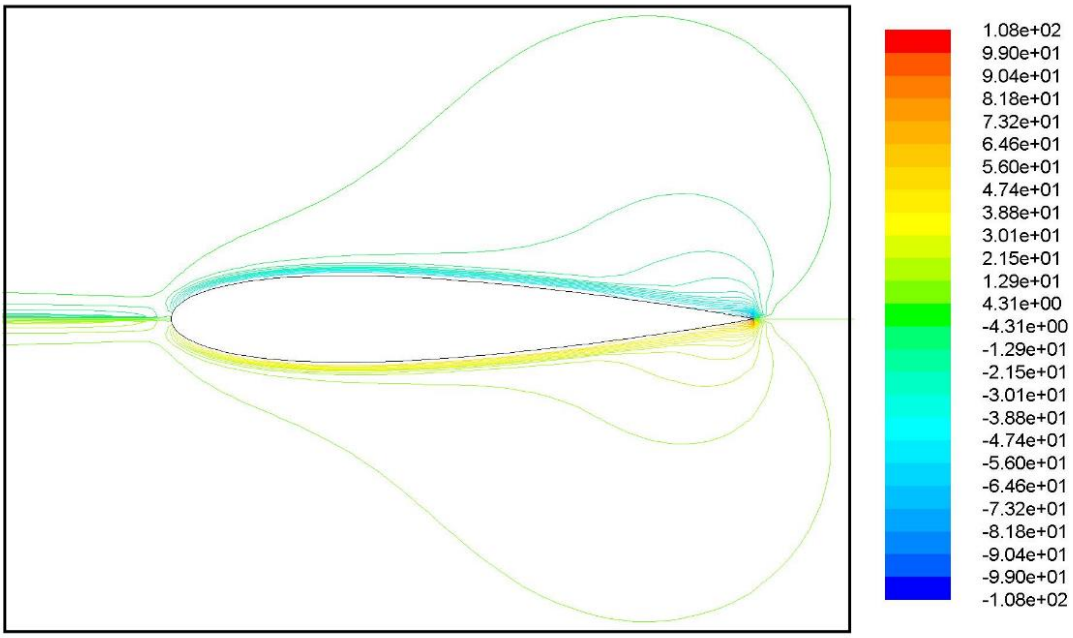


Figure 4.21 Adjoint sensitivity to mass sources contours

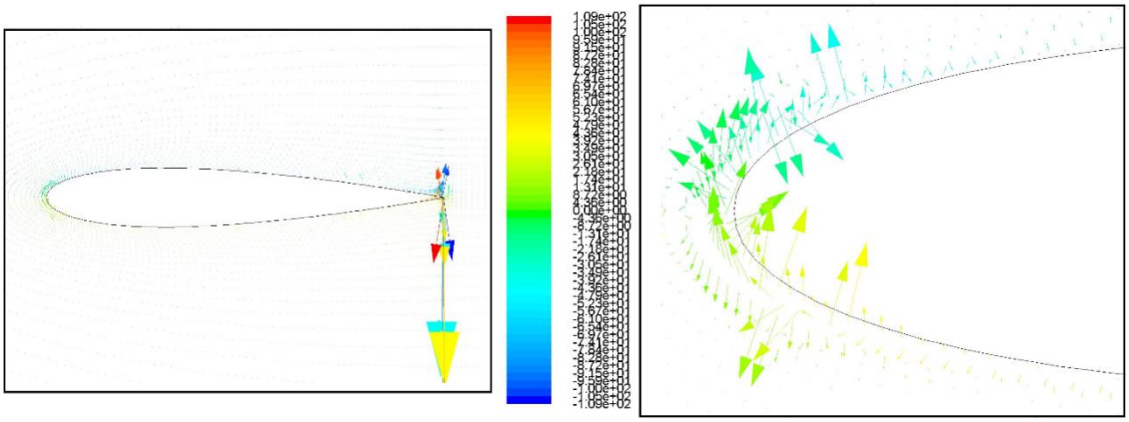


Figure 4.22 Sensitivity of lift to change in shape

Figure 4.22shows how sensitive the lift on the airfoil is to changes in the surface shape. The lift is affected more significantly if the airfoil is given a clockwise rotation on the downstream. The arrows in the right figure will eventually create a camber in the airfoil which as expected will increase the lift.

Then again changes can now be imputed and expected changes are computed. The lift augmentation after each iteration, which is the goal of this analysis, is illustrated in Figure 4.23.

It can also be seen here that after every iteration the lift coefficient increases which validates the optimization technique used.

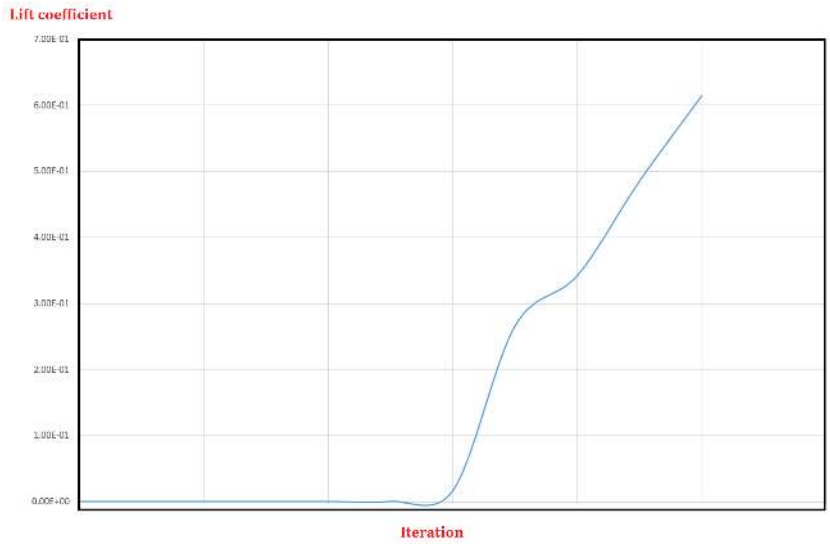


Figure 4.23 Lift coefficient variation throughout lift optimization

4.4.3 Pressure drop analysis

As stated by [59,22], most of the power augmentation in a ducted turbine comes from the pressure drop downstream of the turbine. This drop in pressure will force more flow from upstream which will eventually pass through the turbine. The more this pressure drop, the more the flow rate increase.

