Characterization of Tidal Current Turbine Dynamics Using Fluid Structure Interaction (FSI)

BY

Habibullah

23-FET/MSME/F14

Supervised By

DR. Saeed Badshah

DEPARTMENT OF MECHANICAL ENGINEERING,

FACULTY OF ENGINEERING AND TECHNOLOGY

INTERNATIONAL ISLAMIC UNIVERSITY

ISLAMABAD.

Fluid - Structure interation.

Tidal current turbine.

Thesis entitled

Characterization of tidal current Turbine Dynamics Using Fluid Structure Interaction (FSI)

Submitted to International Islamic University, Islamabad
In partial fulfillment of the requirements
for the award of degree of

MASTER OF SCIENCE

IN

MECHANICAL ENGINEERING

BY

Habibullah

23-FET/MSME/F14

SESSION 2014-2017

DEPARTMENT OF MECHANICAL ENGINEERING

INTERNATIONAL ISLAMIC UNIVERSITY,

ISLAMABAD

Thesis entitled

Characterization of tidal current Turbine Dynamics Using Fluid Structure Interaction (FSI)

Submitted to International Islamic University, Islamabad
In partial fulfillment of the requirements
for the award of degree of

MASTER OF SCIENCE

IN

MECHANICAL ENGINEERING

BY

Habibullah

23-FET/MSME/F14

SESSION 2014-2017

DEPARTMENT OF MECHANICAL ENGINEERING

INTERNATIONAL ISLAMIC UNIVERSITY,

ISLAMABAD

DECLARATION

I, **Mr. Habibullah**, Reg. No. **23-FET/MSME/F14** student of MS mechanical engineering in Session 2014-2016, certify that research work titled **"Characterization of tidal current Turbine Dynamics Using Fluid Structure Interaction (FSI)"** is my own work. The work has not been presented elsewhere for assessment. Where material has been used from other sources it has been properly acknowledged.

Signature of student:_______________________

Dated: 29/03/017

Certificate of Approval

This is to certify that the work contained in this thesis entitled, **"Characterization of tidal current Turbine Dynamics Using Fluid Structure Interaction (FSI)"** was carried out by **Habibullah Registration# 23-FET/MSME/F14,** it is fully adequate in scope and quality, for the degree of MS (Mechanical Engineering).

Viva Voice Committee

Supervisor

Dr. Saeed Badshah

Head of Department (HOD)

DME, FET, IIU, Islamabad

Internal Examiner 29/3/2017

External Examiner 30/3/17

Chairman DME FET IIU, Islamabad

Dean FET, IIU Islamabad

Prof. Dr. Aqdas Naveed Malik

DEDICATION

I dedicate all my efforts to the **Holy Prophet Mohammad (S.A.W.),** who has been sent as a mercy to all mankind, my parents who have always supported me morally and financially, and have always prayed to Almighty ALLAH for my success, to my entire family members and to my respected teachers. I also dedicate it to my parents, brother and sisters for their kind support and prayers.

ACKNOWLEDGEMENTS

Praise to Almighty ALLAH, the most Benevolent, the Omnipotent and Merciful, the Creator of the Universe, who enabled me to accomplish this declaration. Blessing on MOHAMMAD (S.A.W), the seal of the prophets and his pious progeny. I am grateful to International Islamic University, Islamabad for providing me a chance to enhance my skills and knowledge through this thesis and I wish to express my deepest gratitude to my worthy and honorable supervisor **Dr. Saeed Badshah** for his kind supervision with keen interest and valuable guidance. His professional understanding and profound knowledge and kind attitude helped me in compiling this dissertation. His constructive criticism and directional guidance always motivated me to work hard.

I am especially grateful to Engr. Mujahid Badshah and Mr. Inayat Ur Rahman for a helpful discussion during my research work. Also, I am especially grateful to Dr. Nizam Ud Din for guiding me in CFD research And in the end, I would like to pay my special thanks to my friend Mr. Wajid Ali who helped me in preparing my thesis.

I ask ALLAH to help all those who have helped me and guided me from beginning to the end of my thesis.

ABSTRACT

At present time there is a concern over global climate change, as well as a growing awareness on worldwide population about the need of reducing the greenhouse gas emissions. This in fact, has led to an increase in power generation from renewable sources. The tidal current energy has the potential to play a vital role in a sustainable energy future. The tidal current turbines are used to extract the tidal energy.

The main focus of this thesis was to investigate the horizontal axis tidal current turbine (HATCT) dynamics using fluid structure interaction (FSI) using the simulation software package ANSYS 17.0. In order to achieve this aim a number of key steps were performed. The HARP_Opt (Horizontal Axis Rotor Performance Optimization) code was used in this research work for BEM Design of the turbine and result validation. The performance of tidal current turbine was predicted using ANSYS-CFX 17.0 and compare with BEM results. The performance curve, pressure distribution on the blade and velocity streamline was visualized for sex repetitive analysis at different tip speed ratio.

The hydrodynamic loads calculated by CFD analysis was transferred to finite element model (FEM). Finite element method (FEM) was used to investigate the structural response of tidal current turbine. The modal analysis, pre-stressed vibration analysis and forced vibration analysis were performed for the structural response of tidal current turbine. Natural frequencies and mode shapes were visualized for vibration response of tidal current turbine.

In the last section of this thesis, some interesting conclusions were drawn. The recommendation for future work presented at the end is expected to provide a basic guideline to the new researchers.

