UNIVERSITY OF GAZİANTEP
GRADUATE SCHOOL OF
NATURAL & APPLIED SCIENCES

FLOW FIELD AROUND HYDROKINETIC TURBINES

M. Sc. THESIS
IN
CIVIL ENGINEERING

BY
AUMED M. AMEN
JULY 2013

Flow Field around Hydrokinetic Turbines

M.Sc. Thesis
In
Civil Engineering
University of Gaziantep

Supervisor
Assist. Prof. Dr. Mehmet İshak YÜCE

By
Aumed M. AMEN
July 2013

T.C.
UNIVERSITY OF GAZİANTEP
GRADUATE SCHOOL OF
NATURAL & APPLIED SCIENCES
CIVIL ENGINEERING DEPARTMENT

Name of the thesis: Flow Field around Hydrokinetic Turbines

Name of the student: Aumed M. AMEN
Exam date: 02/07/2013

Approval of the Graduate School of Natural and Applied Sciences

Assoc. Prof. Dr. Metin BEDİR
Director

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

Prof. Dr. Mustafa GÜNAL
Head of Department

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

Assist. Prof. Dr. Mehmet İshak YÜCE
Supervisor

Examining Committee Members	Signature
Prof. Dr. Mustafa GÜNAL	
Assist. Prof. Dr. Nihat ATMACA	
Assist. Prof. Dr. Mehmet İshak YÜCE	

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 material and results that are not original to this work.

Aumed M. AMEN

ABSTRACT

FLOW FIELD AROUND HYDROKINETIC TURBINES

M.AMEN, Aumed Rahman
M.Sc. in Civil Engineering
Supervisor: Assist. Prof. Dr. Mehmet İshak YÜCE
July 2013, 71 pages

In the search for clean, renewable and sustainable energy, the hydrokinetic energy which is harnessed from moving water of rivers, streams and oceans is being studied as a predictable and environmentally benign source. Compare to conventional hydropower turbines, hydrokinetic turbines are new technology in electricity generation and need to be improved from efficiency point of view. In this study, a vertical axis cross-flow hydrokinetic turbine, namely, a modified form of Savonius turbine was investigated. The working principles of cross-flow hydrokinetic turbines are different from those of commonly used horizontal axis turbines. The advantages of this type of turbines are; independency from the current direction including reversibility, stacking and self-starting without complex pitching mechanisms. The turbine has been simulated in a three dimensional and fully developed rectangular open channel flow. Computational fluid dynamics (CFD) simulation of the hydrokinetic turbine was performed by computationally solving the Reynolds-Averaged Navier-Stokes Equations (RANS). In computational studies ANSYS FLUENT, which is commercially available software was employed. CFD can be used to support turbine design and performance over a wide range of parameters to minimize the number of prototypes which are used for optimization and experimental studies. CFD can also provide a cost-effective way of evaluating detailed full scale effects, such as mooring lines or local bottom bathymetry features, on both turbine performance and environmental assessment.

Keywords: Computational Fluid Dynamics, Hydrokinetic Turbines, Kinetic Energy, Renewable Energy, ANSYS FLUENT

ÖZET

HIDROKINETIK TÜRBINLERIN ETRAFINDAKI AKIM ALANININ INCELENMESI

M.AMEN, Aumed Rahman
İnşaat Mühendisliği Yüksek Lisans
Danışman:Yrd. Doç. Dr. Mehmet İshak YÜCE
Temmuz 2013, 71 sayfa

Temiz, yenilenebilir ve çevre dostu enerji üretim yöntemlerinden biri de dere, nehir ve okyanuslarda akış halindeki sulardan hidrokinetik enerji türbinleri yardımıyla enerji elde edilmesidir. Hidrokinetik türbinlerin genel verim ve teknolojileri hususunda geleneksel hidroelektrik türbinlerine nazaran geliştirilmeleri gerekmektedir. Hidrokinetik enerji, henüz yeni bir enerji üretme metodu olarak ortaya çıktığı için genel verim ve teknoloji hususunda daha da geliştirilmesi gerekmektedir.Hidrokinetik türbinler, geleneksel hidroelektrik türbinlere nazaran yeni nesil çevreci enerji üretimi yapmaktadır. Bu türbinlerinin verimlerinin artırılması konusunda daha fazla çalışma yapılmalıdır. Bu çalışmada çapraz akışlı dik eksenli hidrokinetik türbinler analiz edilmiştir. Çapraz akışlı türbinler, yatay eksenli türbinlere nazaran farklı prensiplerle çalışmaktadırlar. Bu türbinler, suyun akış yönünden bağımsız olmaları, tüm yönlerden gelen su vektörlerini yakalayabilmeleri, birden fazla türbinin yanyana dizilebilmesi ve kendiliğinden başlayabilme mekanizması gibi avantajlara sahiptirler. Bu çalışmada Gorlov ve Darrieus tarzı çapraz akışlı türbinler 3 boyutlu tam gelişmiş dikdörtgensel açık kanalda hesaplamalı akışkanlar dinamiği metodolojisi kullanılarak analiz edildi. Simülasyonlar numerik RANS (Reynolds averaged Navier-Stokes) denklemleri kullanılarak yapıldı. Numerik çalışmalarda hesaplamalıa kışkanlar dinamiği konusuda dünyada, en, çokkullanılana kademik ve ticari yazılım olan ANSYSFLUENT. İle yapıldı.Türbin dizaynının. HAD ile yapılması, deney ve optimizasyonlarda kullanılan gerçek ölçekli türbin prototiplerine yapılacak masraf ve zahmeti büyük ölçüde azaltmakta olup montajlama, su tabanı topoğrafyası vb. hususlaraçısındankolaylıklarsağlamaktadır.

Anahtar Kelimeler: Hesaplamalı Akışkanlar Dinamiği, Hidrokinetik Türbinler, Kinetik enerji, yenilenebilir enerji, ANSYS FLUENT

ACKNOWLEDGMENT

I express my sincere gratitude to my supervisor Assist. Prof. Dr. Mehmet İshak YÜCE for his perfect guidance and invaluable advice during this research. It was my pleasure to work under his supervision. I would not have been able to complete this work without his guidance and direction.

I would like to thank my relatives for their help, without their support this research project would not have been possible.

My sincere appreciation also extends to my friends and to those gave me the delight of smile especially my friend Rawsht Mustafa.

TABLE OF CONTENTS

CONTENTS **Page**

LIST OF FIGURES

FIGURES **Page**

LIST OF TABLES

TABLES **Page**

LIST OF SYMBOLS / ABBREVIATIONS

A	Cross sectional area a of turbine blade
a	Acceleration
CFD	Computational Fluid Dynamics
Cp	Overall efficiency of turbine
H	Overall head
HEPP	Hydro electrical power plant
K	kinetic energy
L	Length
LES	Large Eddy Simulation
m	Mass
P	Over all power of the dam
Q	Average discharge
RANS	Reynolds average Navier-stokes equation
Re	Reynolds number
RNG	Re-Normalisation Group

SHPP	Small Hydro Power Plant
ρ	Density of water
μ	Overall efficiency of HEPP
V	Stream velocity
2D	Two Dimensional
3D	Three Dimensional
.μ	Viscosity
ΣF	Sigma force
Ɛ	Epsilon
ɯ	Omega

CHAPTER 1

INTRODUCTION

1.1 Energy

Energy is the ability to do work. All life on the earth is based on the energy. Energy exists with different forms. All energy types are accepted to be generated from the sun. Solar power, both in the form of direct solar radiation and indirect forms such as bio energy, water and wind power were the energy sources of early human societies. When our ancestors first used fire, they were actually harnessing the power of photosynthesis, the solar driven process by which plants are created from water and atmospheric carbon dioxide. Until the eighteenth century, the wood fire and animals were the main source of energy. However after the invention of the steam engine, the coal and other greenhouse gas emitting energy sources have been used.

1.2 Fossil Fuels and greenhouse gas emission

Last three or four centuries huge amount of greenhouse gas were emitted by the fossil fuel consumption. Greenhouse gases (GHG) such as carbon dioxide, methane and sulfur dioxide, nitrous oxide and tropospheric ozone are the main greenhouse gases that are responsible from the global warming. These gases are covering the earth’s atmosphere and do not let the sun’s rays to reflect back to the atmosphere and causing an increment in the global temperature. When human being experienced the energy crisis, climate change and global warming, scientists realized the detrimental effects of fossil and nuclear resources. Then, researchers tried to develop new ways of harnessing energy from natural and non-consuming resources. The World Energy Forum has predicted that the fossil fuel type energy resources will consume in less than next 10 decades (Toklu, 2013). In June of 2009 the European parliament and council signed its 20-20-20 climate and energy package. The main scope of this package was to provide at least 20 % of European Union's energy consumption from renewable sources and to reduce the greenhouse gas emission at least 20 % from 1990 levels, until the year of 2020. (Taylor, 2012).In order to meet these energy standards across the globe, researchers should be guided to find new renewable energy production methods. World's primary energy use by different sources is given in Figure 1.1.

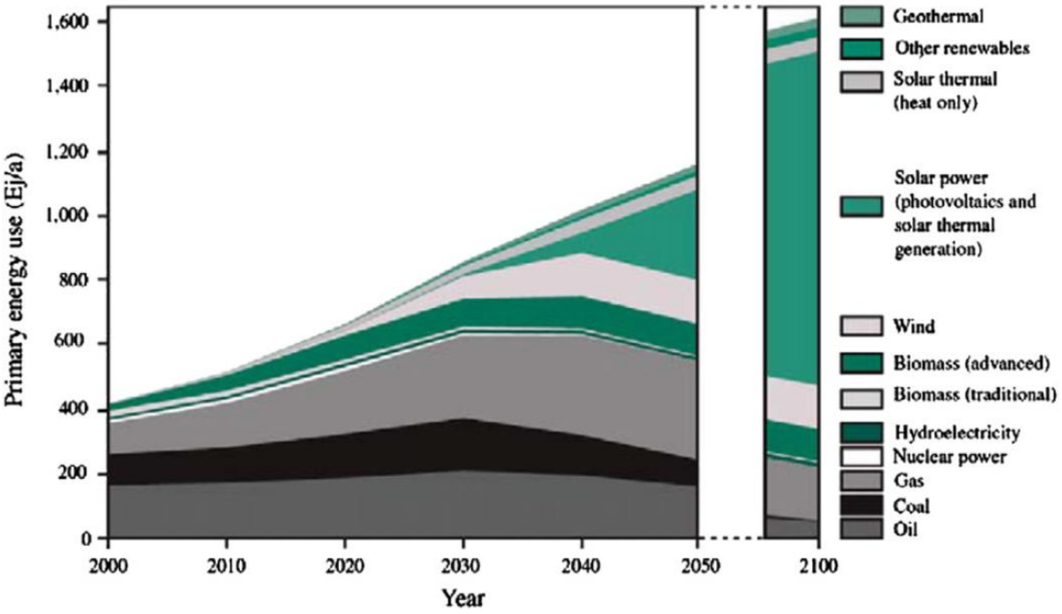


Figure 1.1 World primary energy use by different sources (Toklu, 2013)

1.3 Renewable Energy

The energy is called as renewable if it generated from natural resources (Sunlight, Wind, River Waters, Tides, Waves, and Geothermal) and replenished at the same rate they are used. Renewable energy sources provide more economic and continuous solutions than fossil and nuclear fuels according to the International Energy Agency (IEA, 2011).

Figure 1.2 Renewable Energy Re-sources (www.best-off-grid-computers.com)

Wind energy has emerged as the leader of new energy sources, while other options continue to be investigated by scientists. The kinetic energy of water current in oceans, rivers, and streams is being accepted as predictable and environment friendly sources. Tidal flows have also been recognized as a potential opportunity to harvest clean and predictable type of energy.

1.4 Potential of renewable energy sources

Geothermal energy provides the highest potential among other renewable energy resources with 140 million EJ (Exajoule) and 5000 EJ theoretical and technical potentials, respectively. Hydropower which provides a mechanical conversion of energy has a theoretical potential about 150 EJ. The technically feasible of this amount is estimated as 50 EJ. The wind energy is also a big energy provider with a theoretical potential about 6000 EJ. A 10 percent of this amount is found to be technically feasible. On the other hand, it is estimated the total tidal energy potential of the world is approximately 3 TW with 1 TW in accessible areas for installation of energy extracting devices (Taylor, 2012, Resch at. al. 2008). The technical and theoretical energy potentials are given in Figure 1.3.

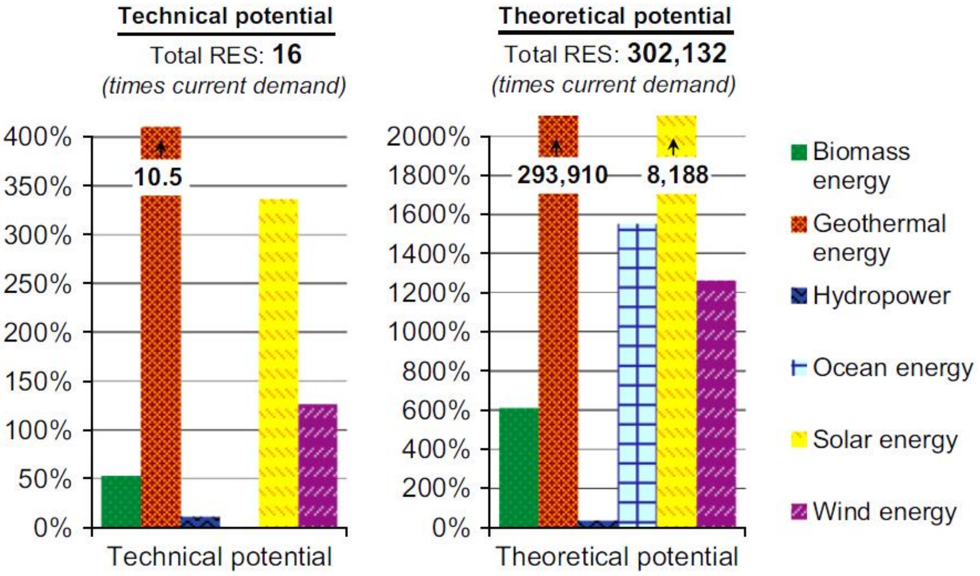


Figure 1.3 Technical and theoretical potential of renewable energy sources (Resch at. al. 2008)

1.5 Renewable energy types

1.5.1 Solar Energy

The energy of the sun is the source of all energies and on the earth. Solar photovoltaic energy is the direct conversion of the sun's rays to electricity. All of the other renewable are produced with the effect of the sun. In the last two decades, scientists began to use direct energy of the sun's radiation using photovoltaic cells and the heat provided by sun in thermal power plants (Godfrey, 2010).

1.5.2 Wind Energy

The kinetic energy of the wind is exploited using wind turbines to generate electricity. Wind energy has been used for thousands of years for milling grain, pumping water and other mechanical power applications. Today, there are hundreds of windmills in operation around the world; many of the mare used for water pumping (Godfrey, 2010).

1.5.3 Ocean Energy

The oceans are huge masses of heats. The temperature difference between surface and deep waters of the oceans can be utilized. Also the potential energy coming from the rising and falling of the tides, and kinetic energy from the wave and water currents can be used. In the last years, there are many studies to exploit the ocean wave and current energy (Godfrey, 2010).

1.5.4 Hydropower

The water power can be harnessed via two ways. The traditional way to produce electricity with dams employs reservoirs to accumulate water in. The storage reservoirs provide a head of water with static pressure. The static energy of stored water is converted to the kinetic energy with high speeds flow inside the turbines. This type of power production using traditional dams is called as hydrostatic. Hydrostatic is the one of the oldest renewable energy extraction techniques to produce hydroelectric power. Small scale hydroelectric power plants provide an alternative way of energy production. They are suitable in tight and off-grid areas where a large HEPP cannot be built. In recent years there are huge studies on small hydroelectric power plants. (Twidell and Weir, 2006).In the last two decades, hydrokinetic energy which is a new technique of producing electricity from water currents is investigated. This approach employs turbines inside the water current to convert kinetic energy of flow directly to the electricity without water impoundment. The energy inside moving tides, flowing rivers and ocean currents can be extracted via this new approach. The advantages, drawbacks and environmental effects of traditional hydropower approach and further information about hydrokinetics is given in following parts.

1.5.5 Geothermal Energy

The historical exploitation of geothermal resources dates back to Greek and Roman times. The early efforts were to harness hot water for medicinal, domestic and leisure applications. It is the accessible thermal energy of heat content of the earth's crust. It is used for industrial and recreational purposes and for heating and cooling of buildings (Andrews and Jelly, 2007).

1.5.6 Bioenergy

It is the general term for energy derived from materials such as wood, straw or animal wastes which are recently living rests in contrast to the fossil fuels. Such materials can be burned directly to produce heat or power, but can also be converted in to biofuels (Godfrey, 2010).

1.6 Open channel

Open channels can either be artificial (for irrigational purposes) or natural channels (rivers). Unlike artificial channels, the geometry of the natural channels is generally varied. The variation of geometry and surface roughness in natural channels makes it difficult to predict the flow characteristics accurately. However this thesis is only concerned with artificial channels having regular cross section.

The flow characteristics in open channels are given below;

a. Uniform flow; the depth and velocity of the flow are constant at every section of the channel.

b. Steady non-uniform flow; depth varies with distance but constant with time.

c. Unsteady flow; varies with both time and distance.

Pipe and open channel flow have some differences. The open channel flow is driven by the gravity; however, pipe flow is under pressure. (H, chanson)Open channel flow has a particular free surface. Because of the existence of the free surface, the channel depth varies with cross-section adding extra difficulties to the flow analyses (White, 2008; Sam, 2010; Das, 2011).

1.7 The purpose of thesis

The main purpose of this thesis is to support the development process of hydrokinetic energy studies by investigating the velocity profile and turbulence around the turbine and revealing the outlines of optimum placement of turbine arrays. In this thesis, the specific objectives are to design and simulate a hydrokinetic turbine inside a 3D channel and investigating the flow around the turbine by changing its position to examine the effect of changing velocity around the turbine.

1.8 Structure of Thesis

This study is composed of totally 8 chapters. The first chapter provides an introduction to the study of the simulation of a hydrokinetic turbine. In this chapter, overall information about the fossil fuels, renewable energy was introduced. The types and potential of different renewable energy sources were introduced. The open channel environment was illuminated and the purpose of the thesis was explained.

Chapter 2 is about the hydropower. The definition water and its amount in different sources were given. The water cycle has been mentioned. The applications of hydropower in the world were supplied. The environmental impacts of hydropower were mentioned and a classification of current hydropower production techniques was introduced.

Chapter 3 provides a general outline of hydrokinetic turbine methodology. In this chapter, the hydrokinetic turbines were evaluated from the power curve to the environmental effects. A classification of hydrokinetic energy turbines was given. Also some pictures for the most popular hydrokinetic turbines were placed.

Chapter 4 explains the Computational Fluid Dynamics terminology which a fundamental approach of this study. The different stage of a CFD simulation was introduced.