To increase the pressure, drop within the downstream part of the turbine, the cross sectional area downstream will be increased. To do that the optimized airfoil obtained from the drag and lift optimization, (Figure 4.24), will be given a full body rotation with a specify angle (θ). Of course the value of this angle is limited by the flow separation in the lower part of the duct. When flow separation starts to appear then the rotation will be terminated.

- About 45% thickness reduction from NACA 0015
- About 3% camber at x = 0.4 m

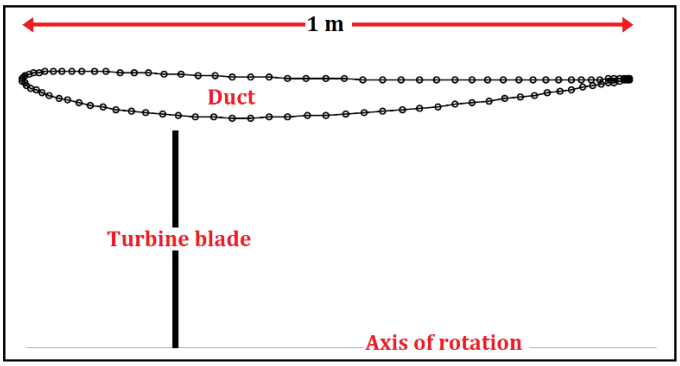


Figure 4.24 Drag and lift optimized airfoil

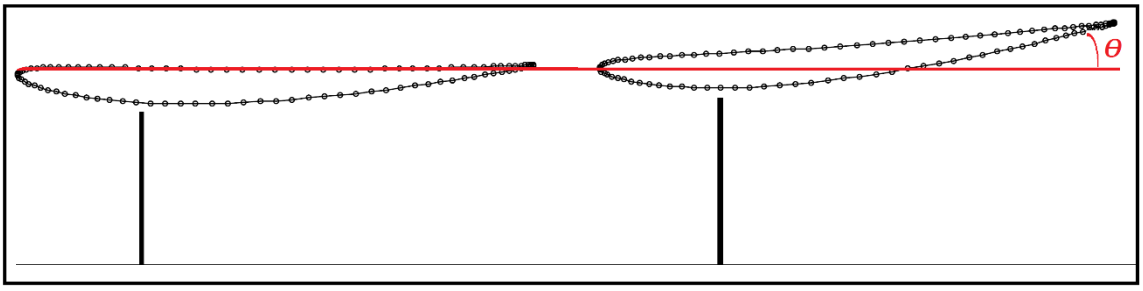


Figure 4.25 Airfoil rotation to increase the drop in pressure

To accomplish the pressure analysis, the airfoil was simulated with 0° rotation then a parametric study was conducted. The x and y components of the flow velocity were assigned as an input parameter, and the pressure drop as an output parameter. The input and output parameter settings are done in Fluent while the rest of the parametric study were done in ANSYS Workbench. The values of x and y velocities were modified based on the chosen rotational angle. Figure 4.26illustrate the parametric study and Figure 4.27shows how the downstream pressure varies with the increasing angle.