TABLE OF CONTENTS

List of Figures

List of Tables

List of Abbreviations

HATCT:	Horizontal axis tidal current turbine
VATCT:	Vertical axis tidal current turbine
BEMT:	Blade element momentum theory
CFD:	Computational fluid dynamics
FEA:	Finite element analysis
FSI:	Fluid Structure interaction
CWC:	Circulating water channel
SST:	Shear stress transport
GGI:	General grid interface
MRF:	Moving reference frame

CHAPTER 1

INTRODUCTION

1.1 Tidal Current Energy

The tidal current energy is a form of hydrokinetic energy extracted from the water flows in the tidal channels. Tidal current is created by gravity of moon. The earth and sun are also the source of generation of tides. Because of the nearness to the earth, the moon plays more role in the creation of tides. The attraction between the sun and earth because of the lunar gravity tilts the ocean to the opposite direction of the earth pull. So the water head raises in ocean. This happens in the place which is in direction of moon and falls in center between these positions. The rise in head causes the movement in water known as tidal current. Then the tidal energy is obtained from water kinetic energy. A different approach is also used for the creation of tidal energy in which water head or potential energy is utilized. For this purpose stream devices known as hydrokinetic turbines are directly installed in the stream of water. A dame like structure is not required in this technology. [1].

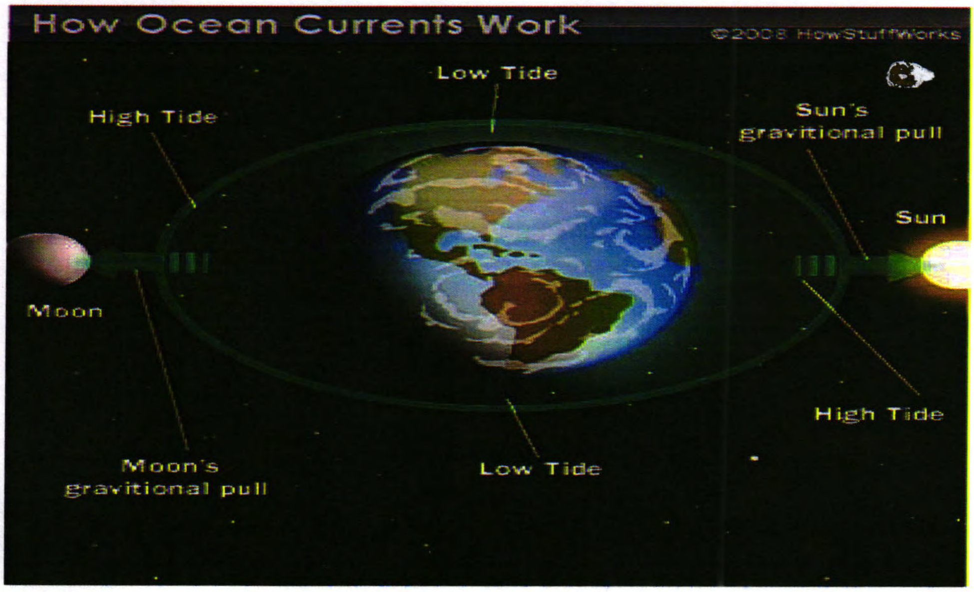


Figure: 1.1 Tidal currents [1]

The tidal energy have more potential than solar and wind energy to become the source of future electricity. In renewable energy sources tidal power is considered the expensive source of energy because of limited sites and design configuration of devices. Although, now a days after the improvement and development in technology in design configuration predict that there is higher availability of sites of tidal energy than earlier assumed, and environmental and economic costs can be decreased to reasonable levels.

1.2 Energy situation in Pakistan

Pakistan is facing a worst electricity crisis now a day. Electricity shortfall has reached up to 4406MW in 2014 and as per state of industry (NEPRA) report 2014 [2], this shortfall is increasing day by day. To overcome this shortfall different conventional and renewable energy techniques should be explored. Due to the environmental effects of conventional energy resources, Pakistan should pay attention towards renewable energy resources. The alternative energy resources, i.e. solar and wind energy have a variety of issues as their power production output is not predictable, but the source of tidal energy is completely predictable as it takes place because of the well-known astronomical phenomenon and have more capacity than any other renewable resource. Tidal power have ability to become the major source of electricity in the universe due to the low operational cost and no fuel cost.

In Pakistan several significant locations of higher tidal velocities are available along the 990 km area of coast. In a survey conducted by institute of occansiongraphy it was estimated that more than 1100 MW power may be generated along the 170 Km coastline of Arabian Sea from Indus Deltaic stream areas [3]. The creek areas from Korangi to Kajhar mainly has a tidal current velocity of 4-5 knots range, but values of higher velocity upto 8 knots were also observed. The difference between tidal hides varies from 2 to 5 km along the coastline of Sindh. The cast area of Kalamat and Sonmianiin Baluchistan are the other significant sites for tidal power [4]. Another oceanographic survey is required to explore the other tidal power sites.

1.3 Tidal Energy in World Prospective

The estimate potential of tidal power varies in several surveys. Although it is strongly accepted that the global tidal power capacity is greater than 120 GW. United Kingdom (UK) has the largest resource estimate of more than 10 GW. The United Kingdom is ideally situated to explore tidal current energy with 10 to 15 % of the global resource and around 50 % of the European resource. A map of world tidal power resources is shown in figure 1.2 [5].

Figure1.2: Global tidal energy resources [5]

A study by Black and Veatch found that within the UK, 58% of the national resource is found off the coast of Scotland, in the Pentland region, 15% around Aldernay, 3.7% around the Mull of Galloway [6].

1.4 Current Status of Technologies

The development of Tidal current technology is presently at the commercial proto type development stage [7]. Real sea testing is under way at different testing sites around the world [8, 9]. Currently there is only one prototype seagen installed by marine current turbine (MCT) in 2008, in Strangford Lough, which can produce a rated power of 1MW. Many other devices have been tested by the European Marine Energy Centre (EMEC) in the Pentland Firth, and as well as other parts of the United Kingdom (UK) [10]. In order to reach the required contribution towards the renewable energy (tidal current energy) target, development in tidal energy technology is in progress for the extraction of electricity from the tides. The tidal current approach is classified in in following two main categories.