Chapter 5 provides a background to the definition of turbulence and turbulence models. The flow chart which shows the methodology of a CFD work is given.

A vertical axis hydrokinetic turbine is analyzed in chapter 6. The simulation process and detailed information for each step is explained.

The result and discussion of the thesis was located in chapter 7. The results were examined.

Finally the conclusion and future works were given in chapter 8.

CHAPTER 2

HYDROPOWER

2.1 Water and its energy

Water is a molecule containing one oxygen and two hydrogen atoms. The 71 % of earth surface is covered with water. There is totally 1.4 billion cubic kilometers of water on the earth including atmosphere. It is the vital substance that all life is depending on. About 97 % of water on the earth is found in the oceans, 1.7 % is under the ground and 1.7 % is in glaciers. Only 0.02 % of water is located inside the rivers and lakes. Nearly 980 liters of water evaporates from each square meter on the earth surface. The water cycle converts about 22 % of the sun's energy coming to the earth. Precipitation rate that comes down on the oceans and lands on the earth is 80 and 20 % respectively. (Wikipedia, 2013; Quaschning, 2010).The global water cycle is given in Figure 2.1. The flowing or elevated water has a static or kinetic energy according to a reference plane. The scope of hydropower engineering is to convert the pressure and kinetic energy of water into more easily used electrical energy (Warnick, 1984). The huge amount of motions in rivers, streams, marine and oceans can be converted to a feasible type of energy. There are various ways to harness the different forms of hydropower. The power of flowing rivers is collected in a reservoir and converted to the energy using hydroelectric power plants (HEPP). The tidal energy which is produced by the gravitational attraction of the moon and sun gives a movement to the water, especially in thigh shores. The tidal movements can be converted to electricity using hydrokinetic turbines and tidal barrages. The wave energy which is originated by the effect of the winds on the sea and oceans can be harnessed via suitable wave energy convertors. Water currents in river and seas can be exploited using hydrokinetic turbines. This chapter concentrates on the background of different methods for hydropower production.

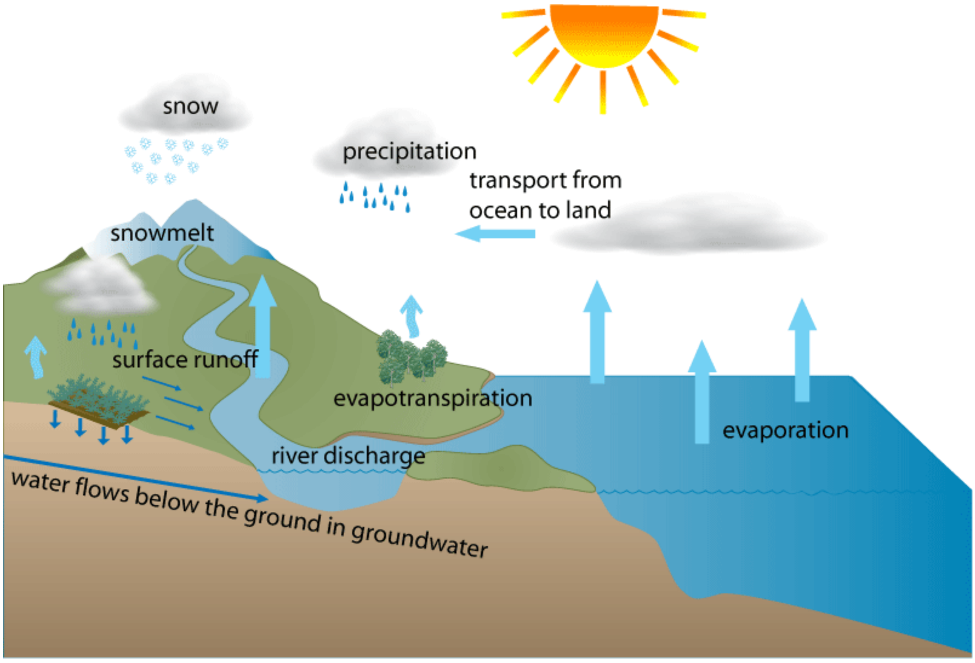


Figure 2.1 The water cycle (www.teachoceanscience.net)

2.2 History of hydropower

The oldest used energy source is hydropower. In the early history, the human being were used the power of water for transportation using boats. Egyptians firstly used the water to rotate water wheels obtaining a shaft power for grinding the grains by lifting a mass of water from lower to higher elevations (Schlager and Weisblatt, 2006, Muratoglu 2011). Ancient Greek, Syrians and Nile River civilizations also used the power of water. The Hama city in Syria is famous with ancient water wheels. Nearly 500,000 windmills were found in the Europe until the beginning of 19th century (Quaschning, 2010). The hydroelectricity firstly used in 1880, in a factory in United States. The first commercial hydroelectric power plant was operated in 1882, on the Fox River. Lester Allan Pelton who is known as the father of hydroelectricity and inventor of Pelton wheel, made a great effort on electric production from water. (Schlager and Weisblatt, 2006). Today there are about 50,000 large dams on the earth for irrigation and domestic purposes. The total area of large dam reservoirs by region is given in Figure 2.2.

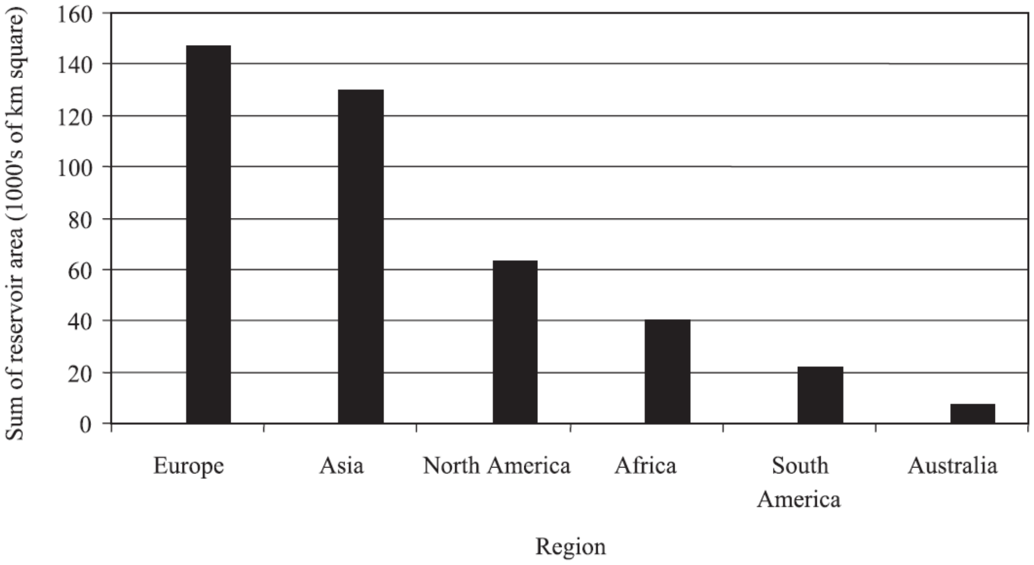


Figure 2.2 The total area of large dam reservoirs by region (Balat, 2006)

2.3 Hydropower in the world

According to (Herzog et. al., 2004) the gross theoretical hydroelectricity potential is about 41,000 TWh/year. Nearly 30% of this amount corresponding about 14,000 TWh/yr is found to be technically feasible. The global installed hydroelectricity capacity is 680,000 MW and the economically feasible hydropower potential is predicted as nearly 7000 TWh/yr. The popular countries for hydropower production are Canada and Norway. Canada is the leader hydropower producer country. On the other hand, Norway which is the cleanest country in the world produces about 99 % of its electricity from water. It is estimated that about 25 % of world's hydropower potential has been exploited with large scale hydroelectric power plants. The hydroelectric power potential for different countries and the world total is given in Table 2.1.

Table 2.1 Theoretically, technically and economically feasible hydroelectric potential by region (Herzog et. al, 2004 cited in Muratoglu, 2011)

Region	Gross theoretical potential (TWh/yr)	Technically feasible potential (TWh/yr)	Economically feasible potential (TWh/yr)	Installed hydro capacity (GW)	Hydropower production (TWh/yr)
North America	5817	1509	912	141.2	697
Latin America and Crbn.	7533	2868	1199	114.1	519
Western Europe	3294	1822	809	16.3	48
Central and Eastern Eur.	195	216	128	9.1	27
Former Soviet Union	3258	1235	770	146.6	498
Middle Est. and Nrt. Afr.	304	171	128	21.3	66
Sub Saharan Africa	3583	1992	1288	65.7	225
Centrally Planned Asia	6511	2159	1302	64.3	226
South Asia	3635	948	103	28.5	105
Pacific Asia	5520	814	142	13.5	41
Pacific OECD	1134	211	184	34.2	129
World total	40784	13945	6965	654.8	2581
Turkey	**435**	**215**	**128**	**12.6**	**45**
Turkey/World total (%)	1.07	1.54	1.84	1.92	1.74

2.4 Environmental impacts of large scale hydropower production

There are several environmental impacts of current hydropower conversion technologies such as inundation, sedimentation, greenhouse gas emission and other regional effects. The biggest environmental effect of large scale HEPPs is the reservoir construction. The proposed reservoir of a dam can be a fertile land for agriculture, forest, alluvial rich land. Building a large reservoir can drown this type of areas. On the other hand, a reservoir area could be a habitat area. The living area for animals or humans can be destroyed. At the same time, the important economic, social and historical heritage lands such as forests, mineral deposits, historical or archeological areas can be swept away. Therefore when a dam site is designed, all of these concerns should be revised.

Figure 2.3 Hasankeyf includes the historical remains of early civilizations and it will be submerged after the construction of Ilısu dam

Changing the sedimentation regime of the river bed is also a big environmental impact of hydropower production, after inundation. The sediment load of a river which is an extremely fertile material moved to the reservoir with flowing water. The transported sediments decrease the fertility of the lands at the downstream region. Also they shorten the lift time period of the dam.

Finally the large scale hydropower production can damage the fish ecosystem and transportation. The chemical that are used in the mechanical equipment of the dam can produce detrimental effects for the water environment. The decomposition debris accumulated at the top of reservoir can become a great source of methane with little effect to the global warming. Land erosion, stimulating seismic activities due to high pressure of water and possibility of terrorist attacks are among the other minor disadvantages of dam construction. The overall list of advantage and disadvantages of hydropower production is given in Table 2.2.

Table 2.2 Advantages and disadvantages of hydropower production (Yuksel, 2010)

Advantages	Disadvantages
Economical aspect	
Provides low operating and maintenance costs	High upfront investment
Provides long life span (50–100 years and more)	Precipitation
Provides reliable service	Requires long-term planning
Includes proven technology	Requires long-term agreements
Instigates and fosters regional development	Requires multidisciplinary involvement
Provides highest energy efficiency rate	Often requires foreign contractors and funding
Creates employment opportunities and saves fuel	
Social aspects	
Leaves water available for other uses	May involve resettlement
Often provides flood protection	May restrict navigation
May enhance navigation conditions	Local and land-use patterns will be modified
Often enhances recreation	Waterborne disease vectors may need to be checked
Enhances accessibility of the territory and its resources	Requires management of competing water uses
Environmental aspects	
Produces no pollutants but only very few GHG emission	Inundation of terrestrial habitat
Enhances air quality	Modification of hydrological regimes
Produces no waste	Modification of aquatic habitats
Avoids depleting non-renewable fuel resources	Water quality needs to be managed
Often creates new freshwater ecosystems with increased productivity	Temporary introduction of methyl mercury into the food chain needs to be monitored/managed
	Species activities and populations need to be monitored
Enhances knowledge and improves management of valued species due to study results	Barriers for fish migration, fish entrainment.
Helps to slow down climate change	Sediment composition and transport may need to be monitored/managed
Neither consumes nor pollutes the water it uses for electricity generation purposes	

2.5 Classification of hydropower production

2.5.1 Large scale hydroelectric power plants

Large scale reservoirs to produce massive amount of power are very important for economic development of a country. Globally, the suitable sites for large scale hydropower are mostly exploited. In this system, huge amount of water is collected and very large head is produced. The stationary water which contains a potential energy according to a reference frame is carried to the turbines with gigantic pipes. The static energy of water is converted to the kinetic energy at the entrance of turbines. Then, kinetic energy is transformed to the machine power with turbine blades. The power of the water accumulated in a reservoir can be calculated with Equation 2.1. The hydroelectric power production method of dams is given in Figure 2.4.

$$P = \rho g Q H \mu \tag{2.1}$$

Where P is the overall power of the dam, ρ is the density of water, Q is the average discharge, H the overall head and μ is the overall efficiency of the HEPP.

The already exploitation of potential large scale hydropower and necessity of power in rural and off-grid areas motivate the scientists and engineers to generate small scale powers with relatively small hydroelectric power plants.

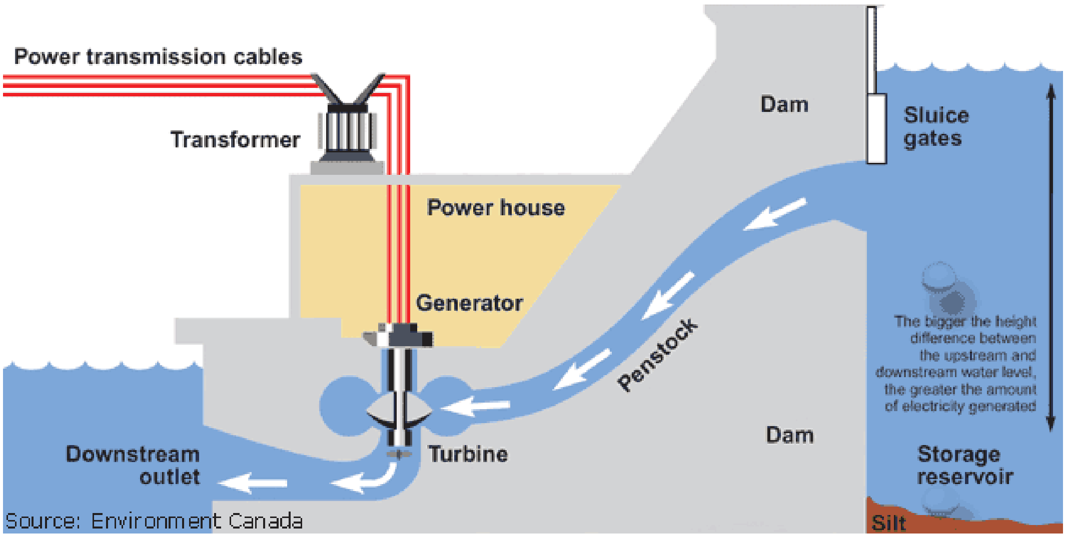


Figure 2.4 The hydroelectric power production method of dams (USGS, 2013).

2.5.2 Small scale hydroelectric power plants

Small scale hydroelectric power plants (SHPP) are relatively small dams that were constructed on rivers which have relatively small reservoir area and head (Muratoglu, 2011). A specific amount of water is taken from the river and transported to lower situated turbines. The energy production method is more or less same to that of dams. However the scales of water accumulation and turbines are lower. These type of structures are more environment friendly compared with Large scale HEPP’s. Blocking the river navigation is the biggest disadvantage of this type of power production. The schematic representation of a small hydropower system is given in Figure 2.5.

The world’s SHPP potential is estimated to be around 500 GW. Today nearly one fifth of this potential is already exploited (Freris and Infield, 2008). It is available 71 SHPP, in Turkey. The gross theoretical potential of SHPP is estimated as 50,000 GWh/yr for Turkey. The technically and economically feasible potentials are 30,000 and 20,000 GWh/yr respectively (ESHA, 2004).

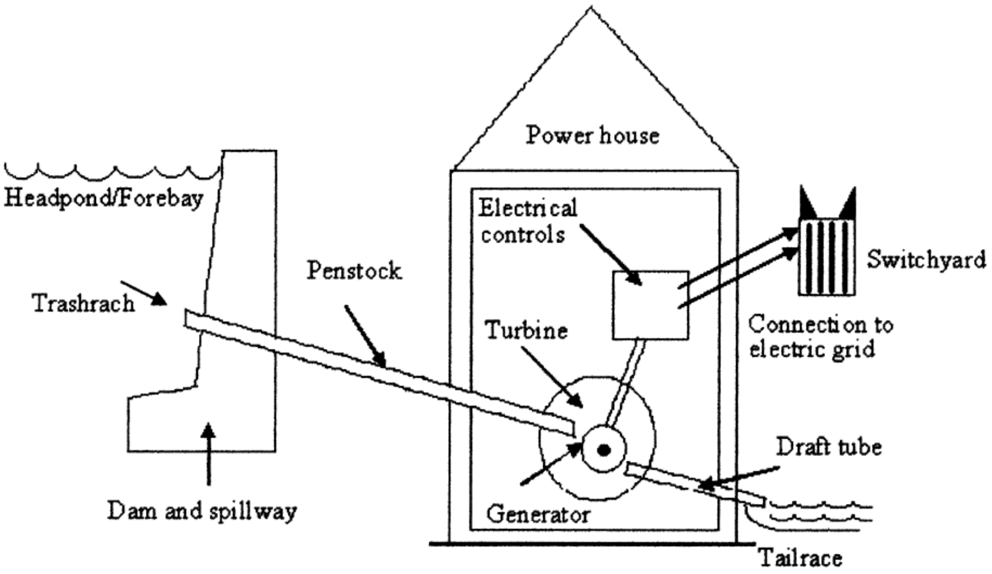


Figure 2.5 Small hydropower system (Balat, 2006)

In small and large scale hydropower production, the efficiency is at the highest levels increasing up to 90 %. The energy production rate is quite predictable and the cheapest energy is provided.

2.5.3 Tidal energy conversion

The gravitational attraction between earth, moon and the sun trigger the water level change on the shore of sea and oceans. These vertical movements of water are called as *tide* (Toossi, 2008). The movement of water relative to the shore is classified as *flood* and *ebb*. The direction of flood tide is toward to the shore and ebb is toward to the ocean. Tidal movements can be sorted according to the frequencies as diurnal, semidiurnal and mixed tides. Diurnal tides observed one times a day with 12 hours duration between high flood and ebb. The period of semidiurnal tides is 6 hours. The categorization of tidal movements along the shores of the earth is given in Figure 2.6.