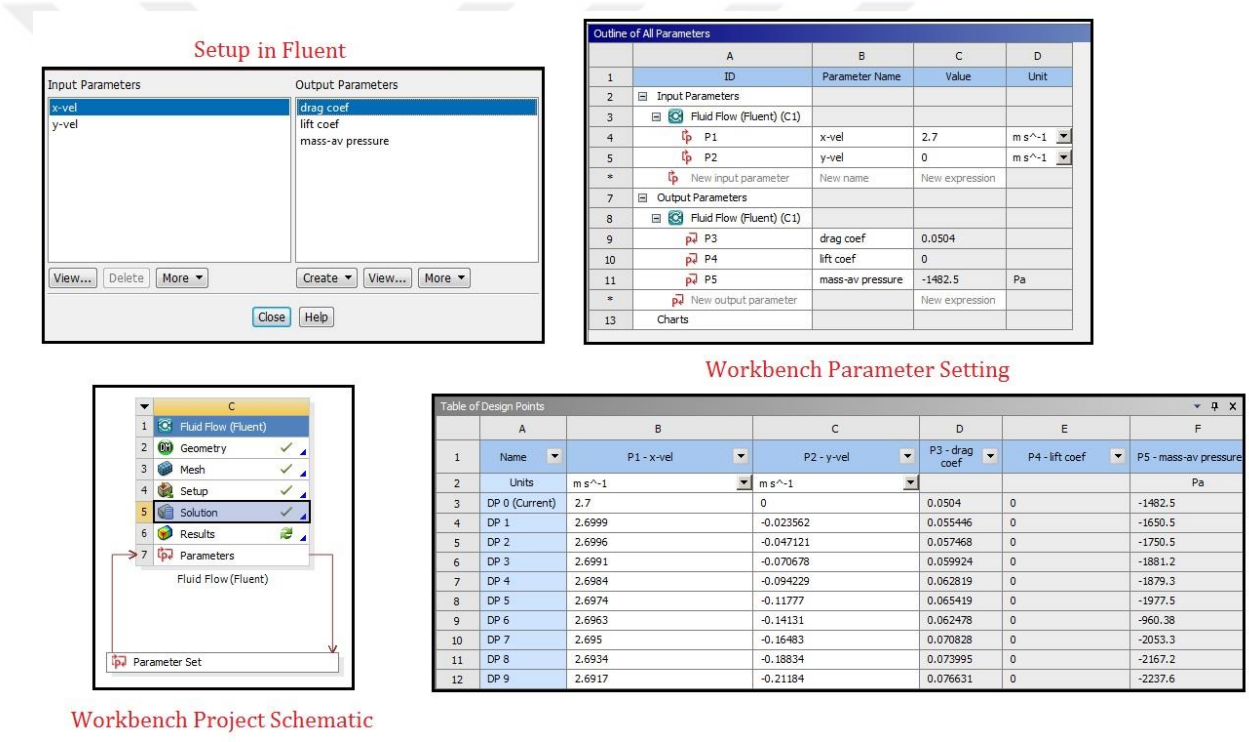


Figure 4.26 Parametric study of pressure drop

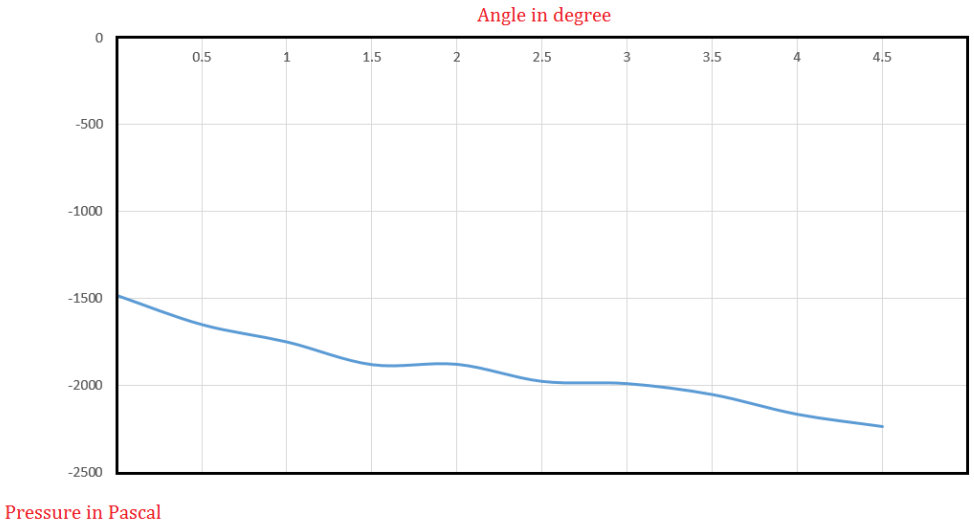


Figure 4.27 Variation of pressure with full body rotation of airfoil

4.5. Ducted turbine simulation

The final shape of the duct obtained from the previous section's work, with the turbine placed inside it, is shown in Figure 4.28 and Figure 4.29. In this section the duct alone is simulated in 2D to see how pressure and velocity varies throughout the duct. Then the ducted turbine is simulated in 3D.

Compared to the bare turbine simulation, the mesh changes in the outer part because the duct was introduced (Figure 4.32), on the other hand the inner part remained the same (Figure 4.33).

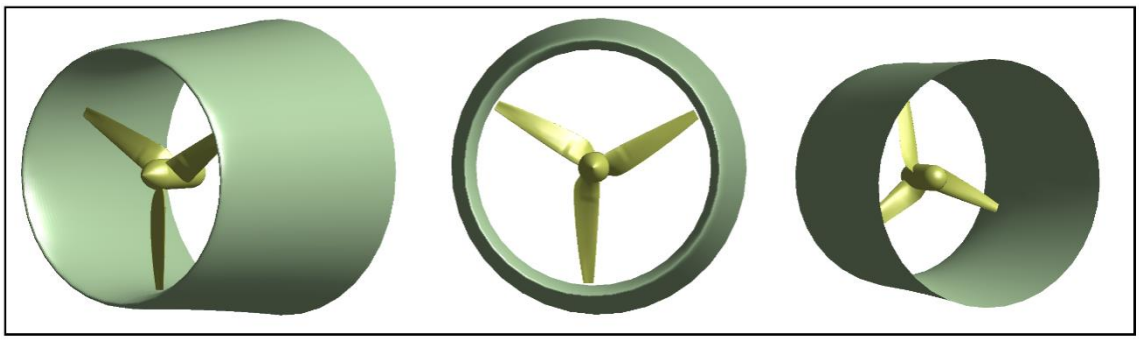
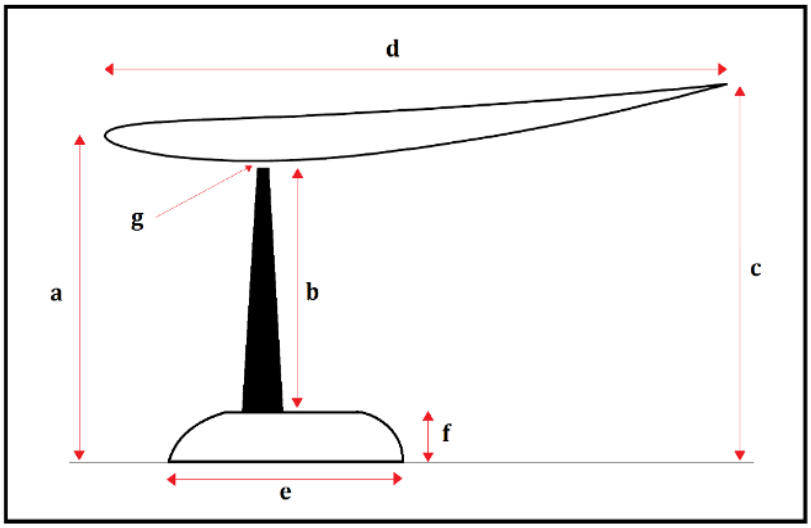
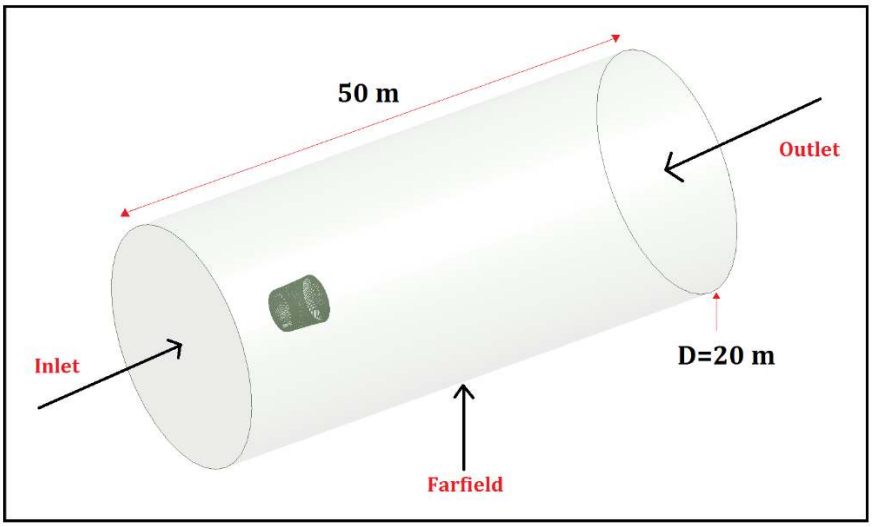