1.4.1 Tidal barrages technology

In tidal barrage potential energy of tidal current is utilizes. A lagoon or dame like structure on the outlet of the creek or river is designed for the development of tidal barrage. The constructed wall separates the tidal reservoir from the ocean and controls the flow of the tidal currents. However tidal steam turbines or other devices are developed to extract the kinetic energy from the tidal currents. The huge investment and operational cost is required for the installment of tidal barrages technology and causes the adverse effects on the environment. On the other hand there is no need of huge capitals for the development of tidal current technology.

1.4.2 Tidal current technology

The tidal current technology is a different methodology as the tidal barrage technology that uses the head of water or potential energy. In this technology tidal power is produced from the water kinetic energy and construction of dam like structure does not require. The instram devices called the hydrokinetic turbine are directly installed in the water. The energy is extracted on the basis of two types of devices such as vertical axis and horizontal axis etc.

1.5 Thesis Scope and Objectives

Vibration in tidal current turbine is produced due to hydrodynamic forces. The vibration causes resonance and dynamic loads on the structures which leads to failure of the structures. The purpose of this research work is to investigate the integrity of the tidal current turbine (TCT) against the influence of vibration caused by the fluid forces. This research work will help to provide base line information to the device designer about the importance of vibration consideration in the design process. The proposed research work will be a contribution to the existing studies on the vibration behavior of TCT. This was met by the following objectives.

1. Computational fluid dynamic (CFD) analysis of tidal current turbine (TCT) operating in tidal channel.
2. Free vibration response of tidal current turbine (Natural frequencies, mode shapes)
3. Development of a computational methodology using FSI in ANSYS workbench to analyze the vibration response of a tidal current turbine under the influence of forces caused by tidal flow.

The research work is proceeded after the observation of higher tidal power resources throughout the country and theses resources may be used to complete the energy requirement as well as to sure the environmental protection from the power plant smoke. This research is the addition of already conducted research by the name of tidal current technology at International Islamic University, Islamabad. On the basis of continuous research, it is expected that after the result of this research IIU, Islamabad be capable to design its own experimental tidal current model and other requirement for the development of tidal current technology. Hopefully this research study will provide the solution of tidal energy to the national industry.

1.6 Research Methodology

The method of the research was software based. A 3-D model of the turbine rotor was developed in Autodesk Inventor Professional 2014. ANSYS workbench 17.0 software was used for the computational fluid dynamic analysis of the turbine rotor. The 3-D geometric model was converted in IGES type format and then imported to ANSYS workbench 17.0 for computational analysis in ANSYS CFX.

1 Exhaustive literature review was carried out on the computational fluid dynamic (CFD) and vibrational behavior of tidal turbine throughout the research work.

2 Investigation for the determination of natural frequencies and mode shapes have been carried out during the free vibration analysis.

3 Computational fluid dynamics (CFD) modeling of TCT operating in a tidal channel.

4 Fluid structure interaction (FSI) analysis of TCT under the influence of fluid loads.

CHAPTER 2

LITERATURE REVIEW

2.1 Types of Tidal Current Turbine

There are many types of tidal current turbine in the development stage. Most of which can be categorized based on whether they produces a motion in the rotary or linear direction. The main categories, into which the tidal current turbine falls, are vertical axis and horizontal axis tidal current turbine.

2.1.1 Horizontal Axis tidal Current Turbine (HATCT)

The main feature of horizontal axis tidal turbine is that, the rotational axis of the turbine is in line to the fluid flow. The rotor of such turbine is of propeller type turbine [11]. The HATCT have higher efficiency, but more complexes in design. The blades of HATCT required twist and taper to achieve the maximum efficiency. Typical peak efficiency range of HATCT is around 39 to 48% [12]. The tidal current turbine is shown in figure 2.1 [13].

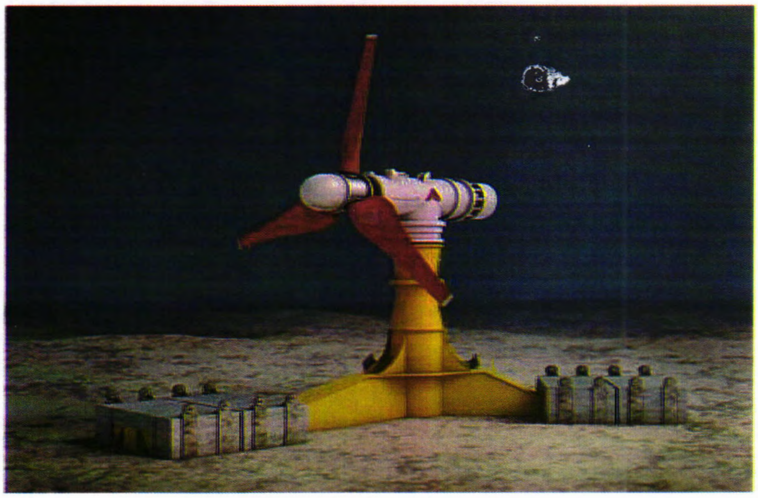


Figure 2.1: Horizontal Axis Tidal Current Turbine [13]

The design of horizontal turbine is same as the design of wind turbine. The HATCT is commonly used for tidal energy conversion. There are many forms of HATCT differing, on how many blades the rotor has and how the device is fixed in position. The most develop HATCT is the 1.2 MW seaGen developed by Marine current turbine (MCT) [14]. It is a first tidal current turbine that used on commercial scale to generate power in UK.