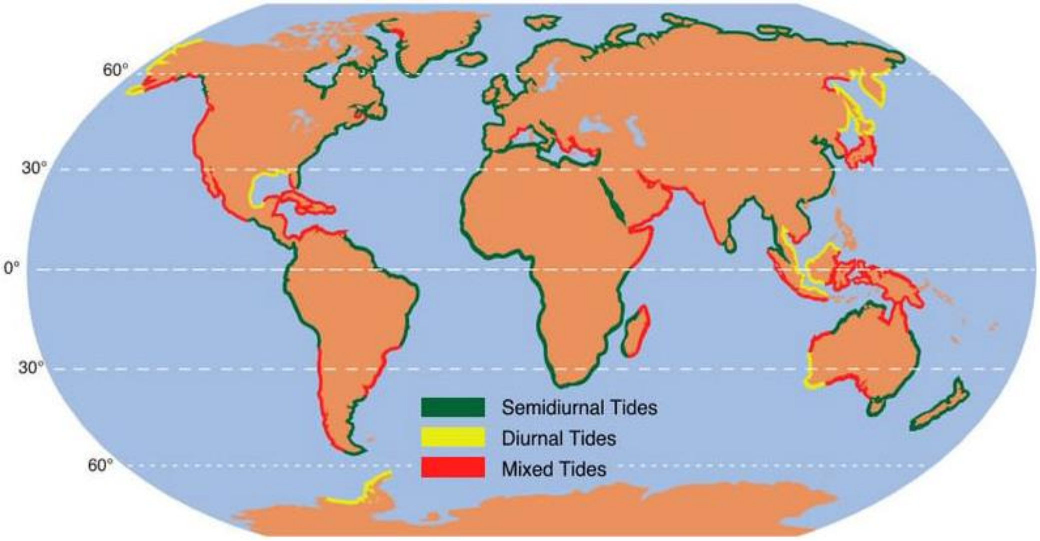


Figure 2.6 Categorization of tides along the coasts of the earth (Kresala, 2013)

Basically, there are two different energy conversion methods for tidal movements. The *tidal barrages* are the traditional approach for energy conversion. The coming tide is accumulated in a reservoir and released when tide moves along the surface. The water level difference of stored water results a potential energy. This energy is converted to the kinetic energy using turbines with the same principle as dams. The most famous tidal barrage is La Rance, in France Figure 2.7.

Figure 2.7 La Rance tidal barrage in France

The list of world’s commercial tidal power plants is given in Table 2.3.

Table 2.3 World's commercial tidal power plants (Gorlov, 2001)

Country	Site	Installed power (MW)	Basin area (km^2)	Mean tide (m)
France	La Rance	240	22	8.55
Russia	KislayaGuba	0.4	1.1	2.3
Canada	Annapolis	18	15	6.4
China	Jiangxia	3.9	1.4	5.08

The tidal energy can be harnessed using in-stream energy converters with no head such as hydrokinetic turbines. The vertical movement of tides produces horizontal currents on the water surface. These currents are exploited via suitable turbines. More information and working principles of hydrokinetic turbines are given in Chapter 3.

2.5.4 Wave energy conversion

Waves are the forward movement of the water at the surface of the ocean or the seas due to the oscillating water particles with frictional drag of wind over the surface of water (Briney, 2013). Wave energy conversion is a very complex stochastic process. The hydrokinetic turbines working with different methodology is used to convert wave energy. The theory of wave energy harnessing is different from that of in-stream hydrokinetic turbines. Wave energy conversion devices can be classified as, oscillating water columns, overtopping devices and wave activated bodies. Figure 2.8.represents the Pelamis device employed for wave energy conversion.

Figure 2.8 Pelamis device for wave energy conversion (http://energy.korea.com/archives/11405)

2.5.5 In stream hydrokinetic energy

In-stream hydrokinetic energy converters employed to harness the kinetic energy of water currents without constructing a reservoir. Extensive information about the hydrokinetic turbines is given in Chapter 3.

CHAPTER 3

HYDROKINETIC ENERGY

3.1 Introduction

Environmental affects like sedimentation transportation, social effects such as inundation, migration, historical effects also, damaging fish habitat, inability to build traditional HEPPs in off-grid regions and other concerns(Baxter, 1977) guide scientists to find more environment friendly power production methods. The main principles of hydrokinetic turbine are employing no reservoir and running with the natural velocity of any stream. The turbines are installed directly inside the water and use the kinetic energy of flowing water. Hydrokinetic turbines generally have compact structure that all systems such as turbine, generator, gearbox, duct, etc. are moving together. These devices can be used in tidal sites, river flows, marine currents and other areas where the water flows with a suitable velocity. It is a new type of technology that came up maximum 2 decades before. The hydrokinetic energy technology aims minimum impact to the original water ecosystem.

There are many new companies who work in the alternative energy sector with particular focus on hydrokinetic power generation. To offer a unique product, companies have identified specific design features that improve efficiencies and/or lower unit costs. These unique designs have been patented and offer specific turbines for specific applications. Kinetic turbine technology is not just about the turbine itself, but about a host of issues including simplicity in turbine deployment and retrieval, safety while handling the turbine, and the ability to reduce cost in spite of various issues: ice, logs, boating traffic, etc.(Shamaz,2009).

3.2 Theory

The technology is very similar to that of wind turbines and used the main principles of wind turbine methodology such as momentum theorem, Betz limit and Blade element method. Hydrokinetic turbines are made up from hydrofoils which produce a pressure difference between upper and lower surfaces and reveals drag and lift forces. The maximum amount of lift force and minimum drag force is desired for maximum performance. The optimum velocity for hydrokinetic turbines is between 1 and 3 m/s. Each turbine type has a characteristic power curve which shows the performance and minimum, maximum velocities to produce power. Cut-in speed is the minimum speed of a hydrokinetic turbine. The turbine idles below the cut-in speed. The rated velocity is the maximum speed that the turbine can produce power. The rated power is the power corresponding to the rated velocity. The power amount of turbine with different speeds can be calculated using the mathematical function of the curve between cut-in and rated speeds. A sample power curve is given in Figure 3.1.

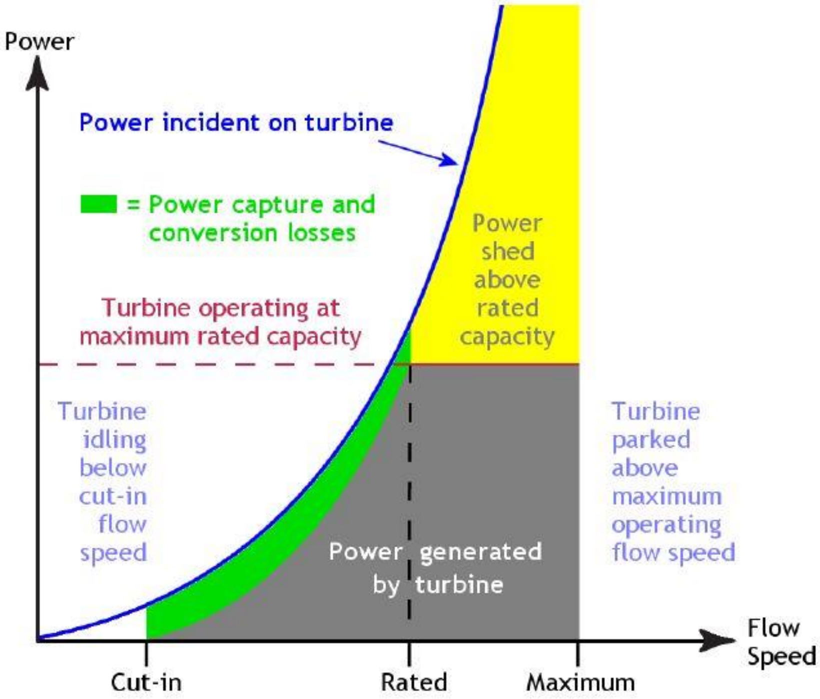


Figure 3.1 A typical power curve for hydrokinetic turbines (EPRI, 2011)

The power output of rotating current energy conversion systems is given below;

$$P = \frac{1}{2} \rho A V^3 C_p \tag{3.1}$$

Where, *P* is the overall power output (watts), ρ is the density of water, A is the cross sectional area of turbine blades (m^2), V is the stream velocity in m/s and C_p is the overall efficiency of the turbine.

As it seen from the Equation (3.1), the power is the proportional with the cube of velocity. Therefore the maximum velocity should be maintained for maximum power. The density of the fluid for hydrokinetic systems is about 1000 times than that of wind turbines (for air). Therefore a water turbine can produce much more power than equally scaled wind turbine.

The theoretical maximum power coefficient of an ideal propeller type turbine is 0.593 which is known as Betz limit (Hansen M., 2008) and the practical power coefficient is between 0.3 and 0.4 for hydrokinetic turbines with low mechanical losses.(Bahaj, A.S and Mayers, L.E., 2003)

3.3 Classification of hydrokinetic turbines

(Muratoglu, 2011) made a broad assessment of hydrokinetic turbines and classified. The hydrokinetic turbines can be classified propeller systems and non-propeller systems. Propeller systems are also called in-stream hydrokinetic turbines that the blades rotate around a horizontal or vertical axis. The tidal, river and marine current applications are inside this group. The horizontal axis turbines are separated into horizontal and vertical axis turbines, helical turbines and ducted turbines.

The non-propeller systems employ a number of moving mechanisms that mainly converts the irregular motion to the regular. Wave energy converters are among the non-propeller systems. The non-propeller systems are out of the purpose of this thesis. The classification of hydrokinetic energy converters is given in Figure 3.2.

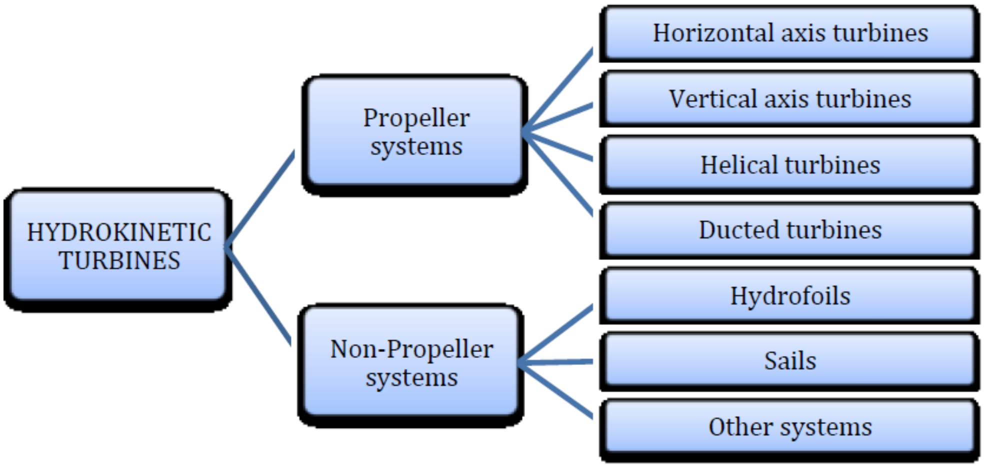


Figure 3.2 Classification of hydrokinetic turbines (Muratoglu, 2011)

3.3.1 Horizontal axis turbines

In horizontal axis hydrokinetic turbines, the rotation axis is parallel to the flow direction. Most of the hydrokinetic and wind turbines are horizontal axis. Seagen, Verdant power, Tidal Stream, Hammerfest Strom, RiverStar, TidalStar and OceanStar turbines are among the horizontal axis hydrokinetic turbines.

Figure 3.3 The Seagen turbine (Lago at. al. 2010)

3.3.2 Vertical axis turbines

The rotational axis of the turbine rotor is perpendicular to the water surface. The firstly designed vertical axis hydrokinetic turbine is Darrieus turbine. The blades are mounted on a vertical shaft mainly works with the darg force of water. The biggest disadvantage of vertical axis turbines is the vibration.

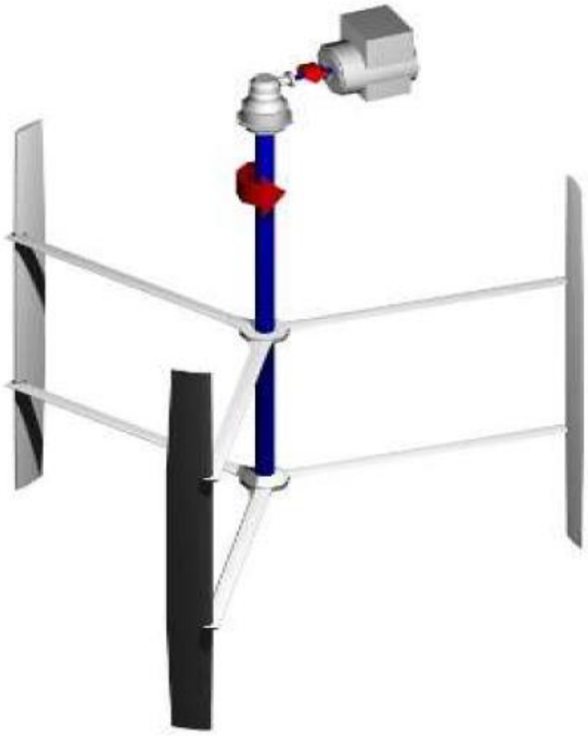


Figure 3.4 A vertical axis hydrokinetic turbine (Ponte de archimede; Muratoglu, 2011)

3.3.3 Helical turbines

Helical turbines are actually, vertical axis turbines with curved blades. The turbine has a helical geometry. The first helical turbine was developed by Alexander Gorlov. It is also called Gorlov turbine. The biggest advantage of this type of turbine is to catch the water flow in every direction and turning at a specific direction for each flow case. On the other hand, high efficiency and practical installation of an array of helical turbine are other advantages.

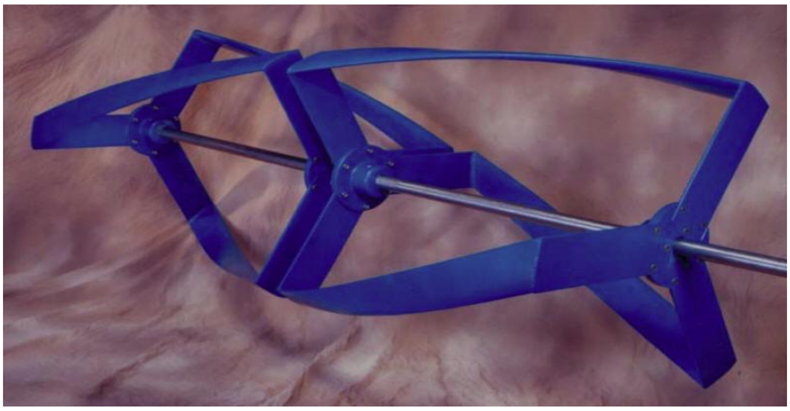


Figure 3.5 Gorlov helical turbine (http://www.builditsolar.com/)

3.3.4 Ducted turbines

The augmentation increases the performance of a turbine by increasing the velocity a decreasing the pressure (Power is proportional with the cube of velocity). High range of efficiency can be obtained in ducted or augmented turbines. The so called Betz limit is not valid for ducted turbines because one-dimensional momentum theory no further usable with the destruction the free flow.

Figure 3.6 The UEK (Underwater Electric Kite) turbine (lucidenergy.com)

3.4 Environmental impacts

The hydrokinetic power has less impact on the environment comparing with other hydroelectric devices, because it does not require building dams. The impacts will vary with the sites and technologies. But still, there are some doubts about the environmental impacts of hydrokinetic turbines. Protecting the fish and other marine life is the main question in construction any hydroelectric project. Sediment transformation, harmful changes in water structure, site impact during installation, conflicts with other uses of the water body, and intrusive visual appearance are among other uncertainties of hydrokinetic energy. In 2009, the U.S. Department of Energy (DOE) wrote a complete report on environmental impacts of hydrokinetic power generation. The report stated that there is no indication that hydrokinetic technologies will cause important environmental impacts on aquatic environments, fish and fish homes, ecological relationships, and other marine and freshwater resources (U.S. DOE, 2009).

Some environmental factors of hydrokinetic power is listed below (Cada and Meyer. 2005; Cada at el, 2007);

- During rotation of the blades, shear stress and turbulence will take place, and it may cause damage of water organisms.

- Turbine installation disturbs the sediment regime of the river, increase the turbidity temporarily. Movement of the rotors and unsecured mooring cables can damage the natural water environment. The extraction of kinetic energy affects the ability of water body to transport sediment; indicating a possible impact on transport and suspension of bed load. Also, sediment size and composition are very important for the organisms that live there, as well a show uniform that sediment is. Some organisms need sediments to resume their life cycle.

- Natural streams provide living place for fish species and all types of water plants and organism. Some type of organisms need silt habitat, but some of them need cobble stream bed to live. The life cycle of the organisms is depend on the other uses of natural streams which might be affected by hydrokinetic power include:
- Commercial navigation.
- Recreational navigation.
- Swimming, skiing, etc.
- Commercial and recreational fishing.
- Removal and discharge of municipal, industrial, and agricultural.
- Electromagnetics field effects.
- Noisy construction.
- Toxic paints.

CHAPTER 4

COMPUTATIONAL FLUID DYNAMICS (CFD)

4.1 What is CFD?

The mechanism of analyzing and design of flow over marine and wind turbines, heat transfer and associated phenomena are called Computational fluid dynamics (CFD)

Industrial and non-industrial applications of CFD are given below;

- Analysis of flow in ocean, river, streams and design of tidal and current turbines.
- Studying the aerodynamics of airplane and wind turbines.
- Medical studies, like flow of blood inside nervous system.
- Electrodynamics and electronic uses e.g. cooling tools like micro circuits.
- Inner air circulation, wind load and heating system of high and multistory buildings.
- Hydrological purpose such as meteorology, precipitation, infiltration and runoff.
- Environmental engineering like studying and analysis of pollutant mater.
- Chemical engineering, it can be used for mixing chemical material and to separate them.
- Mechanical purpose, burning system in gas turbines and engines.

- Ship hydrodynamics

Beginning from the second half of the 20th century, the CFD is used in aerodynamic design, research and development and manufacturing processed of aircraft and jet engines. Then, CFD methodology has been used to design internal burning system engines, combustion chambers of gas turbines and furnaces. Additionally, it is used to predict the drag and lift forces of aircrafts and aerodynamics of mobile cars. Progressively it is used in the design of industrial products and processes as an essential component.

The idea of developments in the CFD field is to provide a capability comparable with other CAE (computer-aided engineering) tools such as stress analysis codes. CFD is a recent upsurge of interest and has entered into the wider industrial community since 1990s because of availability of low cost high-performance computing hardware and the introduction of user friendly interfaces.

There are some incomparable benefits of CFD over experiment-based accesses to fluid systems design, such as;

- Low cost and no need long time.
- Capable to study a huge system that is difficult and impossible with experiment.
- There is no risk or danger during the study.

CFD codes can provide very large volumes of results at effectively small expenses to perform parametric studies. (Versteeg and Malalasekra, 2007).

4.2 How does a CFD code works?

In order to prepare easy access to their solving power, all commercial CFD packages contain advanced user interfaces to input problem parameters and to examine the results CFD codes. The codes include three main elements: (i) pre-processor, (ii) solver and (iii) post-processor.

4.2.1 Pre-processor

The input of a flow problem to a CFD program by means of an operator-friendly interface and the succeeding of conversion of this input into a suitable form for user is called pre-processor. The user should do following things at the pre-processing stage;

- The geometry must be defined in this section as a region of involvement.
- Mesh should be generated which is the division of the domain into a number of smaller and non-overlapping elements.
- The physical and chemical phenomena should be selected that is needed to model.
- All fluid properties should be defined.
- Definition of boundary condition should be made and separated the domain by name.

