UNIVERSITY OF GAZİANTEP
GRADUATE SCHOOL OF
NATURAL & APPLIED SCIENCES

FLOW FIELD AROUND VERTICAL AXIS CROSS-FLOW HYDROKINETIC TURBINES

M. Sc. THESIS
IN
CIVIL ENGINEERING

BY
AZAD DAZAEA
SEPTEMBER 2013

Flow Field around Vertical Axis Cross-flow Hydrokinetic Turbines

M.Sc. Thesis
In
Civil Engineering
University of Gaziantep

Supervisor
Assist. Prof. Dr. Mehmet İshak YÜCE

By
Azad DAZAEA

September 2013

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

Name of the thesis: Flow Field around Vertical Axis Cross-flow Hydrokinetic Turbines

Name of the student: Azad DAZAEA
Exam date: 16/09/2013

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
Assoc. Prof. Dr. Murat PALA	
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.

Azad DAZAEA

ABSTRACT

FLOW FIELD AROUND VERTICAL AXIS CROSS-FLOW HYDROKINETIC TURBINES

DAZAEA, Azad Mohsin
M.Sc. in Civil Engineering
Supervisor: Assist. Prof. Dr. Mehmet İshak YÜCE
September 2013, 77 pages

Renewable energy sources, most notably, wind energy, solar energy and small scale hydropower schemes have undergone major development lately. However the intermittency and weather dependency of solar and wind energies pose a challenge. Another form of renewable energy which has attracted great interest in recent times is hydrokinetic energy. Hydrokinetic energy is harnessed from flow waters of river streams, tidal flows or ocean currents. This energy resource has a great potential to be exploited on a large scale because of its predictability and intensity. It is most likely to be one of the new and clean energy alternatives for the 21st century. In this study, a vertical axis cross-flow hydrokinetic turbine, namely a modified form of Lucid Energy Technology (LET) Gorlov Helical Turbine (GHT) was investigated. The working principles of vertical axis cross-flow hydrokinetic turbines are different from those of commonly used horizontal axis ones. 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, commercially available software was employed. The flow field was studied with the turbine being positioned at two different depths, while the depth and the velocity of the open channel flow were kept constant. The results have shown that placing the turbine closer to the bed of channel causes an increase in water velocity near the free surface. Although the water velocity near the channel bed seems to be decreasing, the turbulence flow takes place just behind the turbine and the burst of the flow directly below the turbine are expected to cause scouring and increase turbidity levels.

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

ÖZET

HIDROKINETIK TÜRBINLERIN ETRAFINDAKI AKIM ALANININ INCELENMESI

DAZAEA, Azad Mohsin
İnşaat Mühendisliği Yüksek Lisans
Danışman:Yrd. Doç.Dr. Mehmetİshak YÜCE
Eylül 2013, 77 sayfa

Yenilenebilir enerji kaynaklarından özellikle, rüzgar enerjisi, güneş enerjisi ve küçük ölçekli hidroelektrik projeler son zamanlarda büyük gelişme gösterdiler. Ancak güneş ve rüzgar enerjilerinin süreksizliği ve hava şartlarına bağımlılıkları sorun oluşturmaktadır. Son zamanlarda büyük ilgi gören yenilenebilir enerji çeşitlerinden biride hidrokinetik enerjidir. Hidrokinetik enerji nehir akımları, gel-git akımları ya da okyanus akımlarından elde edilebilir. Bu enerji kaynağı tahmin edilebilirliliği ve büyük ölçekte bulunuyor olması nedeniyle yararlanılması gereken bir potansiyele sahiptir. Bu kaynağın, 21.yüzyıl için yeni ve temiz enerji alternatiflerinden biri olması muhtemeldir. Bu çalışmada, dikey eksenli çapraz-akışlı bir hidrokinetik türbin olan Lucid Enerji Teknolojisi (LET) Gorlov Sarmal Türbinin (GHT) değiştirilmiş (modified) bir formu incelenmiştir. Dikey eksenli çapraz-akışlı hidrokinetik türbinlerinin çalışma prensipleri yaygın olarak kullanılan yatay eksenli türbinlerinkinden farklıdır. Ters-dönme dahil olmak üzere akım yönden bağımsızlık ve karmaşık mekanizmalar olmadan otomatik olarak kendi-kendine başlama bu tür türbinlerin avantajlarındandır. Türbin üç boyutlu dikdörtgen açık kanal akım ortamında simüle edilmiştir. Hidrokinetik türbinin hesaplamalı akışkanlar dinamiği (CFD) simülasyon hesaplamaları Reynolds- Averaged Navier-Stokes denklemleri (RANS) çözerek yapıldı. Sayısal çalışmalar, ticari bir yazılım olan ANSYS FLUENT programı kullanılarak yapılmıştır. Akım alanı, açık kanal akımının hızı ve derinliği sabit tutularak, türbinin iki farklı derinlikte olması durumu incelenmiştir. Sonuçlar, türbinin kanal tabanına yakın bir şekilde yerleştirilmesi durumunda serbest su yüzeyine yakın olan bölgelerde akımın hızında bir artış olduğunu göstermiştir. Kanal tabanına yakın yerlerde akımın hızı azalıyor olsa da, türbinin hemen mansabında meydana gelen türbülanslı akım alanı ve türbinin altından oluşan akımdan dolayı kanal tabanında oyulma meydana gelecek ve dolayısı ile suda bulanıklılık seviyesinde artış olması beklenir.

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 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 patience and support during my study.

My sincere appreciation also extends to my friends and to those gave me the delight of smile especially Mr. Aumed M. Amen.

TABLE OF CONTENTS

CONTENTS **Page**

LIST OF FIGURES

FIGURES **Page**

LIST OF TABLES

TABLES **Page**

LIST OF SYMBOLS / ABBREVIATIONS

A	Cross sectional area
a	Acceleration
B	Width
CFD	Computational Fluid Dynamics
C_p	Power coefficient
D_m	Mean hydraulic depth
EARSM	Explicit algebraic Reynolds stress model
g	Ground acceleration coefficient
GHT	Gorlov helical turbine
H	Overall head
HEPP	Hydro electrical power plant
K	kinetic energy
Ks	Roughness coefficient
L	Length
LES	Large Eddy Simulation
LET	Lucid Energy Technologies inc.

M	Mass
P	Power, wetted perimeter
Q	Average discharge
R	Hydraulic radius
RANS	Reynolds average Navier-stokes equation
Re	Reynolds number
RNG	Re-Normalisation Group
RSM	Reynolds stress model
SHPP	Small Hydro Power Plant
y	Depth of water
Z	Vertical distance from the surface
P	Density
μ	Overall efficiency, dynamic viscosity
V	Velocity
2-D	Two Dimensional
3-D	Three Dimensional
ΣF	Sigma force
Ɛ	Turbulent dissipation
ω	Specific dissipation

CHAPTER 1

INTRODUCTION

1.1. Energy Background

The ability to do work is the most common definition of energy. Energy is a conserved quantity, it can neither be created nor destroyed, however it can be converted into different forms. All forms of energy are originated from the sun. Solar power, both in the form of direct solar radiation and indirect forms such as bio energy, hydropower and wind power were the energy sources of early human societies. In the middle of 21st century the population of the world is estimated to be double and the need for energy is estimated to be increased by at least 70 % over the next 30 years (Jennings, 1996).With the increasing of the population the need for energy is also expected to increases. Currently most of the world's energy demand is generated by fossil fuels. Increasing energy demand and limited natural resources forces researchers to investigate alternative sources of energy generation. World energy consumption in 2005 was 100.2 quadrillion Btus and expected to increase by approximately 1.1 per-cent each year. It is expected that by 2030 reliance on renewable resources will almost double (EIA, 2011).

1.2. Renewable energy and its Resources

Renewable energy is generally defined as energy that comes from resources continually replaced. Around 3.7% of global final energy consumption comes from hydropower while the share of hydropower in electricity production is around 16%.

Renewable energy sources derive their energy from the sun, either directly or indirectly. Hydro and wind are expected to supply energy for almost another 1 billion years at which point the predicted increase in heat from the sun is expected to make the surface of the Earth too hot for liquid water to exist. Renewable energy resources do not produce greenhouse gases and other environmentally harmful pollutants, unlike fossil fuels. Renewable energy sources which are sunlight, wind, conventional hydropower, flowing river waters, tidal flows, sea waves, biomass, and geothermal, provide more economic and continuous solutions than fossil and nuclear fuels (IEA, 2011). Wind energy, solar energy, and small scale hydropower schemes are most notably renewable energy sources that have submitted main developments. Their potential to meet the world's energy demand for the 21st century and beyond, cleanly, safely, and economically is high (Twidell and Weir, 2006). Hydrokinetic energy is harnessed from flowing waters of river streams, ocean currents and tidal flows. The types of the renewable energy resources and the estimated renewable energy share of global final energy consumption are given in Figures 1.1 and 1.2 respectively

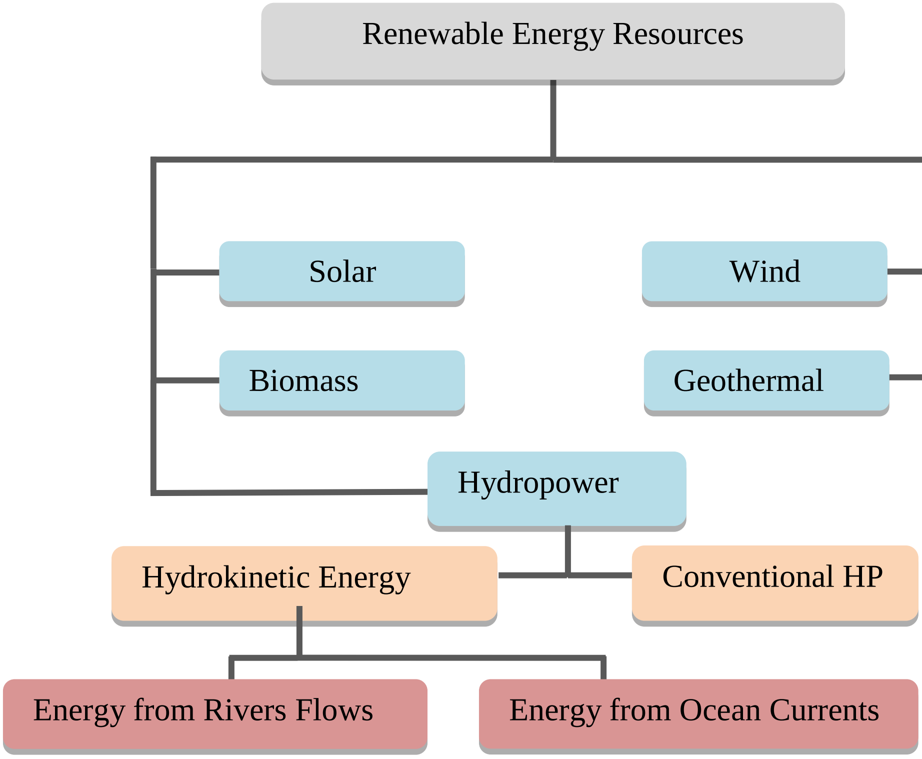


Figure 1.1 Types of renewable energy resources

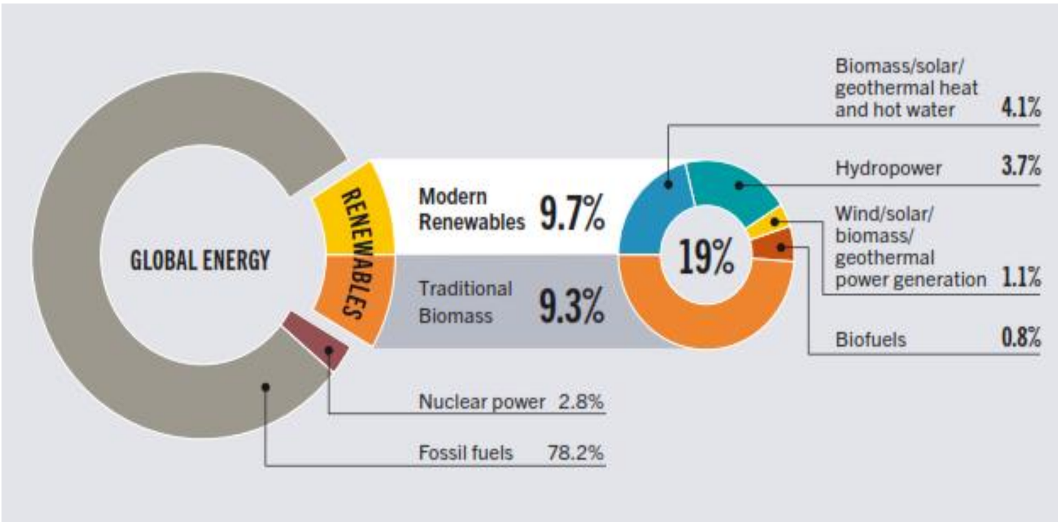


Figure 1.2 Estimated renewable energy share of global final energy consumption (Renewables, 2013)

1.2.1. Solar Energy

Sun is the main source of energy. A massive amount of energy spreads from the Sun on daily bases. The earth receives only a very small part that. Even that small part is so much which is enough to meet the world's energy demand. Solar energy is harnessed from the direct radiation of light or heat by using solar photovoltaic panels or solar thermal collectors. The semiconductor silicon cells directly convert the sun's light into electric energy by solar photovoltaic panels. However the efficiency of these panels is very low and their cost is high because of the problem of high requirements of purity of material. Solar thermal collectors are used to concentrate the sun light at a point to heat-up air, water or oil like fluids by parabolic dishes or deflectors. Since the heated air is expected to rise electrical energy could be generated by this natural flow of hot air by using a turbine. Likewise, the heated water will be evaporated and help turn a turbine. Heated oil is used to boil water in a container; the steam then will turn the turbine to generate electricity (Twidell and Weir, 2006).

1.2.2. Wind Energy

Windmills are used to grind wheat and pump water since long time. Wind turbines are modern windmills. Wind turbines transform the energy in the wind into mechanical energy, which can then be used to produce electricity. A minimum wind velocity required for the economical usage of wind energy is about 7 m/sec. The conversion of wind energy into electricity has increased dramatically in the last decade which is making wind energy the leader of new energy sources in terms of development rate (Twidell and Weir, 2006).

1.2.3. Energy from Rivers

The most environmentally friendly traditional renewable energy resource is hydropower never the less it has problems in rehabilitation. The uncertainty of rainfall and regional problems of water use and distribution are never ending. Energy from water can be used by two ways. The traditional way which is producing electricity by dams which utilizes reservoirs to store water. The storage reservoirs provide ahead of water with static pressure and the static energy of stored water is converted into kinetic energy with high speed flow inside turbines. This is called hydrostatic method. One of the oldest renewable energy production methods to produce hydroelectric power is hydrostatic technique. Small scale hydropower plants (up to 25 MW), included in the category of renewables has a large share in the total achievement. In tight and off grid areas where large dams cannot be built small scale hydropower is suitable. Figure 1.3 shows Ataturk dam which is the largest dam and the biggest hydroelectricity power plant in Turkey (Twidell and Weir, 2006).

Figure 1.3 Ataturk dam

The second way is hydrokinetic energy which has drawn the attention of the scientists and investors in the last 20 years. It’s a new technique of producing electricity which employs turbines inside water current to convert kinetic energy of flowing water directly into electricity without building dams and reservoirs.

1.2.4. Energy from Oceans

There are three ways to use the energy inside the ocean. First is the ocean thermal energy conversion method. The difference in temperature of about 25° C between the surface and deep water can be used in a heat engine to produce electric power. Second method is employing tides which can be used as a source of energy from the oceans. Large structures like barrages can be built which allow tidal water to pass through large turbines for producing power. The third way of harnessing energy in oceans is the use of sea waves (Twidell and Weir, 2006).

1.2.5. Energy from Biomass

The energy derived from materials such as wood, straw, municipal and industrial wastes or animal wastes is called bioenergy and classified as a type of renewable energy in contrast to the energy generated from fossil fuels. The biomass can be converted into biogas or bio-liquid and used as a source of energy (Godfrey, 2010).

1.2.6. Geothermal Energy

Greek and Roman are thought to be the first who used exploitation of geothermal resources in history. They used hot water for medicinal, domestic and leisure applications (Andrews and Jelly, 2007).The heat in the core of the earth is the source of the geothermal energy. In some countries geothermal is used for industrial, recreational purposes and heating buildings (Twidell and Weir, 2006).The development of the primary energy demand of the world is given in Figure 1.4.

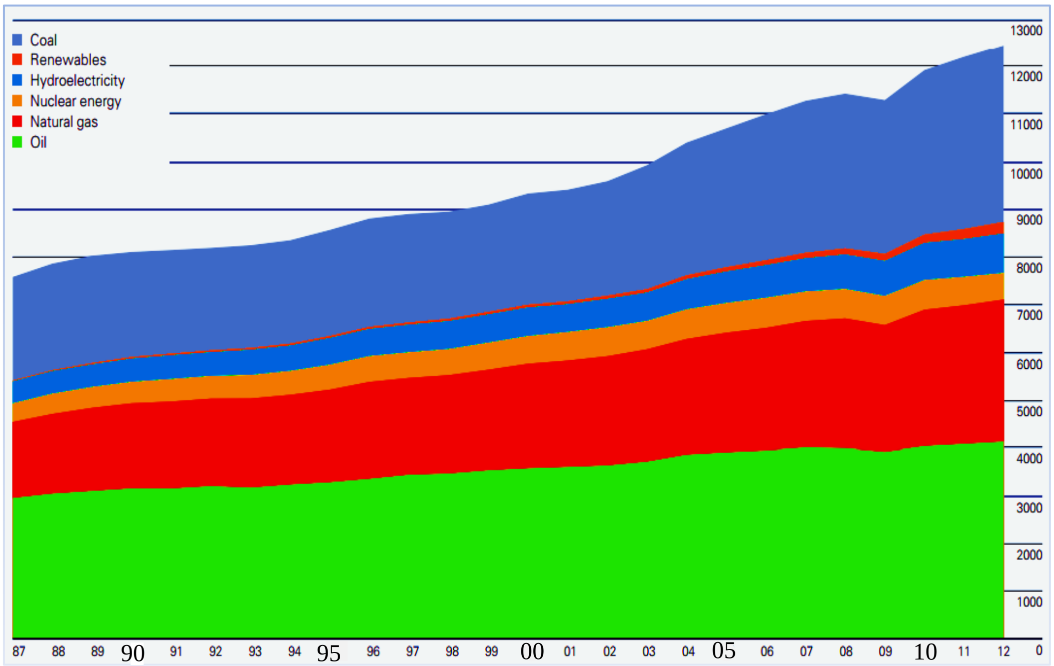


Figure 1.4 Development of primary energy demand (BP, 2013)

1.3. The purpose of thesis

Supporting 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 is the main purpose of the thesis. In this thesis, the specific objectives are simulating a hydrokinetic turbine inside an open channel and investigating the flow around the turbine by changing its position to examine the effect of changing velocity around the turbine.

1.4. Structure of Thesis

This study is composed of totally 7 chapters. The first chapter provides an introduction to the study of the simulation of a hydrokinetic turbine. In this chapter, overall information about the renewable energy is given. The types and potential of different renewable energy sources are introduced. The purpose of the thesis was explained. Chapter 2 is about the power from water. The definition hydropower and its total potential in the world are 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 is about open channel flows, definition of open channel flows, types of open channel, and types of flow and velocity profile of open channel is described.

Chapter 4 explains the Computational Fluid Dynamics (CFD) terminology which is a fundamental approach of this study. The different stage of a CFD simulation was introduced and turbulence with its models is described. Chapter 5 is about simulation of turbine inside the open channel, selection of a certain type of hydrokinetic turbine which is modified Lucid Energy Technology (LET) Gorlov Helical Turbine (GHT), and its' procedure of simulating. In chapter 6 the results are examined. Finally the conclusion and future works are given in chapter 7.

CHAPTER 2

POWERFROM WATER

2.1. What is hydropower?

The energy that comes from the force of moving water is called hydropower. Water in oceans and rivers evaporates by energy from the sun and move sit upward as water vapour. The water vapour concentrates and forms clouds when reaches a cooler air front in the atmosphere. The humidity finally falls as rain or snow, replenishing the water in the oceans and rivers. This operation is called the water cycle. Water flows from high grounds to low grounds by gravity. Power can be extracted from the force of flowing water. Hydropower is categorised as a renewable energy source. This energy source will not be finished, as long as the water cycle continues (need, 2013).