2.1.2 Vertical Axis tidal Current Turbine (VATCT)

VATCT also works in same way as the HATCT and extracts energy from the tidal current in a similar way to HATCT, but the rotational axis of vertical axis turbine are at perpendicular to the flow direction. In such devices both axis of rotation and surface of water are vertically to each other. The VATCT are shown in below figure 2.2.

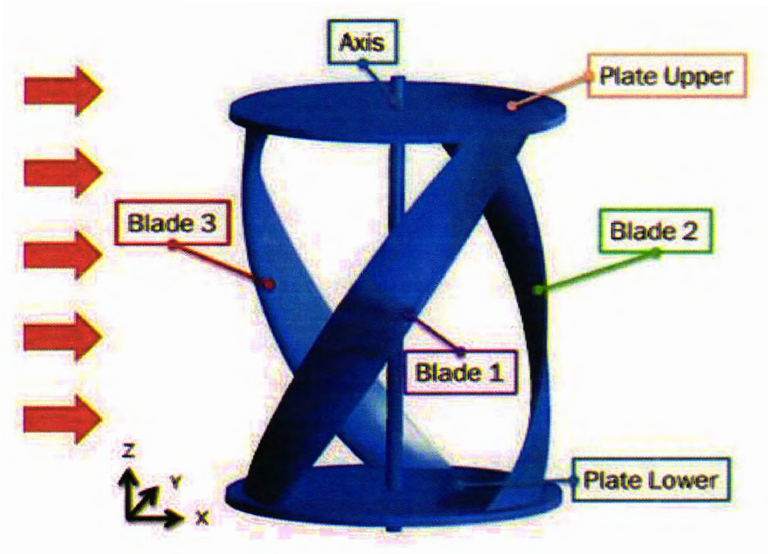


Figure 2.2: Vertical Axis Tidal Current Turbine [15]

The VATCT can be operating regardless of the directions of tidal current flow without loss of operational efficiency. Such turbines have straight blades reducing the design and manufacturing cost as compared to HATCT blades. The rotational velocity of VATCT is lower than HATCT due to which they produce less noise. But they produce less energy and they are less efficient. The efficiency range of VATCT is around 37 to 40% [16].

2.2 Turbine Location

The selection of proper location and proper type of turbine is very important for the extraction of maximum energy from the tidal currents. For the selection of turbine location the study about the weather conditions, depth of sea and installation mechanism is very important. In general a tidal current device is placed at the sea bed. However the device can be operating at varying depths with the application of floating mechanism to the device. The turbine can also be installing with a fixture near the free surface of water. The installation procedure must thoroughly be planned and well executed as poorly installed devices could result in large costs. For example, the estimated cost to rectify 164 poorly installed offshore wind turbines was £13 M [17]. The horizontal axis tidal current turbine (HATCT) is suitable for all location configurations with same success. Whereas, the vertical axis tidal current turbine (VATCT) can only be installed closer to the free surface. The reason for this is the preference of keeping the generator and other moisture sensitive parts out of the water.

2.2.1 Environmental Impacts

It is considered an optimal tidal current turbine configuration extract the maximum energy. The principal of extracting energy from moving water is same, as energy extracting from wind. However some fundamental differences may be consider with regard to density, compressibility and boundary condition. The characteristic behavior of the ocean depends on many factors such as depth, weather and water flow speed. These factors contribute to the decision of where to install the turbine that may only be suitable for a given depth and other environmental conditions. The design, installation and operation of several number of stream devices will have direct and indirect effects on the surrounding environment and on the life within it.

The tidal current devices are still in the development stage and hence there have been few studies into the environmental effects of these technologies. The studies which have been carried out up to date are mostly either based on predictions or are unverified [18], with the exception of Marine Current Turbines (MCT). The marine current turbine was independently studied by Haskoning, for the deployment of SeaGen [19]. This study found that there were no major environmental impacts as a result of the seaGen project.

The environmental impacts of both horizontal axis and vertical axis turbines for both orientations are mostly the same. Tidal turbine have no significant impact on the water level and depth after the installation. However it is important to consider the impact of such turbines on quality of water (Flow speed of downstream and upstream), transportation of sediments and risk of immediately whirling of water flow around the device that causes the scouring. Several other effects include noise pollution, the striking of sea habitants with moving parts or rotor blades, migration of ocean organism and generation of electromagnetic field [20]. No evidence has yet been found that electromagnetic fields from generators and cables affect the migratory and feeding habits of marine mammals and fish, however this is still a controversial area [21]. The effect on shipping is a concern and this may limit the location and size of device arrays. Pollution from the machinery used should also be considered such as the release of hydraulic fluids, lubricants, etc are very harm for the marine life [22]. There are also concerns of the cavitations effects produces in tidal current turbines, causing sudden pressure changes. This sudden pressure changes could harm the marine mammals and fish [23].

2.3 Advantages and Disadvantages

The evolution of tidal current technology is increasing. In this a effort have been done to explain the several issues and benefits of tidal current turbine. Advantages and disadvantages of vertical axis tidal current turbine (VATCT) and horizontal axis tidal current turbine (HATCT) based on technical features are given below.

2.3.1 Advantages of VATCT

The vertical axis tidal current turbine is an old technology. VATCT rotor rotates in vertical direction around its axis instead of horizontal direction. Advantages of vertical axis tidal current turbine are given below.