The accuracy of the CFD solution is depending on the number of nodes and elements in the grid. Therefore maximizing the number of cells provides better conversion and accuracy. The solution to a flow problem (turbulence, temperature, pressure, velocity, etc.) is defined at inside each cell. The fineness of grid is important because both of accuracy of solution and its cost is related with required computer hardware. The calculation time for a simulation increases with the fineness of the grid, decreases with the power of the hardware. The desirable mesh generally is non-uniform which means it is not same in all points. The regions of the mesh which are sensitive to the variation should be fine meshed, larger meshes are sufficient in other regions. Products are in progress to develop CFD codes with a character meshing capability. Finally such programs automatically separate the areas that large variation is occur and refine it to less variated areas

More than 50% time of a CFD project is required to define the domain of the geometry and to mesh it. All the major codes now contain their private CAD-style interface and/or facilities to import data from proprietary surface modelers and mesh generator such as PATRAN and I-DEAS in order to increase productivity of CFD personnel. Up-to-date user access to libraries of material properties for common fluids and a facility to invoke special physical and chemical process models (e.g. turbulence models, radiative heat transfer, combustion models) alongside the main fluid flow equations are provided by pre-processors(Versteeg and Malalasekra, 2007).

4.2.2 Solver

Finite difference, finite element and spectral methods are the three different methods of numerical solution techniques. The finite volume method is used for most of the CFD codes.

Numerical algorithms contain the following steps:

- Convert or approximate the unknown flow variables to a simple function.

- Substitution the function of unknown flow variables to the mathematical. manipulation in order to discrete.
- The algebraic equations by an iterative method are solved.

The first step the control volume integration, categorizes the finite volume method from all other CFD techniques. The exact conservation of relevant properties for each finite size cell is shows by resulting statements. The main responsibility of the finite volume method is the close relationship between the numerical algorithm and underlying physical conservation forms and makes its concepts much simpler to understand by users than the finite element and spectral methods.

The key transport phenomena, convection (transport due to fluid flow) and distribution (transport due to variation of variables from point to point) as well as for the source terms (associated with creation or distribution of variables) and rate of change with respect to time are treated by CFD codes because it include adequate separation techniques. The underlying physical phenomena are complex and non-linear so an iterative solution approach is required. To ensure an acceptable linkage between pressure and velocity, process of solution is done by the TDMA (tri-diagonal matrix algorithm) line-by-line solver of the algebraic equations and SIMPLE algorithm because it has the most popular calculation process (Versteeg and Malalasekra, 2007).

4.2.3 Post-processor

Recently, the engineering workstations are well developed in all branches especially in mechanics and hydraulics. Therefore lots of development is taken place in post-processor field like pre-processor many of which have outstanding graphics abilities.

The leading CFD package is now armed with all around data visualization tools. The post-processor contain:

- Display the geometry and mesh generation of whole domain.
- Display and show vectors.
- Show the surfaces 2D and 3D.
- View sections and surfaces and domain streams.
- Show rotating domain, translation, scaling.
- Separate the regions of domain by color.

More recently these facilities may also include animation for dynamic results with graphics. (Versteeg and Malalasekra, 2007). The flow chart which shows the methodology of CFD work is given in Figure 4.1.

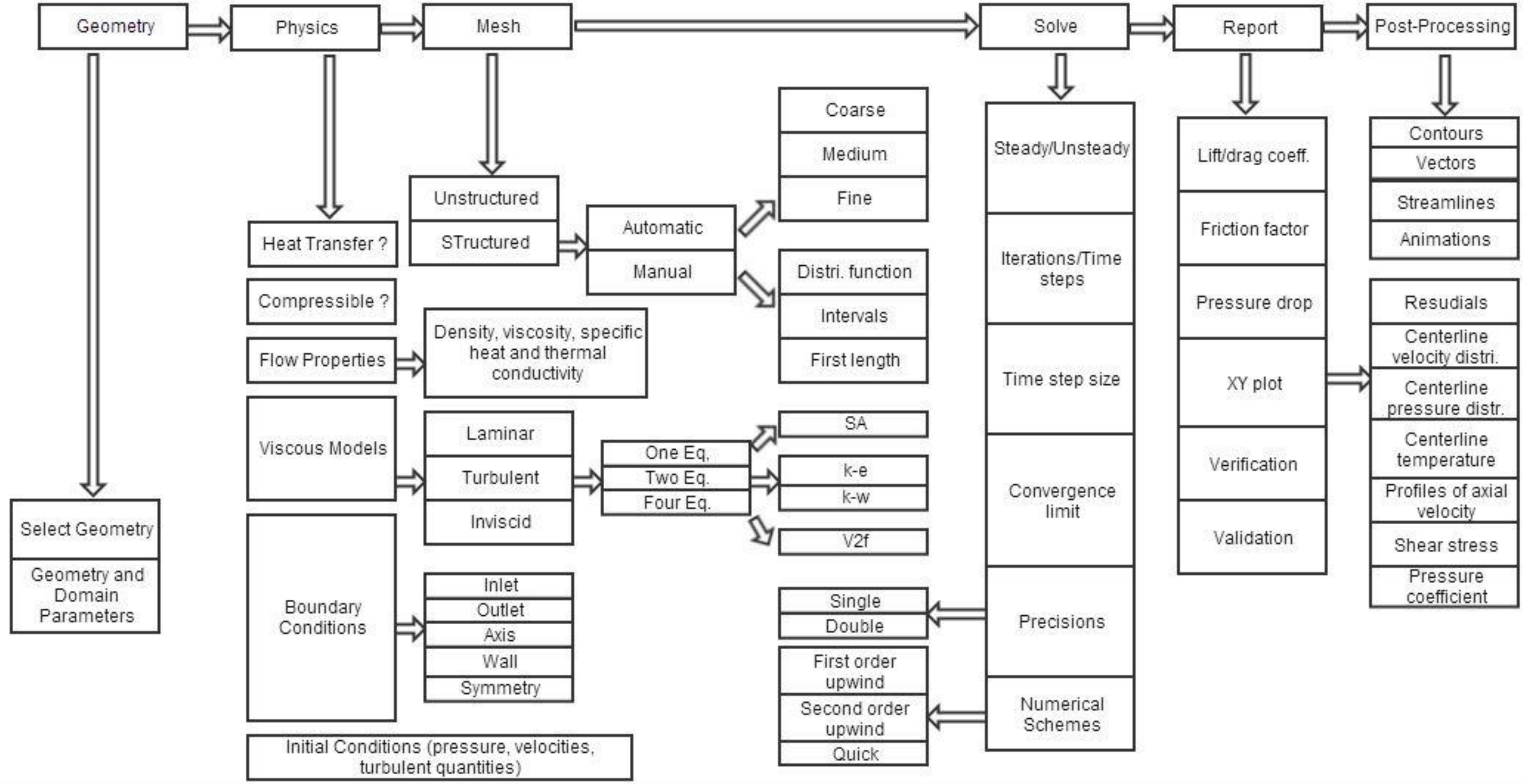


Figure 4.1 The flow chart which shows the methodology of CFD work

CHAPTER 5

TURBULENCE

5.1 Introduction

Turbulence is a flow characteristic in which the viscous forces are small compared to the inertial forces. In turbulent flow the viscous force has no ability to damp out small perturbations in boundary and initial condition. Instead, these perturbations are amplified causing rapid variation in pressure and velocity in space and time. The criterion to determine the dominating force (inertial or viscous) is the Reynolds number

$$R_e = \frac{\rho V L}{\mu} \tag{5.1}$$

Where ρ is the density of the fluid, V is the velocity, μ is the viscosity and L is the length.

For open channel there is no clearly limit at which Re the turbulence developed in flow but it usually taken more than 2000. (Sam, 2010).

Figure 5.1 A view of turbulence

5.2 Turbulence Models in CFD

The turbulence models are the computational process or models that are used to simplify the solution of governing equations in an engineering application especially for mechanical processes. It is needed to represent scales of the flow that are not resolved.

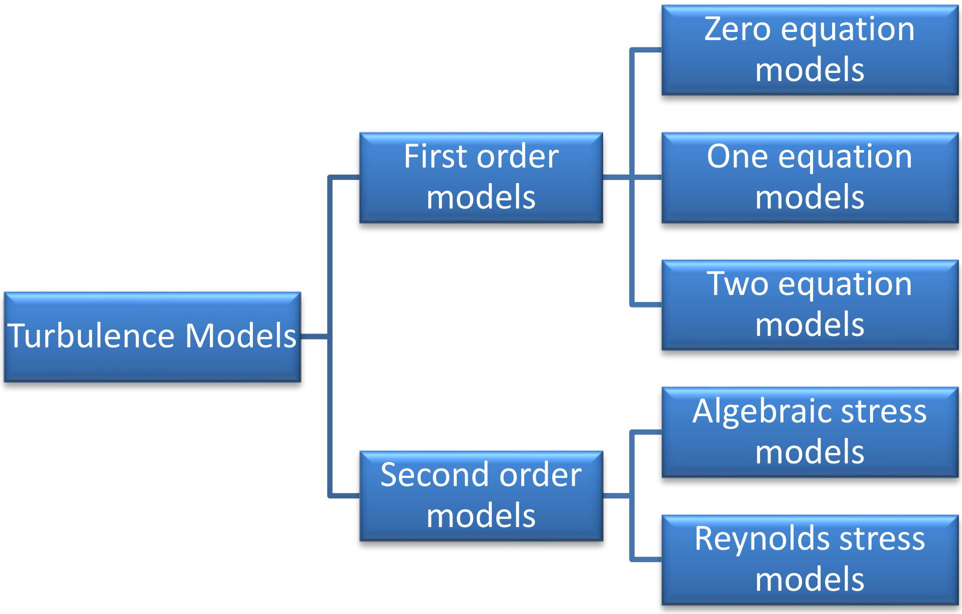


Figure 5.2 Classification Turbulence Models (Miller, 2010)

5.2.1 Reynolds Average Navier-Stokes Equations (RANS)

Turbulent flows contain velocity fields that are chaotically fluctuating over a large range of temporal and spatial scales. While direct numerical simulation is possible, capturing these fluctuations requires a highly resolved flow field and computationally intensive work. The Reynolds Averaging approach is a very popular alternative in CFD for modeling turbulent flow fields. In Reynolds Averaging the variables in the Navier-Stokes equations are decomposed into their mean and fluctuating components, known as Reynolds Decomposition (Pope, 2008).

According to Newton’s second law of motion for fluid flow, the continuity equation can be written as (Taylor, 2012);

$$\Sigma \bar{F}_B + \Sigma \bar{F}_S = m\bar{a} \tag{5.2}$$

From continuity and the incompressible Navier-Stokes, equations can be written in vector form as:

$$\nabla . \bar{V} = 0 \tag{5.3}$$

$$\rho \left\{ \frac{\partial \bar{V}}{\partial t} + u \frac{\partial \bar{V}}{\partial x} + v \frac{\partial \bar{V}}{\partial y} + w \frac{\partial \bar{V}}{\partial z} \right\} = \rho \bar{B} - \nabla P + \mu \left\{ \frac{\partial^2 \bar{V}}{\partial x^2} + \frac{\partial^2 \bar{V}}{\partial y^2} + \frac{\partial^2 \bar{V}}{\partial z^2} \right\} \tag{5.4}$$

5.2.2 The standard k-epsilon model

The terms k and ε refer to the turbulence kinetic energy which are the variance of fluctuations in velocity. Turbulence eddy dissipation is equal to the viscosity multiplied by the fluctuating vorticity. An exact transport equation for the fluctuating vorticity defined as rate of dissipation of velocity fluctuations (Sarvan Mamaidi, 2009).

5.2.3 The RNG k-ε model

It is employed for incompressible flows. It is similar to the standard *k-e* model, but includes some improvements as follows (Miller, 2010);

- It has an additional term in the ε equation to improve the accuracy.
- The RNG models have enhanced accuracy for swirling flows.
- Instead of the user-specified, constant-value turbulent Prandtl numbers implemented with the standard *k-ε* model.

5.2.4 The realizable k-ε model

Improvements in the realizable *k-ε* are;

- Shares the same turbulent kinetic energy equation as the standard *k-ε* model.
- The equation for ε is improved.
- Variable $C\mu$ is used, instead of constant.
- Improved performance for flows involving planer and around jets. The model predicts the round jet spreading correctly.

5.2.5 The standard k-omega Model

The terms *k* and *omega* stand for two transport equations turbulent kinetic energy and turbulent frequency, respectively. The values are computed based on the concept of eddy-viscosity (Sarvan Mamaidi, 2009).

5.2.6 Large Eddy Simulation (LES)

Large-eddy simulations (LES) differ from the other fluid flow computation methods. In this simulation method, the largest eddies are explicitly resolved and the smaller eddies are modeled. Modeling the ocean and atmosphere inspired the first large-eddy simulations, but true large-eddy simulations of the ocean and atmosphere are surprisingly rare. The LES equations for incompressible flow describe the evolution of the large scale eddy in the flow-field (Goldstein, 2004).

CHAPTER 6

SIMULATION OF HYDROKINETIC TURBINES

6.1 Introduction

This chapter contains the procedure of modeling and simulation of the domain from pre-processing to the post-processing stage. Firstly, the assumptions to input data to the software were described. The simple analytical method used to validate the model. Good modeling qualification is important to get the correct selection in all simulations, in order to reduce the complexity of the problem and retaining the important characteristics. Identification and formulation of the flow problem in terms of computational domain, the physical and chemical properties of flow are necessary to be considered. Proper operation is made and suitable results were obtained from the CFD simulations.

6.2 Modeling and Simulation Procedure

In this part the modeling and simulation procedure will described from pre-processing to solving and post-processing stages.

6.2.1 Pre-processing

It is the first and very important part of the study. It contains the preliminary creation and generation of the model. The geometry is identified the program using the Design Modular which is a part of the ANSYS workbench. Then mesh is generated on the pre-described geometry. Furthermore the physical and chemical properties and necessary boundary conditions are defined to the software.

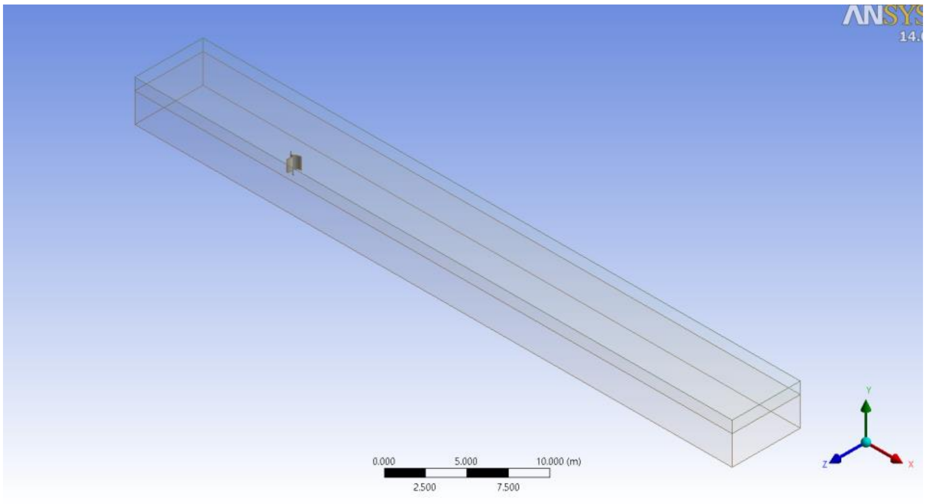


Figure 6.1 The whole domain

6.2.1.1 Creating the model and defining the geometry

The model was created in DESIGN MODELER. There are two steps to create geometry regions. The first step is creating a model which represents an open channel with rectangular cross-section. The top surface of channel represents the free surface, while the bottom represents the channel bed. The second step is creating the hydrokinetic turbine which is a modified (Savonius) vertical axis turbine with three curved rotor blades and circular shaft in the middle. The turbine is taken from original (Savonius) which has two circular blades, as in Figure 6.2.

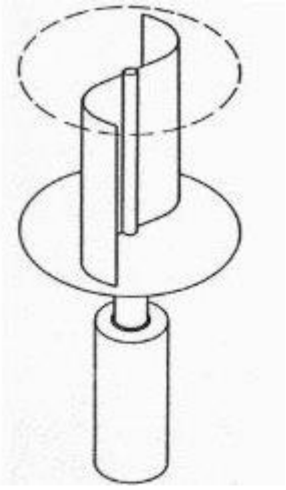


Figure 6.2 The original Savonius turbine.

The hydrokinetic turbine characteristics are based on the information given in previous chapters. Turbine design includes three circular blades, each sweeps 120 circumferential degrees. The 360̊ combined spans of the circular blades reduces the total torque of the turbine to a constant value, improving the life of the turbine as well as the self-starting capabilities. The outer diameter of blades is 1 m, diameter of shaft or axis rotation is 0.1 m with 1.4 m length. The front view, top view of the turbine is given in Figures 6.3, and 6.4. The situation of the turbine inside the open channel is seen in Figure 6.5.The dimensions of the channel and turbine were given in Table 6.1 and 6.2respectively.