Figure 4.28 CAD model of ducted turbine



a	1.66 m
b	1.30 m
c	1.92 m
d	3.32 m
e	1.20 m
f	0.20 m
g	0.03 m

Figure 4.29 2D representation of ducted turbine with dimensions



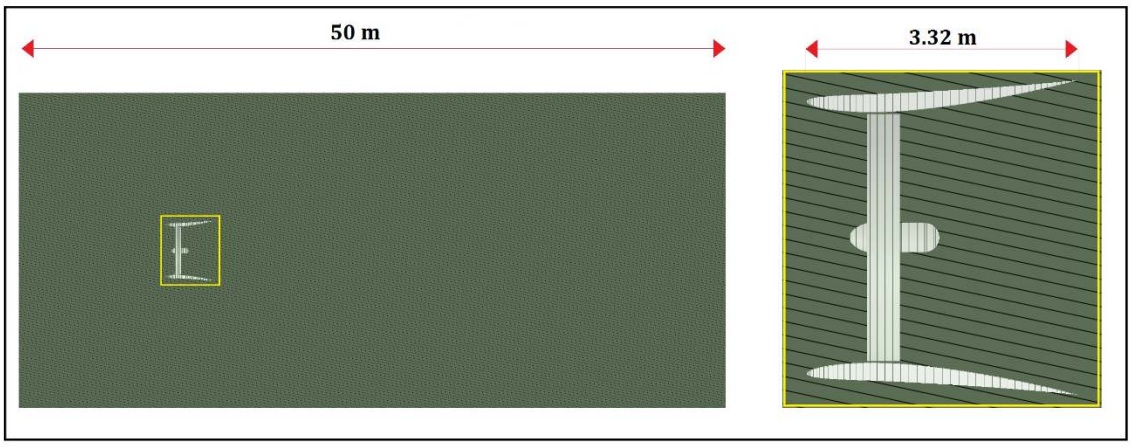


Figure 4.30 Detail of ducted turbine outer domain

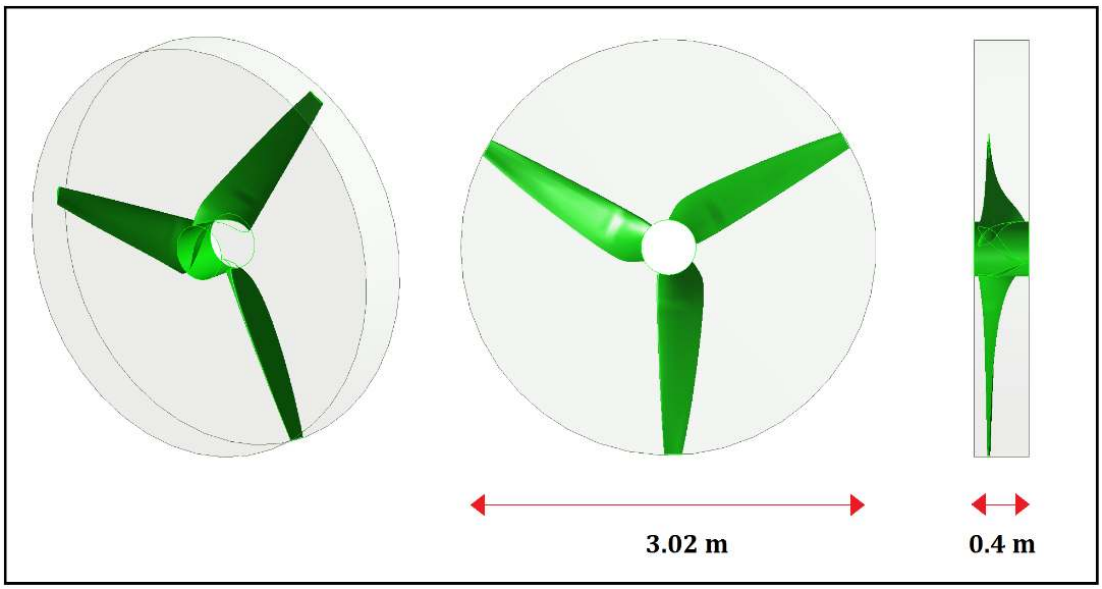


Figure 7.31 Detail of ducted turbine inner domain

Table 4.4 Ducted turbine meshing details

		Outer Part	**Inner Part**
Size function		Proximity and Curvature	Curvature
Relevance Center		Medium	Fine
Smoothing		Medium	Fine
Curvature Normal Angle		18.0°	18.0°
Min Size		Default (1.32 e-2 m)	Default (6.26 e-4 m)
Max Face Size		Default (1.32)	Default (6.26 e-2 m)
Max Tet Size		Default (2.65 m)	Default (0.13 m)
Growth Rate		1.2	1.2
Inflation	**Transition Ratio**	0.272	0.272
	Maximum Layers	5	5
	Growth Rate	1.2	1.2
Statistics	**Nodes**	1391409	1532295
	Elements	4837959	6289847