1) The vertical axis tidal current turbine has smaller cost of manufacturing, installation mechanism, and easy transport from one place to other.
2) They can produce electricity in any flow direction.
3) It is not required to change the direction of turbine with the variation of direction of flow so pitch mechanism and yaw drive is not needed.
4) The floating structure and mooring mechanism in of vertical axis turbine is very easy.
5) The blades of VATCT rotates relatively at low speed so they do not harm the marine life

2.3.2 Disadvantages of VATCT

1) These are very less efficient as compared to horizontal axis tidal turbine, because of the additional drag force during the rotation of blades.
2) These types of turbines are not typically well suited for use in areas where the flow velocity is high.
3) Such turbines have relatively high vibration due to the turbulent flow of water.
4) Maintenance cost is high due to the increase of vibration bearing wear.
5) Such turbines produce noise pollution.

2.3.3 Advantages of HATCT

In horizontal axis tidal turbine axis of the rotor and flow stream are parallel to each other. Multi blades HATCT are using now a days. Many drawbacks of vertical axis tidal current turbine can be eliminated from horizontal axis tidal turbine. Some benefits of such turbines are listed below.

1) The HATCT collects maximum amount of tidal energy by allowing the angle of attack to be remotely adjusted.
2) The horizontal axis tidal current turbines are self-starting.
3) It is more cheaper because of higher production volume.
4) The HATCT can be controls easily for both stall regulated and active pitch control. Both controls provide flexibility against over speeding and ensure efficient operation.

2.3.4 Disadvantages of HATCT

1) They have complicated design and blade manufacturing is very difficult.
2) Placement of moisture sensitive parts, generator and gear box is under water.
3) It is difficult to place the power supply cables for HATCT under the water.
4) Installation procedure is relatively difficult due to the transportation of long blades and tall tower.

2.4 Previous Study on Performance prediction of TCT

Now a days variety of tidal current devices have been developed, However closest to commercially available turbines are usually open-bladed horizontal axis turbines, such as current installed turbines for tidal current generation Ltd's Deep Gen, SeaGen and Andritz Hydro Hammerfest's HS1000 are horizontal axis turbines (Renewable UK, 2012) [24]. The "Seaflow" is considered as the first tidal energy project throughout the world on commercial scale [25]. In 2008 a new commercial 1.2 Mega Watt turbine "SeaGen" was developed by a company known as marine current turbine and then installed in Strangford lough, Northern Ireland that is connected to the national grid station for eletricity [26]. The technology of both "SeaGen" and "Seaflow" is similar. The Marine Current Turbine limited tried to introduce the sea current turbine into the market for the future commercial purpose. The company aim was to do experiment on this turbine on large scale [26]. Verdant Power Company developed a turbine for the Roosevelt Island tidal energy (RITE) project in New York east river [27].

A tidal current turbine (TCT) system utilizes the tidal current and kinetic energy is converted into the rotational energy of turbine to produce the electricity. Therefore, turbine design and performance verification is important in developing TCT systems. The capability to develop the optimal tidal current turbines (TCT) design and prediction of turbine performance and life is critical for the success of the tidal current turbine industry. The performance of a TCT can be predicted in two ways. One way is the experimental procedure, which uses the towing tank and circulating water channel (CWC). Bahaj and Joo et al. [28] developed an experimental setup for the investigation of the performance of tidal current turbine, and Jo et al. [29] carried out the experimentation for the comparison of three different types of tidal turbines. National Nature Science Foundation of China funded the Zhejiang University (ZJU) in 2005 for the investigation of horizontal axis tidal turbine and installed a 5 kW model in 2006 [30]. It was ability this model to produce 2 KW power at a 1.8 m/s flow speed of water. A multi blade turbine system was developed by Northeast Normal University (NNU) for the generation of electricity at lower flow rate (< 1 m/s) and it was predicted the efficiency of can be improved by reducing the power loss at the rotating seal [31]. In January, 2013 IEEE student member developed the experimentation as well as numerical simulation model for the determination of the performance of tidal current turbine [32] The performance of turbine was evaluated on the basis of efficiency of the entire design. The turbine was installed in a circulating water channel and the torque, lift, drag, and thrust were calculated at different flow velocity. The dynamometer was used to measure these values, from which the performance of the entire system can be calculated [33].

The performance of horizontal tidal turbine can be estimated on the basis of two approaches, computational fluid dynamics and blade elements momentum theory [34]. The blade element momentum theory is based on two theories that are blade element theory and momentum theory. Momentum theory is based on the principle of conservation of momentum in which control volume analysis of the forces at the blade is considered. On the other hand in Blade element theory a specific section of blade is selected for the analysis of forces. The performance of Horizontal tidal turbine is predicted from BEMT after combining the results of these two theories. However in Computational Fluid Dynamics (CFD) a commercially available tool such as ANSYS is used to predict the performance of tidal turbine after the examination the fluid flow along the turbine rotor three dimensionally. Velocity and pressure variation around the blade surface can be examined in pictorial view. This theory is also used to investigate the cavitation (pressure on the blade surface reduces below the vapor pressure of fluid) initiation on the blade area of tidal turbine [35].

Many earlier researches have been carried out for the investigation of performance on the basis of blade element momentum theory Batten et al. [36], Campos and Baltazar [37] and Bum-Suk Kim et al [38] designed a 1 MW ocean current turbine on the basis of blade element momentum theory (BEMT) and computational fluid dynamics (CFD) considering cavitation effects to predict the performance of the turbine. According to the performance prediction the blades for ocean current turbine can experience cavitation near the tip area according to its installation depth and rotational speed. In 2012 the 26th International Association of hydro-environmental and research (IAHR), held a symposium at Tsinghua university Beijing China on hydraulic machinery. They discussed the structural safety requirements and prediction of performance of 50 KW ocean current turbines in this conference. The unsteady CFD simulations of the rotor configuration of turbine is presented for the prediction of cavitation and performance [39]. The wake effect on the performance of horizontal axis tidal Current turbine was studied by Kang-Hee Lee et al. [30], and they investigated the interference of tidal turbines for axial configuration through experimentation and CFD simulation. In a current study Co-workers and McSherry used CFD to model horizontal axis marine current turbine (HAMCT) performance, and it was concluded that the proper meshing is so much important for better result [40].