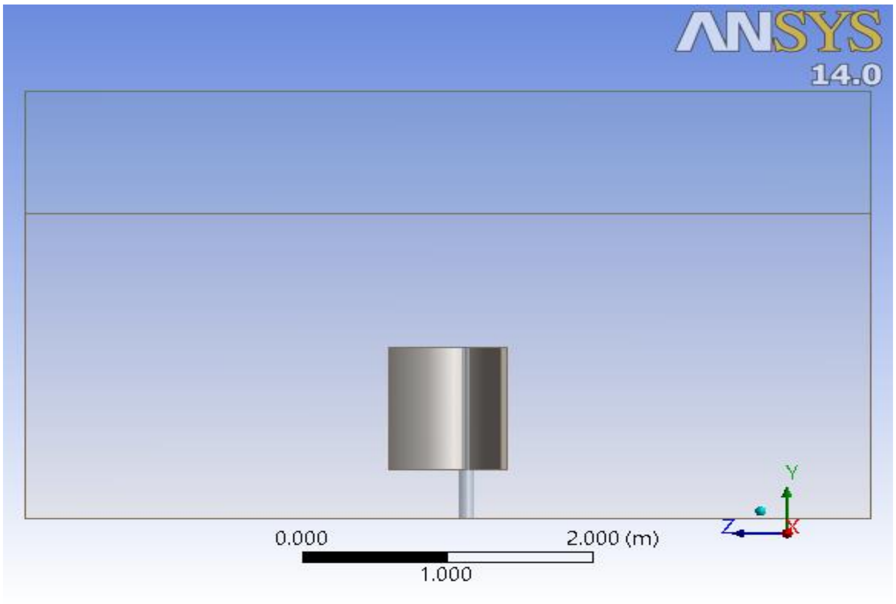


Figure 6.3 Front view of the domain

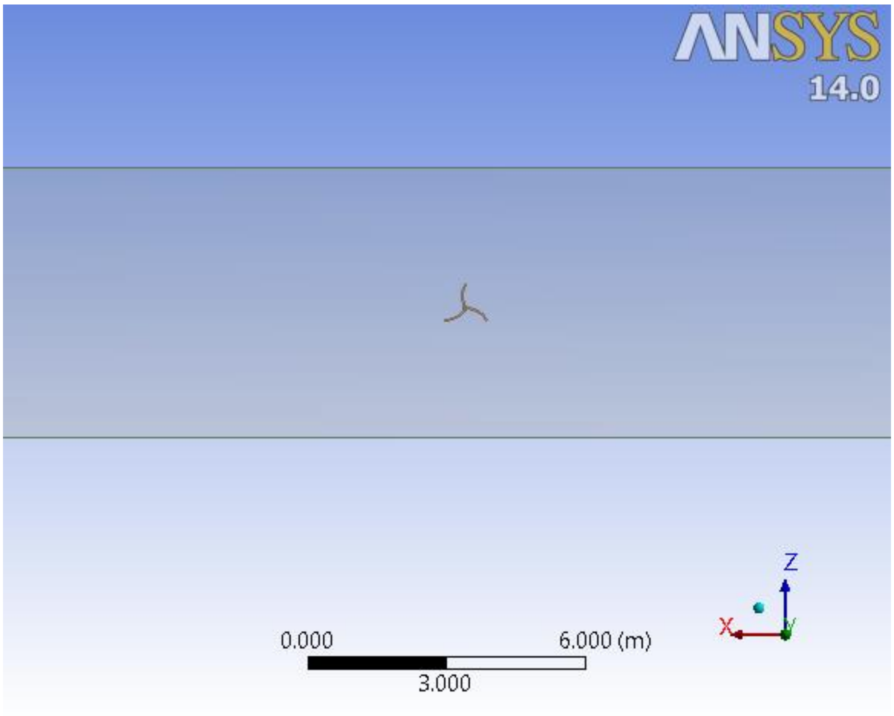


Figure 6.4 Top view of domain

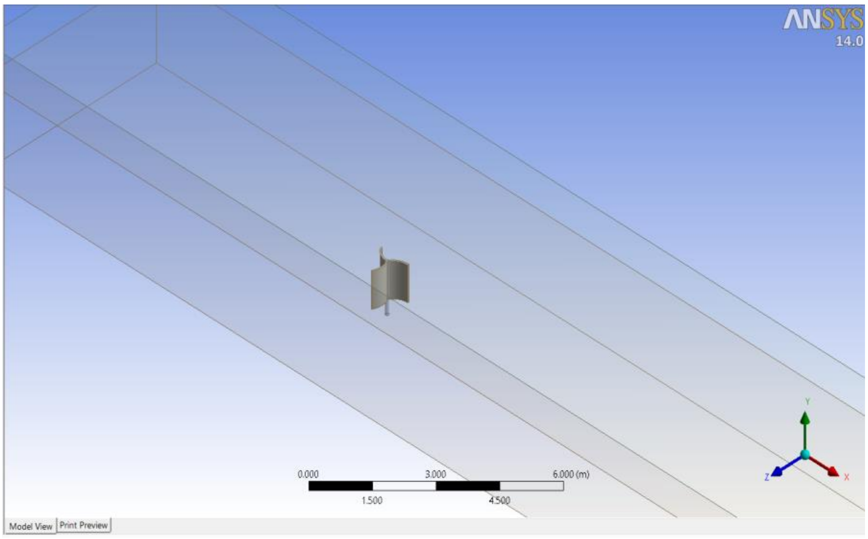


Figure 6.5 The modified Savonius turbine inside open channel

Table 6.1 Channel dimensions

Dimension	**Length(m)**
Length	50
Width	5
Height	3

Table 6.2 Dimensions of the turbine

Description	Dimension
Diameter of turbine	1.0m
Diameter of shaft	0.10m
Number of blades	3
Length of shaft	1.40m
Height of turbine without shaft	1.0m
Shape of blades	Circular

6.2.1.2 Creating the boundary layer and meshing

When the model was created and defined, it should be divided into a number of cells for accuracy and control of the results. This operation is known as meshing. On the other hand, putting boundary layers on all parts of model such as walls, surfaces, inlet, outlet, and turbine in the open channel as in Figure 6.6

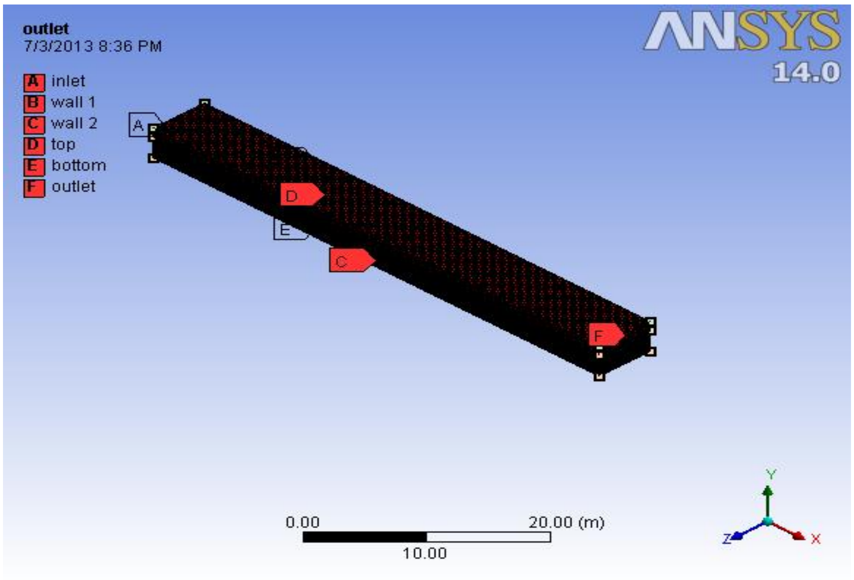


Figure 6.6 The boundary conditions

Provide more accurate results close to the boundary regions. The mesh should be examined and refined especially on the rotor blade surface. It should be highly smoothed in the free surface and open channel bed. In this study, the model was divided into 196213 nodes and 585312 elements. Figure 6.7 show the mesh metric

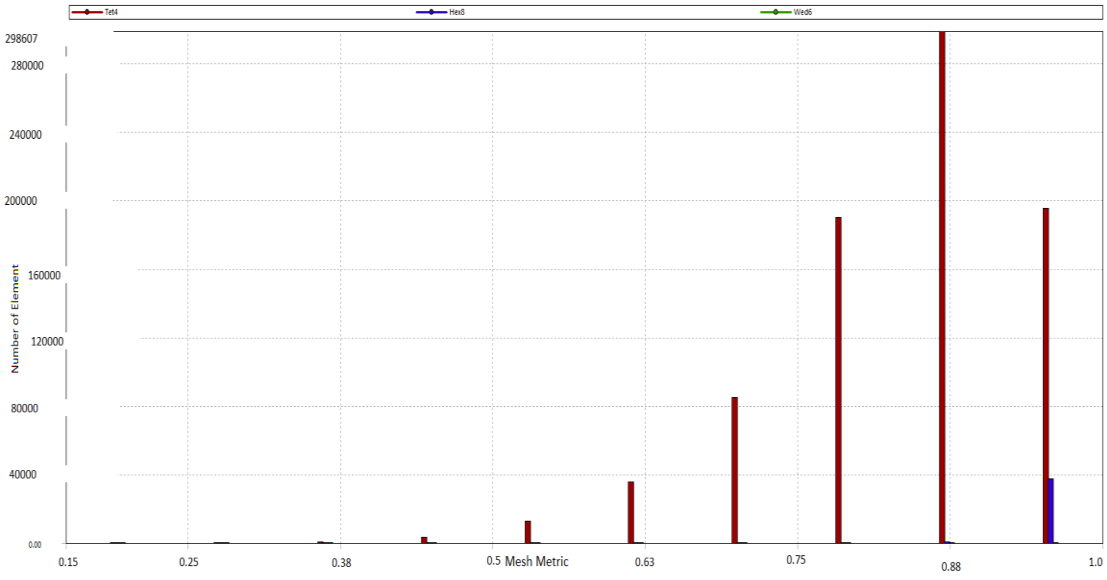


Figure 6.7 Mesh metric

Defining the boundary conditions is the final step of the meshing. Each boundary condition in the model should be defined. In this model, the bed of the open channel is defined as bottom, sides are specified as the walls or symmetries, the water level in the open channel flow is defined as surface, the entrance and exit of the water is defined as velocity inlet and velocity outlet. Finally the turbine is specified as to be a rotary domain. Before entering the SETUP process, the mesh should be closed and updated. The different views of meshing are given in following figures.

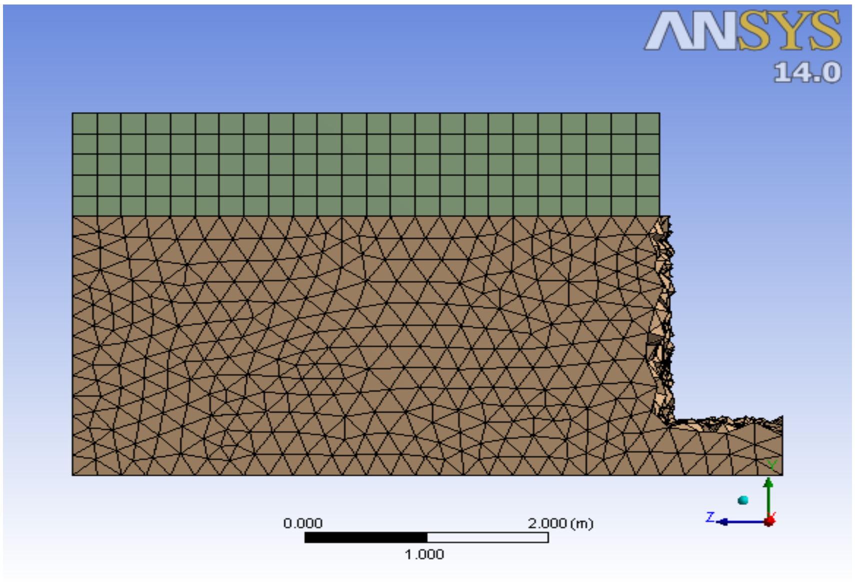


Figure 6.8 The cross-section showing the mesh

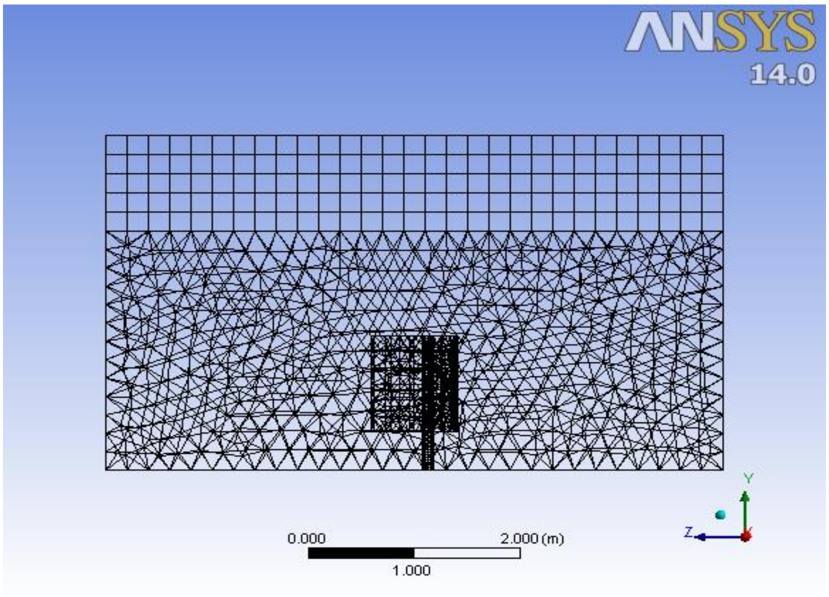


Figure 6.9 The wireframe mesh

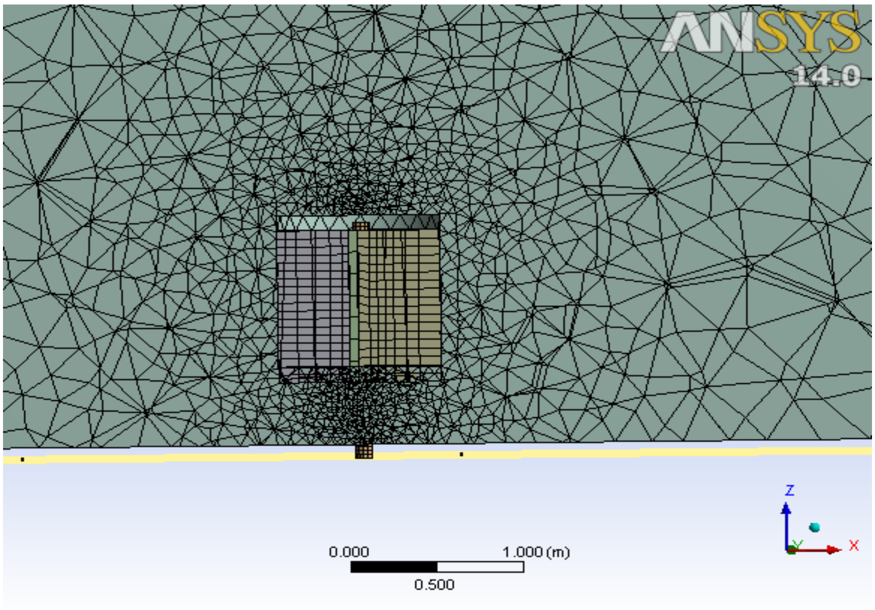


Figure 6.10 Mesh generation near turbine

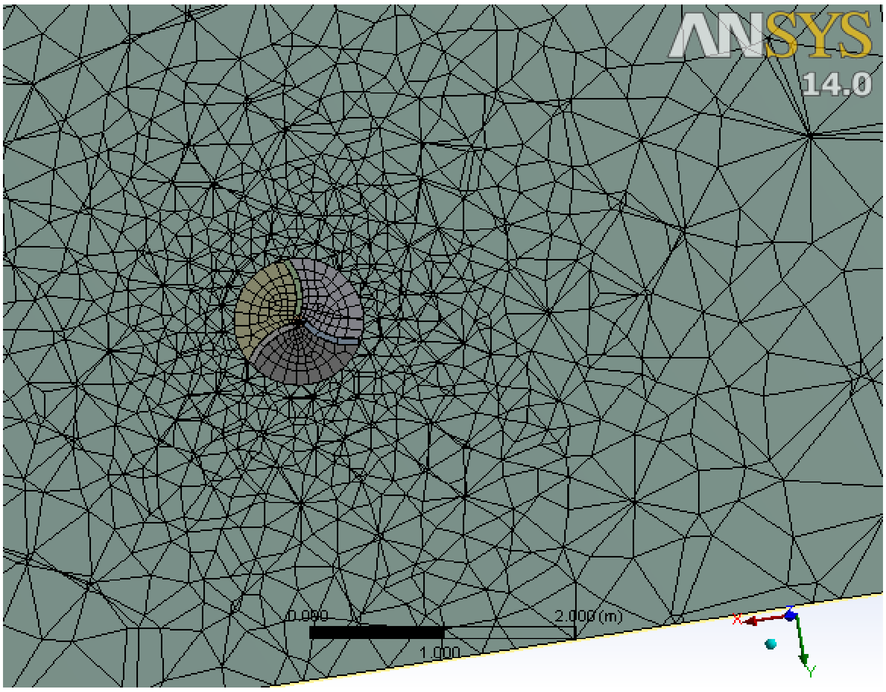


Figure 6.11 Top view of mesh generation

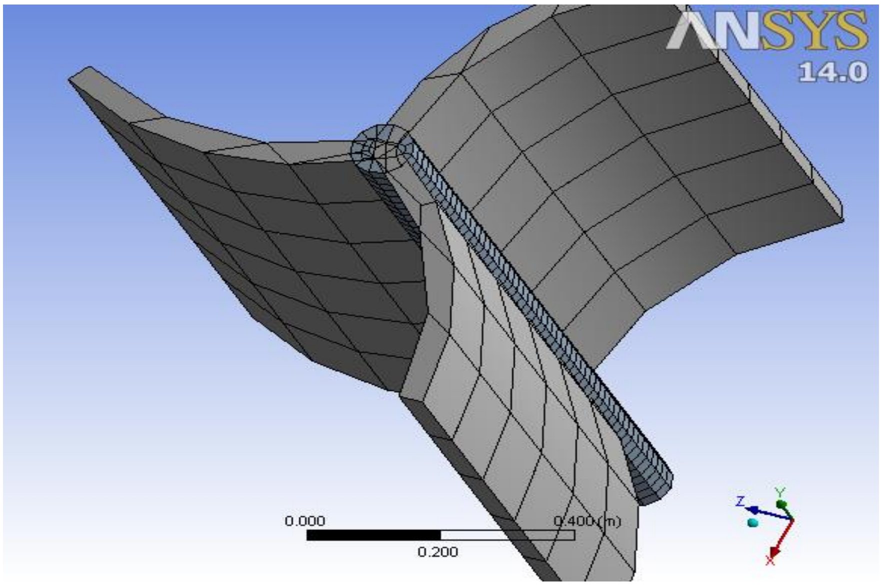


Figure 6.12 The mesh on the turbine

6.2.2 Simulation

The generated mesh with boundary layers is entered to the FLUENT software. Then the quality of the mesh was checked. Finally the physical phenomena and fluid properties were specified. The steps to simulate the model are defined in following parts.

6.2.2.1 The physical procedure

Separate solver and implicit formulation was selected for the 3D model. Then the multiphase model is chosen. Furthermore, k-ε realizable (two equation) viscous model with standard wall functions is picked for the turbulent flow. All these properties are assumed constant over the model.

6.2.2.2 Defining the fluid properties

The fluid is selected as liquid water and necessary changes for viscosity, density, gravitational acceleration, etc. were introduced. The air is defined for the free surface so as to separate two different phases of matter as air and water. The fluid properties were entered to the software which is given in Table 6.3.

Table 6.3 Properties of open channel

Property	Value
Density	998.2 kg/m^3
Viscosity	0.001003 kg/ms
Temperature	20c

6.2.2.3 Specification of boundary conditions

In this stage, cell zone and boundary conditions are defined. The model has two cell zones which are air and water. The boundary layers of pressure outlet and velocity inlet is selected. The fluid and wall and other parts are defined as inlet and outlet. The list of the boundary conditions is given in Table 6.4.

Table 6.4 The boundary conditions

Surface	Boundary type
Inlet	Velocity inlet
Outlet	Pressure outlet
Free surface	Surface
Turbine	Stationary domain

6.2.3 Initialization and solving the model

In this step, the properties of the turbulence were defined and other factors were introduced. The inlet was taken as reference to initialize the solution. The active reference frame was chosen relative to the cell zone. The input velocity in the x-direction was introduced as 3m/s. The number of iterations was set as 200 for good conversion. In this step, it is important to save the results in order to prevent from any trouble with computer. The diagram which shows residuals vs. number of iterations is given in Figure 6.13.