2.2. Background of hydropower

Water wheels have been used in Mesopotamia to grind wheat into flour long ago. American and European factories used the water wheel to power machines in the beginnings of 17th century. The water wheel is a simple machine it is located below a source of flowing water and captures water in pails attached to the wheel and the weight of the water causes the wheel to turn. The movement comes from the potential energy (gravitational energy) of the water by the water wheels. That energy can then be used for grinding grain. The force of falling water was used to generate electricity in the late 19th century (need, 2013).Hydroelectricity consumption by regions is given in Figure 2.1 in million tons of oil equivalent (Mtoe)

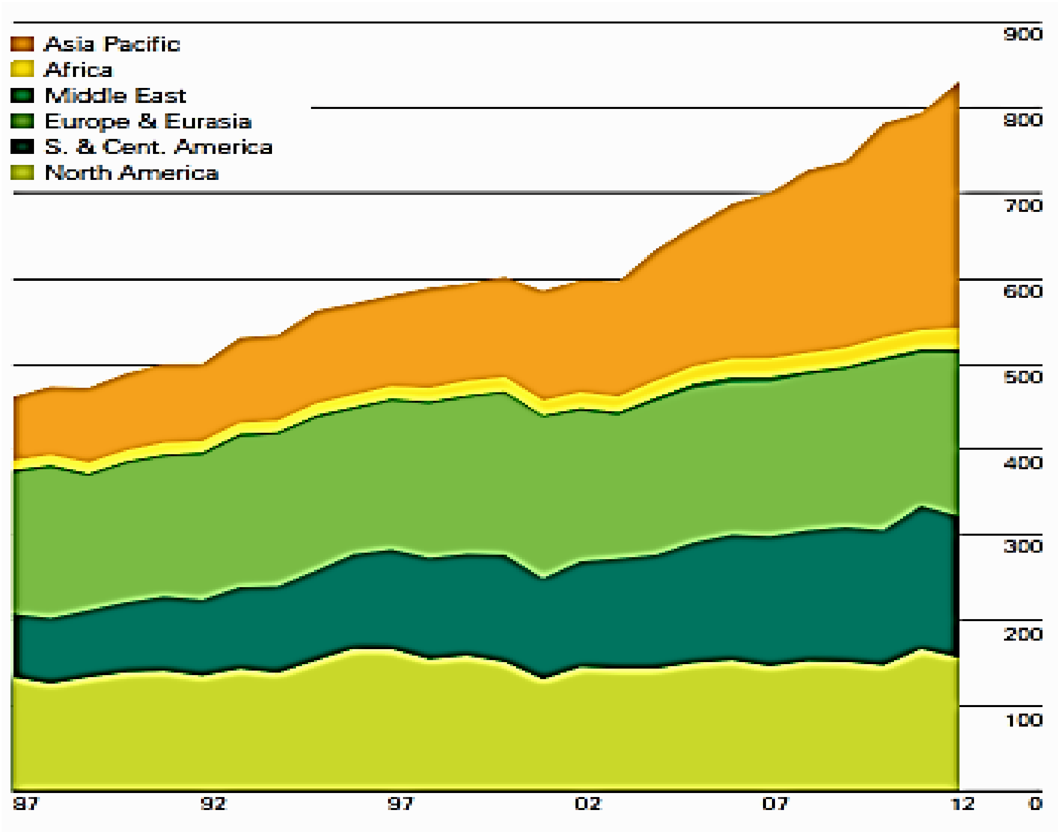


Figure 2.1 Hydroelectricity consumption by region Mtoe (BP, 2013)

2.3. Types of hydro power

Hydropower is categorised into two main types which are conventional hydropower and hydrokinetic energy. Types of hydropower are detailed in Figure 2.2

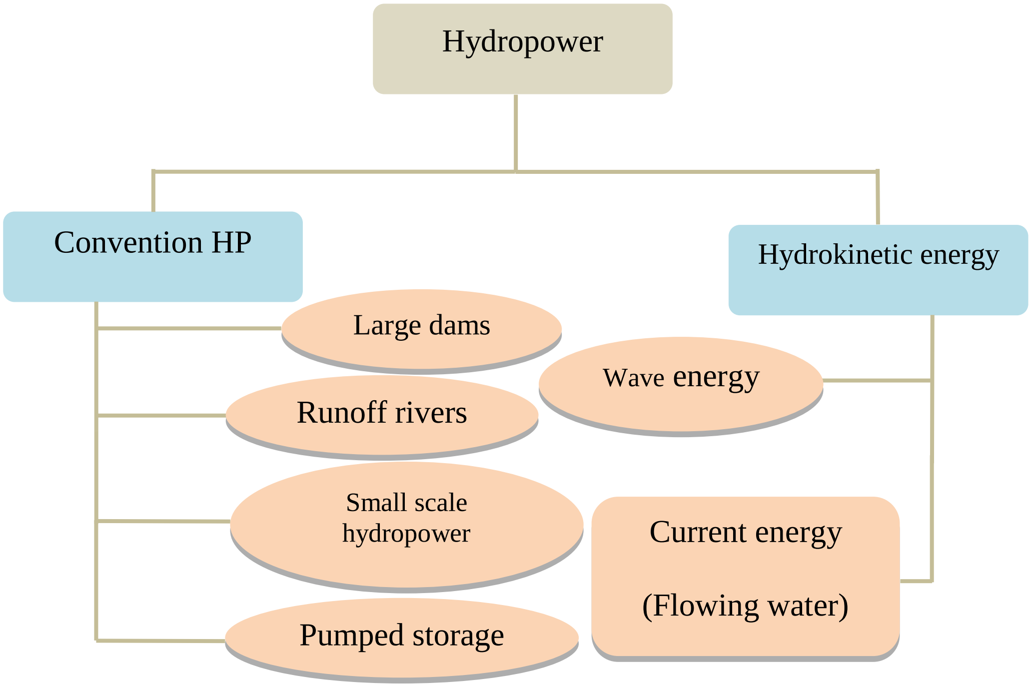


Figure 2.2 Types of hydropower plants

2.3.1. Conventional hydropower

Conventional hydropower has four types as large scale hydropower, run off river schemes, small scale hydropower and pumped storage. Large scale hydropower is producing high amount of electricity by using dam or big reservoir to reserve water from rivers. The storage is built in a suitable place where a sufficient amount of water could be collected without too much seepage. The water is taken by a number of pipes and conveyed to the hydrostatic turbines producing electricity by using the high drop of water. In small hydropower plants dam or barrage is smaller than large scale usually just a weir and almost little or no water stored. Small systems give low degree of power. The effects of the small scale systems are smaller than the big systems. The energy that can be produced from the water depends on the head and the discharge of flow (Freris and Infield, 2008). In these systems, energy can be obtained by the following ways:

- Building a small dam (Small hydroelectric power plants)
- Diverging the water (River run off schemes)
- Placing turbine directly into the water flow (Hydrokinetic turbines,).

The advantages of small hydropower plants over the other power technologies (big scale hydropower, wind, solar, wave etc.).

- Maximum efficiency is obtained (efficiency is between 70-90% for run off river conversion schemes)
- The flow is highly predictable. That provides capability of long term planning, the energy cost can be guaranteed comparing with the other means of energy generation.
- Technology is long term and advancing.
- It is one of the cheapest energy sources. Due to the falling prices, small scale hydropower plants are becoming more demanded than ever.
- Energy can easily be supplied to remote and off grid areas.

In small hydropower projects, 75% of the cost is determined by the location and site conditions, while only about 25% of the cost is relatively fixed, being the cost of manufacturing the electromechanical equipment (NRC, 2004a).Runoff river schemes are a kind of small barrage without a reservoir. In this type of systems, generally a relatively small mass of water is diverged from the natural river and a fall is tried to obtain. The runoff river power plants are useful where the head is enough to run the turbines efficiently, the sufficient head is a few meters. The water is taken from the river and reached until the turbines through a weir. Then the turbines run and generator produces electricity. Pumped-storage hydroelectricity is a type of hydroelectric power generation used by some power plants for load balancing. The method stores energy in the form of potential energy of water, pumped from a lower elevation reservoir to a higher elevation. Low-cost off-peak electric power is used to run the pumps. During periods of high electrical demand, the stored water is released through turbines to produce electric power. Although the losses of the pumping process makes the plant a net consumer of energy overall, the system increases revenue by selling more electricity during periods of peak demand, when electricity prices are highest. At times of low electrical demand, excess generation capacity is used to pump water into the higher reservoir. When there is higher demand, water is released back into the lower reservoir through a turbine, generating electricity. Reversible turbine/generator assemblies act as pump and turbine (usually a Francis turbine design).

2.3.2. Hydrokinetic energy

Hydrokinetic power can be generated from two main sources: marine and rivers. Marine hydrokinetic power deals with extracting energy in the ocean from tides and ocean currents. Tidal energy comes from the expected rise and fall of tides generated by the gravitational pull of the sun and the moon whereas ocean currents are large convection systems generated by temperature differences. The kinetic energy of the flow leads to extracted hydrokinetic power from river streams. Unlike dams that use potential energy from a stored reservoir and utilize the difference in elevation from inlet to outlet to create a hydraulic head, river hydrokinetic uses the energy from flowing water, thus extracting power from the kinetic energy within the flow using the same principles as wind energy. Extracting power in this way is possible even if the river stream has a low velocity (Kassam, 2009).

2.4. Hydrokinetic turbines

Many ideas for devices to extract the kinetic energy from the flow were created throughout the development of the wind industry. In the development of tidal energy extraction, the industry is still in early stages where several designs for energy extracting devices are being introduced and investigated.

2.5. Classification of hydrokinetic turbines

Hydrokinetic turbines are classified into two systems propeller systems and non-propeller systems. The turbines that the blades rotate around a horizontal or vertical axis they are propeller systems, also called in-stream hydrokinetic turbines which include the tidal flow, river stream and marine current applications. The horizontal axis turbines are separated into horizontal and vertical axis turbines, helical turbines and ducted turbines.

When the turbines have a number of moving mechanisms that mainly converts the irregular motion to the regular motion they are called non-propeller systems. Wave energy converters are such systems (M. Amen, 2013; Muratoglu, 2011). The classification of hydrokinetic energy converters is given in Figure 2.3. Also Figures 2.4-2.8show the propeller system and non-propeller system hydrokinetic turbines respectively.

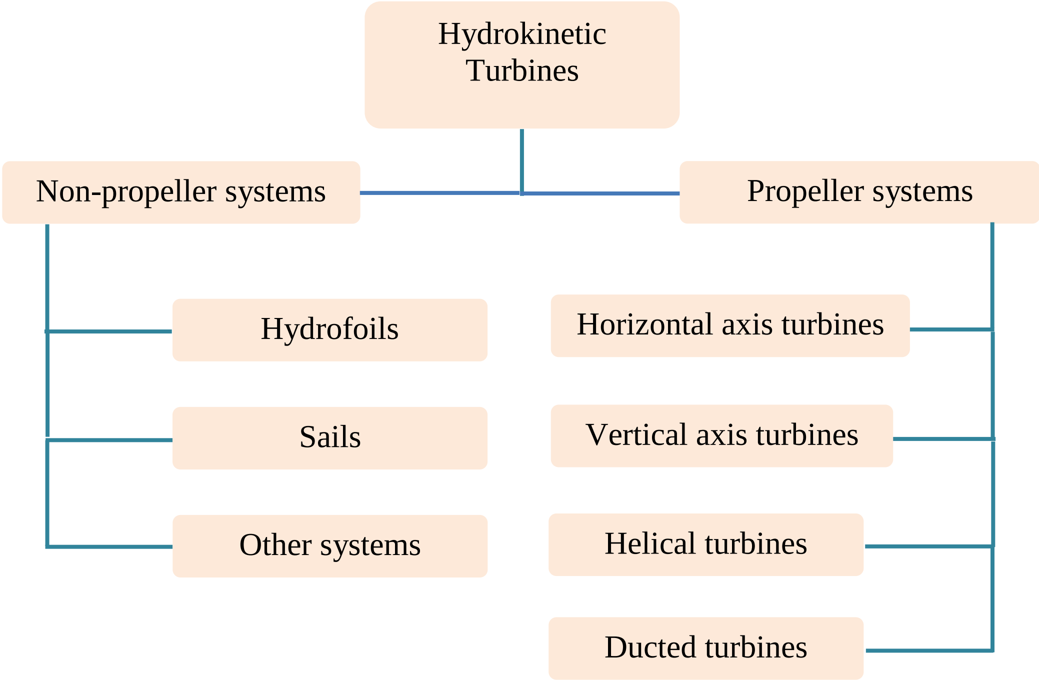
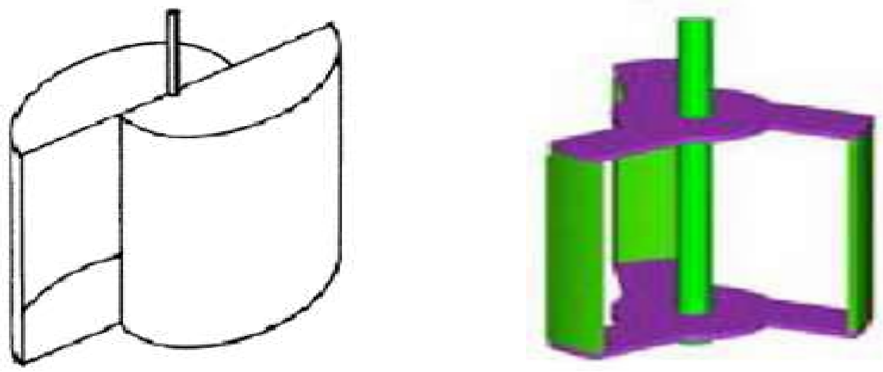


Figure 2.3 Classification of hydrokinetic turbines

a) Marine current turbines

b) Verdant power

Figure 2.4 Horizontal axis hydrokinetic turbines



a) The Exim tidal turbine b) Darrieus turbine

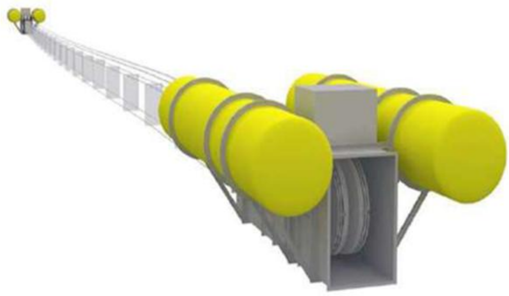
Figure 2.5 Vertical axis hydrokinetic turbines

a) GCK Gorlov helical turbine b) LET Gorlov helical turbine

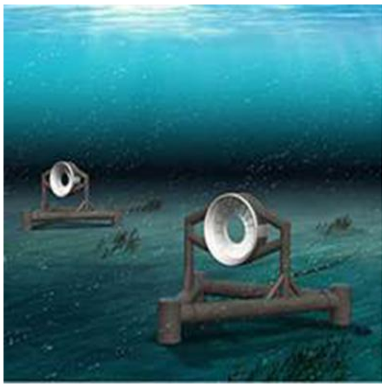
Figure 2.6 Helical turbines

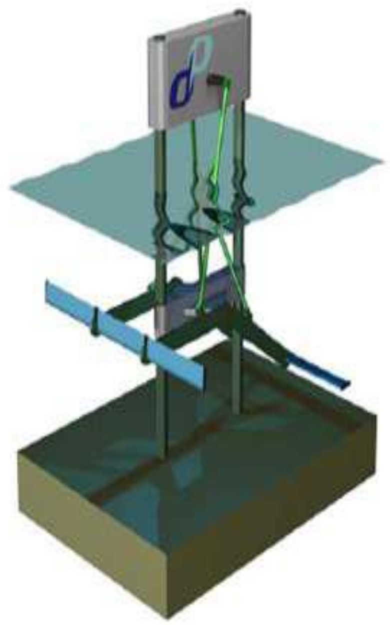
a) UEK turbine b) Shrouded turbine

Figure 2.7 Ducted hydro kinetic turbines



b) Tidal sails





a) Pulse generation Hydrofoils

c) Open hydro turbines

Figure 2.8 Non-propeller system turbines

2.6. Theory of hydrokinetic energy

Technology of hydrokinetic turbines is very close to wind turbines and uses the main principles of wind turbine methodology such as momentum theorem, Betz limit and Blade element method. Hydrokinetic turbines are industrialized from hydrofoils which produce a pressure difference between the upper and lower surfaces and experiences drag and lift forces. The maximum amount of lift force and minimum drag force is desired for the maximum performance of a turbine. Flow velocities from 1 to 3 m/s are the optimum velocities for hydrokinetic turbines (M. Amen, 2013). Each turbine type has a characteristic power curve which shows the performance of the turbine, minimum and maximum velocities to produce power. The minimum speed of a hydrokinetic turbine is called cut-in speed. The turbine idles below the cut-in speed. The maximum speed that the turbine can produce powers called the rated velocity. The rated power is the power corresponding to the rated velocity. The power amount of a turbine with different speeds can be calculated using the mathematical function of the curve between cut-in and rated speeds.

The power output of rotating current energy conversion systems is given the following equation;

$$P = \frac{1}{2}\rho A V^3 C_p \tag{2.1}$$

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.

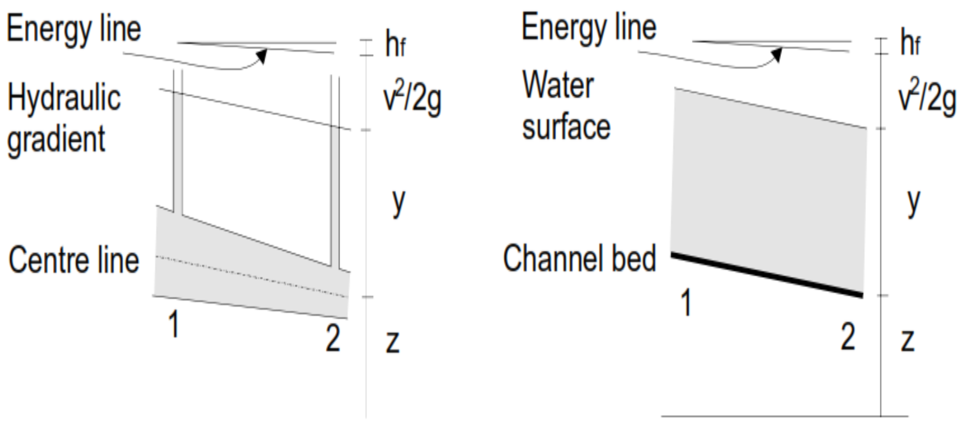
The power is proportional with the cubical power of velocity. Maintaining maximum velocity is important for maximum power. A hydrokinetic turbine can produce much more power than an equally scaled wind turbine due to the difference between the density of water and air. The theoretical maximum power coefficient 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)

CHAPTER 3

OPEN CHANNEL FLOW

3.1. Introduction

Open channel flow or pipe flow are flows of water in a conduit. Both kinds of flow are similar in many ways but differ in one important point. Free surface must be available in open channel flow and there is no free surface in pipe flow. A free surface is subject to atmospheric pressure. There is no direct atmospheric pressure in pipe flow only hydraulic pressure exists. The two kinds of flow are compared in the figure below (Goodwill and Sleigh, 2008).



a) Pipe flow b) Open chneel flow

Figure 3.1 Comparison of pipe and open channel flow

Two piezometers are placed in the pipe at sections 1 and 2. The water levels in the pipes are maintained by the pressure in the pipe at elevations represented by the hydraulics grade line or hydraulic gradient. The pressure exerted by the water in each section of the pipe is shown in the tube by the height ‘y’ of a column of water above the centre line of the pipe. The total energy of the flow of the section (with reference to a datum) is the sum of the elevation z of the pipe centre line, the piezometric head ‘*y’* and the velocity head $V^2/2g$, where V is the mean velocity. The energy is represented in the figure by what is known as the energy grade line or the energy gradient. The loss of energy that results when water flows from section 1 to section 2 is represented by ‘h’. A similar diagram for open channel flow is shown. This is simplified by assuming parallel flow with a uniform velocity distribution and that the slope of the channel is small. In this case the hydraulic gradient is the water surface as the depth of water corresponds to the piezometric height. Despite the similarity between the two kinds of flow, it is much more difficult to solve problems of flow in open channels than in pipes. Flow conditions in open channels are complicated by the position of the free surface which will change with time and space. And also by the fact that depth of flow, the discharge and the slopes of the channel bottom and of the free surface are all inter dependent (Goodwill and Sleigh, 2008).

3.2. Types of flow

Steady flow, unsteady flow, uniform flow and non-uniform flow are main types of open channel flows.

Steady and Unsteady: Flow is said to be steady if the flow parameters (velocity, depth, etc.) do not change with time. The flow is unsteady if they change with time.

Uniform Flow: Open Channel flow is said to be uniform if the velocity does not change in any section of the channel. Hence the uniform flow can only occur in prismatic channels.

3.3. Properties of open channels

3.3.1. Artificial channels

Channels which have been built by human are called artificial channels. They include irrigation canals, navigation canals, spillways, sewers, culverts and drainage ditches. They are usually constructed in a regular cross-section shape in the field they are commonly constructed of concrete, steel or earth and have the surface roughness reasonably well defined.

3.3.2. Natural channels

Natural channels, such as rivers have irregular cross-sectional areas. The surface roughness will often change with time distance and even elevation.

3.4. Geometric properties necessary for analysis

For analysis various geometric properties of channel cross-sections are required. For artificial channels these can usually be defined using simple algebraic equations given y the depth of flow. The commonly needed geometric properties are shown in Table 3.1and defined as:

Depth (*y*): The vertical distance from the lowest point of the channel section to the free surface.

Stage (*z*): The vertical distance from the free surface to an arbitrary datum

Area (*A*): The cross-sectional area of flow, normal to the direction of flow

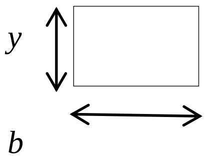
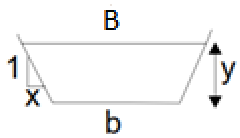
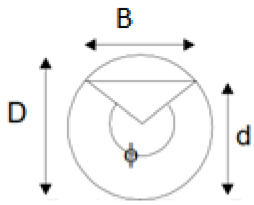
Wetted perimeter (*P*): The length of the wetted surface measured normal to the direction of flow.

Surface width (*B*): Width of the channel section at the free surface

Hydraulic radius (*R*): The ratio of area to wetted perimeter (*A/P*)

Hydraulic mean depth (D_m): The ratio of area to surface width (*A/B*)

Table 3.1 Equations for rectangular, trapezoidal and circular channels.

	Rectangle	Trapezoidal	Circle
			
Area, A	*by*	*(b+xy)y*	$\frac{1}{8}$(Ø-sinØ)D^2
Wetted perimeter, P	*b+2y*	$b + 2y\sqrt{1 + x^2}$	$\frac{1}{2}$ØD
Top width, B	*B*	*b+2xy*	(sinØ/2)*D*
Hydraulic radius, R	*By/(b+2y)*	$\frac{(b + xy)y}{b + 2y\sqrt{(1 + x^2)}}$	$\frac{1}{4}\left[1 - \frac{sinØ}{Ø}\right]D$
Hydraulic mean depth, D_m	*Y*	$\frac{(b + xy)y}{b + 2xy}$	$\frac{1}{8}\left[\frac{Ø - sinØ}{\sin(\frac{1}{2Ø})}\right]D$

3.5. Velocity profile in open channel flow

In open channel flow, the velocity is not constant with depth. It increases from zero at the bed of the channel to a maximum value close to the water surface. The velocity difference results from the resistance to flow at the bottom and sides of the channel. Figure 3.2 displays how the velocity increases from zero at the bed to the maximum value near the free surface.

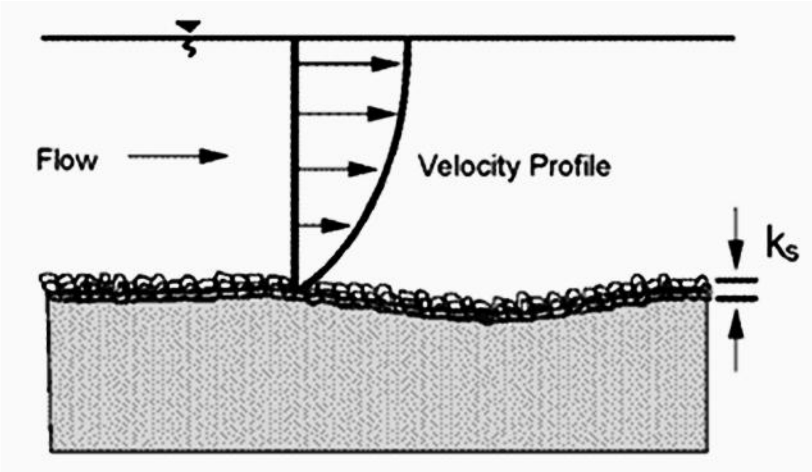


Figure 3.2 Velocity profile in open channel flow

CHAPTER 4

COMPUTATIONAL FLUID DYNAMICS (CFD)

4.1. Definition and history of CFD

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing researches yields software that improves the accuracy and speed of complex simulation scenarios such as turbulent flows (Versteeg and Malalasekera, 2007). The fundamental bases of almost all CFD problems are the Navier–Stokes equations, which define any single-phase fluid flow.

4.2. Applications of CFD

The first adopters of CFD were the automotive, aerospace and nuclear industries (Bakker et al., 2001). More development and growth of CFD and its capability to model complicated phenomena over with speedy increase in power of computer have continuously expanded the range of implementation of CFD. CFD is implemented in wide domain industries such as power, biomedical, mechanical, process, petroleum, metallurgical, pharmaceutical and food industries. CFD techniques have been implemented in a large scale in the industry procedure of vision into different flow phenomena, check different designs of equipment drawn an analogy performance under various conditions.