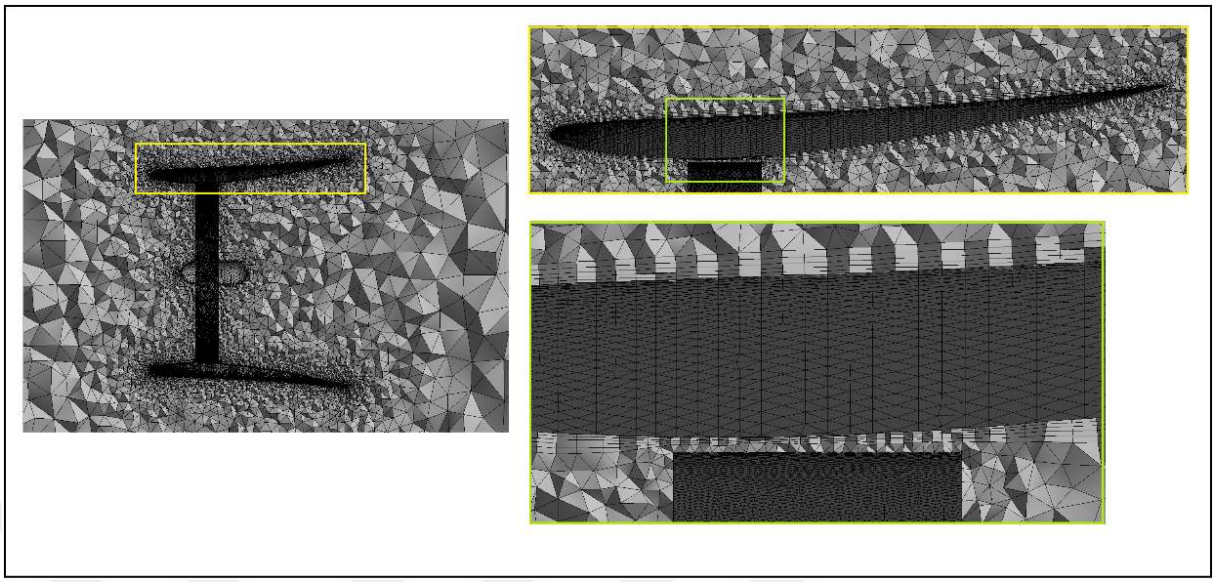


Figure 4.32 Mesh of the outer domain of the ducted turbine

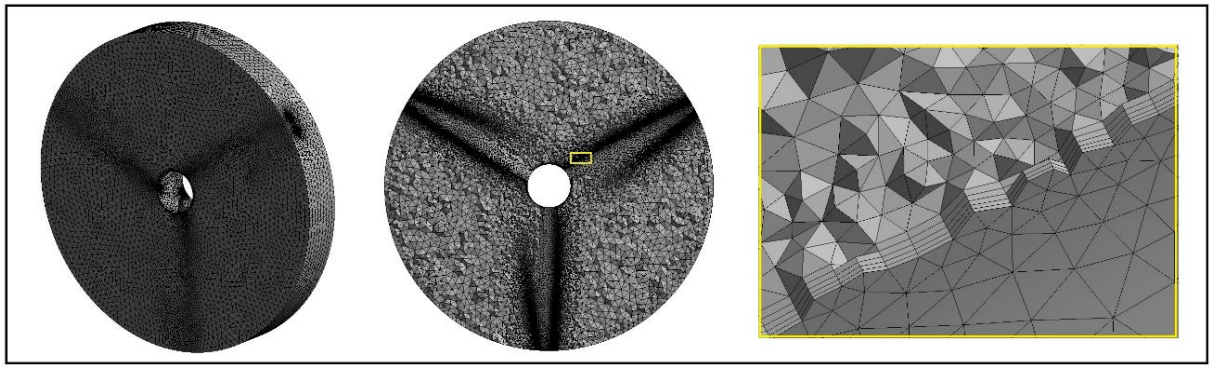


Figure 4.33 Mesh of the inner domain of the ducted turbine

Table 4.5 Ducted turbine simulation details

Options		Double Precision
Processing options		Parallel Solver
		8 Processes
Solver		Steady Pressure-Based
Model		k-ε, Standard Wall Fn
Cell Zone Conditions (Inner)		Frame motion, 45 rpm (-x)
Boundary Conditions	**Blades**	Wall
	Duct	Wall
	Farfield	Free Slip Wall
	Inlet	2.7 m/s
	Outlet	148452 Pa
Solution Methods	**Scheme**	SIMPLE
	Gradient	Least Squares Cell Based
	Pressure	Second Order
	Momentum	Second Order Upwind
Solution Initialization		Hybrid Methods
Number of Iterations		700