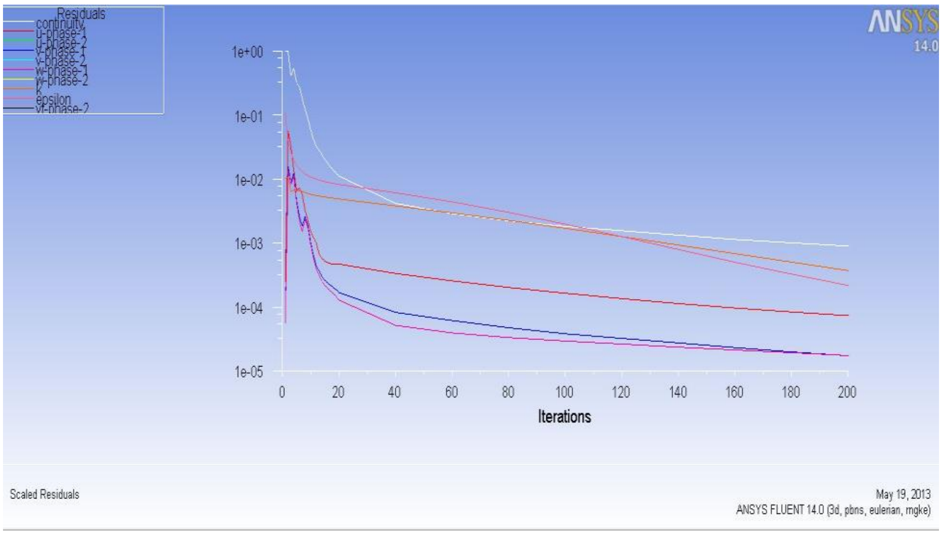


Figure 6.13 The residuals versus number of iterations

6.2.3 Post-processing

In the post-processing stage, the results can be screened and the velocity, pressure and other variables can be plotted. The required representation style such as, vector or contour plot, is surfaces and other visual preferences can be arranged. The post-processing results were given in Chapter 7.

CHAPTER 7

RESULTS AND DISCUSSIONS

This chapter provides the post-processing visualization and simulation results. The velocity and pressure profiles and other physical properties are presented.

7.1 The top-view velocity distributions

The velocity at the beginning is seemed clearly to be constant in the channel. When the flow reaches to the turbine the flow deforms to find the line of least resistance. The deformation takes place at the midway between the turbine and open channel boundaries. The turbine works as an obstruction and runs with minimum velocity, approximately between 0 and 0.5 m/s. The top view velocity distributions of open channel and near the turbine is given in Figures 7.1 and 7.2 respectively.

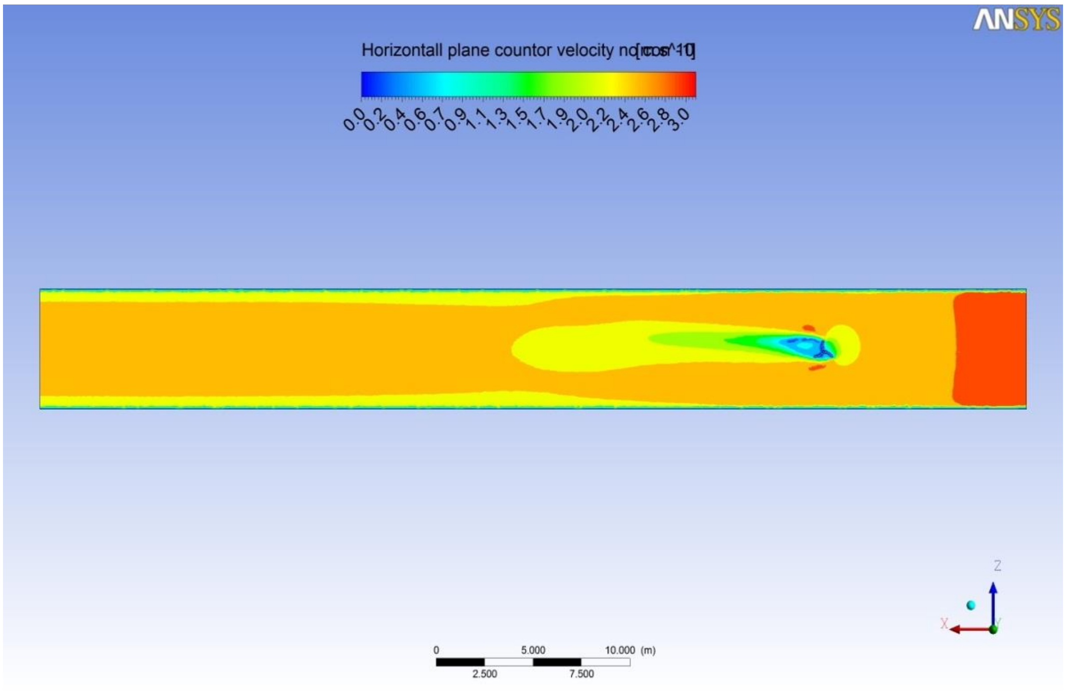


Figure 7.1 The velocity distribution on the open channel

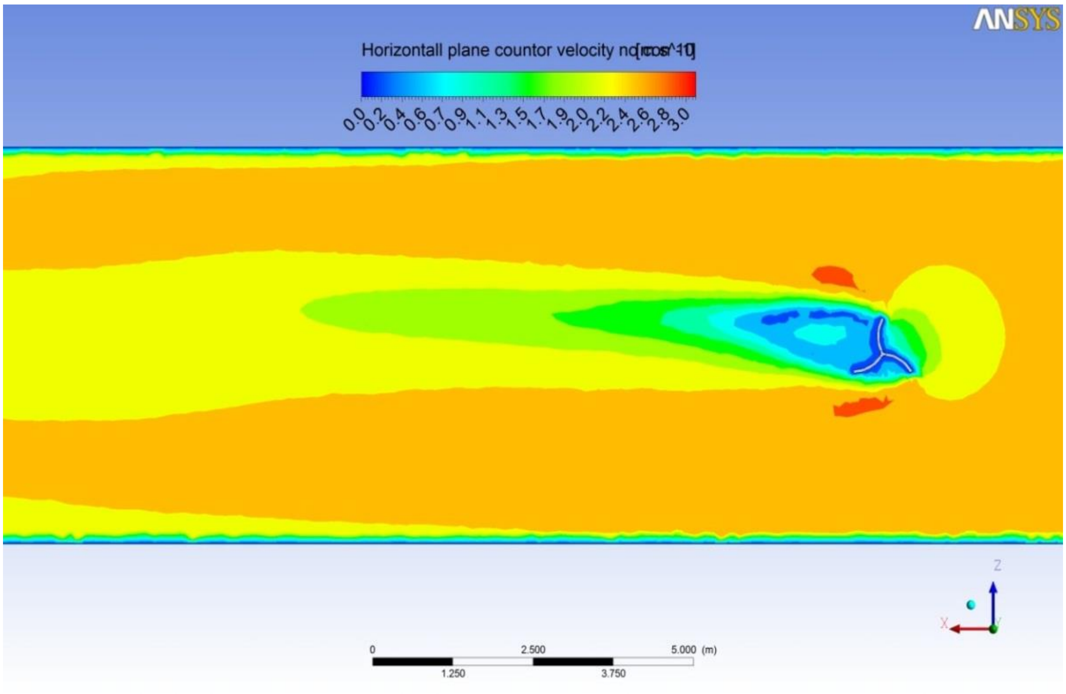


Figure 7.2 The velocity distribution near the turbine

7.2 Vertical velocity distributions

The deformity of the vertical velocity distribution along the length of the open channel has been studied. The velocity is almost constant at the beginning. The biggest decrease in the velocity occurs near the turbine. This is an evidence of energy extraction from the system by the rotating turbine blades. The vertical plan view of velocity contour for the channel and near the turbine is given in Figure 7.3 and 7.4 respectively.

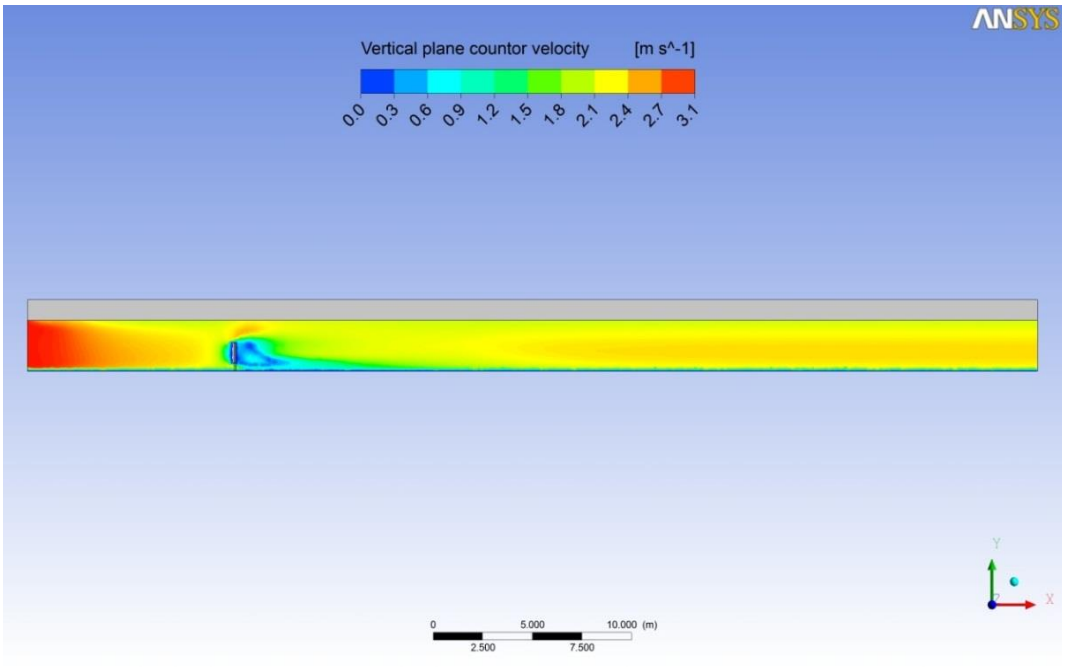


Figure 7.3 Vertical plan view of velocity contours for the channel

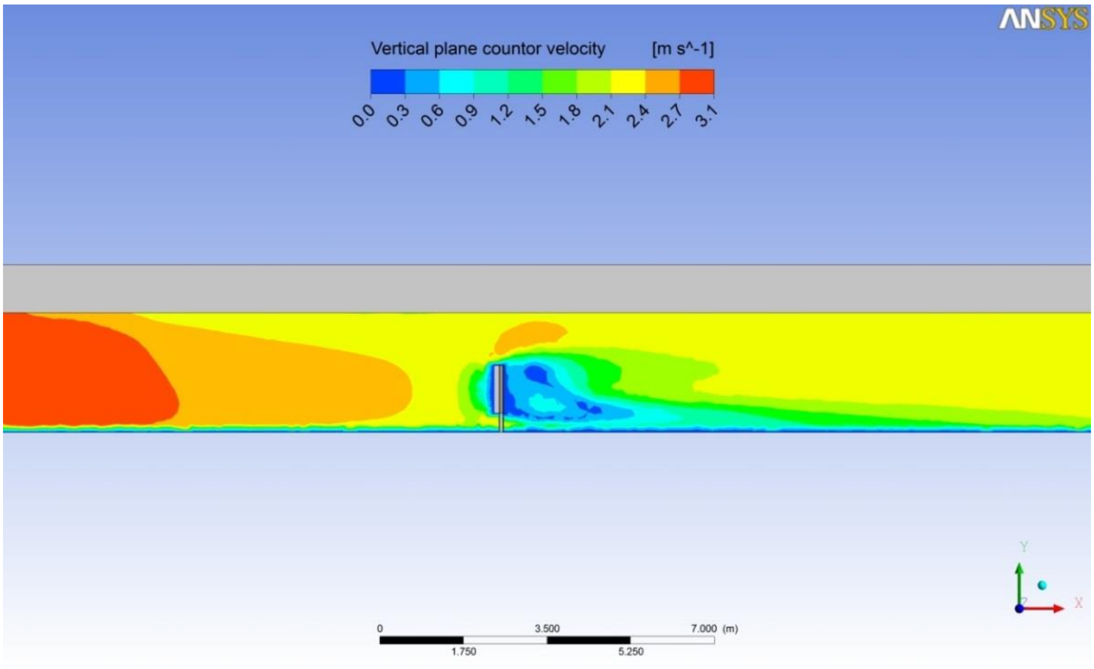


Figure 7.4 Vertical plan view of velocity contours near the turbine

7.3 Velocity streamlines

Streamline is flow path of fluid relative to solid body in which the fluid is moving in smooth line without turbulence. Figure 7.5 and 7.6 shows the horizontal and vertical stream lines along the channel. At the very inlet of the channel there is no turbulence in the flow and the streamlines are straight. When the flow reaches to the turbine the turbulence takes place and flow becomes unsteady due to the resisting forces. A vortex is produced at the turbine wake. Then the flow path becomes steady again.

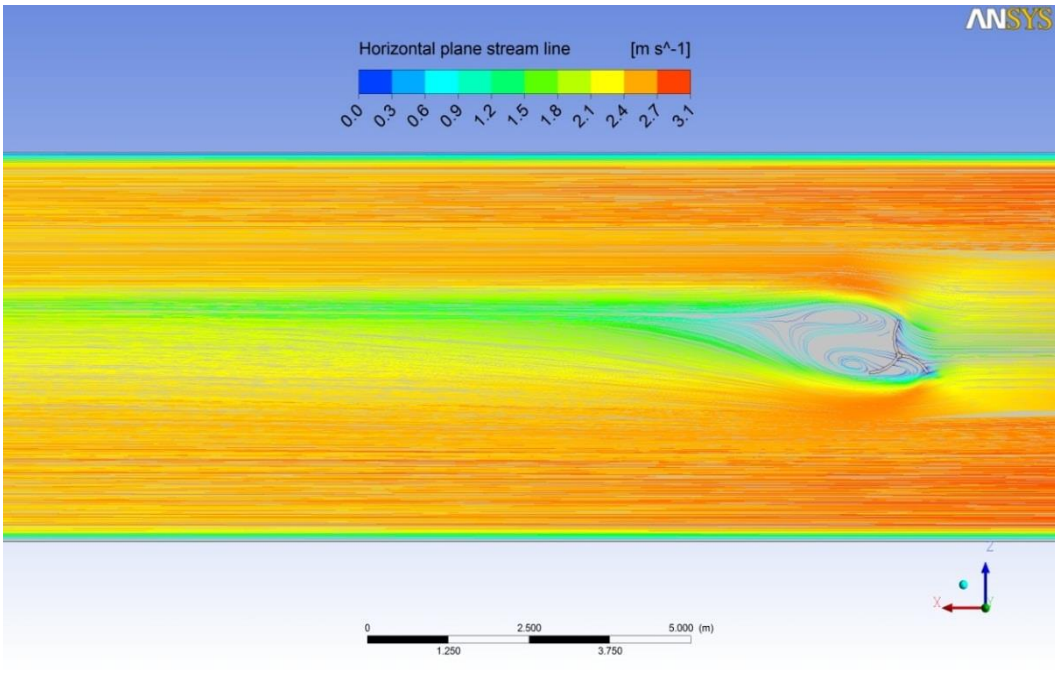


Figure 7.5 Plan view of velocity streamlines inside the channel

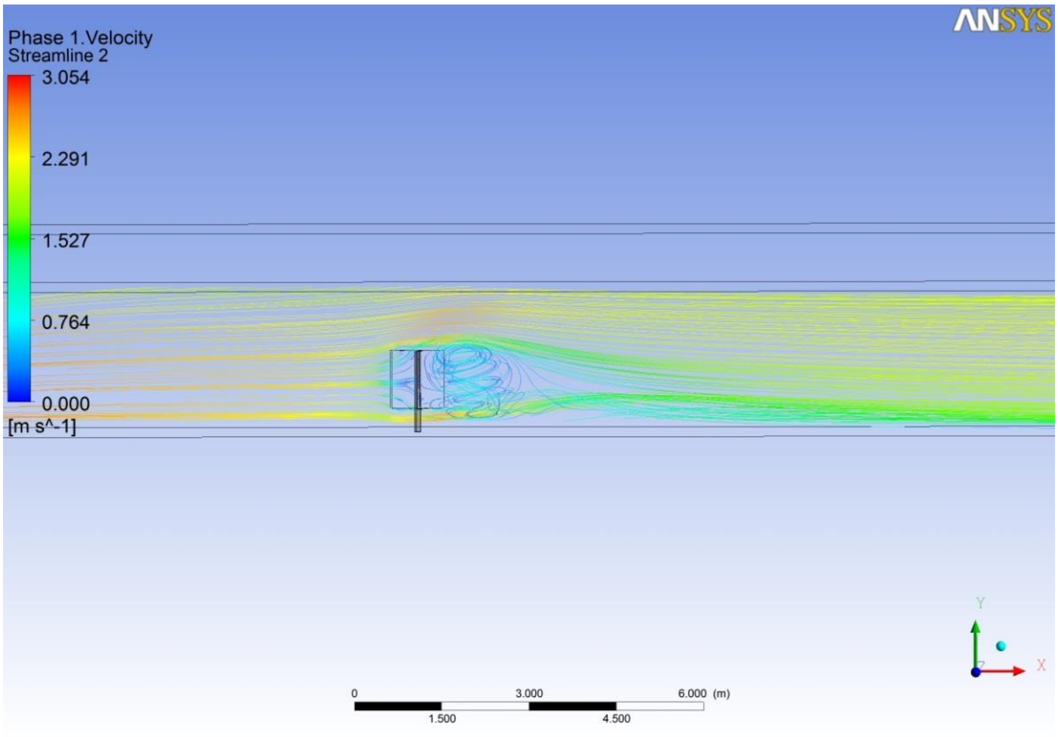


Figure 7.6 Vertical view of velocity streamlines

7.4 Velocity vectors

Figure 7.7, 7.8.and 7.9 shows the velocity vectors inside the fluid domain. The vector view in the fluid at the rotor blade sections due to the rotation is seen.