4.3. Advantages of CFD

Although the use of CFD may have a number of pitfalls, which can mostly be reduced to the user's inexperience and are therefore not fundamental, these pitfalls are far overweighed by its benefits (Bakker et al., 2001). There are situations, however, where complete fundamental knowledge of all the underlying physics may not exist, thus leading to inherent assumptions in the mathematical model adopted which give rise to possible inaccuracy. Nonetheless, CFD enjoys a number of advantages which contribute to the growing application of general-purpose CFD codes, including the following:

(i) Capability of study systems where controlled experiments are not feasible.

(ii)The data range that proved the experiments may sometimes be restricted because of technique or equipment limitation, CFD can provide a big range of inclusive data where such restrictions are usually appear.

(iii) The complicated physical interactions which happen in flow situation can be paradigm in the same time since no restricting assumptions are usually needed.

(iv) CFD can provide over all flow ideation. In many industrial applications CFD is more widely applied as a flow ideation than a score of absolute quantitative data.

4.4. Outline of CFD process

Computational Fluid Dynamic codes are constructed around numerical algorithms that can solve fluid flow problems. All CFD codes which are ready in the market have three basal elements which divide the entire analysis of the numerical experiment to be used on the specific domain or geometry. The three basal elements are the Pre-processor, the Solver and the Post-Processor.

4.4.1. Meshing and pre-processing

The primary treatment of the CFD process consists of the input of a flow problem by methods of user-friendly programs or software and the following transformation of this input into a style is made suitable to use by the solvers. The primary treatment is the connection between the user and the solver. The user activity at the primary treatment stage of the CFD process includes the following:

1) *Introduction to Geometry or region of Interest*: this process deals with many computer aided design (CAD) software like ANSYS, CATIA, Solid works, Pro-E and much more. By the use of CAD software, the structure and the topology of the fluid flow region of benefit is defined. This software takes a main part of the design and optimization process in research analysis.

2) *Grid Generation or Meshing*: As the CFD process is a numerical parataxis way of using finite volume method, the given field or region of interest needs to be split into several structured elements. All the elements or cells are linked to each other by nodes to form the required region of flow. For this aim, special meshing or grid descent software are used. This point is the key element in the CFD finite volume numerical simulation and it also contributes to the accuracy of the final results.

3) *Definition of Fluid properties*: Each fluid region or surface has its own distinguished property. The properties of the fluid used in the CFD region are defined at this point of the CFD process. Generally the CFD code has this facility.

4) *Boundary Conditions*: Each different system of the CFD range needs to have a configuration, which is achieved by the boundary conditions input. The CFD code generally easiness to define the boundary conditions of the CFD problem, where each cells at particular limit are given finite values.

4.4.2. Numerical solver

The numerical solver is the main element of the CFD process and contains the major part of the CFD process. In the present market, the solvers generally use three distinguished ways to calculate the solutions, which are by names, the finite difference method, finite element method and the finite volume method.

The finite difference and finite element method are generally appropriate for stress and structure analysis and it is not appropriate for the requirements of the CFD process. The finite volume method is the most appropriate method for the CFD process. Where the name denotes, finite volume method is the calculation process of numerical algorithm dealing with the use of finite volume cells. The steps implicated in this solving process are generally executed in the following sequence;

1) Official integration of the controlling equations of fluid flow on the entire controlled volumes or finite volumes of the solution range.

2) The transformation of the whole forms of the equations into a system of algebraic equations.

3) Computation of the algebraic equations by a repeated method.

4.4.3. Post processing

The added treatment is the last stage of the CFD process which involves data conception and results analysis of the CFD process. This stage uses the versatile data conception tools of the CFD solver to present range geometry, grid presentation, vector plots, line plots, shaded contour plots, 2-D & 3-D surface plots, particle tracking, XY plots and relevant graphs of results

4.5. Turbulence

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

$$R_e = \frac{\rho V L}{\mu} \qquad (4.1)$$

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

For open channel there is no clear limit at which Re the turbulence develops in flow but it usually taken more than 2000 (Sam, 2010).A view of turbulence drawn by Da Vinci is given in Figure 4.1.

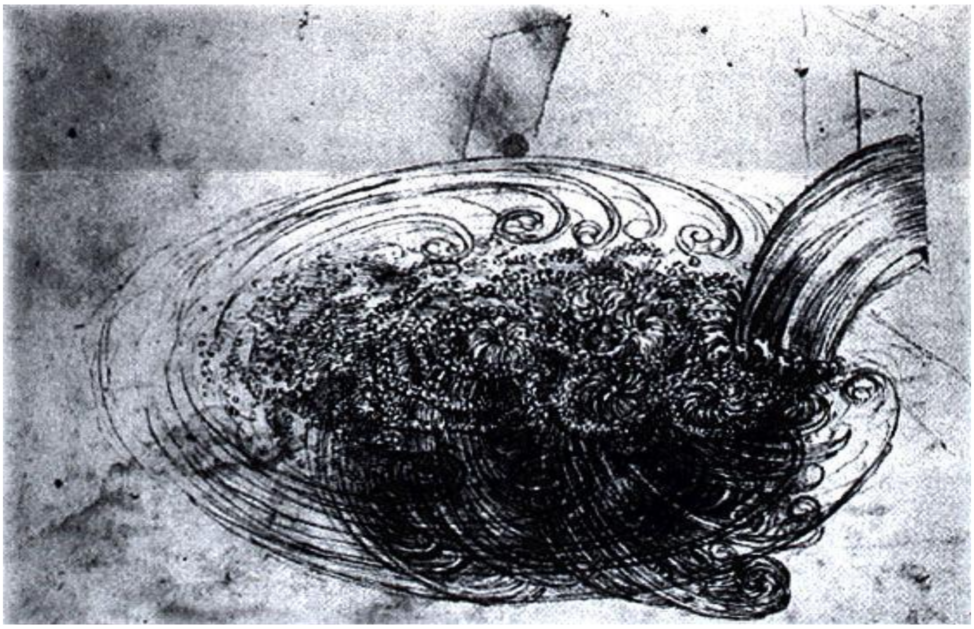


Figure 4.1 A view of turbulence (Da Vinci)

4.6. Turbulence Models in CFD

The turbulence models are the computational process that is used to simplify the solution of governing equations in an engineering application especially for mechanical processes. They are needed to represent scales of the flow that are not resolved. Classification of Reynolds-Averaged Navier-Stokes turbulence models are shown in Figure 4.2.

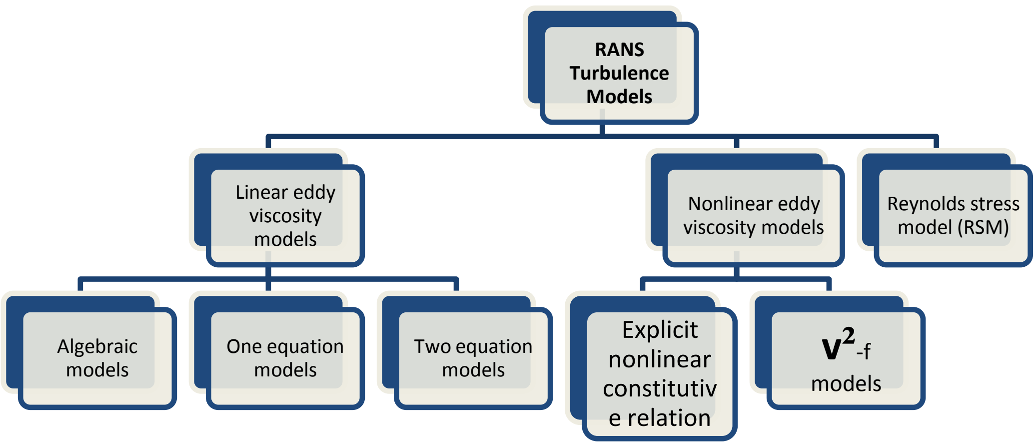


Figure 4.2 Classification turbulence models (CFD-online)

4.7. 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 modelling 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).

$$\nabla.\vec{V} = 0$$

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

$$\Sigma\vec{F}_B + \Sigma\vec{F}_S = m\vec{a}$$

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

4.7.1. Linear eddy viscosity models

These are turbulence models in which the Reynolds stressesare modeled by a linear constitutive relationship with the mean flow straining field.

There are several subcategories of linear eddy-viscosity models, depending on the number of transport equations solved to compute the eddy viscosity coefficient, which are algebraic models, one equation models and two equation models.

4.7.1.1. Algebraic models (Zero equation models)

Algebraic turbulence models or zero-equation turbulence models do not require the solution of any additional equations. Algebraic models are calculated directly from the flow variables. These models are easy to implement, fast in calculation times and provide good predictions for simple flows. However they are completely incapable of describing flows where the turbulent length scale varies anything with separation or circulation. These models calculate only mean flow properties and turbulence shear stress. Cebeci-Smith, Baldwin-Lomax and Johnson-King are the most known algebraic models

4.7.1.2. One equation models

These models are only solving one turbulent transport equation, which is usually the turbulent kinetic energy. The original one-equation model is Prandtl's one-equation model. Other common one-equation models are: Baldwin-Barth Spalart-Allmaras and Rahman-Agarwal-Siikonen Models.

4.7.1.3. Two equation turbulence models

One of the most common types of turbulence models is two equation turbulence models. Two equation turbulence models are; k-epsilon models, k-omega models. Two equation turbulence models have become industry standard models and are commonly used for most types of engineering problems. Two equation models are still being developed (Bredberg, 2000).

4.7.1.3.1. *k*-ε models

One of the most commonly used turbulence models is the k-ε model. The k-ε model is a two equation model that means it includes two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy. Although it just doesn't carry out well in cases of large adverse pressure gradients. The first transported variable is turbulent kinetic energy, *k* and the second transported variable is the turbulent dissipation, ε.

The terms *k* and ε refer to the turbulence kinetic energy which are the variance of fluctuations in velocity. $\mathcal{E}$ is the dissipation rate of *k*. Turbulence eddy dissipation is equal to the viscosity multiplied by the fluctuating vortices. An exact transport equation for the fluctuating vortices defined as the rate of dissipation of velocity fluctuations (Sarvan Mamaidi, 2009). k-ε model is simple to implement, leads to stable calculation and reasonable predictions for many flows. It is poor in predictions for swirling and rotating flows, flows with strong separation and axis symmetric jets. Well known k-$\mathcal{E}$ models are; Standard k-ε, Realisable k-ε and RNG k-ε models.

4.7.1.3.2. k-ω models

The first transported variable is turbulent kinetic energy, *k*. The second transported variable in this case is the specific dissipation, ω. It is the variable that determines the scale of the turbulence, whereas the first variable, *k* determines the energy in the turbulence. The most known k-ω models are; Wilcox's k-ω, Wilcox's modified k-ω and SST k-ω models.

4.7.2. Nonlinear eddy viscosity models

There are two models in nonlinear eddy viscosity models which are:

1. Explicit nonlinear constitutive relation: these models are cubic k-ε models and explicit algebraic Reynolds stress models (EARSM).

2. V^2-f models.

4.7.3. Reynolds stress model (RSM)

The higher level of elaborate turbulence model is Reynolds stress model (RSM). It is usually called a Second Order Closure. In RSM, the eddy viscosity approach has been discarded and the Reynolds stresses are directly computed. The exact Reynolds stress transport equation accounts for the directional effects of the Reynolds stress fields (Goldstein, 2004).

4.8. Large Eddy Simulation (LES)

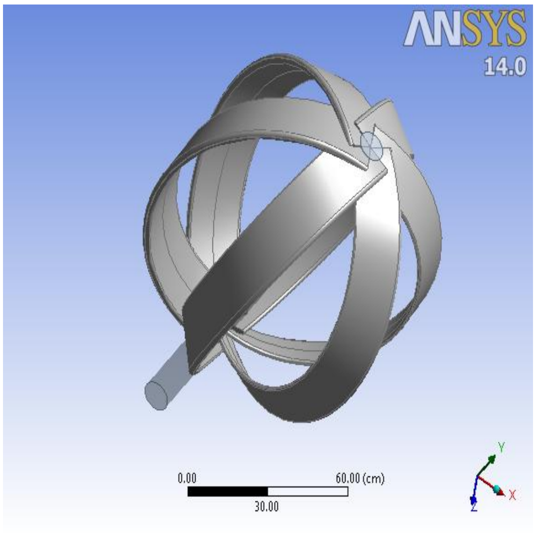
Large-eddy simulations (LES) differ from the other fluid flow computation methods. In this simulation method, the largest eddies are explicitly resolved and the smaller eddies are modelled. Modelling the ocean and atmosphere inspired the first large-eddy simulations, but true large-eddy simulations of 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 5

SIMULATION OF HYDROKINETIC TURBINES

5.1. Introduction

This chapter contains the simulation of flow field around a modified Lucid Energy Technology (LET) Gorlov Helical Turbine (GHT) (Figure 5.1) by CFD codes using ANSYS FLUENT 14.0 software. To simulate the model for getting results close to the experiment results, the physical and chemical properties of the fluid should be appointed and the model should be gridded to small parts.



a) Modified LET GHT

b) Original LET GHT

Figure 5.1 Modified LET GHT and the original LET GHT

5.2. Geometry creation

In this section the whole domain which includes the rectangular open channel of 50m in length 3m in width and 2.5m in depth and the modified LETGHT positioned inside the channel by using work bench ANSYS FLUENT design modeller. Then the domain has to be meshed. Figures 5.2 – 5.6 show the simulation domain, the position of the turbine in the open channel and blades of the turbine.

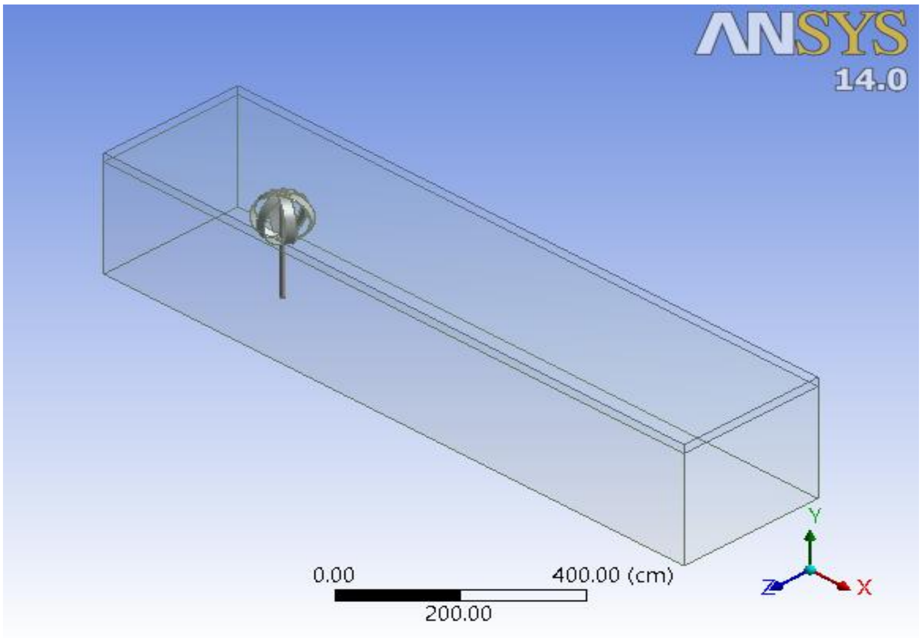


Figure 5.2 The whole simulation domain

The model formed in Design Modeller. Geometry of the domain was created in two steps. First, the open channel model with rectangular cross-section was formed. The top surface of channel represents the free surface of the open channel flow, while the bottom represents the channel bed. Second, the hydrokinetic turbine which is a modified LET Gorlov Helical turbine with six curved rotor blades and a circular shaft in the middle was produced.

The turbine design includes six circular blades; each blade has a semi-circular shape. Blades are bended to create a spherical shape. Figure 5.3showsthe shape and the cross-section of each blade, while Figure 5.4 demonstrates the circular shape of the turbine with circular shaft in the middle.

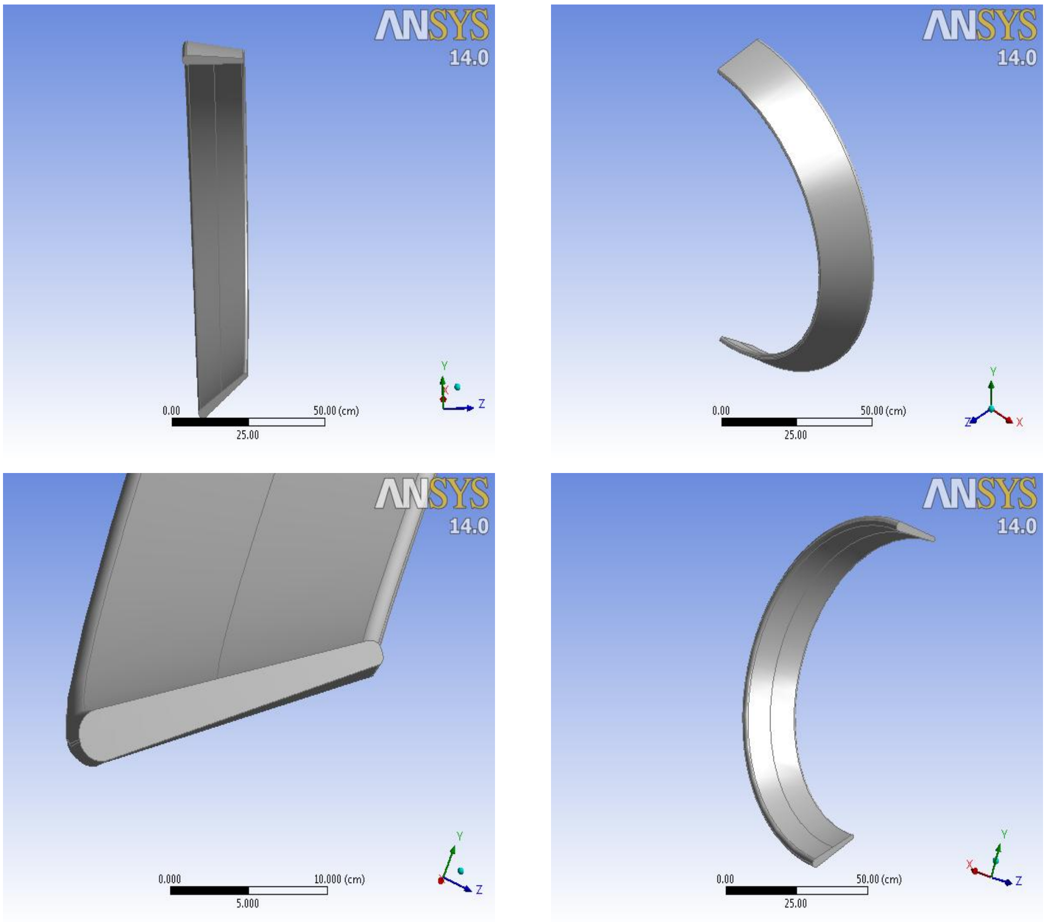


Figure 5.3 Section and shape of the modified LET GHT's blade

The diameter of the spherical turbine is 1m, whereas the diameter of the shaft or axis rotation is 0.1m with a 2.4m length. The front view and the top view of the turbine in the flow domain are given in Figures 5.4, and 5.5. The position of the turbine inside the open channel is seen in Figure 5.6. The dimensions of the channel and turbine are given in Table 5.1 and 5.2 respectively.

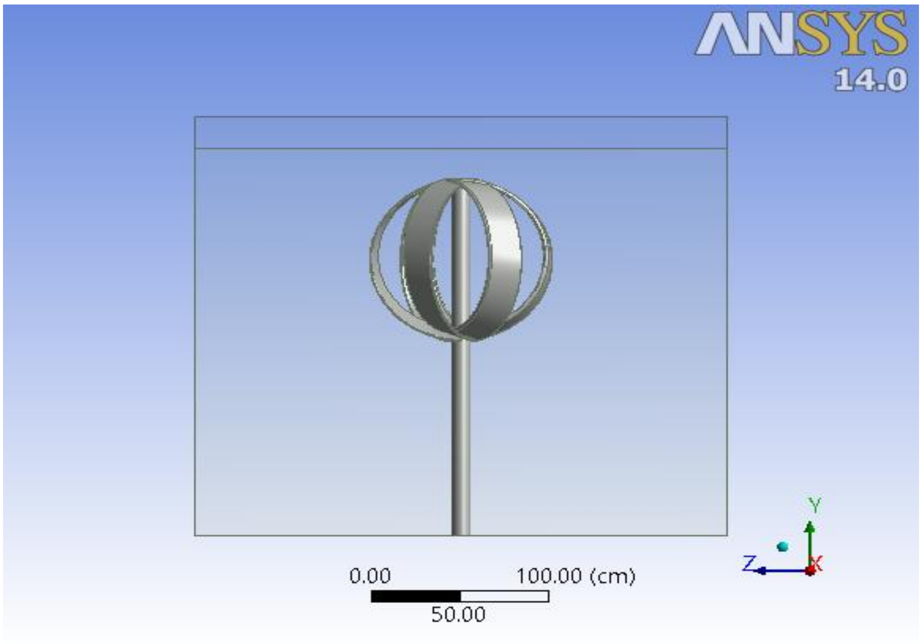


Figure 5.4 Front view of the domain

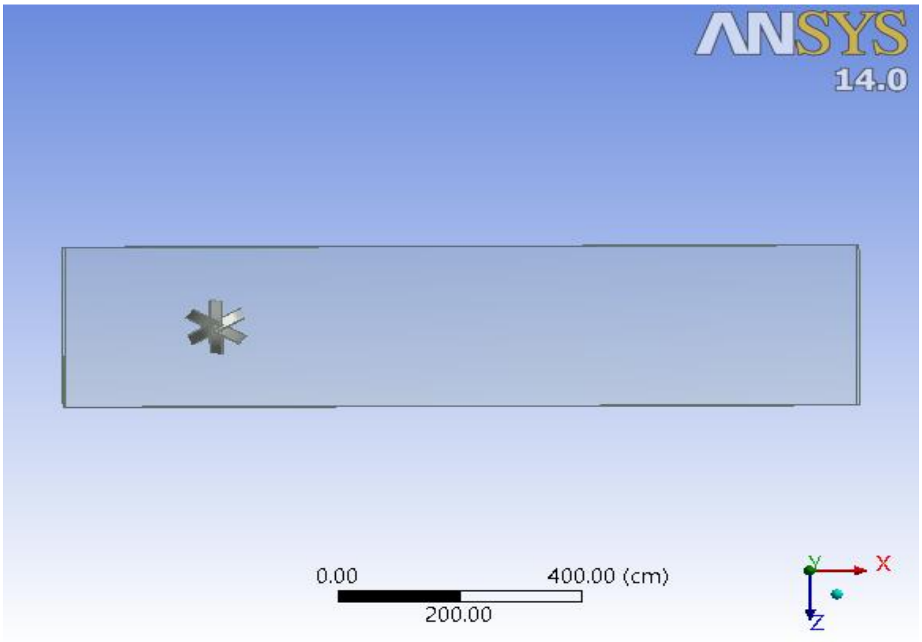


Figure 5.5 Top view of the domain

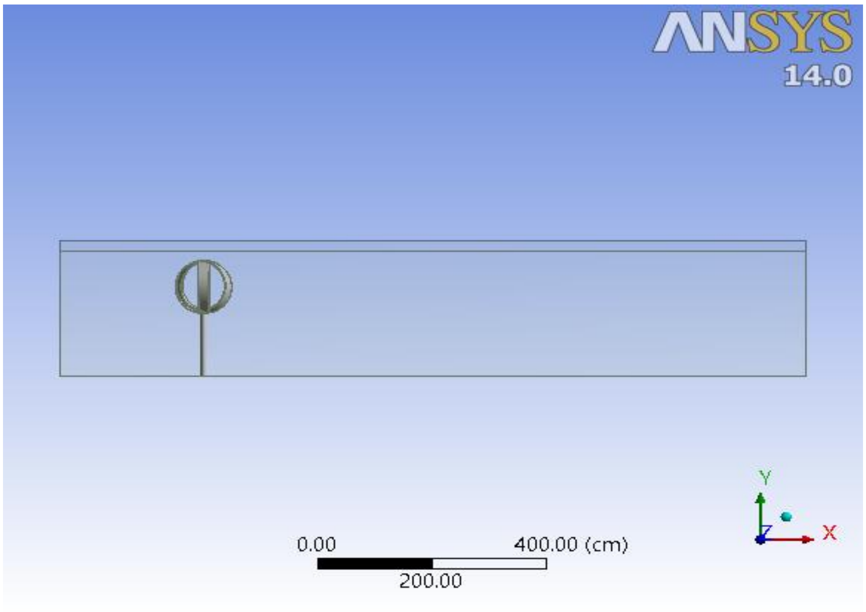


Figure 5.6 The position of the turbine in the open channel

Table 5.1 Channel dimensions

Dimension	**Length(m)**
Length	50
Width	3
Height	2.5

Table 5.2 Dimensions of the turbine

Description	Dimension
Diameter of turbine	1.0m
Diameter of shaft	0.10m
Number of blades	6
Length of shaft	2.40m
Shape of blades	Circular

5.3. Mesh generation

After creating the geometry of the model and defining it, the domain should be divided into a number of grids for accuracy in calculations and controlling the results. This operation is called meshing. On the other hand, determining the boundary conditions, such as, walls, surfaces, inlet, outlet, and turbine in the open channel, enables to achieve accurate results at the boundary regions (Figure 5.7).