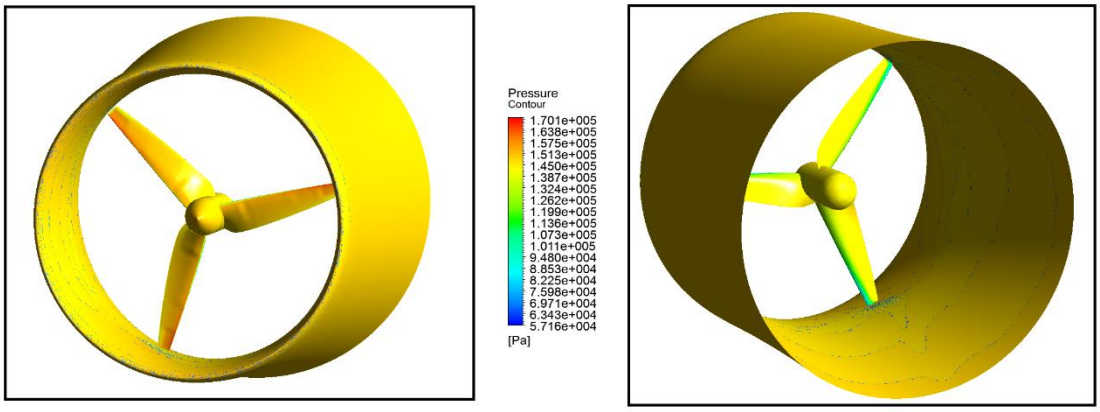


Figure 4.34 Pressure contour on the ducted turbine

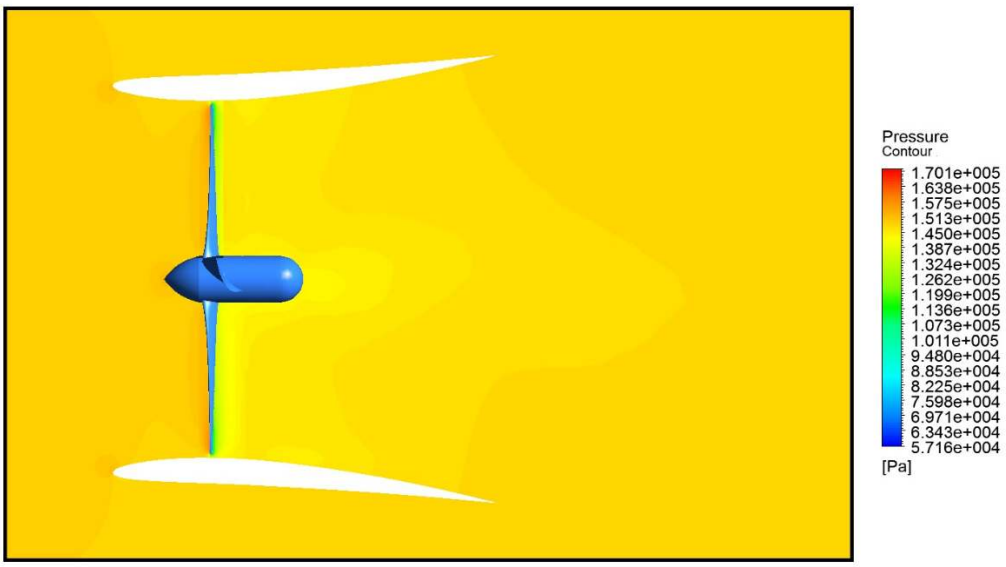


Figure 4.35 Pressure contour on all flow domain

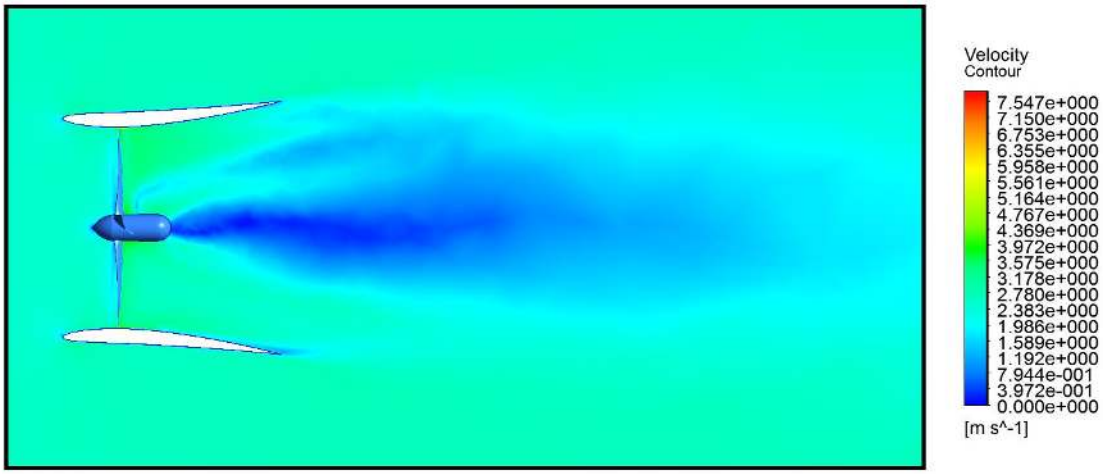


Figure 4.36 Velocity contour on all flow domain

CHAPTER 5
RESULTS AND CONCLUSION

5.1. Discussion of results

During the study, CFD was used to solve all the numerical flow problem. Because the quality of mesh is very important in all CFD simulations, the meshing techniques were first pre-validated before the study even begun. Two meshing techniques were used and the best one which is the triangular with inflation was chosen. The pressure coefficient plot from the two techniques were plotted together with the one obtained from JAVAFOIL. As seen in Figure 4.8 the simulation went well, which validates the meshing techniques.

To validate the 3D simulation technique used in the thesis, the bare turbine was first simulated using FLUENT then the results were compared against the simulation results obtained from different simulation. The details of that comparison are illustrated in Table 5.1.

Table 5.1 Rotating bare turbine simulation results (2.7 m/s stream velocity)

	Theoretical (Muratoğlu)	**CFX simulation (Muratoğlu)**	**FLUENT simulation (Tsiry)**
Power (kW)	27.48	28.99	26.95
Torque (kN.m)	5.83	6.17	5.72
Thrust (kN)	17.05	17.48	17.6

Figure 4.19 and Figure 4.22 illustrate the shape sensitivities in drag and lift of the airfoil section of the duct. These figures illustrate the real optimization because even without them one can intuitively conclude that to reduce the drag, the tickness should be reduced.

And Also to increase the fift of an airfoil, camber should be added. After the lift and drag optimization, the obtained shape was rotated to force an expansion downstream. Though this body rotation has a big range, the flow separation limits it. In this work the airfoil cross section was rotated by 4.5 degree because flow separation starts to occur beyond that value.

After all the modification was done, the turbine and the duct were put together, because the duct was originally designed using a unity length, it was later scaled so that it can enclose the turbine. Characteristics of the designed duct are listed in Table 5.2. Results from the ducted turbine 3D simulation are presented in Table 5.3, a 40% increase in power output can be observed when the designed duct is used alongside the turbine.

Table 5.2 Turbine characteristics

Original airfoil	0.44
Thickness reduction coefficient	45 %
Camber at 40% L (from left)	3%
Expansion ratio (A3/A2)	2.08
Contraction ratio (A1/A2)	1.56
Tilt angle (a)	4.5°