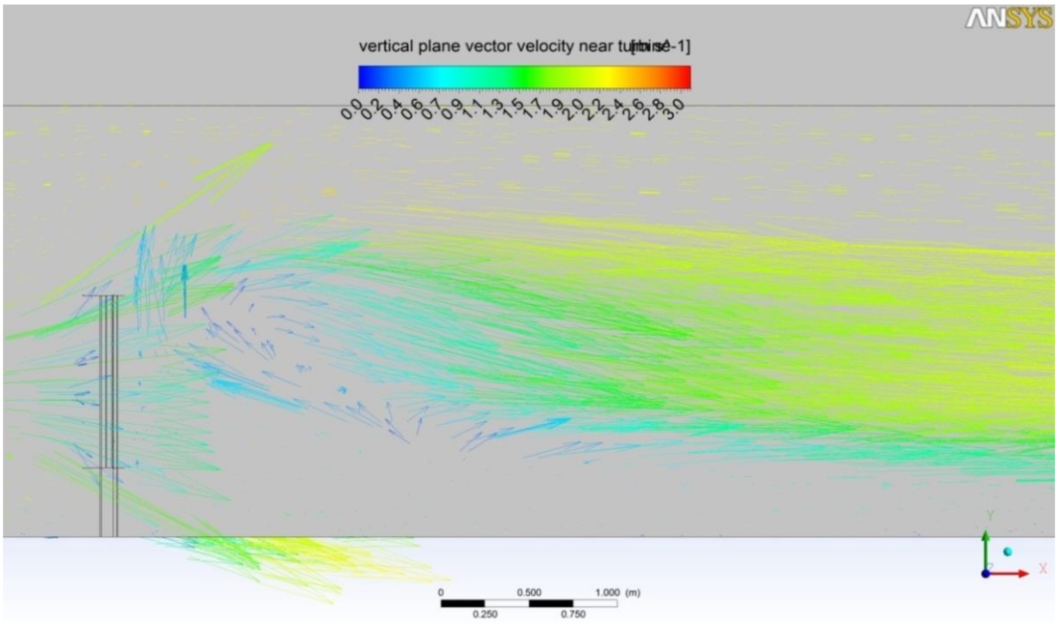


Figure 7.7 Vertical cross-section view of velocity vectors near the turbine

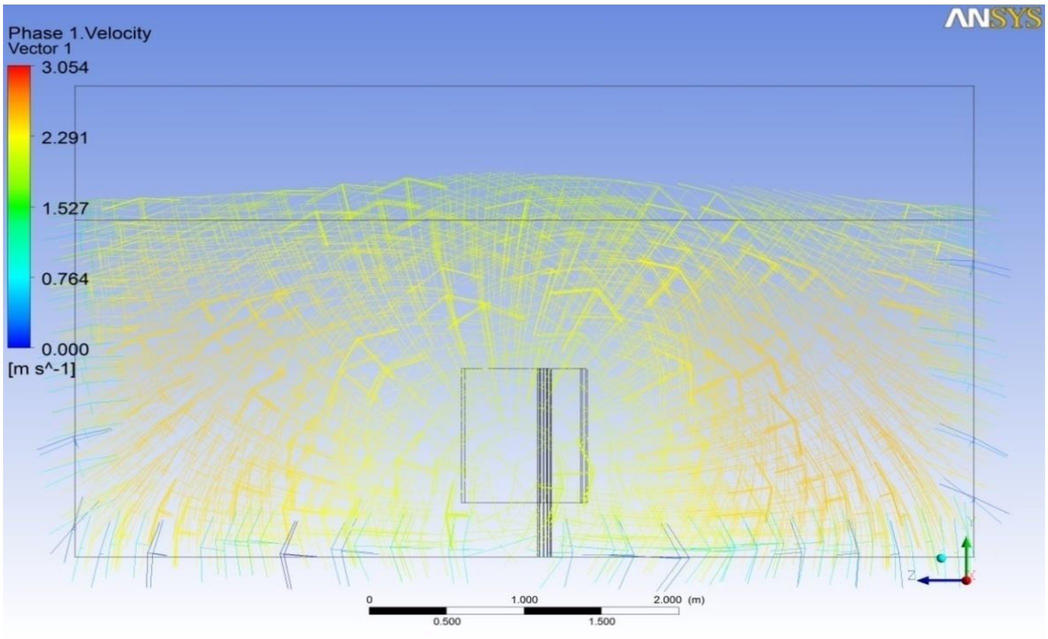


Figure 7.8 Vertical cross-section view of velocity vectors near the turbine

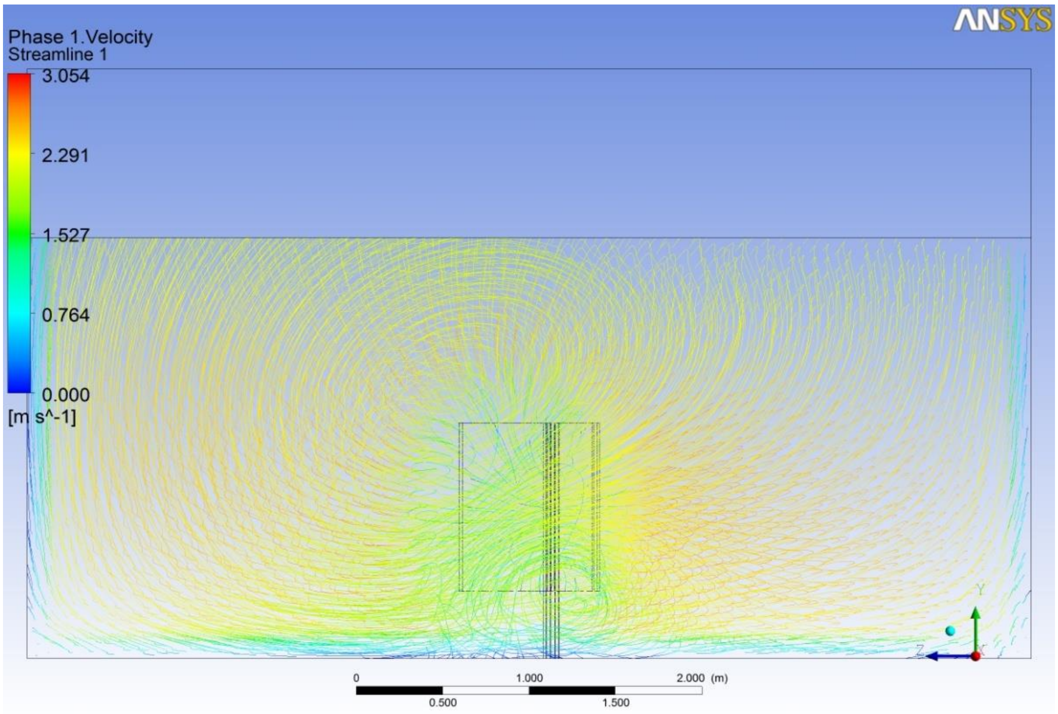


Figure 7.9 Vertical cross-section view of velocity streamlines near the turbine

The volume rendering of the whole domain is given in Figure 7.10.

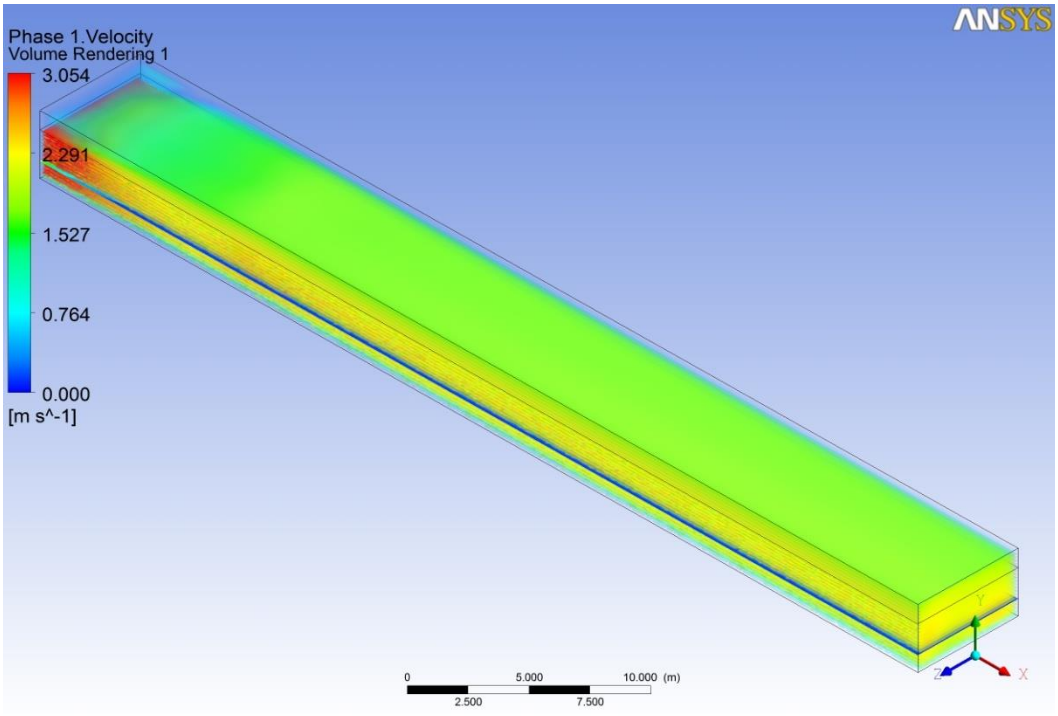
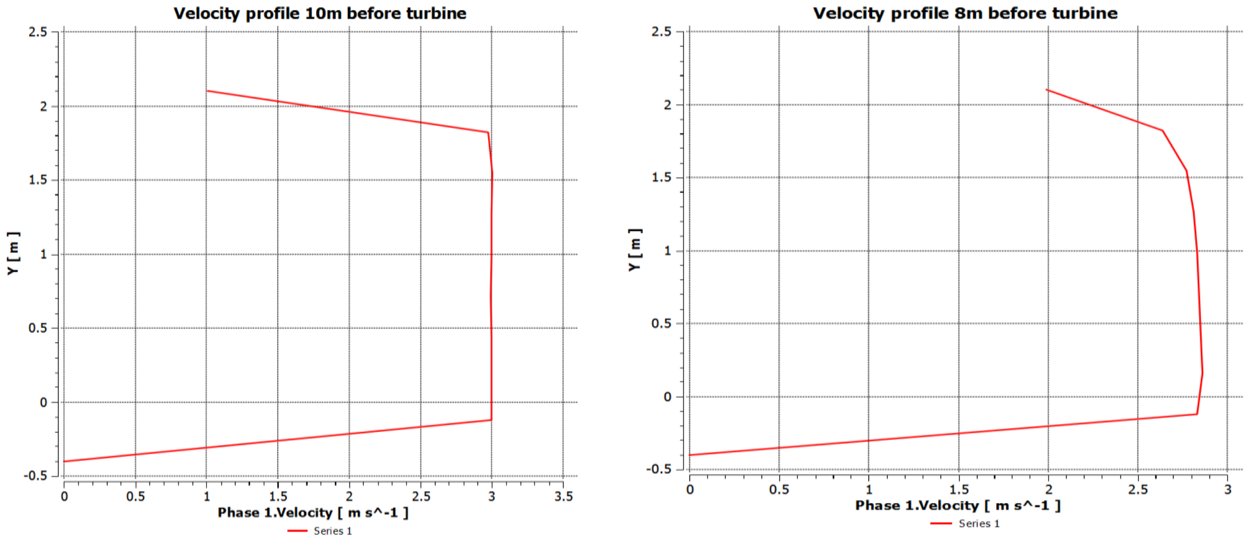


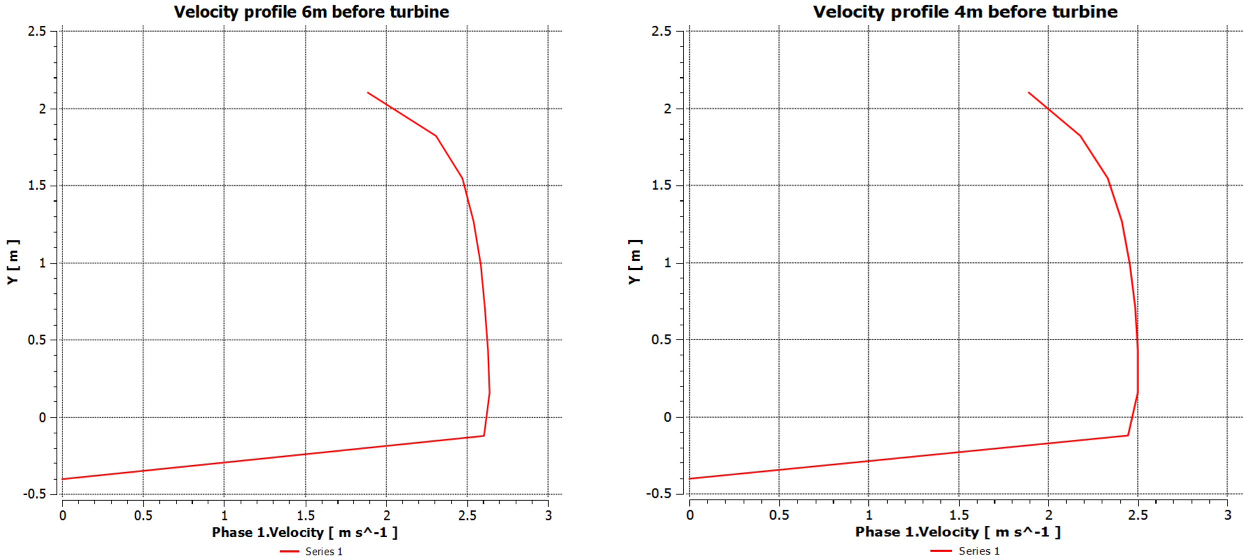
Figure 7.10 Volume rendering of whole domain

7.5 Velocity profiles

The velocity profile diagrams illustrate the velocity change in the channel along the position. The velocity variation at 7 m distance from the turbine is very limited. When the flow moves toward to the turbine the changes in velocity are observed. The velocity profiles at different positions are given in Figure 7.11.

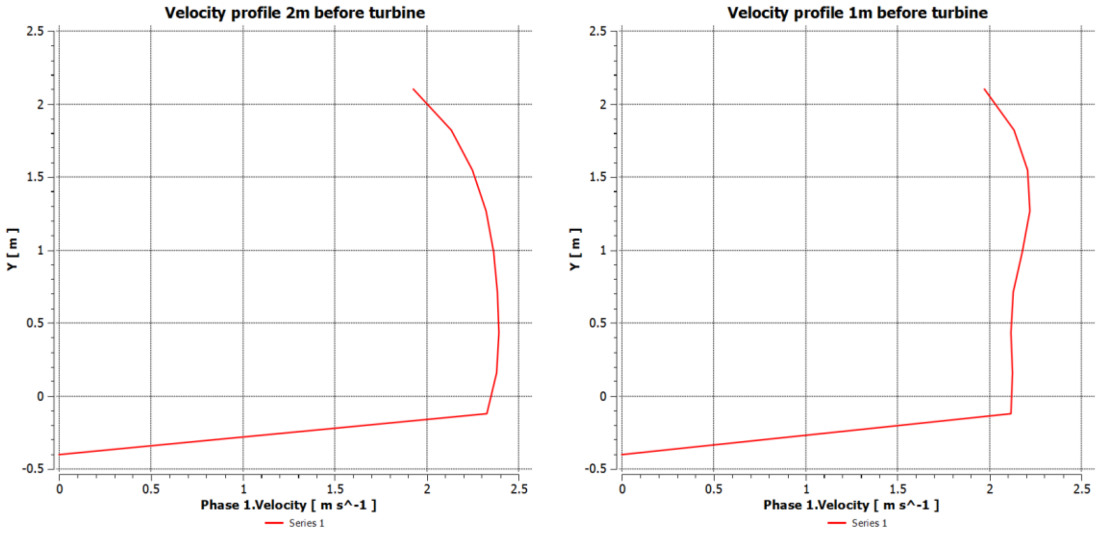


Velocity profile at 10 m upstream   Velocity profile at 8 m upstream



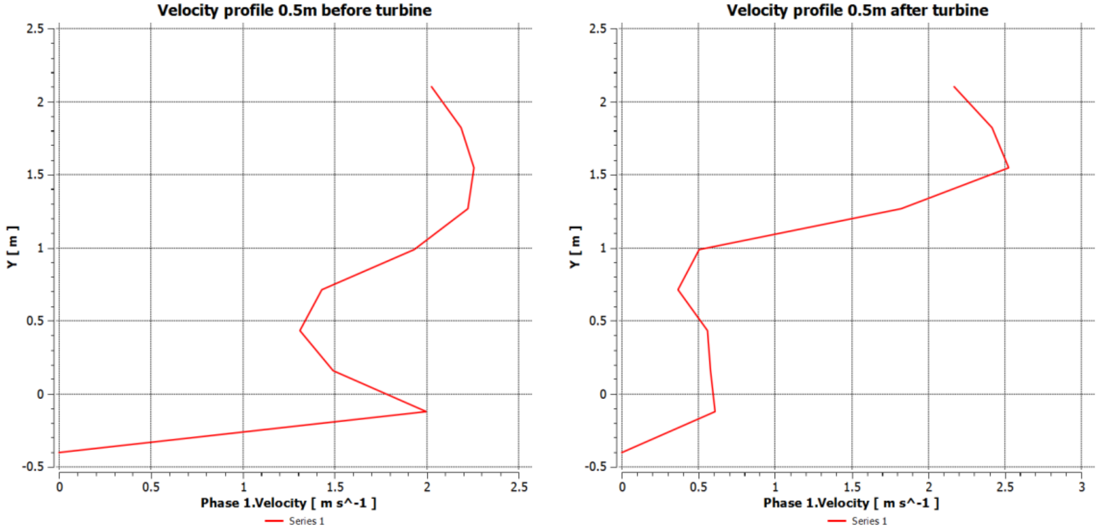
Velocity profiles at 6m upstream   Velocity profile at 4 m upstream