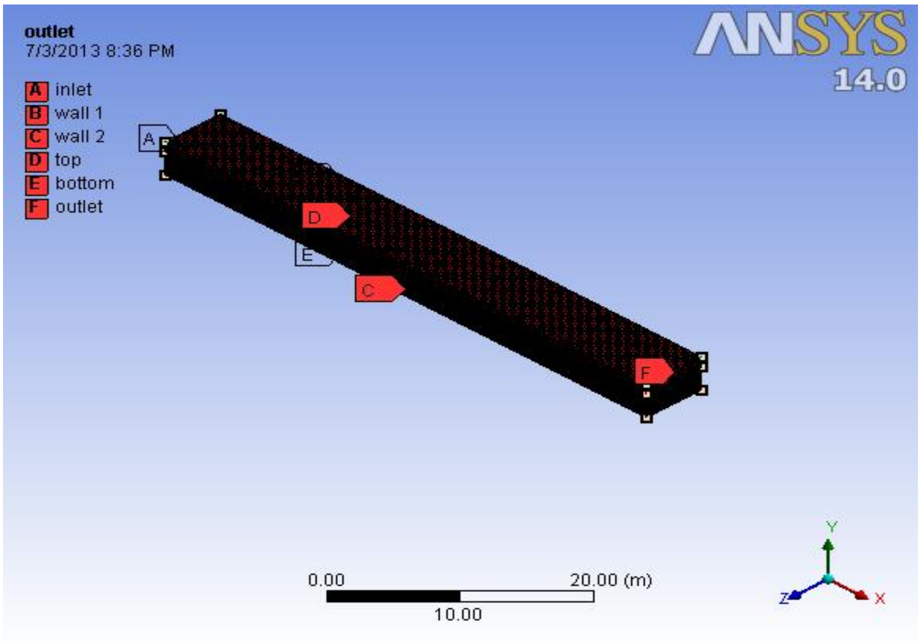


Figure 5.7 The boundary conditions

Figures 5.8–5.11 illustrates meshing of the turbine and the whole domain. The mesh should be examined with care and if it is necessary it should be refined especially on the rotor blade surface. It should be highly smoothed at the free surface and at the open channel bed.

The final step of the meshing is selection the name of each boundary condition such as bed, side walls and the surface of the channel and the turbine. Each boundary condition in the model should be defined. In this model, the bed of the open channel was defined as bottom, sides were specified as the walls or symmetries, the water level in the open channel flow was defined as surface, and the entrance and exit of the water were defined as inlet and outlet. Finally the turbine was specified as to be a rotary domain. Before entering the SETUP process, the mesh should be closed and updated.

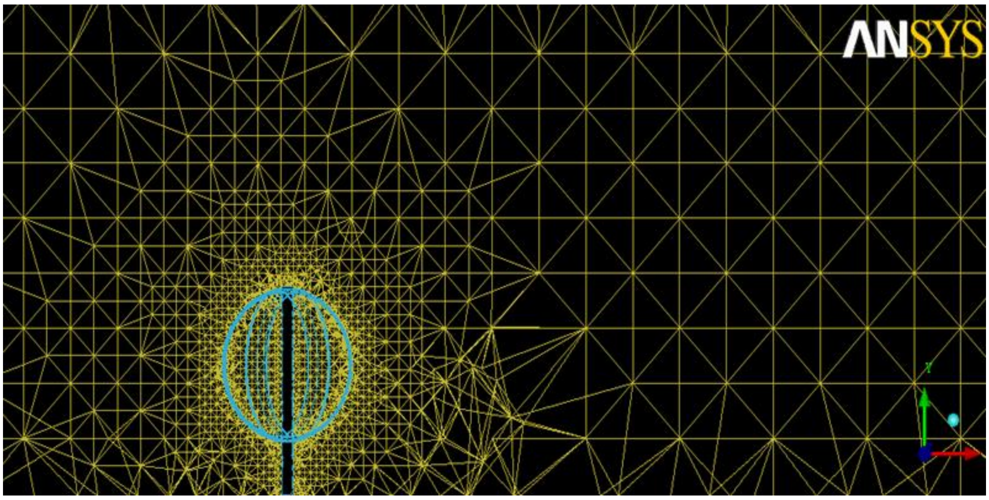


Figure 5.8 Meshing around the turbine

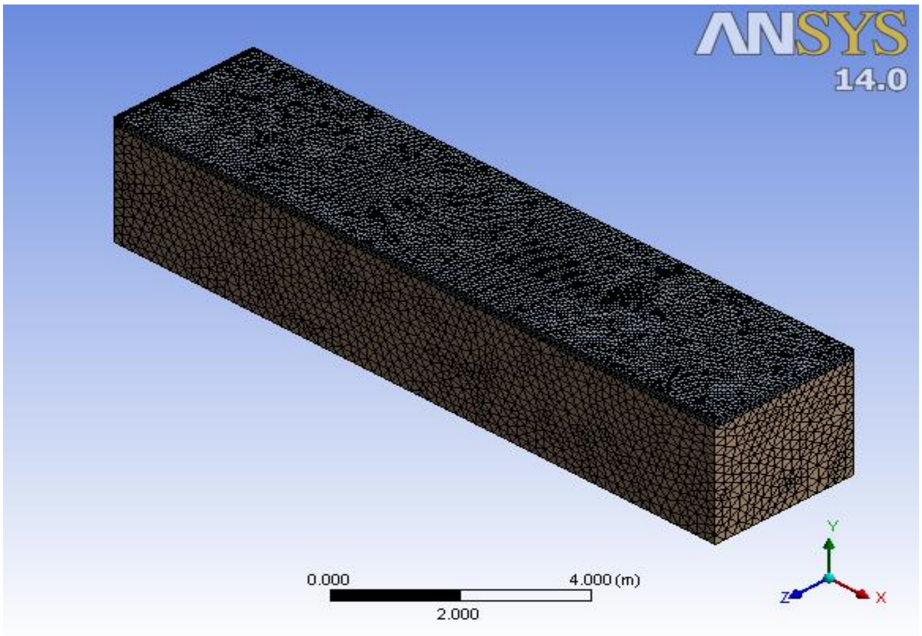


Figure 5.9 The whole domain mesh

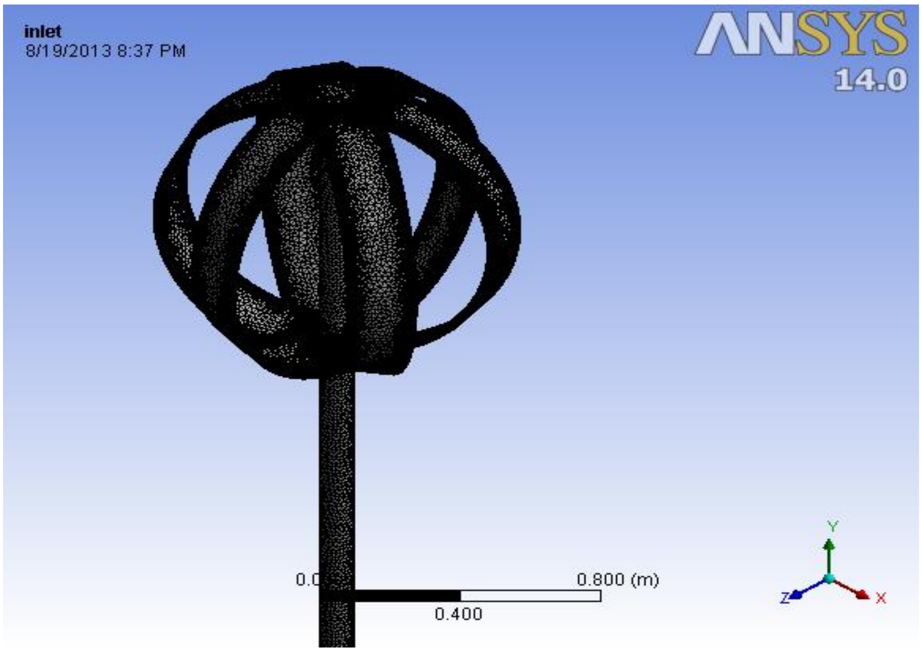


Figure 5.10 Turbine mesh

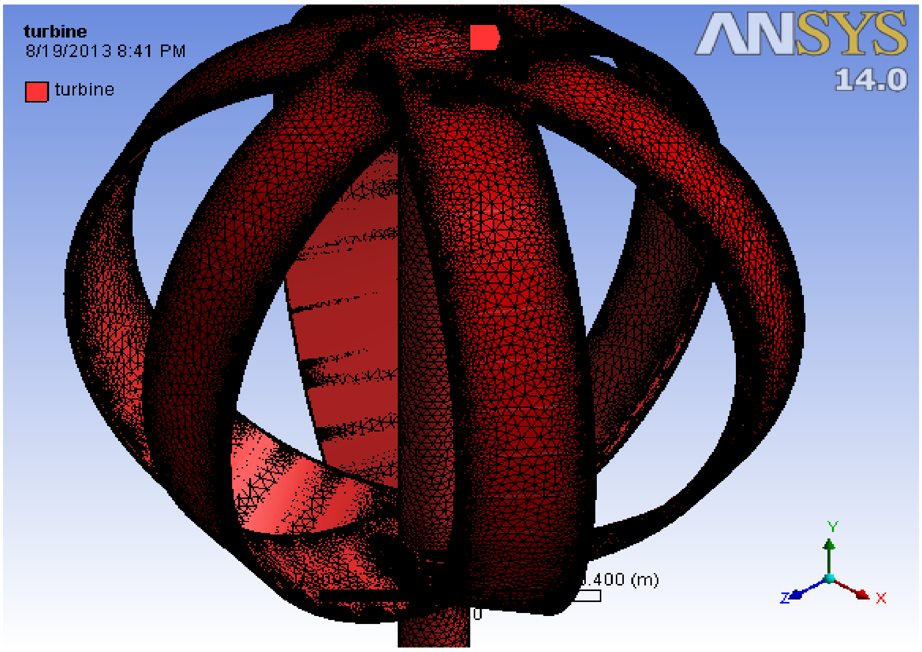


Figure 5.11 Mesh of blades

5.4. Numerical Simulation

Numerical simulation is inputting the whole domain with its geometry, meshing and boundary conditions into the FLUENT software. Then the quality of the mesh will be checked. Finally the physical properties and fluid properties will be specified. The steps to solving and simulating the model are defined in following parts.

5.4.1. Physical properties

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

5.4.2. Defining the fluid properties

The fluid was selected as liquid water and necessary changes for viscosity, density, gravitational acceleration, etc. were introduced. The air was defined for the free surface in order that to separate two different phases of matter as air and water. The fluid properties (Table 6.3) were inputted to the software.

Table 5.3 Properties of open channel

Property	Value
Density	998.2 kg/m^3
Viscosity	0.001002Pa s
Temperature	20c

5.4.3. Specification of boundary conditions

In order to satisfy the free surface open channel conditions a two phase flow was identified, one being water and the other air with atmospheric pressure. The inlet to the channel was chosen to be velocity inlet and the outlet was chosen to be pressure outlet where pressure is 1 atmosphere. The channel bottom and sides were selected to be walls while the turbine was chosen to be a stationary domain (Table 5.4).

Table 5.4 The boundary conditions

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

5.4.4. Initialization and solving the model

In the simulations the realizable k-ε turbulence model was selected as the closure. To initialize the solution the inlet was taken as the reference. The active reference frame was chosen relative to the cell zone. The input velocity in the x-direction was taking as 3m/s. The number of iterations was set to1200. The diagram which shows residuals vs. number of iterations is given in Figure 5.12.

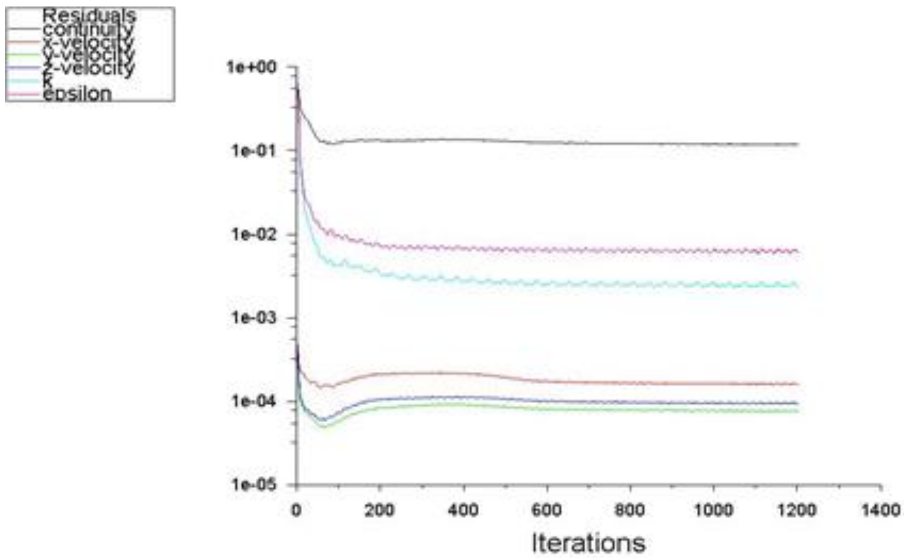


Figure 5.12 The residuals versus number of iterations

5.5. Result and validation

Reports of the results either in graphs, charts, tables or any other available formats could be obtained at the last stage of the simulation. Representations could be in vector, contour, streamline or other visualization formats. The post-processing results are given in Chapter 6.

CHAPTER 6

RESULTS AND DISCUSSIONS

6.1. Introduction

Computational Fluid Dynamics (CFD) codes and simulations have shown many reasonable results and detailed visualizations which economically cost a lot less than experimental works. The time and the effort that numerical simulation studies need compare to experimental studies are insignificant. In some flow conditions the results produced by CFD simulations are almost as accurate as experimental or field studies. In some scientific investigation before conducting any experimental studies, performing CFD simulations will help reduce the number of experiments thus reduce time, effort and money spent. This chapter deals with three different cases of Hydrokinetic turbine simulations. In the first case the turbine was positioned at about 1.4m above the channel bed, in the second case the turbine was positioned at 0.1m above the channel bed. In the third case two turbines were positioned one after another one at the same water depth as the first case at the centre line of the flume. Comparison of the first to cases will reveal the influence of the position of the turbine on any scouring which may take place at the bottom of a natural channel. The results of the last case are expected to show how far one turbine could be placed from another one in order to harness the maximum amount of energy from a flowing stream. In all three cases the velocity of the flow was set to be 3m/s.

6.2. Case I:

Installing modified LETGHT at 1.4m above bed of channel is shown in Figure 6.1.

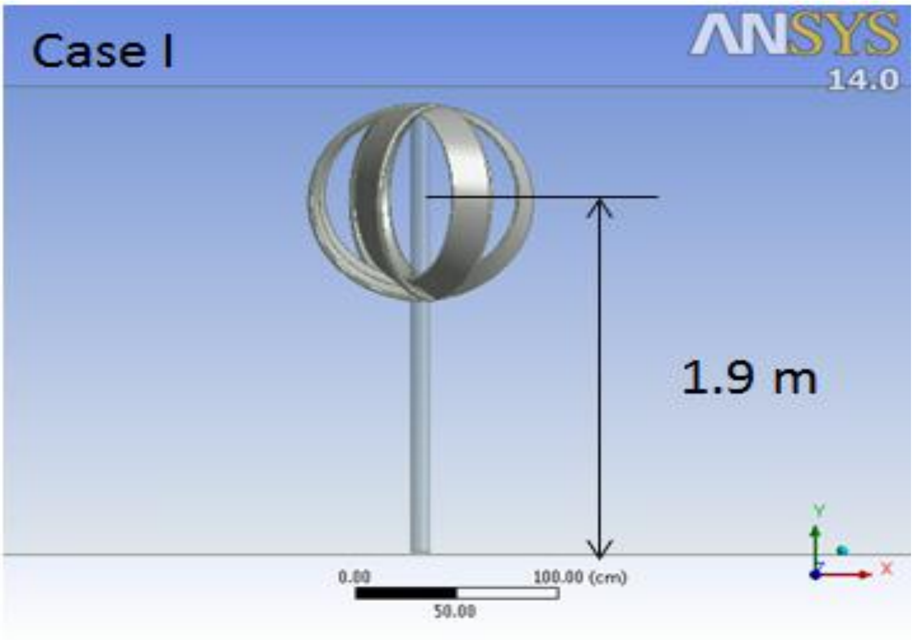
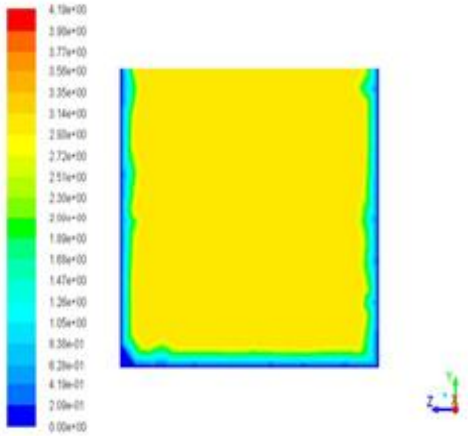


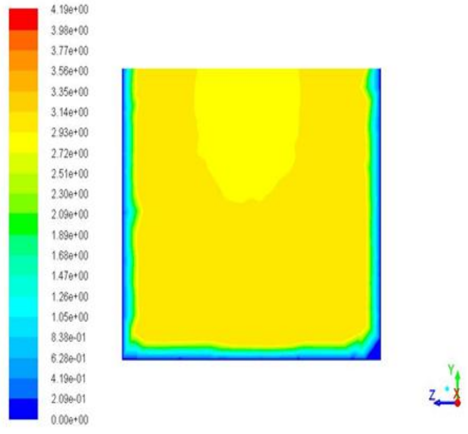
Figure 6.1 Installing modified LETGHT at 1.4m above bed of channel

6.2.1. Velocity distributions

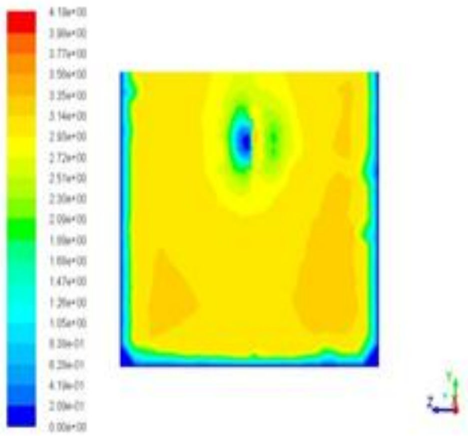
Velocity of the flow in the channel is changed in some sections. At the beginning the velocity is almost constant which is about 3m/s, after 6m the velocity reduces due to the effect of the turbine, the velocity decreases gradually until reaching the turbine. At around of the turbine the velocity has its minimum value where it is about 0m/s, because the turbine works as an obstacle. Here the flow changes its path; it moves in every direction to find a suitable path. The velocity gradually increases again at the downstream of the turbine and it reaches 3m/s about 17-20m downstream of the turbine. Figures 6.2-6.6 show the changes of velocity in different cross-sections of the open channel.