The development of experimental setup for the actual performance of tidal turbine design is more reliable but for this purpose a lot of experience, time and huge capital is required. However numerical analysis not only reduces the time for analysis and almost no cost is required for analysis. For this reason many industries of fluid machinery are using the numerical analysis [41]. Also the scaling issue is not applying in numerical methods, so the model may be specified at any dimension [42]. The BEMT approach was adapted from wind turbine technologies and has low computational demand as compared to CFD but due to assumption of uniform flow, the result of BEMT is not reliable and accurate as compared to CFD.

2.5 Previous Study on FEM Analysis of TCT

Extensive researches have been carried out to investigate the structural loading effect on wind and gas turbine blade. Modal vibration analysis of rotating turbine for the comparison of two experimental approaches on the basis of dynamic response of turbine was developed by Tcherniak, S. Y. D. and Allen, Matthew S. [43]. Edgewise mode shapes of the blade on the basis of time and blade variation have been examined in this study. Alshroof, Osama N. et al., [44] used the commercial software ANSYS 12.1 for the development of the one way fluid structure interaction (FSI) model of turbine blade and investigated the interaction between the blade motion and pressure on the turbine surface. The accurate motion of vibrating blade was considered and pressure around the blade is also investigated. The results show that there is no significant variation of pressure in between rotor blade tip and casing surface. Gunjit, B., and Jonkman, J. [45] also carried out the dynamic analysis of wind turbine by considering the three configurations of turbine floating platform supported turbine, land based turbine and floating platform supported turbine. It is observed that the influence of elastic foundation and hydro-dynamic plays significant role in dynamic analysis.

Flap-wise and edgewise model on the basis of variable length (different position of blade) of wind turbine blade was developed by Tartibu, L. K., et al., [46]. Torsional natural frequencies for ten different position of blade is investigated. A program for the prediction of natural frequencies is also developed in MATLAB. Results show that natural frequency does not remain constant due to the variation of the length of the blade and there is no cause of resonance. On the basis of finite element analysis (FEA) Ashwani K., et al., carried out the vibration analysis of wind turbine blade made of Al 2024 [47]. Modal and structural analysis was also performed on the ANSYS 14.0. The analysis results were compared with the experimental results and the recommended for the use. Similarly linear vibration analysis of rotating blade on the basis of degree of freedom (DOF) and linearized equation of motion was performed by Park, J-H., et al., [48]. Constrained multi body approach is used for the linearization and derivation of EOM and multi degree of freedom system. The proposed method is recommended for the prediction of vibration and dynamic response of rotating blade. Numerical problems were solved to verify the accuracy of the proposed method. The proposed study will be conducted on the characterization of tidal current turbine dynamics by using fluid structure interaction (FSI).

Chapter 3

Numerical Modeling and Hydrodynamic of TCT

3.1 Modeling

Modeling of a full scale tidal current turbine numerically or by scaled prototype is a fundamental part of the design process, providing developers and investors with estimates of loads, power and flow characteristics. This section considers different modeling techniques and hydrodynamics of tidal current turbine.

3.1.1 Physical Modeling

The physical scale modeling has been carried out in towing tanks, recirculating flumes, rivers and offshore. The towing tank experiment has been carried out by Faudot et al. in 2013 and Clarke et al in 2008 [49]. In this experiment they describe and analyze the behavior of 1 MW horizontal axis tidal current turbine subjected to runaway situation. The results obtained from the Towing tank tests can provide a controlled environment but can only be used with plug flows and cannot simulate the velocity profiles. The recirculating flumes experiment has been carried out by several research groups, Tedd and Olczak in 2013 and they include the velocity profiles [50]. However the recirculating flumes have smaller cross section than towing tanks and therefore the rotor diameter can be limited by the blockage effect.

The river and offshore testing include work by Orme and Masters in 2004 and Starzmann in 2013 [51]. Unlike the towing tank and recirculating flumes,river and offshore testing does not provide a controlled environment. Therefore it is less suitable for validating the numerical models, but it gives an insight into the effect of variable flow velocity variable turbulence and provide valid information regarding the performance of tidal current turbine in a more realistic environment.

3.2 Numerical Modeling

In order to reduce the time and cost to move from initial concept to commercial development of TCT, the numerical modeling can be used to reduce the number of design iteration in the described physical prototype. The numerical approach depends on time and computational resources and is often based on one or more of the following methods.

3.2.1 Blade element momentum Theory

In 1892 Drzwiecki proposed the blade element momentum theory (BEM) for the investigation of airplane propeller. It is faster and very less expensive computationally as compared to other numerical methods. BEMT is considering as a basis for the commercial wind and tidal current turbine design, due to the number of design iterations that can be performed in a very short period of time.

BEM theory is the combination of the drag and lift coefficient of a blade profile (blade element theory) and momentum theory. According to the blade element theory blade is subdivided into several small parts by considering that these can move hydrodynamically independent from surrounding elements as 2D airfoils. The total momentum and forces experienced on the tidal turbine may be estimated from the resultant forces of all elemental forces in the direction of the rotating blade. On the other hand, the loss of momentum in rotor due to the work done by the water flow on blade element is assumed. With BEM theory, the induced velocities can be calculated from the loss of momentum in the axial and tangential flow directions. It is because of the effect of the inflow in rotor plane by induced velocities and this also alter the calculated forces.