Figure 7.11 Velocity profiles before turbine



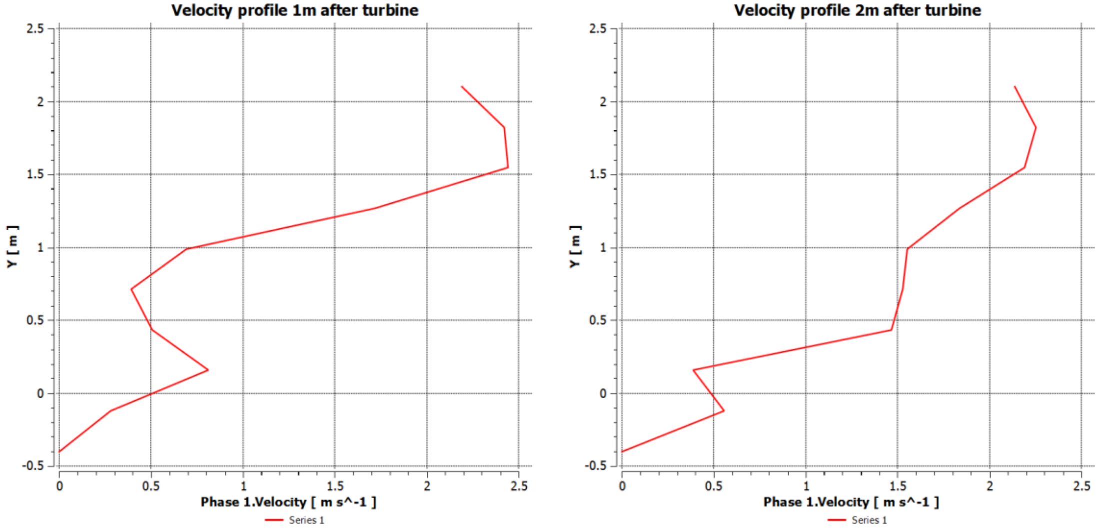
Velocity profile at 2upstream

Velocity profile at 1 m upstream



Velocity profile at 0.5 m upstream

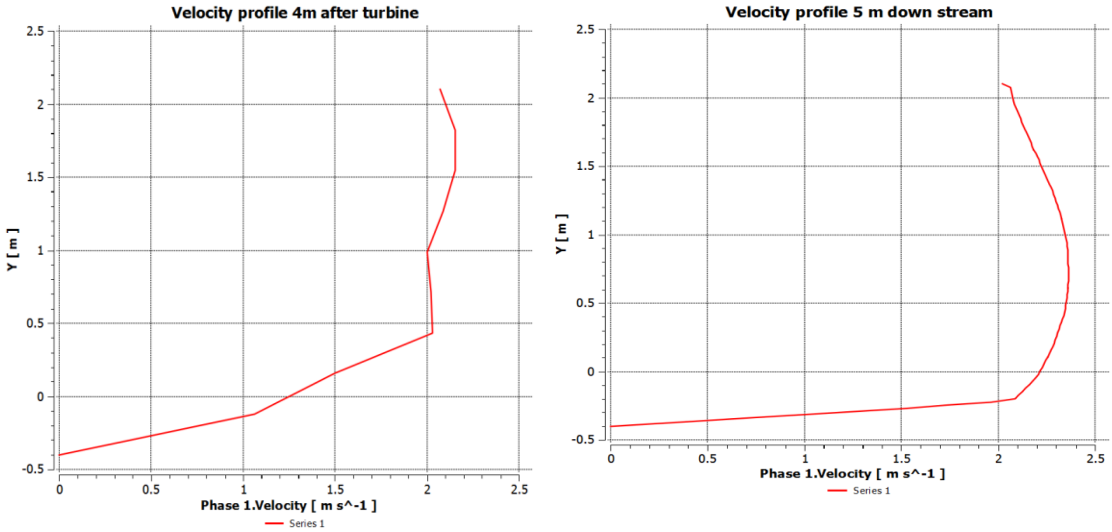
Velocity profile at 0.5m downstream



Velocity profile at 1 m downstream

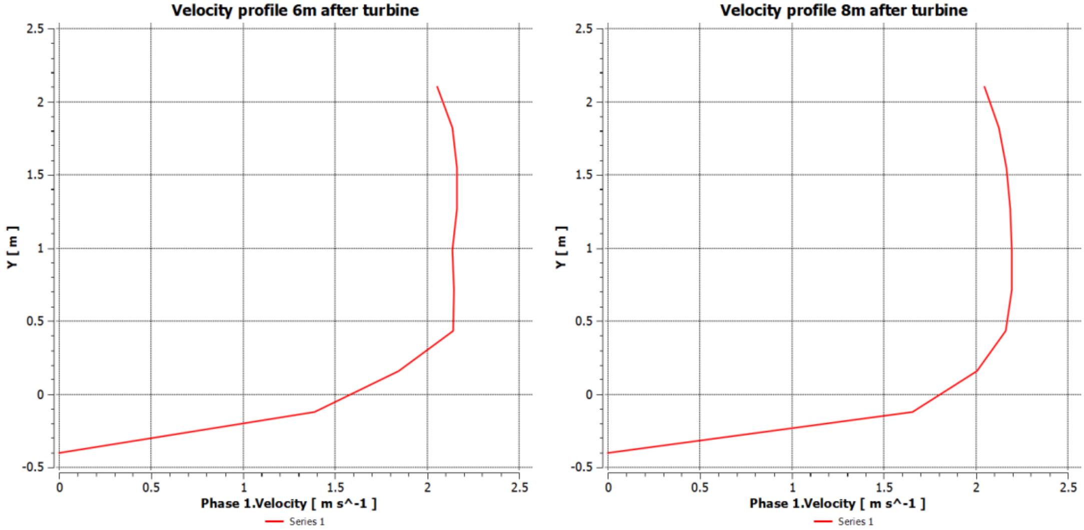
Velocity profile at 2 m downstream

Figure 7.12 Velocity profiles around turbine



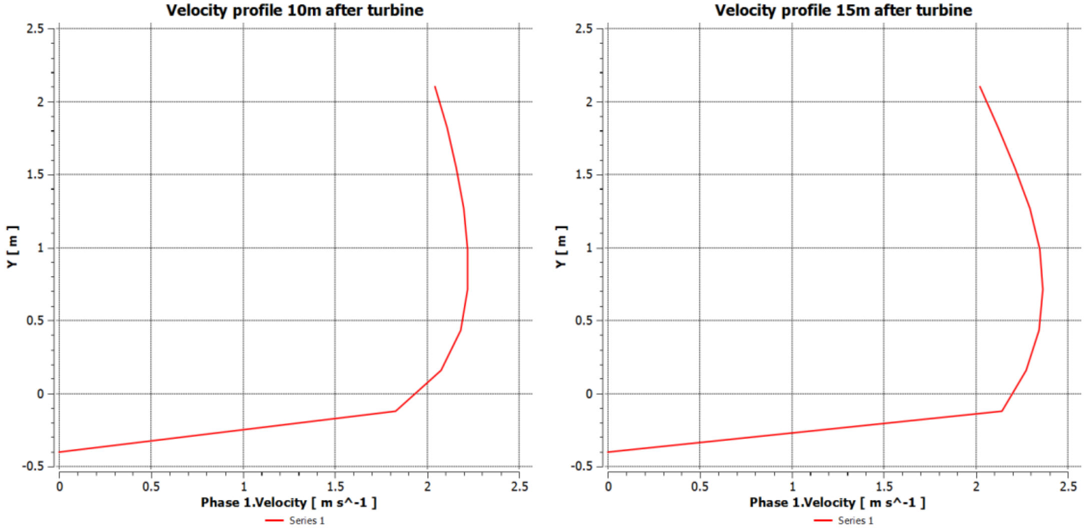
Velocity profile at 4m downstream

Velocity profile at 5 m downstream



Velocity profile at 6 m downstream

Velocity profile at 8 m downstream



Velocity profile at 10 m downstream

Velocity profile at 15 m downstream

Figure 7.13 The velocity profile at downstream

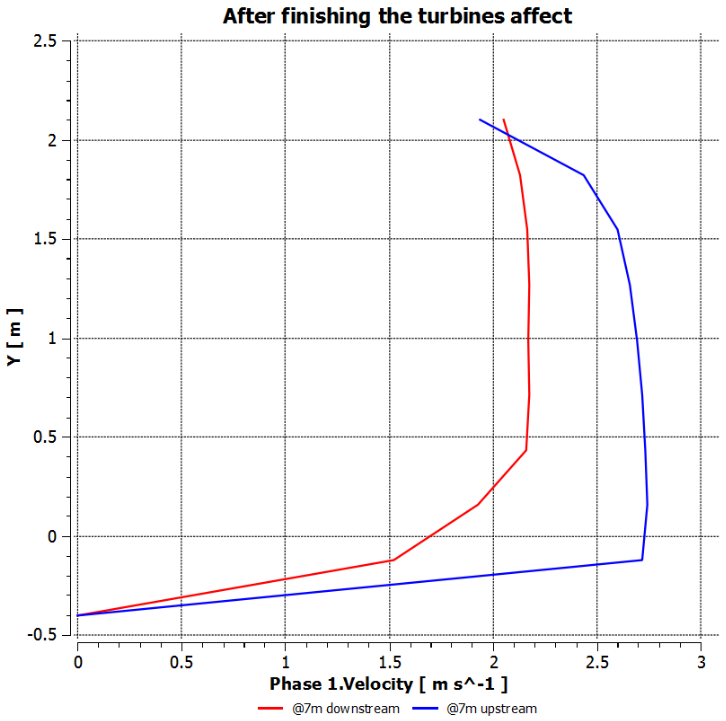


Figure 7.14 Velocity profiles at 7m upstream and 7m downstream of the turbine

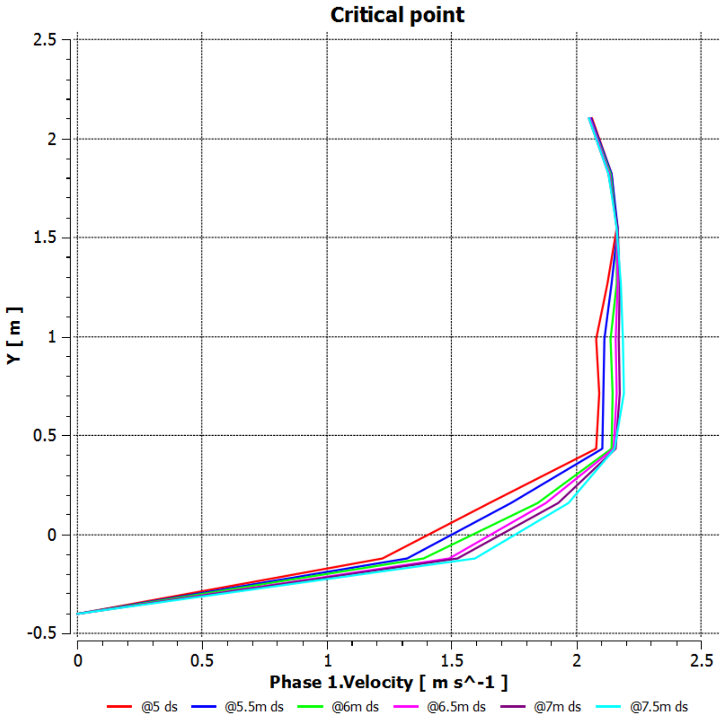


Figure 7.15 Velocity profiles in different points at the downstream

CHAPTER 8

CONCLUSIONS

The hydrokinetic energy which is harnessed from moving water of rivers, streams and oceans is being considered as a predictable and environmentally benign sourceof clean, renewable and sustainable energy. The technology of generating electrical energy by hydrokinetic turbines is new, compare to conventional hydropower turbines, and needs to be improved from efficiency point of view. In this study, the flow field around a vertical axis cross-flow hydrokinetic turbine, namely, a modified form of Savonius turbine was simulated by using ANSYS FLUENT which is commercially available CFD software. The working principles of cross-flow hydrokinetic turbines are different from horizontal axis turbines. The advantages of this type of turbines are; independency from the current direction including reversibility, stacking and self-starting without complex pitching mechanisms. The turbine has been simulated in a three dimensional fully developed rectangular open channel flow. Computational fluid dynamics (CFD) simulation of the hydrokinetic turbine was performed by computationally solving the Reynolds-Averaged Navier-Stokes Equations for an incompressible Newtonian fluid (RANS). CFD can be used to support turbine design and performance over a wide range of parameters in order to minimize the number of prototypes to be built which are used for optimization and experimental studies. CFD can also provide a cost-effective way of evaluating detailed full scale effects, such as mooring lines or local bottom bathymetry features, on both turbine performance and environmental assessment.

The vertical velocity profile was a typical fully developed steady open channel flow with the magnitude of the velocity being zero at the boundaries and increasing with the depth of the water. A slight decrease was observed in the velocity profiles due to the surface tension water experiences at the free surface. In order to harness the maximum amount of power from such a flow the hydrokinetic turbines need to be positioned at a level close to the free surface. In this research, it was aimed to find out the effect of the turbine on the upstream and the downstream flow velocity profiles, thus the depth of the water that the turbine was positioned in was not the primary concern. The simulation work was based on a stationary (non-rotating) turbine.

The study provides a framework to use the CFD simulations in order to improve the efficiency of the hydrokinetic turbines. The vertical velocity distribution along the central line of the steady open channel flow under goes significant changes. The shape of the velocity profile alters about 1m upstream of the turbine. The velocity of the flow decreases in the flow direction in the vicinity of the turbine, while it was observed to increase in the right and left hand sides of the turbine with respect to the flow direction. The wake emerges at the downstream of the turbine was observed to be turbulent for a length of around 6 to 7 m from the center line of the turbine. In case of positioning another turbine at the downstream of the first turbine, the distance between the two turbines should be at least 6 to 7 m in the direction of the flow, in order to harness the maximum quantity of electricity. The disturbed velocity profiles near the turbine causes turbulence thus, may lead scouring in the channel bed.

8.1 Future work

The hydrodynamic analysis of hydrokinetic turbines is still in its early stages. The flow areas at the upstream and downstream regions of hydrokinetic turbines and the variations in the turbine blade should be further studied. This important and new technology should not be neglected.

In this study simulation and analysis were performed for a stationary turbine; however simulations and analyses and experimental studies should be performed for a rotational turbine. Moreover, the longitudinal and cross-wise distances between turbines in an array configuration with the objective of optimizing the energy generation should be studied, with God’s help, in Ph.D.

REFERENCES

Andrews, J., Jelley, N. (2007). Energy science, principles, technologies and impacts. New York: Oxford University Press.

Bahaj, A.S., Myers, L.E. (2003). Fundamentals applicable to the utilization of marine current turbines for energy production, Renewable energy 28, pp.2205-2211.

Balat, M. (2006). Hydropower systems and hydropower potential in the European Union countries. Energy Sources Part A: Recovery, Utilization and Environmental Effects.28; 10, pp.965-978.

Balat, H. (2007). A new perspective for sustainable energy development in Turkey: The case of small hydropower plants. Renewable and Sustainable Energy Rewievs.Vol.11, pp. 2152-2165.

Baxter, R.M. (1977). Environmental effects of dams and impoundment: Annual Review of Ecological systems 8.

BHA (British Hydropower Association). (January 2005). A guide to UK mini hydro I

Briney, A. (2013) Ocean waves. Available at: http://geography.about.com/od/physicalgeography/a/waves.htm/.

Cada, Meyer. (2005). Resource concerns associated with natural streams. In: Proceedings of the Hydrokinetic and Wave Energy Technologies Technical and Environmental Issues. U.S. Department of Energy, Washington, D.C. pp 42-44.

Cada, G. J., Ahlgrimm, M., Bahleda, T., Bigford, S., Stavrakas, D,. Hall, R., Moursund and Sale, M. (2007). Potential impacts of hydrokinetic and wave energy conversion technologies on aquatic environments. Fisheries. Vol 32 (4) pp 174-181.

Cruden, G. (2005). Energy alternatives. Thomson Gale.

Das, M. M. (2011). Open Channel Flow. UK.

Dixon, S. L. (2005). Fluid Mechanics and Thermodynamics of Turbomachinery (5th edition).

Electric Power Research Institute (EPRI). (2011). Mapping and assessment of the United States ocean wave energy resource: USA.

ESHA, (European Small Hydropower Association (2004). Small hydropowersituation in the new EU member states candidate countries, thematic network on small hydropower (TNSHP) project, Renewable Energy House, Rue du Trone 26, 1000 Burssels, Belgium.

Freris, L., Infield D. (2008). Renewable Energy in Power Systems. United Kingdom: Wiley.

Godfrey, B. (2010). Renewable Energy. Second Edition, UK.

Goldstein, D.E. (2004). Stochastic Coherent adaptive Large Eddy Simulation.

Gorlov, A.M. (2001). Tidal Energy, Academic Press, Boston.

Hansen, M. (2008). Aerodynamics of wind turbines. (2ndedt.). London: Earthscan.

.Chanson, H. (2004). The Hydraulics of Open Channel Flow: An Introduction. Oxford, UK. (2ed).

International Energy Agency (IEA). (2011). Annual Report.

Kaya, D. (2004). Renewable energy policies in Turkey.Renewable and Sustainable Energy Reviews.Vol.10, pp.152-163

Kresala, Alternative Marine Energy Resources. (2013). Available at: http://kresalaenergia.wordpress.com/..

Lago, L.I, Ponta, F.L, Chen, L, (2010). Advances and trends in hydrokinetic turbine systems. Energy for Sustainable Development. Vol 14; pp.287-296.

NRC. (Natural Resources Canada). (2004a). Clean energy project analysis: RETScreen Engineering & Cases Textbook, chapter Small Hydro Project Analysis.

NRC. (Natural Resources Canada). (2004b). Micro hydropower systems.Abuyer's guide.

Pop, S. B. (2008). Turbulent Flow, Chapter 10. Cambridge University.

Quaschning, V. (2010). Renewable energy and climate change. (HedyJourdan, Trans.) Berlin University of Applied Systems HTV, Germany: Wiley.

Resch, G., Held, A., Faber, T., Panzer, C., Toro, F., Haas, R. (2008). Potentials and prospects for renewable energies at global scale.Energy Policy. 36, pp. 4048-4056.

Sam, A. (2010). Water Wheel CFD Simulation, Msc. Thesis, Graduate School of Engineering of Lund University, Sweden.

Sarvan Mamaidi, M.S. (2009). Analysis of a mini hydrokinetic turbine, Northern Illinois University.

Schlager, N., Weisblatt, J. (2006). Alternative energy.Vol.3. Thomson Gale.

Shamez, K. (2009). In-Situ Testing of a Darrieus Hydro Kinetic Turbine in cold climates, Msc Thesis, Graduate School of Engineering of Manitoba Winnipeg University, Canada.

Schlager, N., Weisblatt, J. (2006). Alternative energy. China. Thomson Gale.

Taylor, J. H. (2012). Numerical Simulation of a cross Flow Marine Hydrokinetic Turbine, Msc Thesis, Graduate School of Engineering of Washington University.

Toklu, E. (2013). Overview of potential and utilization of renewable energy sources in Turkey. Renewable Energy.Vol.50, pp.456-463.

Toossi, R. (2008) Energy and the environment; sources, technologies and impacts, VerVe Publishers.

Twidell, J., Weir, T. (2006) Renewable Energy Resources. Second Edition, UK.

U.S. Department of Energy (DOE). (2009). Energy Efficiency and Renewable Energy (EERE), Report to Congress on the Potential Environmental Effects of Marine and Hydrokinetic Energy Technologies.

Muratoglu, A. (2011). Assessment of Tigris River Hydropower Potential, Msc Thesis, Graduate School of Engineering of Gaziantep University, Turkey.

U.S. Geological Survey (USGS). (2013). Hydroelectric power water use.Available at: http://ga.water.usgs.gov/edu/wuhy.html..

Versteeg, H. K., Malalasekera, W. (2007).An Introduction to Computational Fluid Dynamics.2nd Edition. UK.

Warnick, C. (1984). Hydropower Engineering. New Jersey. Prentice Hall.

Wikipedia. (2013). Available at: http://en.wikipedia.org/wiki/Water.

Yuksek, O., Komurcu, M.I., Yuksel, I., Kaygusuz, K. (2006). The role of hydropower in meeting Turkey's electricity demand.Energy Policy, 34, 3093-3103.

Yuksel, I. (2010). As a renewable energy hydropower for sustainable development in Turkey. 14, 3213-3219.

White, F.M. (2008). Fluid Mechanics, 6thedn, McGraw-Hill, New York.

Veronica Miller. (2010). Dynamic modeling and environmental analysis of hydrokinetic energy. PhD thesis University of Pittsburgh.