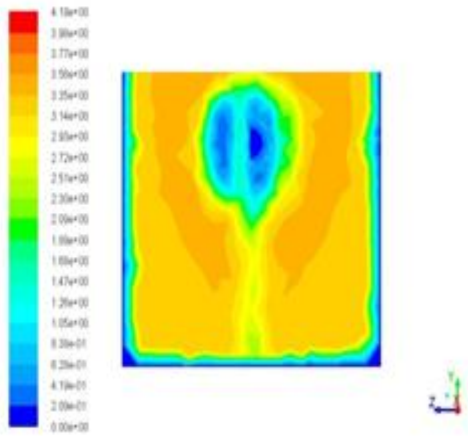
a) 5m upstream of the turbine



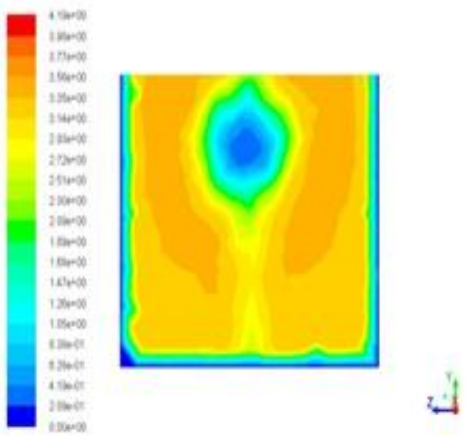
b) 1m upstream of the turbine



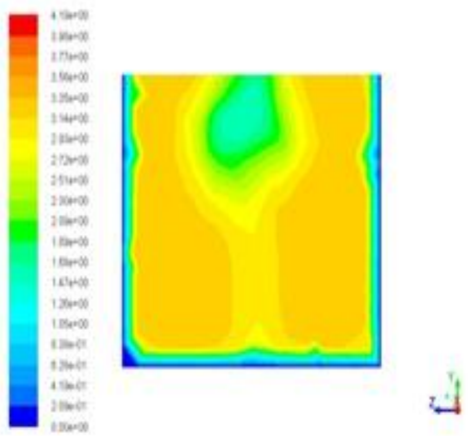
c) 0.5m upstream of the turbine



d) 0.5m downstream of the turbine



e) 1m downstream of the turbine



f) 5m downstream of the turbine

Figure 6.2 Velocity contours at different cross sections

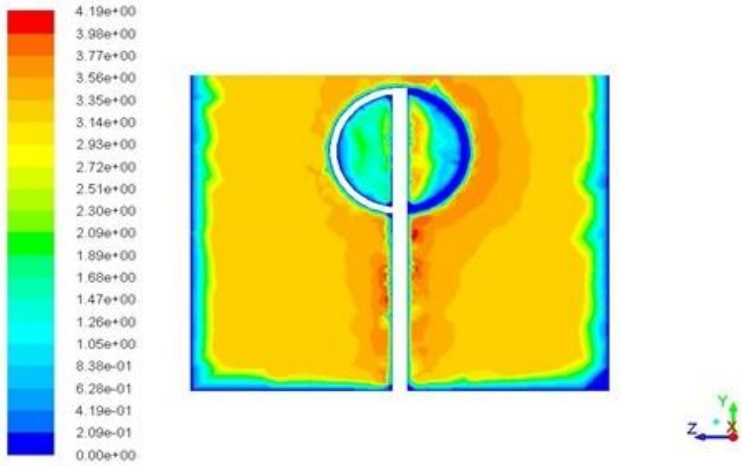


Figure 6.3 Velocity contours at the turbine

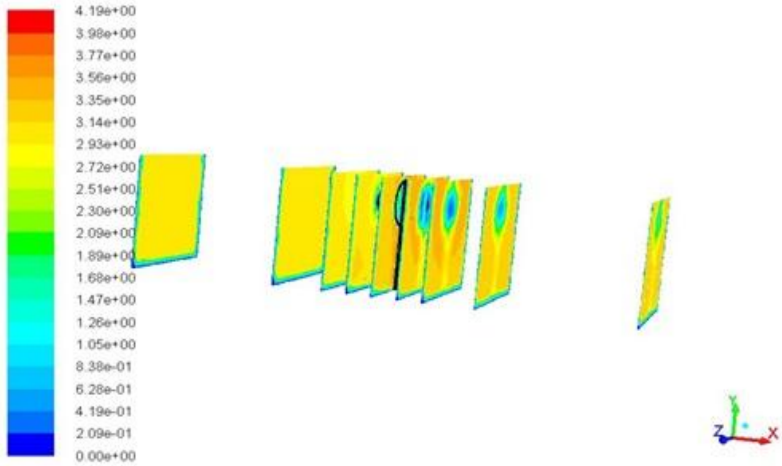


Figure 6.4 Velocity contours at all different sections

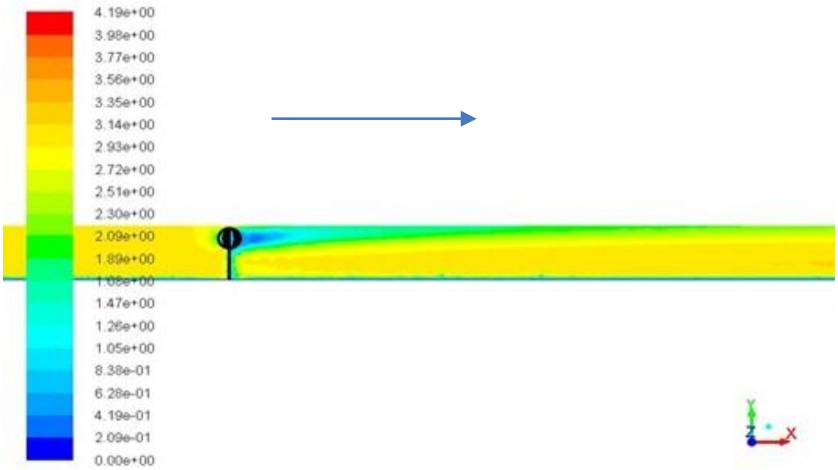
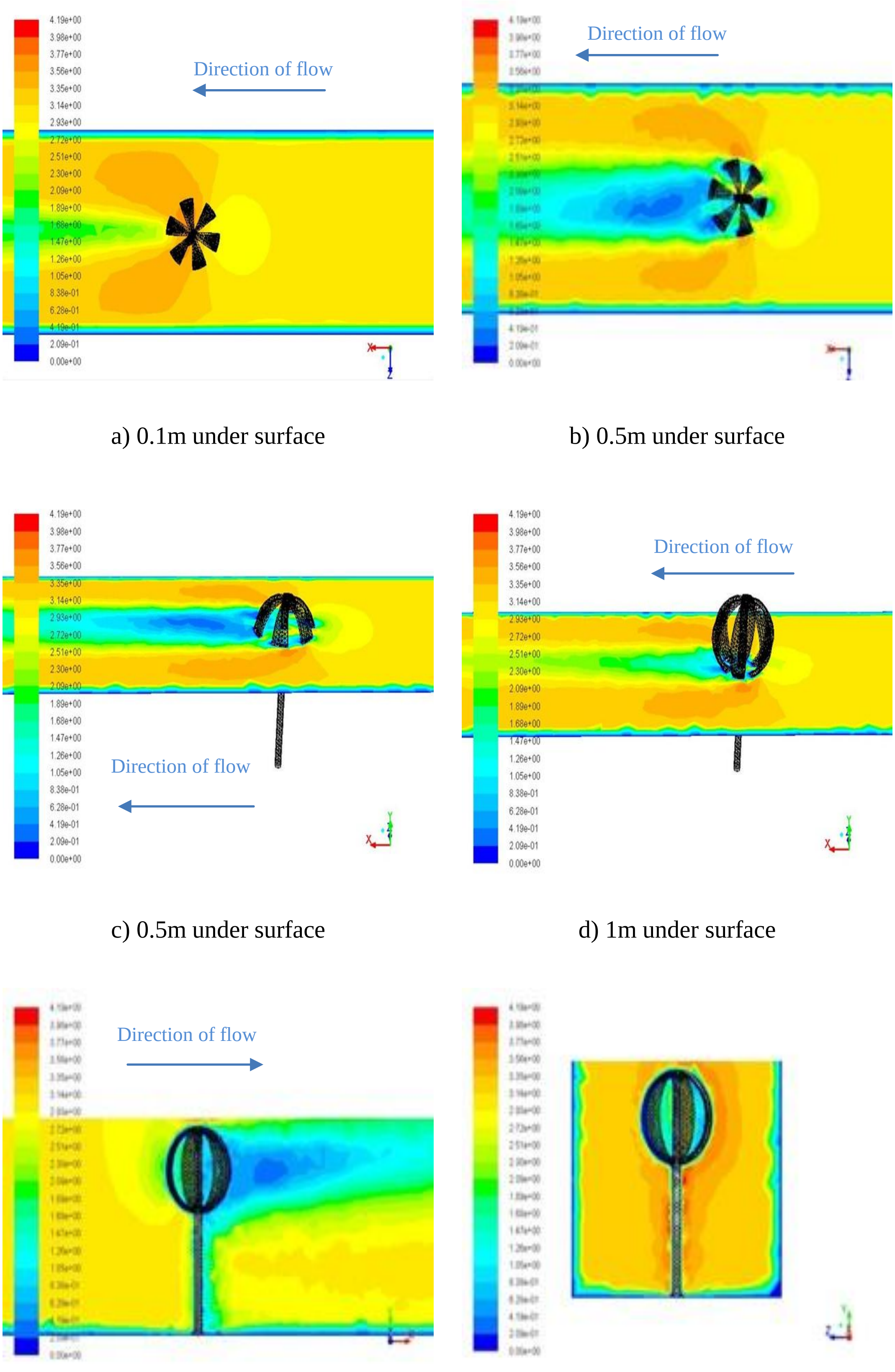


Figure 6.5 Velocity contours in longitudinal cross-section of the channel



a) 0.1m under surface

b) 0.5m under surface

c) 0.5m under surface

d) 1m under surface

e) Longitudinal section

f) Cross-section at the middle of turbine

Figure 6.6 Velocity Contours in some sections of the channel

6.2.2. Velocity vectors

Velocity vectors are the directions of particles of water in the channel. When the flow reaches to the turbine some of the vectors go in all directions due to the reduction of magnitude of the velocity, after turbine vortex takes place and some vectors goes opposite direction which is called recirculating flow. The effect of vortex reduced after turbine by about 7m. Figures 6.7-6.9 display velocity vectors at different cross-sections of the channel.

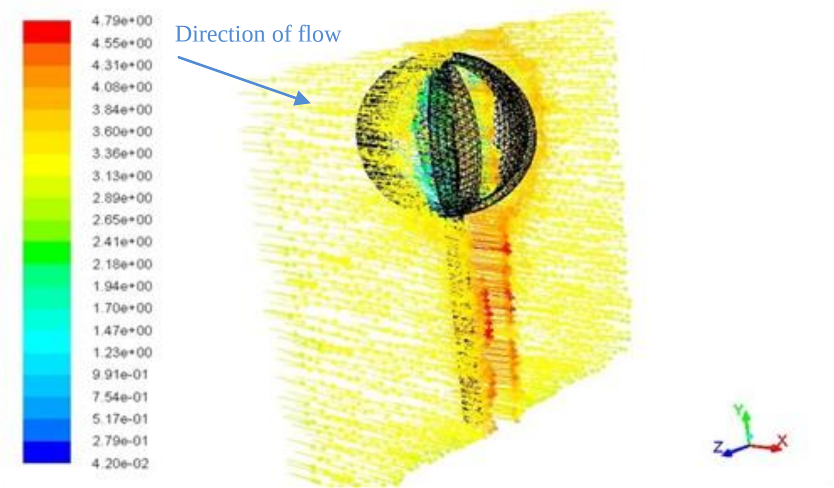


Figure 6.7 Vertical cross-section view of velocity vectors at the turbine

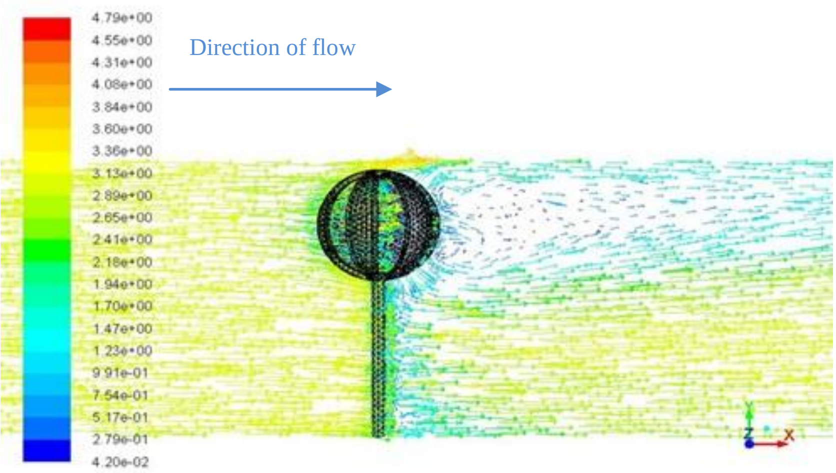
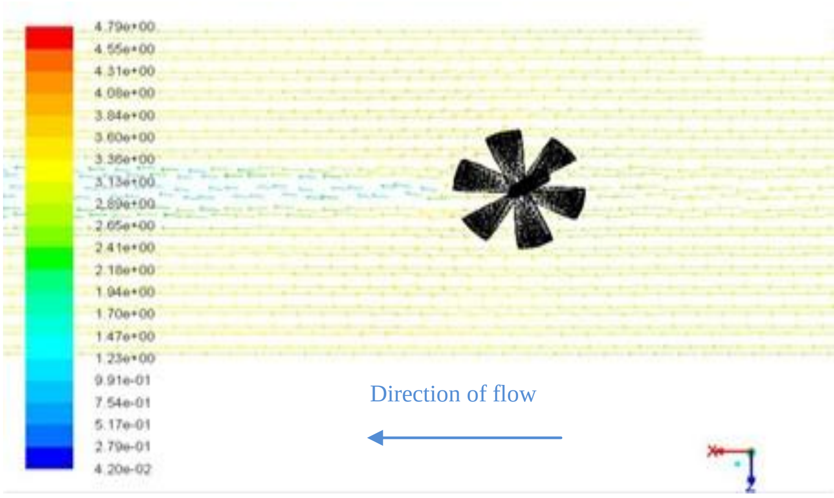
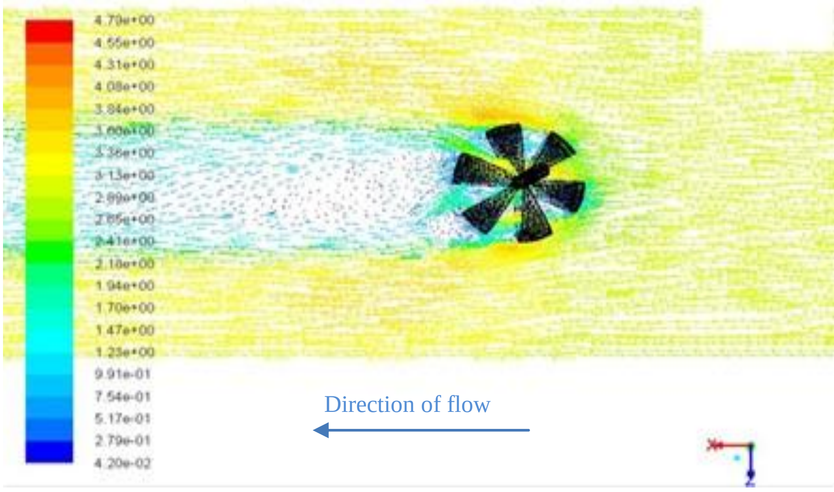


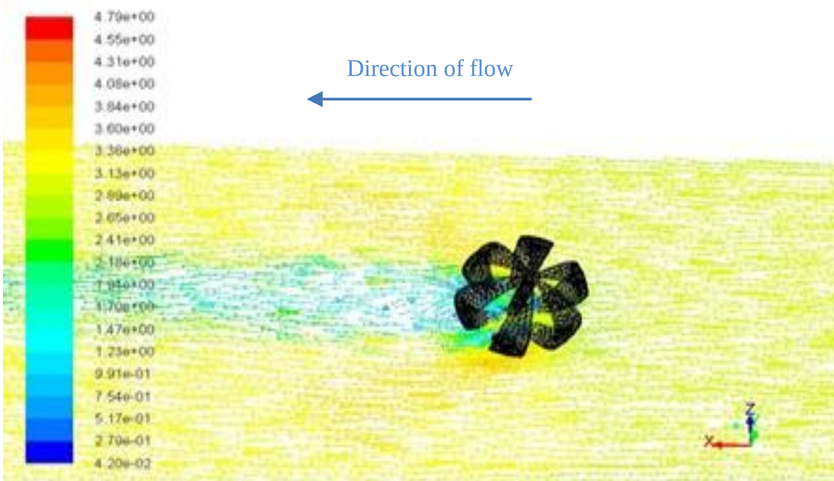
Figure 6.8 Longitudinal section of velocity vectors



a) At the surface



b) 0.5m under surface of water

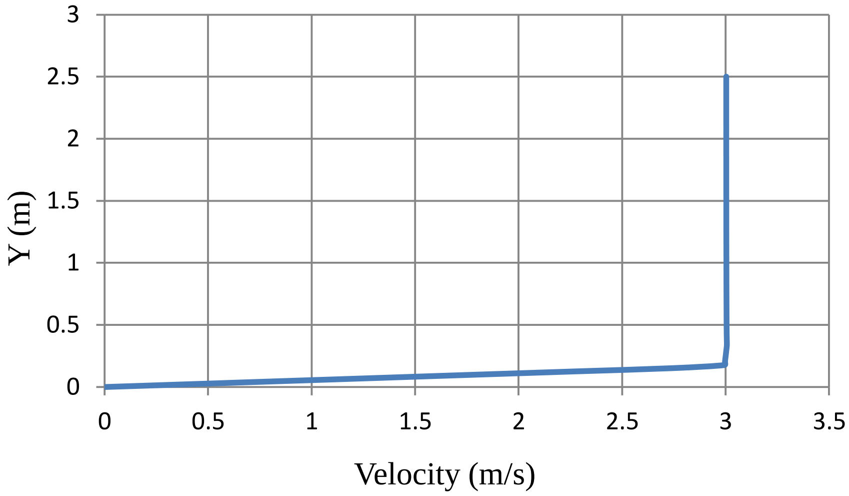


c) 1m under surface of water

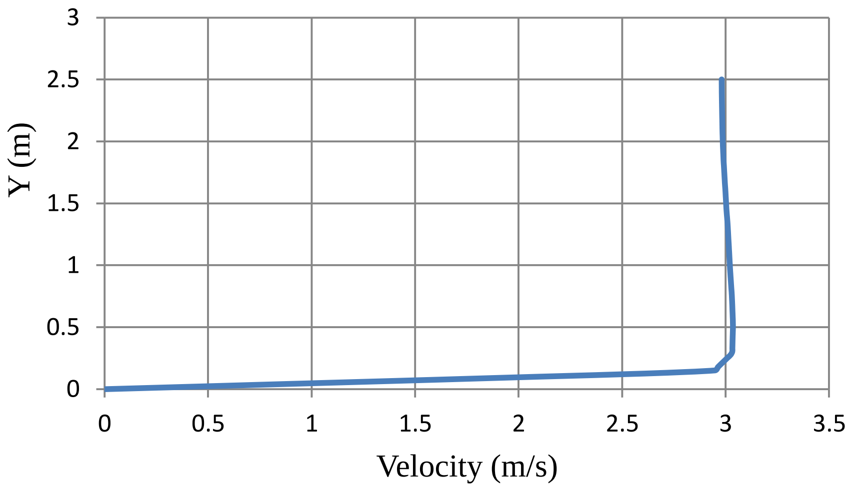
Figure 6.9 Top view sections of velocity vectors

6.2.3. Velocity profiles

The velocity profile diagrams illustrate the velocity change along the channel. The velocity variation at 8m distance before the turbine is very limited. When the flow moves towards the turbine the changes in velocity are observed. The velocity profiles at different positions are given in Figures6.10-6.14.

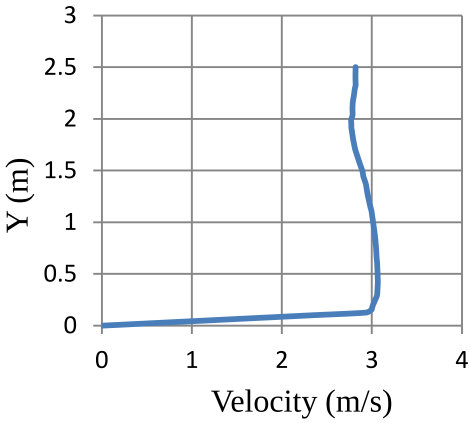


a) 8m upstream of the turbine

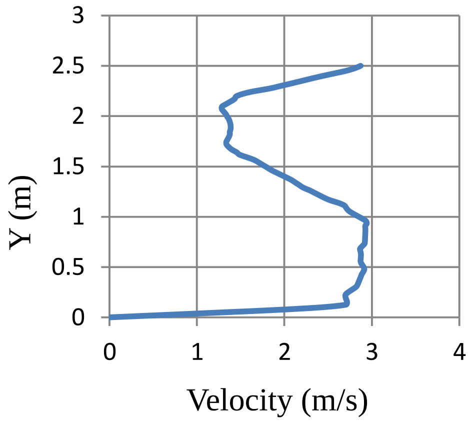


b) 2m upstream of the turbine

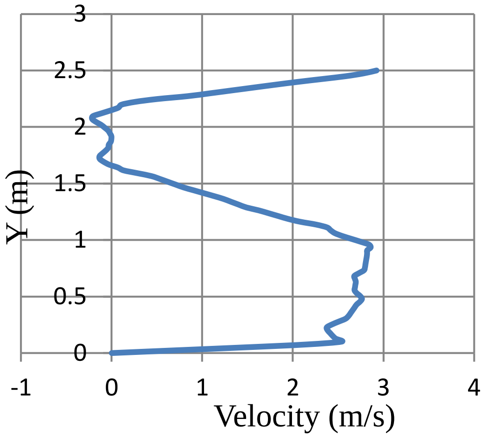
Figure 6.10 Velocity profiles at upstream of the turbine



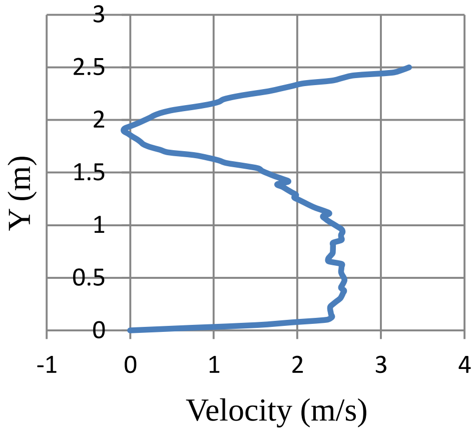
a) 1m upstream of the turbine



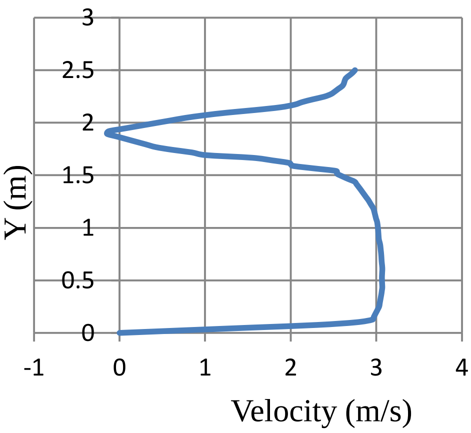
b) 0.75m upstream of the turbine



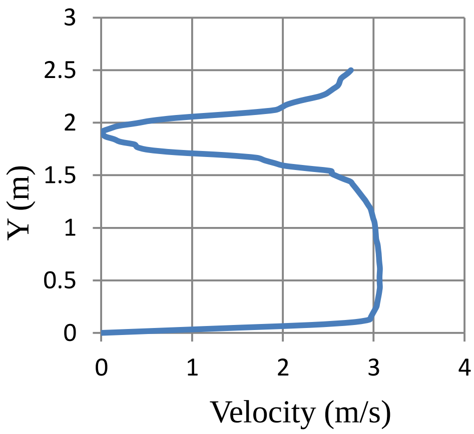
c) 0.5 m upstream of the turbine



d) At center of turbine

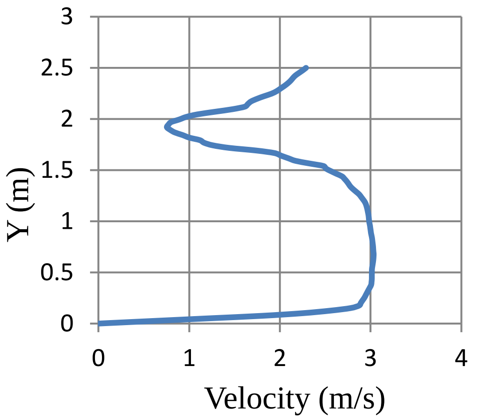


e) 0.5m downstream of the turbine

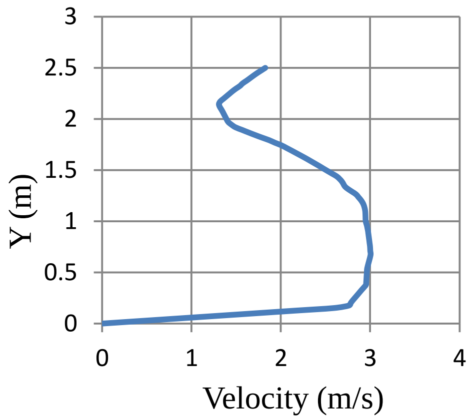


f) 1m downstream of the turbine

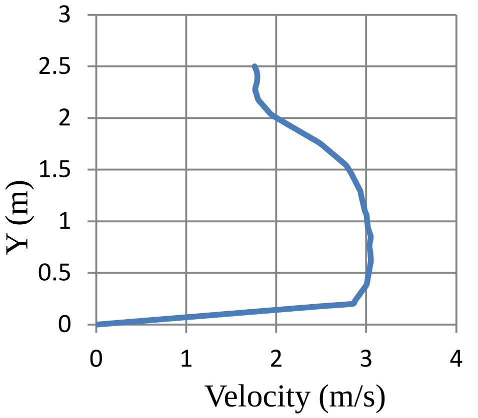
Figure 6.11 Velocity profiles around the turbine