The blade element momentum theory is based on two theories that are blade element theory and momentum theory. Momentum theory is based on the principle of conservation of momentum in which control volume analysis of the forces at the blade is considered. On the other hand in Blade element theory a specific section of blade is selected for the analysis of forces in which the blade is split into a number of segments and each section of the blade is analyzed separately. The BEMT is faster and very less expensive computationally as compared to other numerical methods. And it is considering as a basis for the commercial wind and tidal current turbine design, due to the number of design iterations that can be performed in a very short period of time. With BEM theory it is possible to calculate the steady loads, thrust and power for different setting of flow speed, rotational speed and different pitch angle.

3.2.2 Computational Fluid Dynamics (CFD)

Computational fluid dynamics (CFD) is the branch of fluid mechanics which deals the algorithms and numerical methods to analyze and solve the problems that involves fluid flows. This method can give more detailed information and more accurate results of forces on tidal current turbine. CFD is a numerical tool that is used to analyze the horizontal axis tidal current turbine (HATCT) through discretsation method. This has the advantage of saving time and the cost associated for the initial design of tidal current turbine. For all the work presented in this thesis, the available software ANSYS CFX is used for the CFD analysis in which the finite volume method is used to solve the governing equation for a fluid flow for a predefined or user defined material properties for 2D and 3D domains. Finite volume method (FVM) is the standard and classical approach and most widely used in research codes and commercial software. In FVM a set of partial differential equation of Navier stroke equation is converted in conservative form and then discretize this equation. All the governing equations are solved on discrete control volumes.

The CFD code is used for the working of CFD. These codes are developed from the algorithms that are used to solve a problem. The user guide and input parameters are defined in all commercially available CFD packages. So it is easy to use this software for problem solving. Hence the simulation process of any problem is completed in four different phases.

1. Design of Geometry and Meshing
2. Assigning the Physics of the geometry.
3. Problem solution
4. Analyzing the results in postprocessor

3.2.2.1 Geometry and Mesh generation

Geometric modeling and mesh generation are the pre-processing processes. First of all a model is designed and mesh is generated to provide an input to preprocessor. The geometry can be designed in any design software and mesh is generation of mesh is done using meshing command. In geometry the region of interest can be defined, the surface, boundary name and the region of the fluid flow can be specified. In ANSYS CFX the designed model in native format can also be imported from Major CAD software, and then meshing of control volume is generated automatically.

3.2.2.2 Assigning Physics of the Model

This step is also the pre-processing stage that is used to provide the input to the solver. In physics preprocessor the mesh geometry is loaded. The physical models for the simulation is selected and then boundary condition and fluid properties are assigned in this step.

3.2.2.3 Solving the CFD problem

CFD problem is solved in solver that produces the results in a non-interactive process. The CFD solver integrated control volume of interested area in the form of partial differential equations. Then this set of integral eqns. is converted into a system of algebraic eqns. And the algebraic equations are then solved iteratively.

3.2.2.4 Analyzing the results in postprocessor

The postprocessor is the CFD element used to visualize, analyze and present the results of the simulation. Any result obtained from initial values to complex animated sequences is included in postprocessor. The postprocessor of the CFD includes grid and geometry display, line plots, vector and shaded contour plots, two and three dimensional surface plots, particle tracking and scalar variable such as pressure, speed and temperature and charts showing graphical plots of variables.

3.3 Fluid Structure Interaction (FSI)

Fluid-structure interactions (FSI), is the interfaces of some deformable and moving structure with a surrounding or internal fluid flow. They are the most challenging multiphysics problems. In order to gain an insight and correct information of the hydro elastic behavior of a tidal current turbine, actual experimentation can be required to visualize and quantify the behavior of the device under different circumstances. However the experimental investigation of tidal current turbine in Pakistan is time consuming, extremely costly and difficult to develop due to the limited technology in order to gain useful and credible information from the experimental setup. Alternatively, computational analysis has the ability to be much less expensive, with regard to both money and time and have more accurate results.

Hence when a deformable solid structure interacts with internal fluid pressure is exerted because of fluid that causes the deformation in structure and fluid flow alters then Fluid-structure interactions (FSI) occurs. It is highly suitable to integrate the both structure analysis and CFD analysis to solve the complex problem. Mainly three methods of joint fluid structure are interacted in time domain modeling, that are solving the main equation in integrated, uncoupled or coupled manner. The uncoupled simulation is computationally cheap but is limited to a small perturbation and minimum non-linearities. The coupled method are solving the equation separately but in an iterative manner, while integrated method solves both the fluid and structural equations simultaneously [52].

The coupled method is further divided into one and two way fluid structure interactions. The information exchange depends upon the coupling method. In one way fluid structure interaction, only fluid pressure over the structure is transferred to the structure solver. However for two way coupling analysis, displacement of structure is transferred from structural solver to fluid solver, and then the result is analyzed. The two way FSI solution is more reliable, especially for higher deflection where the fluid field is highly influenced by structural deformation. But they have more computational cost as compared to one way FSI. And also the deformation of fluid mesh needs to be calculated which is a tedious job. But on the other hand one way FSI is computationally very less expensive and also they provide a mesh of constant quality, so fluid mesh deformation is don't need to be calculated.

In this research work one way fluid structure interaction (FSI) method was used. CFD models were coupled with FEA models to predict deformation of the blades and the resulting change in hydrodynamic forces. The hydrodynamic forces were calculated by CFD and it was transferred to FEA model where TCT deformations were then calculated.

3.4 Hydrodynamic of Tidal Current Turbine

3.4.1 Turbine Power

The energy of tidal current can be extracted from the blades of tidal turbine. Theoretical and actual power of the blade can be calculated using the formulas [53].