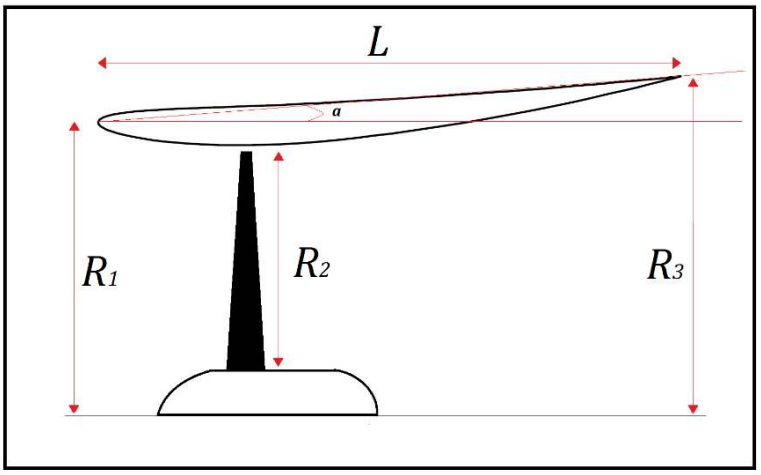


Table 5.3 Final comparison between bare and ducted turbine

	Bare turbine			Ducted turbine
	Theoretical (Muratoğlu)	**CFX simulation (Muratoğlu)**	**FLUENT simulation (Tsiry)**	**FLUENT simulation (Tsiry)**
Power (kW)	27.48	28.99	26.95	40.28
Torque (kN.m)	5.83	6.17	5.72	8.55
Thrust (kN)	17.05	17.48	17.6	22.27

5.2. Conclusions and future works

During this work, an attempt has been made in increasing the power output of an hydrokinetic turbine. In order to achieve that goal, the turbine was put inside a duct. The design of the duct was the focus of this work. At the end of the design an increase of 40% of power output from the bare turbine was observed.

Though this work can be said to be successful, many engineering aspects were not considered. So in the future, details such as material design and cost of the duct should be analyzed. Environmental impact of the device should also be considered in future works.

At the end of the study, it was concluded that if a turbine was designed specifically to function inside a duct then even more output power can be obtained. So in the future, to design a better and more efficient ducted turbine, apart from the duct, the turbine should be built from scratch and its design should take in consideration the presence of the duct around it.

REFERENCES

[1] Business Dictionary <http://www.businessdictionary.com/> (access date June, 2016)

[2] Current World Population <http://www.worldometers.info/world-popul/> (access date June, 2016)

[3] World Electricity <http://www.indexmundi.com/world/electricity_con> (access date June, 2016)

[4] Wikipedia <https://en.wikipedia.org/wiki/Hoover_Dam/> (access date 6, 2016)

[5] Not a lot of people know that <https://notalotofpeopleknowthat.word> (access date July, 2016)

[6] Total Energy <http://www.eia.gov/totalenergy/> (access date July, 2016)

[7] Global Trends in Renewable Energy Investment 2016 <http://www.fs-unep-centre.org> (access date July 6, 2016)

[8] Awan, A.B., and Khan, Z.A. (2014). Recent progress in renewable energy – Remedy of energy crisis in Pakistan. Renewable and Sustainable Energy Reviews, 33, 236-253.

[9] Wikipedia <https://en.wikipedia.org/wiki/Watermill> (access date July, 2016)

[10] Riverine Hydrokinetic Technology: A Review Long Pham, Student Member, IEEE.

[11] Energy without carbon <http://www.energy-without-carbon.org/sites/def> (access date July, 2016)

[12] Nautic Expo <http://www.nauticexpo.com/pro> (access date July, 2016)

[13] Schottel <http://www.schottel.de/fr/schottel-hydro/> (access date July, 2016)

[14] ABS Alaskan <http://www.absak.com/catalog/index.php/cPath/33_89> (access date July, 2016)

[15] Alternative Energy Tutorials <http://www.alternative-energy-tutorials.com/hydro-energy/waterwheel-design.htm> (access date July, 2016)

[16] Yuce, M. I. and Muratoglu, A. (2014). Hydrokinetic energy conversion systems: A technology status review, Renewable and Sustainable Energy Reviews, 43, 72-82.

[17] Hansen, M.O.L. (2000). Effect of placing a diffuser around a wind turbine, Wind Energy, 3, 207-213.

[18] CADDET Centre for Renewable Energy, Water Current Turbines Pump Drinking Water, Technical Brochure No 83.

[19] Clean Current <http://www.cleancurrent.com/> (access date July, 2016)

[20] Hydroquest <http://www.hydroquest.net/wp-media/uploads/2015/> (access date July, 2016)

[21] Khan, M. J., Bhuyan, G., Iqbal, M.T. and Quaicoe, J. E. (2009). Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review, Applied Energy, 86(10), 1823–1835.

[22] Shives, M.R. (2011). Hydrodynamic Modeling, Optimization and Performance Assessment for Ducted and Non-Ducted Tidal Turbines. M.Sc. Thesis, University of Victoria

[23] Quora <https://www.quora.com/What-is-the-difference-between-kin> (access date July, 2016)

[24] Wikipedia <https://en.wikipedia.org/wiki/Eddy_(fluid_dynamics)> (access date July, 2016)

[25] Encycl. of Life Support Syst <http://www.eolss.net/eolsssamplechapter> (access date July, 2016)

[26] Durham University Community <https://community.dur.ac.uk/suzanne> (access date July, 2016)

[27] Gorban', A.N., Gorlov, A.M. and Silantyev, V. M. (2001). Limits of the Turbine Efficiency for Free Fluid Flow, Journal of Energy Resources Technology, 123(4), 311-317.

[28] Kulunk, E. (2011). Aerodynamics of Wind Turbines, Fundamental and Advanced Topics in Wind Power, Dr. Rupp Carriveau (Ed.) ISBN: 978-953-307-508-2.

[29] Magdi Ragheb and Adam M. Ragheb (2011). Wind Turbines Theory - The Betz Equation and Optimal Rotor Tip Speed Ratio, Fundamental and Advanced Topics in Wind Power, Dr. Rupp Carriveau (Ed.).

[30] MIT <http://web.mit.edu/2.972/www/reports/airfoil/airfoil.html> (access date July, 2016)

[31] Daniel M., Busra A., Carlos S. F., Tonio S., Gerard van Bussel (2014). The origins of a wind turbine tip vortex, Journal of Physics: Conference Series 555.

[32] Spera, D.A. (1994). Wind Turbine Technology, Fundamental Concepts of Wind Turbine Engineering. ASME Press, New York.

[33] Wikipedia <https://en.wikipedia.org/wiki/Mathematical_optimization> (access date July, 2016)

[34] Venter, G.: Review of optimization techniques (2010). In: Blockley, R., Shyy, W. (eds.)
Encyclopedia of Aerospace Engineering, pp. 5229–5238. Wiley, New York.