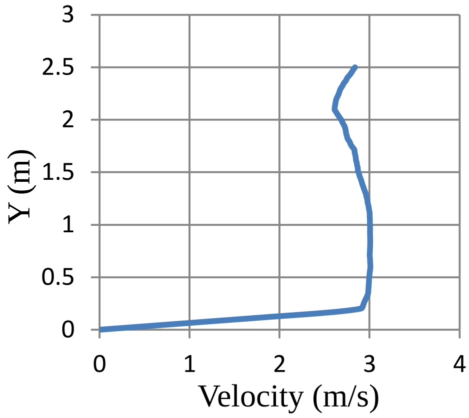
a) 2m downstream of the turbine



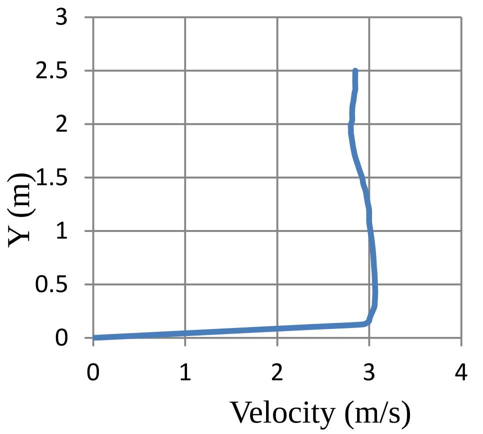
b) 4m downstream of the turbine



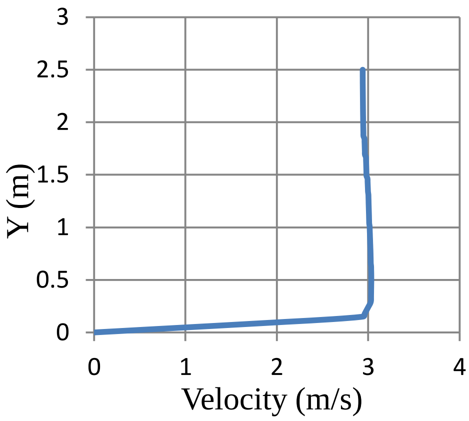
c) 8m downstream of the turbine



d) 15m downstream of the turbine



e) 17m downstream of the turbine



f) 20m downstream of the turbine

Figure 6.12 Velocity profile at downstream of the turbine

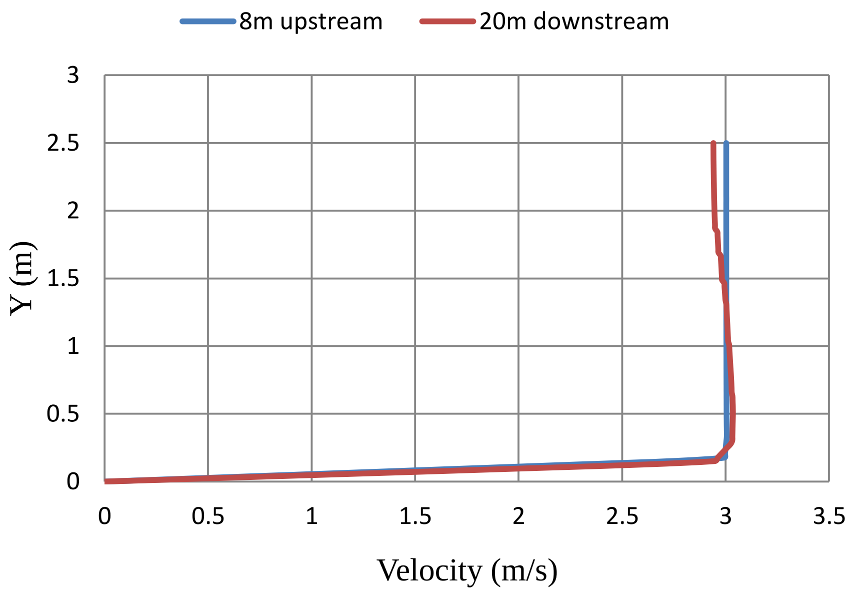


Figure 6.13 Velocity profiles at 8m upstream and 20m downstream of the turbine

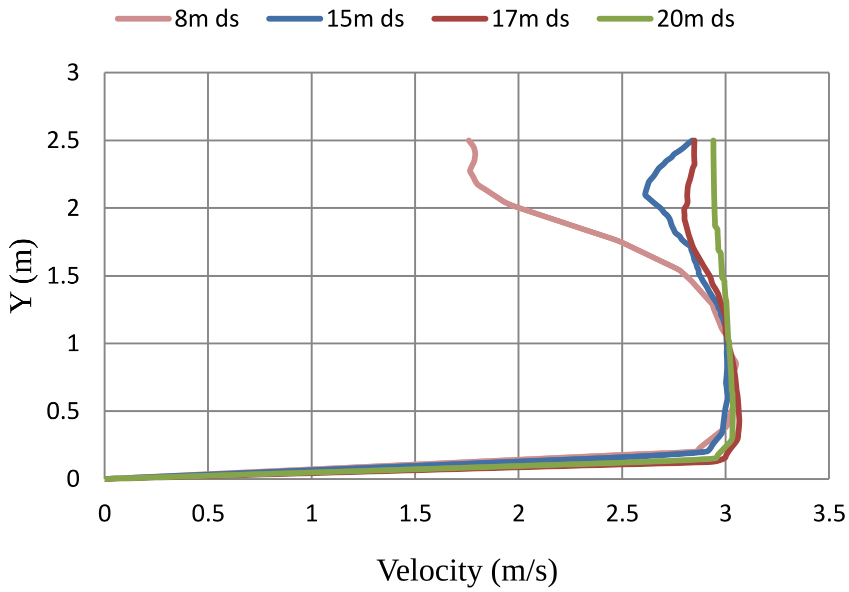


Figure 6.14 Velocity profiles in different points at the downstream of the turbine

6.3. Case II:

Installation of the modified LETGHT at 0.1m above bed of channel is shown in Figure 6.15.

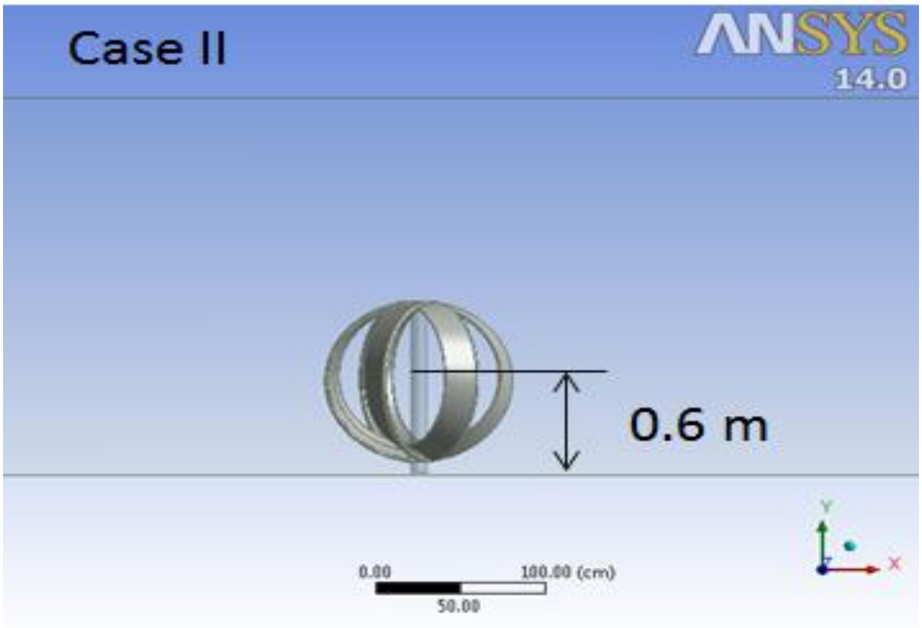
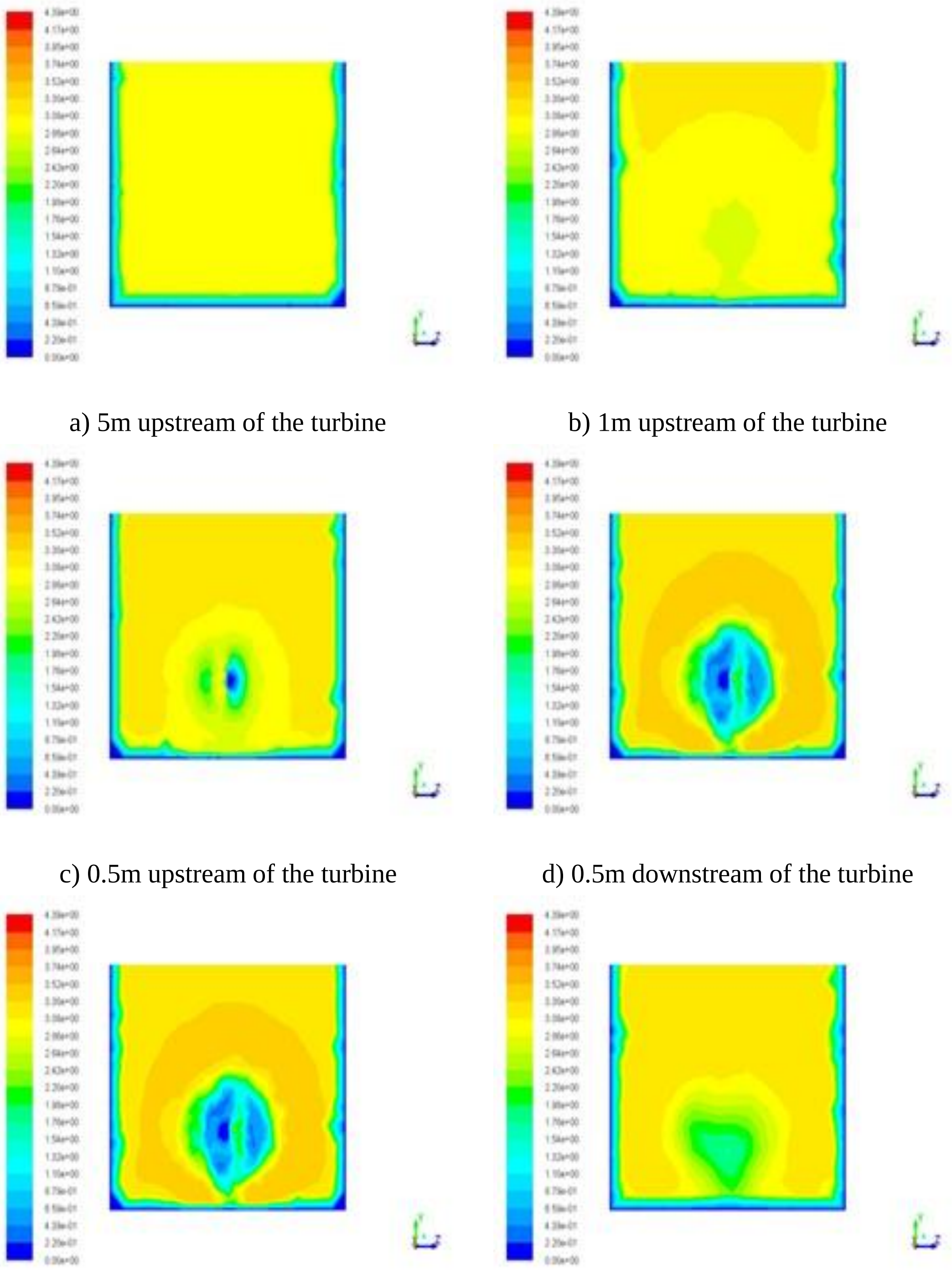


Figure 6.15 Installing modified LETGHT at 0.1m above bed of channel

6.3.1. Velocity distributions

The velocity of the flow is the same as case I which is 3m/s, the turbine works as an obstacle it makes the velocity reduce at about2m upstream of the turbine. At around the turbine the velocity records the minimum amount which is near 0m/s. After the turbine the effect of the turbine will decrease step by step at about 17m the value of the velocity returns to 3m/s. Figures 6.16-6.19 show the changes of velocity contours in different sections.



a) 5m upstream of the turbine
b) 1m upstream of the turbine
c) 0.5m upstream of the turbine
d) 0.5m downstream of the turbine
e) 1m downstream of the turbine
f) 5m downstream of the turbine

Figure 6.16 Velocity contours at different cross sections

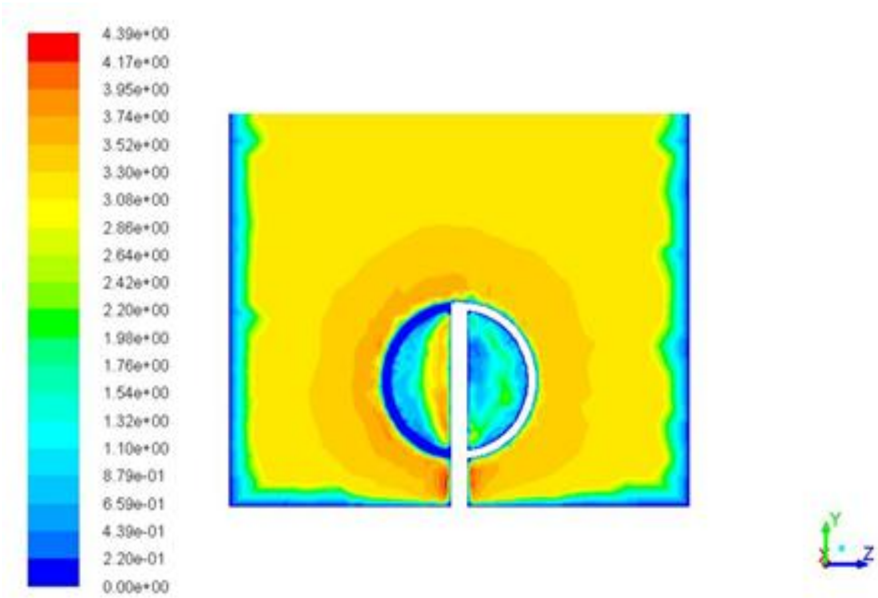


Figure 6.17 Velocity contours at the turbine

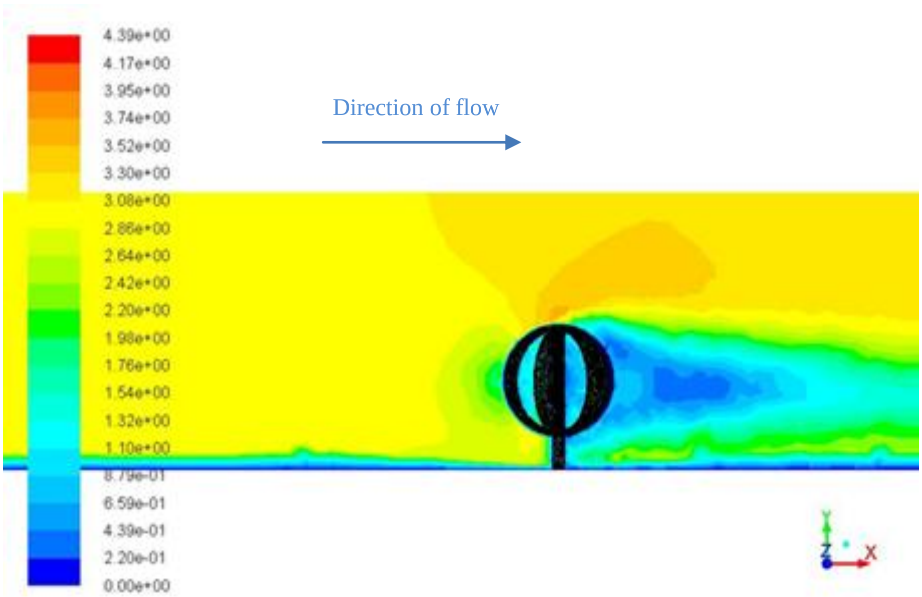


Figure 6.18 Velocity contours longitudinal section of the channel

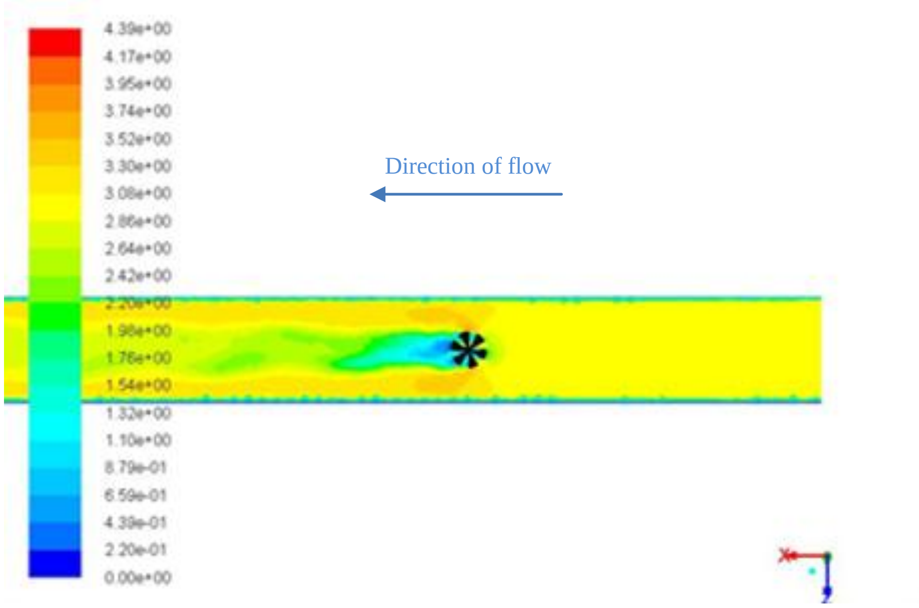


Figure 6.19 Velocity Contours horizontal section at 1m above bed of channel

6.3.2. Velocity vectors

Figures 6.20-6.22 illustrate the directions of velocity in different cross-sections of the open channel. When the flow reaches the turbine some of water flows in all directions due to reduction of velocity, downstream of the turbine vortex takes place and some vectors goes opposite direction creating a recirculation flow. The effect of the vortex reduces at the downstream of the turbine by about 7m. In this case because the turbine is near the bed therefore when the vectors reach the turbine some of them go upward and downward. The turbine is above the bed by 0.1m that makes the downward flow to cause scouring at the bottom of the channel which may lead to the collapse of the turbine.

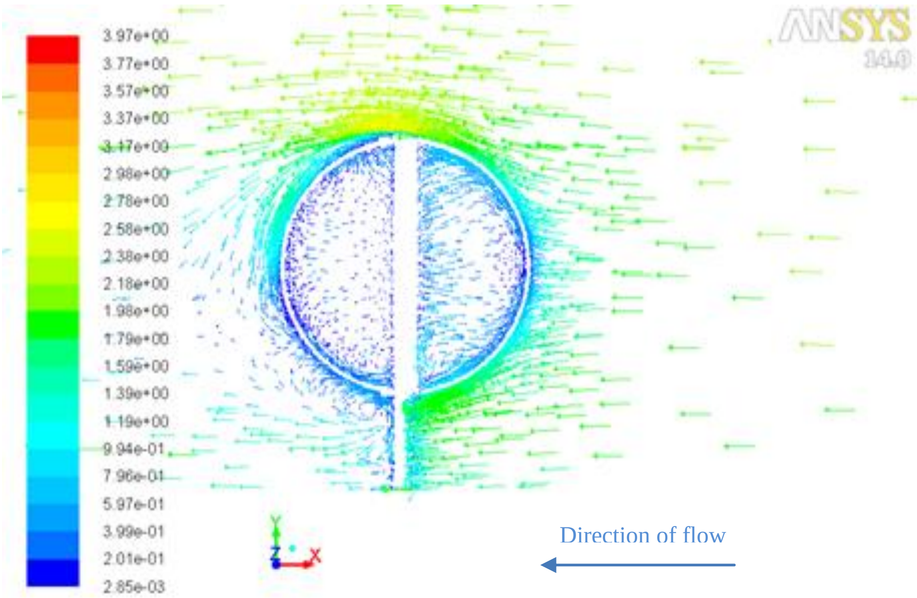


Figure 6.20 Side view of velocity vectors at the turbine

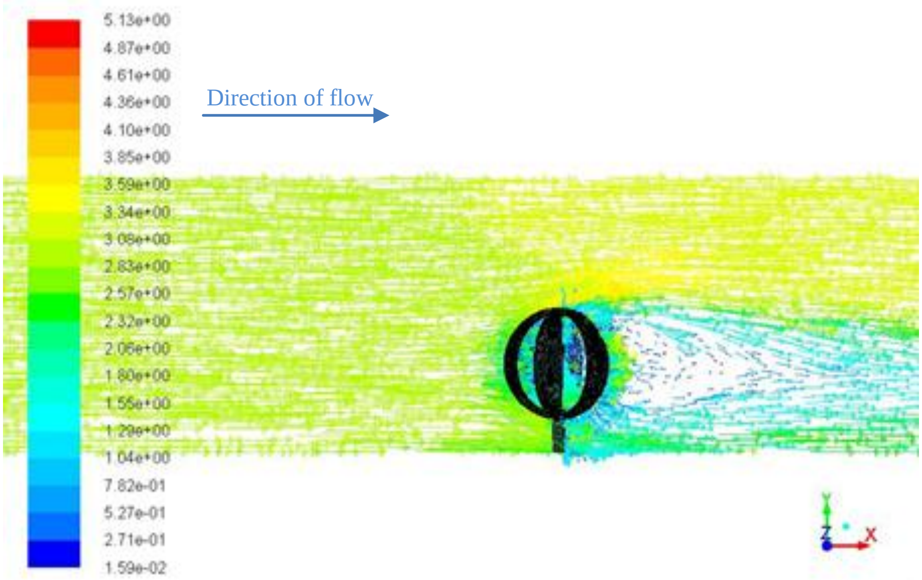


Figure 6.21 Longitudinal section of velocity vectors

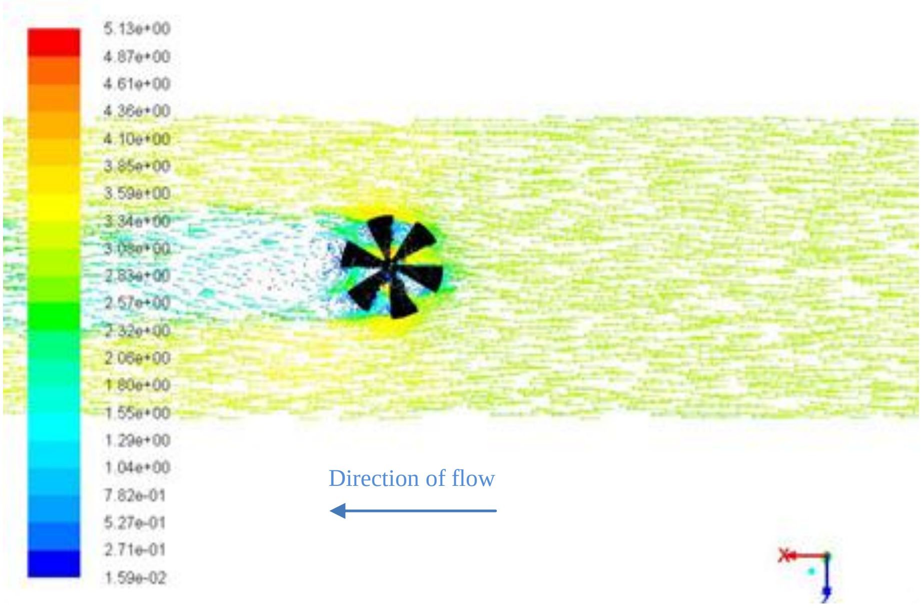
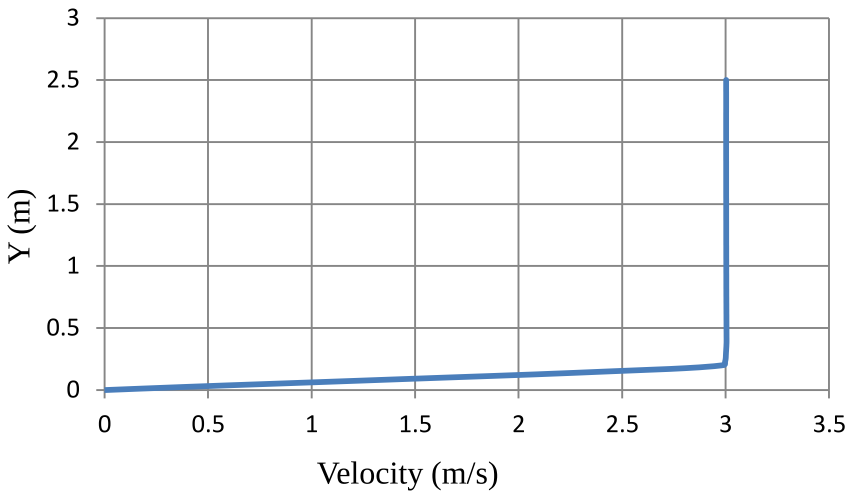


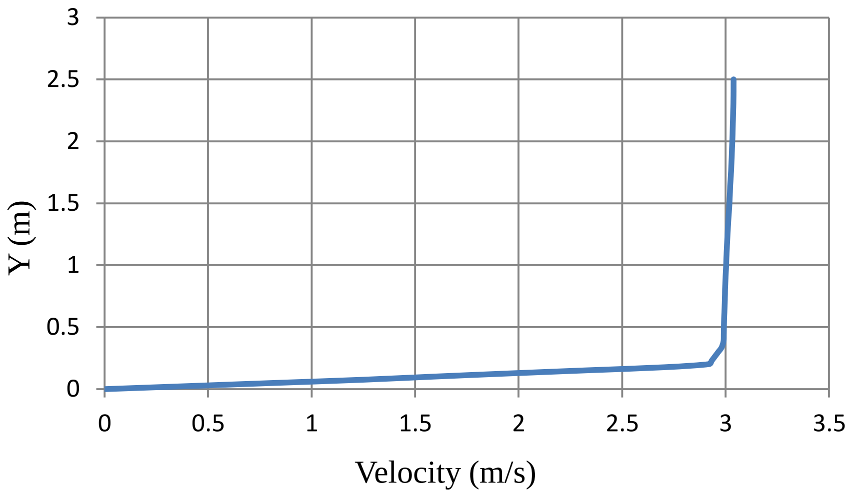
Figure 6.22 Top view of velocity vectors at a section 1m above the bed of channel

6.3.3. Velocity profiles

The velocity variation at 8m upstream of the turbine is steady. 2mupstreamof the turbine change in velocity starts. The velocity profiles at different positions are given in Figures 6.23-6.27.

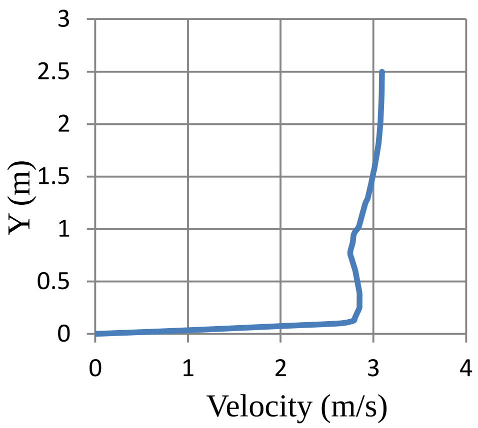


a)8m upstream of the turbine

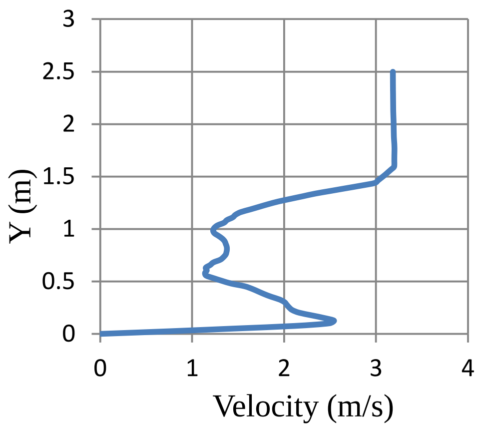


b) 2m upstream of the turbine

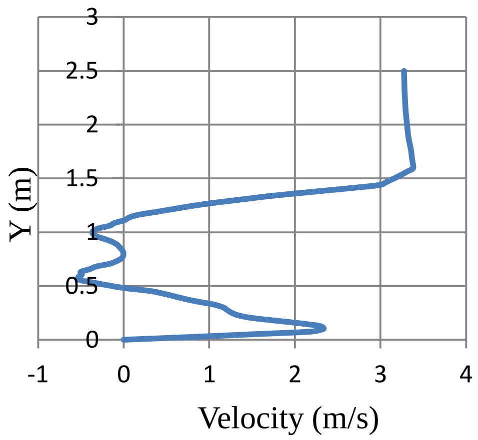
Figure 6.23 Velocity profiles at upstream of the turbine



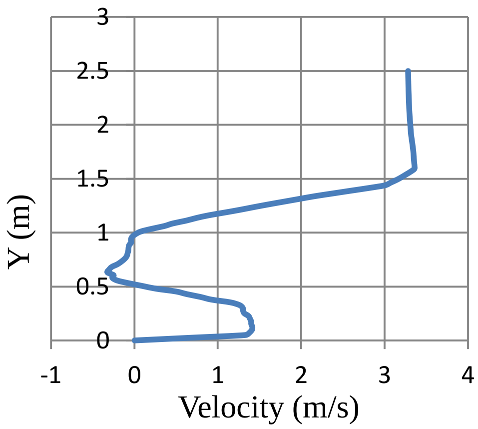
a) 1m upstream of the turbine



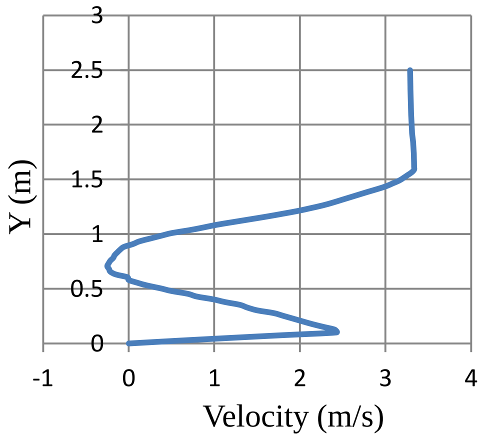
b) 0.75m upstream of the turbine



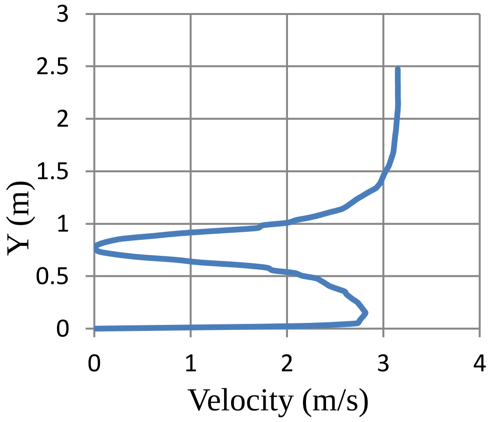
c) 0.5m upstream of the turbine



d) At the center of turbine

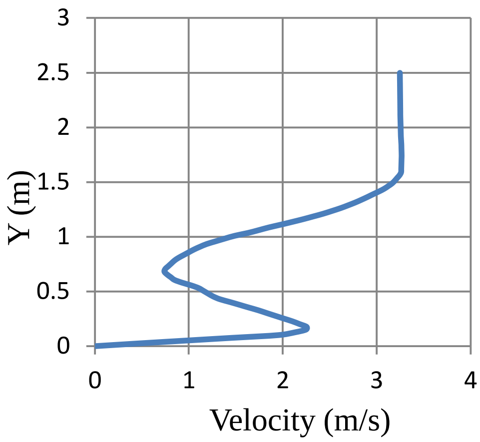


e) 0.5m downstream of the turbine

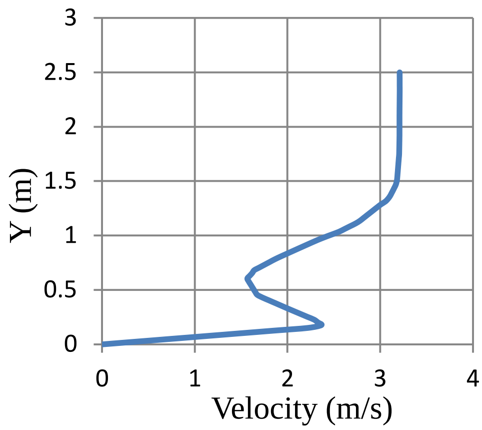


f) 1m downstream of the turbine

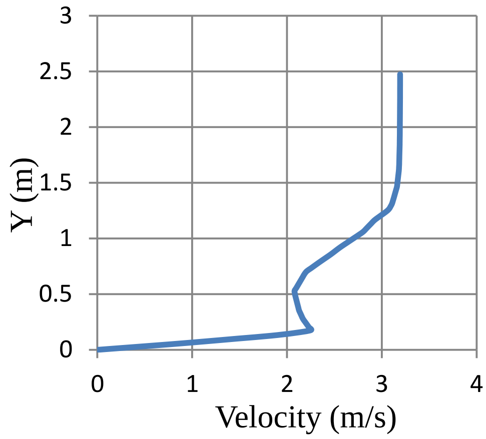
Figure 6.24 Velocity profiles around the turbine



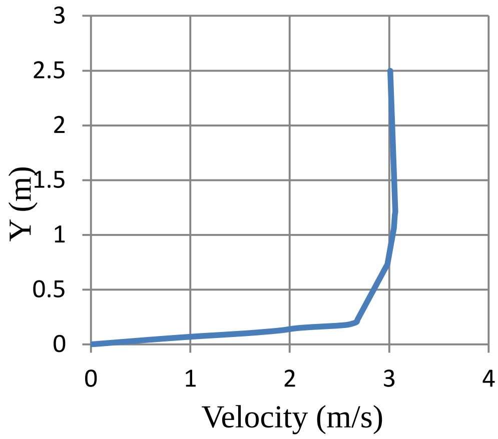
a) 2m downstream of the turbine



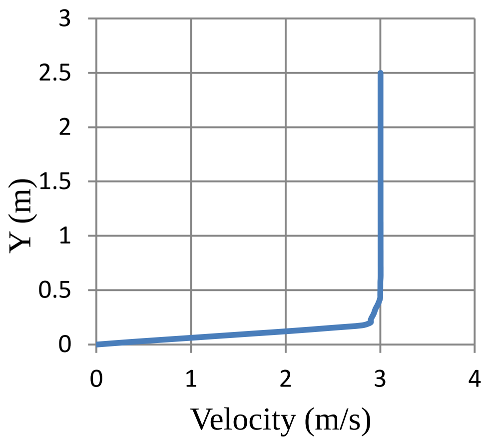
b) 4m downstream of the turbine



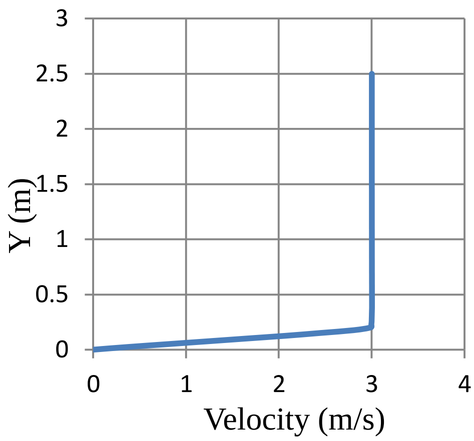
c) 8m downstream of the turbine



d) 15m downstream of the turbine



e) 17m downstream of the turbine



f) 20m downstream of the turbine

Figure 6.25 Velocity profile at downstream of the turbine

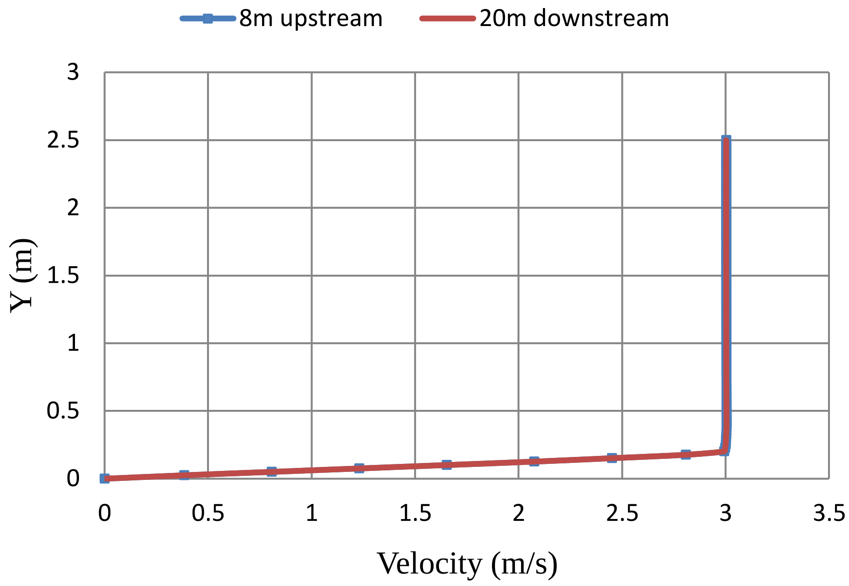


Figure 6.26 Velocity profiles at 8m upstream and 20m downstream of the turbine

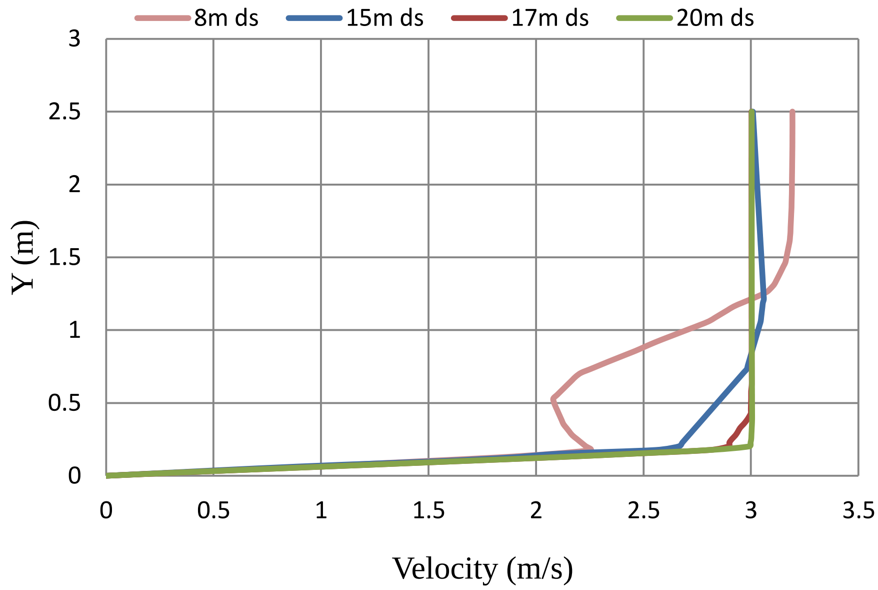


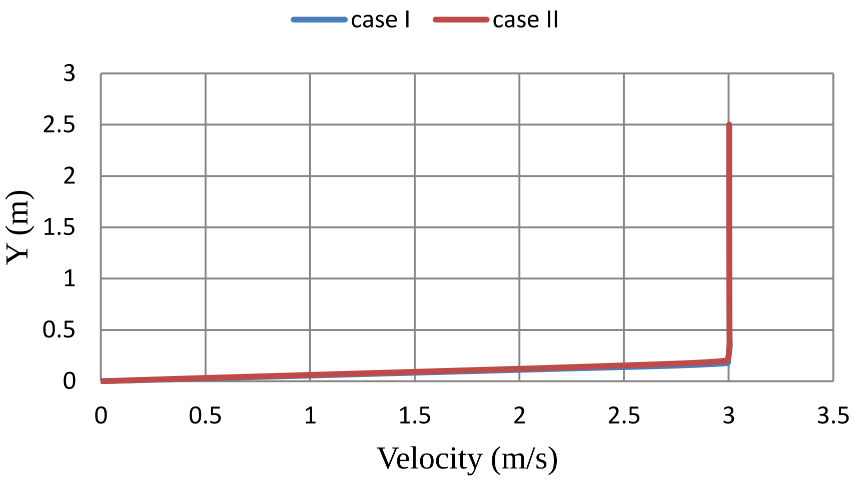
Figure 6.27 Velocity profiles in different points at downstream of the turbine

6.4. Comparison between case I and case II

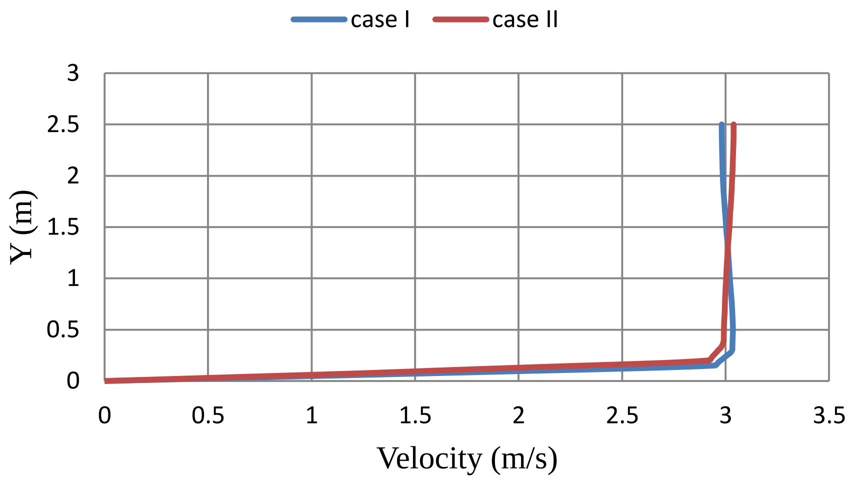
After simulating the flow field around the turbines in case I and case II showed that there are some advantages of case I over case II, which makes case I to be preferred .

1. The vertical velocity profile of flow in the channels is not constant in every level, because near the bed there are frictions between the bed of channel and water particles, which leads to reduce the velocity to the minimum value, but near the surface there is no any friction the velocity is close to maximum value. In order to increase the energy generation with hydrokinetic turbines, turbines should be installed as close as possible to the free surface of water in the open channel.
2. In natural channels the bed is not lined with concrete if the turbine is installed near the bed when flow pass the turbine may cause scouring of the bed around the turbine, this is risk for the structure, which may lead to damage the whole structure.
3. In case of installing turbine near the bed may damage it in shorter period because of sediment transportation in the bed.
4. Installing turbine near the bed may increase turbidity at the downstream of the turbine during operation, which may damage environment of the channel and create risk for aquatic life.

Figures 6.28-6.30 show the velocity profiles of case I and case II at difference cross-sections.