[35] Holland J.H. (1975). Adaptation in Natural and Artificial Systems. University of Michigan Press.

[36] Kennedy J. and Eberhart R.C. (1995). Particle swarm optimization. In Proceedings of the 1995 IEEE International Conference on Neural Networks, Perth, Australia, 1995; 1942–1948.

[37] Vanderplaats G.N. (2007). Multidiscipline Design Optimization. Colorado Springs, Vanderplaats Research and Development, Inc.

[38] Errico, R. M., 1997: What is an adjoint model? Bull. Amer. Meteor. Soc., 78, 2577–2591,

[39] Steven G. J. (2012). Notes on Adjoint Methods for 18.335 <http://math.mit.edu/~stevenj/18.336/adjoint.pdf> (access date August, 2016)

[40] ANSYS, Fluent Adjoint Solver Module Manual, ANSYS CFD release 17.0, 2015.

[41] Wizelius, T. (2007). Developing Wind Power Projects Theory & Practice.

[42] Prowind <http://www.prowind.ca/downloads/Resources/Report.pdf> (access date August, 2016)

[43] Poul La cour Museum <http://www.poullacour.dk/index-uk.htm> (access date August, 2016)

[44] The Guardian <https://www.theguardian.com/environment/2008/> (access date August, 2016)

[45] Muscle Horsepower <http://www.musclehorsepower.com/the-most-powerful-wind-turbine-in-the-world-enercon-e126/> (access date August, 2016)

[46] Hydrovolts <http://hydrovolts.com/wp-content/uploads/2011/06/In-Str> (access date July, 2016)

[47] CADDET Centre for Renewable Energy, Water Current Turbines Pump Drinking Water, Technical Brochure No 83.

[48] Muratoğlu A. (2014) Design and Simulation of a Riverine Hydrokinetic Turbine, PhD thesis.

[49] Chica, E., P´erez, F., Rubio-Clemente, A and Agudelo, S. (2015). Design of a hydrokinetic turbine, Energy and Sustainability VI, 137.

[50] André, L.A.M., Alexandre, L.A.M., Felipe, C. P., Jerson, R.P.V., Marcus, V.G.M. and Carmo, G. (2014). A methodology for the transient behavior of horizontal axis hydrokinetic Turbines, Energy Conversion and Management, 87, 1261–1268.

[51] Laurens, J.M., Ait-Mohammed, M. and Tarfaoui, M. (2015). Design of bare and ducted axial marine current turbines, Renewable Energy 89, 181-187.

[52] Kolekar, N., Hu, Z., Banerjee1, A., and Du, X. (2013) Hydrodynamic Design and Optimization of Hydro-kinetic Turbines using a Robust Design Method, Proceedings of the 1st Marine Energy Technology Symposium Washington, D.C.

[53] Ismail, K.A.R. and Batalha, T.P. (2015). A comparative study on river hydrokinetic turbines blade profiles, Int. Journal of Engineering Research and Applications, 5(5), 01-10.

[54] Silva, P.A.S.F.; Shinomiya, L.D.; de Olivera, T.F.; Vaz, J.R.P.; Mesquita, A.L.A.; Brasil, A.C.P., Jr. (2009). Analysis of cavitation for the optimized design of hydrokinetic turbines using BEM. Appl. Energy 2016, in press. 55- Sale, D., Jonkman, J. and W. Musial.

[55] Sale, D., Jonkman, J. and W. Musial (2009). Hydrodynamic optimization method and design code for stall-regulated hydrokinetic turbine rotors. National Renewable Energy Laboratory, United States.

[56] Romero-Gomez, P. and Richmond, M.C. (2014). Simulating blade-strike on fish passing through marine hydrokinetic Turbines, Renewable Energy, 71, 401-413.

[57] Sanuki, M. Studies on biplane wind vanes, ventilator tubes and cup anemometers. 1950 Pap Meteorol Geophys. 1.1 81-132

[58] M.Iwasaki. The experimental and theoretical investigation of windmills. Reports of Research Institute for Applied Mechanics, 2(8): 181 – 229, dec. 1953.

[59] Lilley, G.M and Rainbird, W.J. (1957). A preliminary report on the design and performance of ducted windmills. College of Aeronautics, Cranfield.

[60] Foreman, K. M., Gilbert, B., and Oman, R. A., Diffuser Augmentation of Wind Turbines. Solar Energ, 20: pp. 305-311, 1978

[61] Gilbert, B.L., Oman, R.A., Foreman, K.M. 1978. Fluid dynamics of diffuser-augmented wind turbines. Journal of Energy 2 (6): 368–374.

[62] Foreman, K.M., Gilbert, B.L. 1979. Technical development of the diffuser augmented wind turbine concept. Wind Engineering 3 (3): 153–166.

[63] Phillips, D.G., Flay, R.G.J., Nash, T.A. “Aerodynamic Analysis and Monitoring of the Vortec 7 Diffuser Augmented Wind Turbine”, IPENZ Trans., Auckland, New Zealand, November 1999.

[64] Phillips DG, Richards PJ, Mallinson GD, Flay RGJ (1998). Computational modelling of diffuser designs for a diffuser augmented wind turbine. 13th Australasian Fluid Mechanics Conference, Melbourne.

[65] Phillips, D.G., Richards P.J. and. Flay R.G.J. Diffuser Development for a Diffuser Augmented Wind Turbine Using Computational Fluid Dynamics.

[66] K. Abe, Y. Ohya, An investigation of flow fields around flanged diffusers using CFD, J. Wind Eng. Indust. Aerodyn. 92 (2004) 315–330.

[67] Altair <http://www.altair.com/ResourceLibrary> (access date July, 2016)

[68] ANSYS <https://support.ansys.com/staticassets/ANSYS/Conference/> (access date July, 2016)

[69] RBF-Morph <http://www.rbf-morph.com/> (access date July, 2016)

[70] Symscape <http://www.symscape.com/blog/moving-reference-frame-for-cfd> (access date July, 2016)

[71] Javafoil <http://www.mh-aerotools.de/airfoils/javafoil.htm> (access date July, 2016)