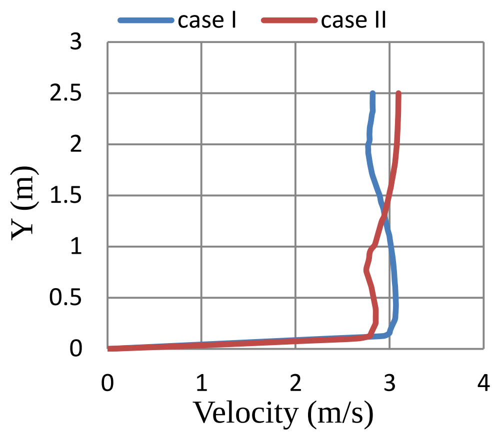


a) 8m upstream of the turbine

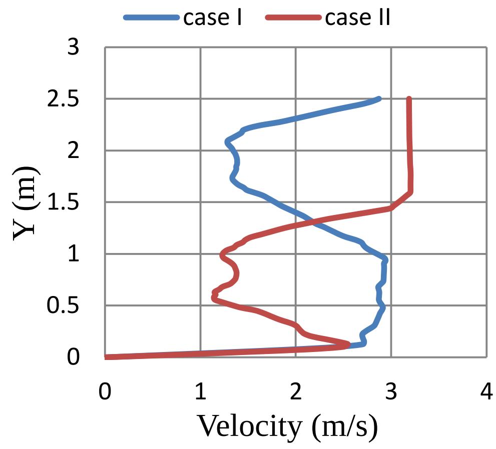


b) 2m upstream of the turbine

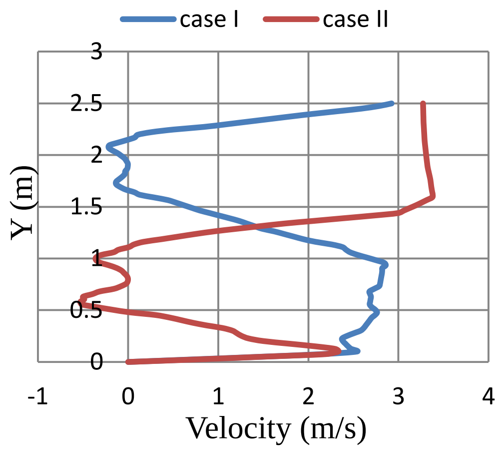
Figure 6.28 Velocity profiles at upstream of the turbine



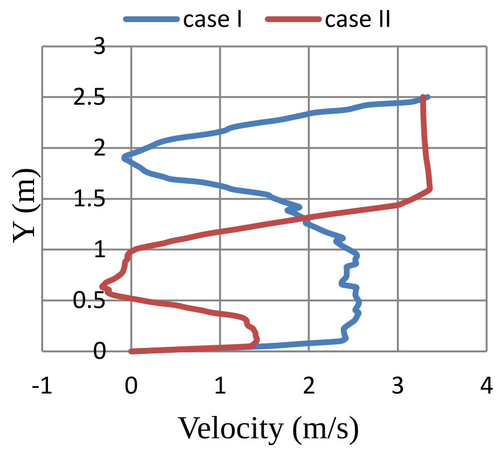
a) 1m upstream of the turbine



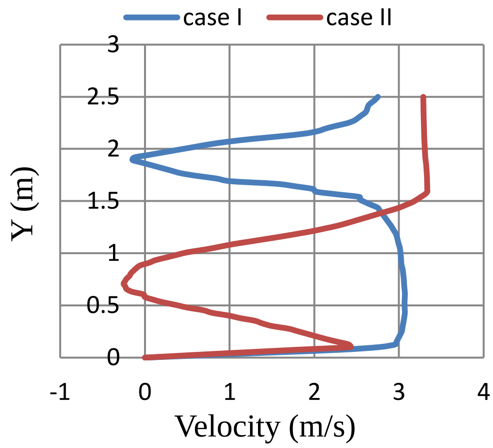
b) 0.75m upstream of the turbine



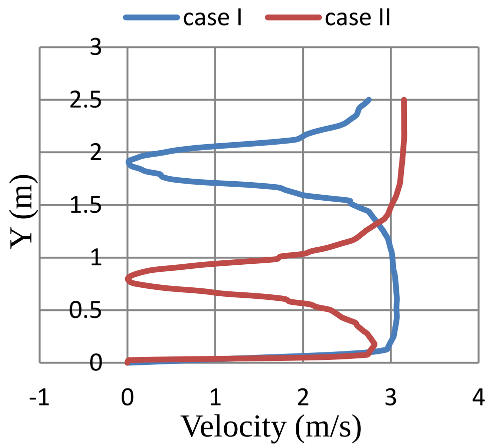
c) 0.5m upstream of the turbine



d) At centre of the turbine

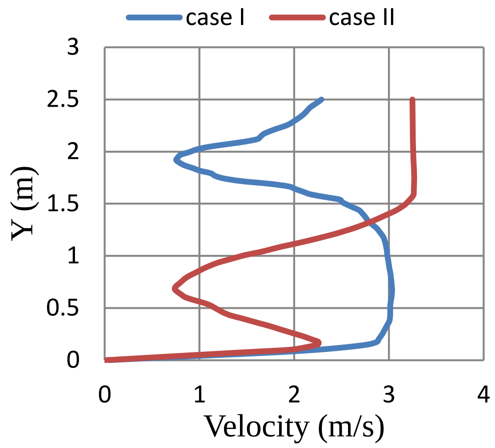


e) 0.5m downstream of the turbine

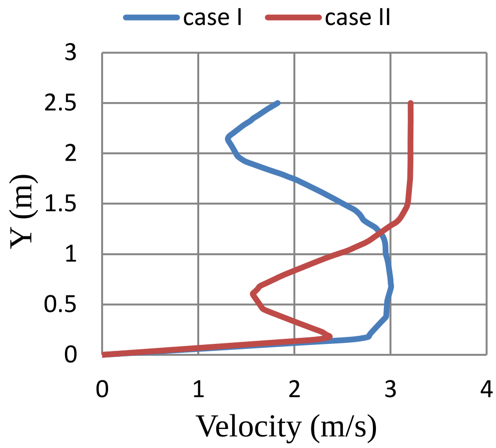


f) 1m downstream of the turbine

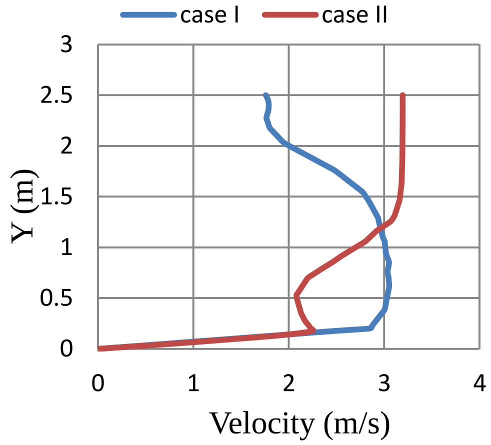
Figure 6.29 Velocity profiles around the turbine



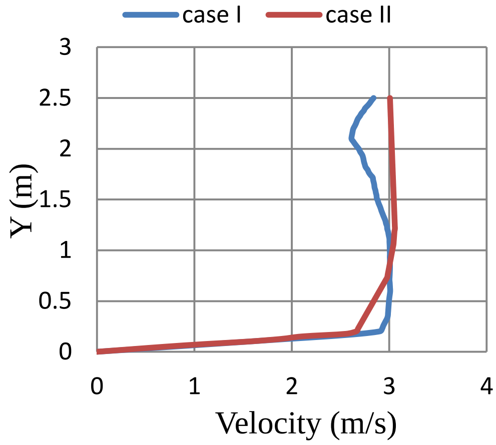
a) 2m downstream of the turbine



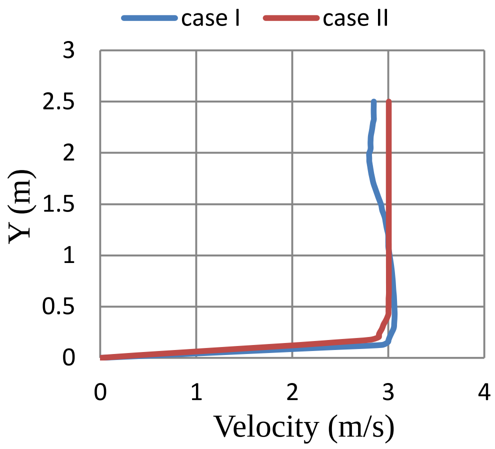
b) 4m downstream of the turbine



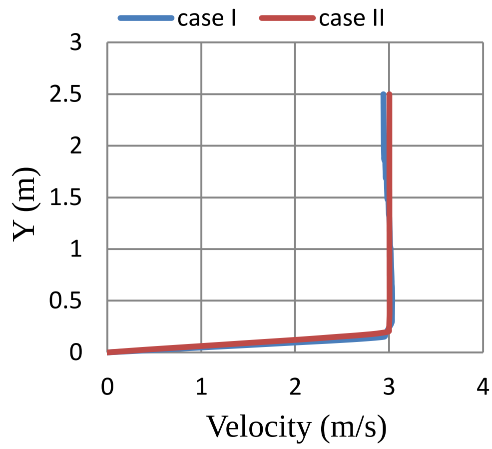
c) 8m downstream of the turbine



d) 15m downstream of the turbine



e) 17m downstream of the turbine



f) 20m downstream of the turbine

Figure 6.30 Velocity profile at downstream of the turbine

6.5. Case III:

In this case two turbines were installed at a level of 1.4m above the bed of the channel at the centre line of the channel one at the downstream of another one. This case indicates simulating more than one turbine with suitable positioning in order to extract maximum possible energy. In simulation studies of case I and case II it was noted that the effect of the turbines on the flow characteristics diminishes at about 17 - 20m downstream of the first one. Also positioning turbines in arrays may increase the amount of the energy that can be harnessed from a flowing water body. Velocity distributions, velocity streamlines and velocity vectors in this case are the same as in case I for each turbine which are shown in Figures 6.31-6.36.

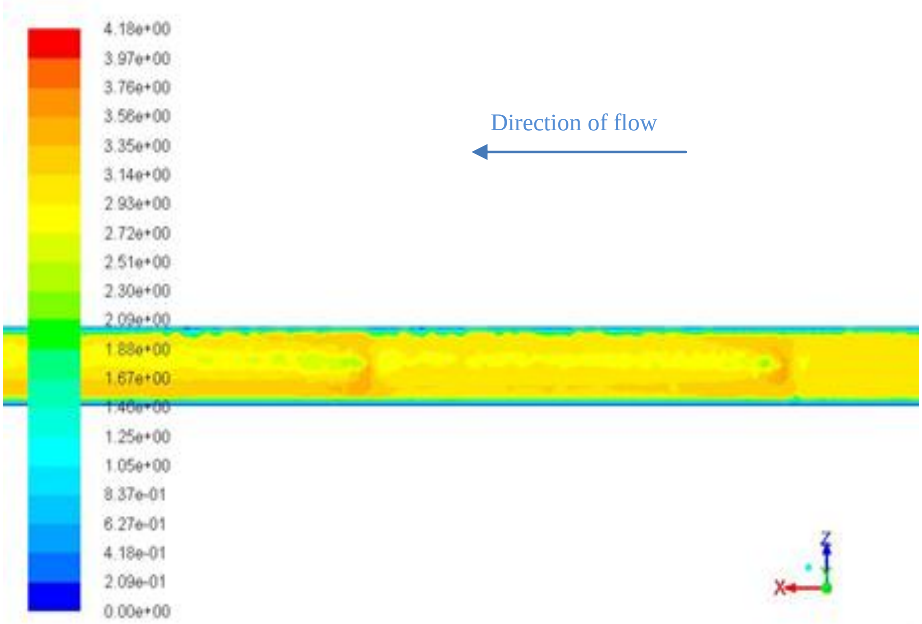


Figure 6.31 Velocity Contours at the surface

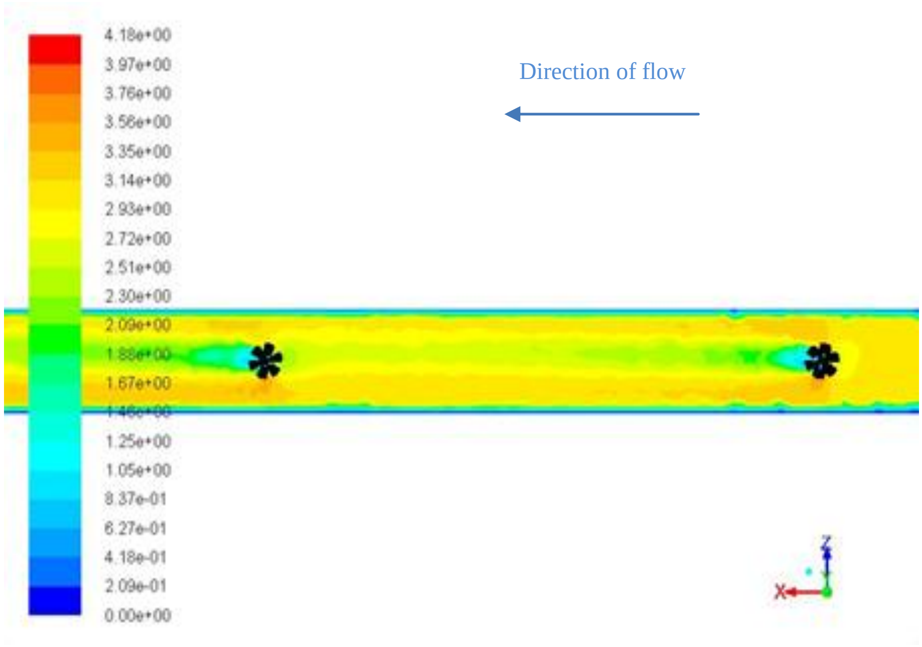


Figure 6.32 Horizontal section of velocity contours at centre of channel

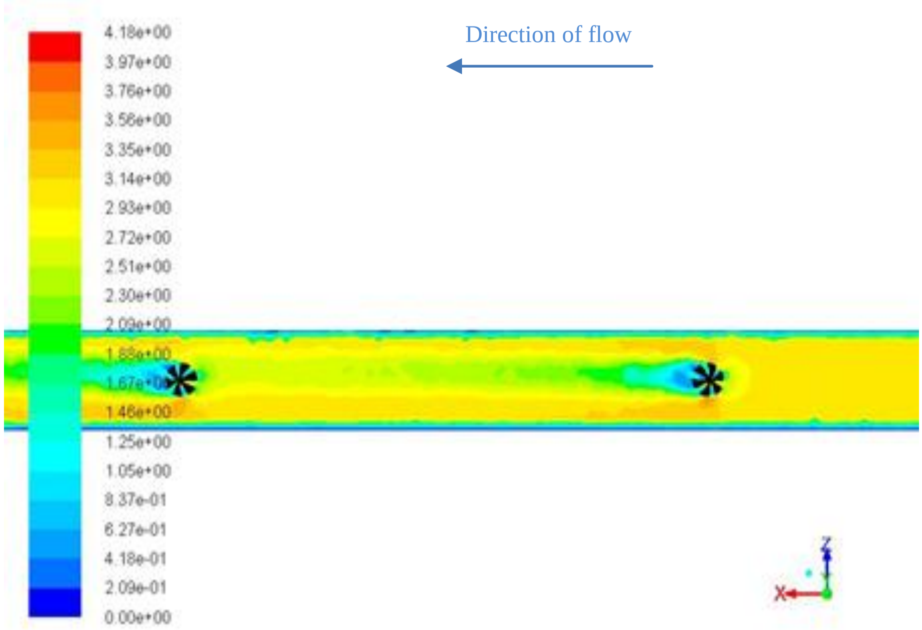


Figure 6.33 Horizontal section of velocity contours at 1m above the bed

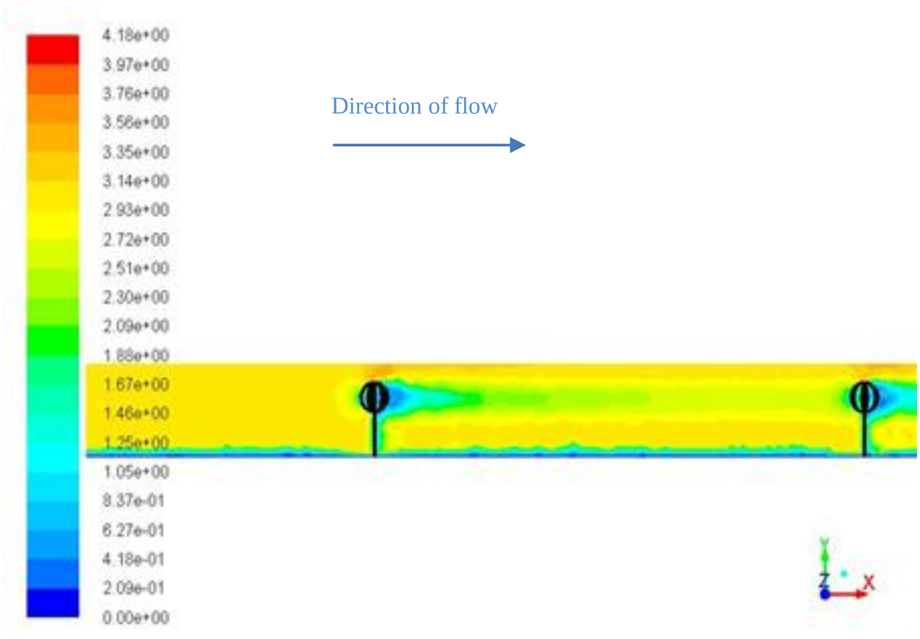


Figure 6.34 Longitudinal section of velocity contours

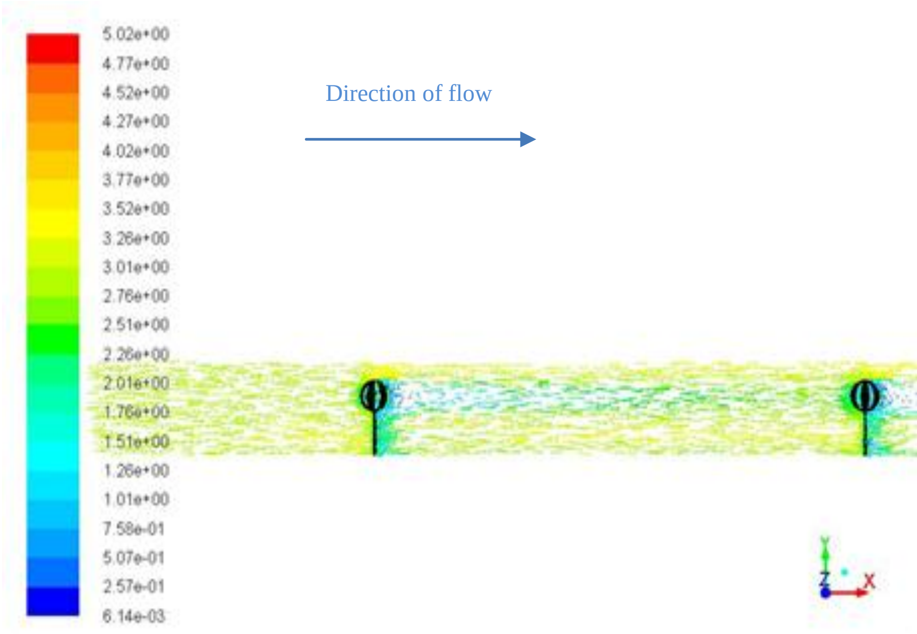


Figure 6.35 Longitudinal section of velocity vectors

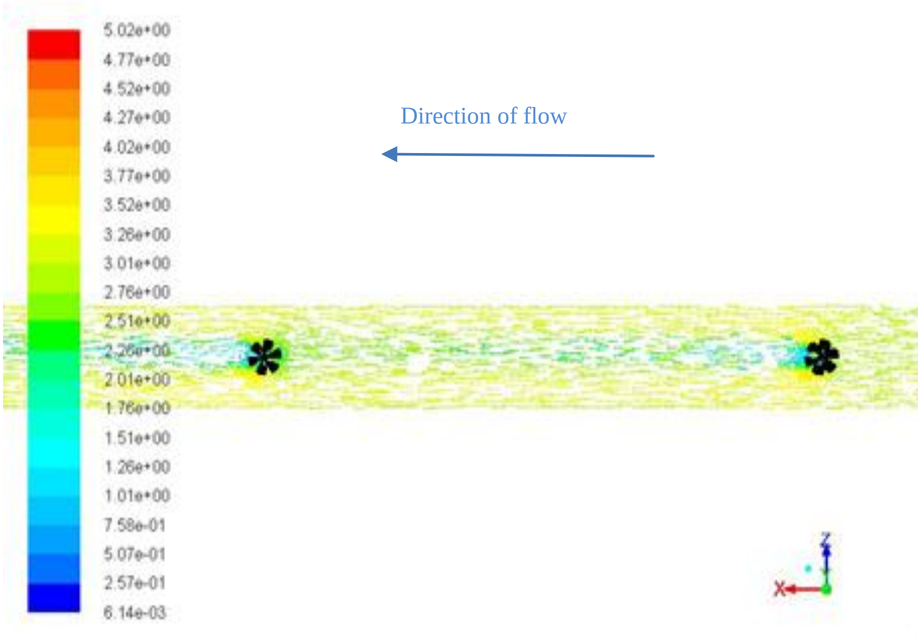


Figure 6.36 Horizontal section of velocity vectors at 1m above the bed

CHAPTER 7

CONCLUSIONS

The hydrokinetic energy which is harnessed from flowing water of rivers streams, tidal flows and ocean currents is being considered as a predictable and environmentally benign source of 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, specifically, a modified form of Lucid Energy Technology (LET) Gorlov Helical Turbine (GHT) was simulated by using ANSYS FLUENT which is commercially available CFD software. The working principles of vertical axis 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 (RANS) Equations for an incompressible Newtonian fluid. 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 profiles observed well before the turbine was a typical fully developed steady open channel flow with the magnitude of the velocity being zero at the boundaries due to no-slip condition 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 make a preliminary design of a hydrokinetic turbine and simulate it in an open channel flow. The flow field was studied with the turbine being positioned at two different depths, while the depth and the velocity of the open channel flow were kept constant. The simulation work was based on a stationary (non-rotating) turbine. The results have shown that placing the turbine closer to the bed of channel causes an increase in water velocity near the free surface. Although the water velocity near the channel bed seems to be decreasing, the turbulence flow takes place just behind the turbine and the burst of the flow directly below the turbine are expected to cause scouring and increase turbidity levels. This may harm the aquatic life in the natural channel and may also cause the failure of the structure that the turbine in attached to.

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**, 2205-2211.

Bakker et, al. (2001). Realize grate benefits from CFD, *Fluid Solids Handling* **34**, 45-53.

BP (2013). British Petroleum Statistical Review of World Energy. Available at: bp.com/statisticalreview

Bredberg, J. (2000). On the wall Boundary Condition for turbulence Models.Department of Thermo and Fluid dynamics, Internal Report, Chalmers university of Technology, Gotebory, Sweden.

Cfd-Online. (2013). Turbulence Modeling. Available at: http://www.cfdonline.com /Wiki/ Turbulence modeling

Da Vinci, eFluids Gallery image and Discription. (2013). Avilable at: http://www.efluids.com/ efluids/gallery/gallery_ pages/da_vinci_page.htm

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 Method. *Phys. Fluids* ***16***, 99-106.

Goodwill and Sleigh. (2008). Fluid Flow in Pipes, CIVE2400 Fluid Mechanics.

Hansen, M. (2008).Aerodynamics of wind turbines. 2nd edition. London: Earth scan.

International Energy Agency (IEA). (2011). Annual Report. Available at: www.iea.org.

Jennings, P. J. (1996). Education in the Australian CRC for Renewable Energy.In proceedings of solar96. Darwin, Australia.

Kassam, S. (2009). In-Situ Testing of a Darrieus Hydro Kinetic Turbine In Cold Climates. MSc. Thesis. Faculty of Graduate Studies, University of Manitoba, Canada.

M.Amen, O. (2013). Flow Field around Hydrokinetic Turbines. MSc. Thesis. Graduated school of engineering, Gaziantep University, Turkey.

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

National Energy Education Project. (2013). Available at: www.need.org/needpdf /infobook_ activities/PriInfo/Sources.pdf.

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

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

Renewable Energy Policy Network for the 21St century. (2013). Renewables. Global Status Report. Available at: http://www.ren21.net/Portals/0/documents/Resources /GSR/2013/GSR2013_lowres.pdf.

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.

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

Twidell, J., Weir, T. (2006). Renewable Energy Resources. 2nd Edition. UK: Taylor & Francis.

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

Xia B. and Sun D. W. (2002). Applications of Computational Fluid Dynamics (CFD). In the food industry: a review, *Computers and Electronic in Agriculture*, **34**, 5-24.