$P_{theoretical} = \frac{1}{2} \rho A V^3$ -- (1)

$P_{actual} = T * W$ -- (2)

Where we have in equation (1)

"P" is power in watt

"$A = \frac{\pi}{4} * D^2$ " is Turbine Area =196.25 mm^2

"ρ =1025" is the density of water in kg/m^3

"V =1" is the water velocity in m/s

And in equation (2) we have

"T" is torque of the turbine

"W" is the angular velocity for specific tip speed ratio (TSR).

By substituting the above values in equation (1), the theoretical power of the turbine can be calculated which is,

$P_{theoretical} = 100.57$ watt.

$P_{theoretical}$ is the power available in tidal current flow. And the actual power of the turbine is calculated by taking the product of the coefficient of power and the theoretical power, or by using equation 2 [53].

3.4.2 Tip Speed Ratio (TSR)

The tip speed ratio is the ratio of the actual water velocity of the blade tip to the rotational speed at the tip of the blade. It is characterized by a non-dimensional factor known as tip speed ratio (TSR) or lambda. We can calculate the tip speed ratio (TSR) from below formula [53].

$$TSR = \frac{\text{Blade tip speed}}{\text{Incomminf flow speed}}$$

$$\lambda = \frac{wr}{v} \text{ -- } (3)$$

where

"w" is the rotational or angular speed of the turbine (rad/sec)

"r" is the radius of the turbine = 0.25 (m)

3.4.3 Angular Speed

The speed at which the tidal current turbine rotates to convert the tidal energy into electricity is known as rotational speed. The rotational speed can be calculated by using equation 3. The angular speed can be calculated by changing the tip speed ratio value from two to seven respectively, while the incoming flow speed and radius of turbine is kept constant. The values of angular speed for different tip speed ratio are shown in table 3.1.

3.4.4 Torque Calculation

The torque values were calculated from the function calculator of ANSYS CFX. These values were calculating by selecting each blade and hub independently. The torque was calculated for six repetitive analysis. The net torque values for different TSR and different angular velocity are shown in table 3.1

3.4.5 Calculating actual power

The actual power of the turbine was then calculating by multiplying the net torque to the rotational speed of the turbine. But it should be noted that the theoretical upstream power is 59.3 % (Betz limit) of the available upstream power [53]. The actual power calculated by using equation 3 for six repetitive calculations is shown in table 3.1.

3.4.6 Power Co-efficient (Cp)

The power co-efficient (Cp) is not a fixed or static value. Its value changes with the with the change of TSR of the turbine. The power coefficient is used to measure the efficiency of tidal current turbine. It is the ratio of actual power produced by a tidal current turbine at a specific speed divided by the theoretical power of the tidal current turbine. The power coefficient (Cp) can be calculated by using the following formula,

$$Cp = \frac{Actualpower\ of\ turbine}{Theoritical\ power\ of\ turbine}$$

$$Cp = \frac{\text{Pactual}}{\text{Ptheoretical}} \quad \text{--} \quad (4)$$

The actual power of the turbine was calculated by using equation 2, and shown in table 3.1. While the theoretical power of the turbine was calculated by using equation 2. By putting these values in equation 4 the power coefficient is calculated for the tidal turbine and is shown in table 3.1.

Tip speed ratio (λ)	Angular velocity (Rad/sec)	Torque (N.m)	Power Actual (Watt)	Power coefficient (Cp)
2	8	0.52	4.16	0.0414
3	12	1.17	14.04	0.14
4	16	1.83	29.28	0.291
5	20	2.00	40	0.398
6	24	1.72	41.28	0.41
7	28	1.23	34.44	0.343

Table 3.1: Hydrodynamics of tidal Current Turbine

As discussed in section 2.1.1 the peak efficiency value of the tidal current turbine is in range of 39 to 48%. So the values shown in table 3.1 are in the prescribed range. And the maximum efficiency of the tidal turbine used in this work is 41%.

CHAPTER 4

CFD Analysis and Results

4.1 Turbine Design Parameter

The HARP_Opt (Horizontal Axis Rotor Performance Optimization) code is used in this research work for BEM Design of the turbine. The HARP_Opt uses a multiple objective blade element momentum (BEM) and optimization algorithm flow model to design the tidal current turbine. Optimization algorithms to perform optimization and the WT_Perf BEM theory code to predict rotor performance metrics.

Various types of rotor control arrangements may be constructed using HARP_Opt, such as variable or fixed rotor speed and variable or fixed blade pitch operation. Turbine blades having non-circular or circular roots may be designed using HARP_Opt. In this thesis fixed speed and fixed pitch configuration were used for the designed turbine. The turbine design parameters used in this work are given in below table 4.1 and table 4.2 shows the BEM chord and twist distributions for the blade.

Design Parameters	**Values**
P: Rated Power (W)	36.23
Cp: Estimated power coefficient	0.4
Efficiency	0.9
Rated current velocity (m/sec)	1.0
Sea water density (kg/m^3)	1025
Tip speed ratio	5
Turbine diameter (m)	0.5
Number of Blades (EA)	3
Angular velocity (RPM)	191

Table 4.1: Turbine Design Parameters

Fig 4.1: Solid 3D model of TCT

station	r/R	radius (mm)	Chord (m)	Twist Angle (degrees)
1	0.22	0.055	0.055364	16.95306
2	0.26	0.065	0.053791	15.57584
3	0.3	0.075	0.052075	14.27066
4	0.34	0.085	0.050243	13.04085
5	0.38	0.095	0.048318	11.88887
6	0.42	0.105	0.046322	10.8164
7	0.46	0.115	0.044273	9.824262
8	0.5	0.125	0.042188	8.912484
9	0.54	0.135	0.040077	8.080252
10	0.58	0.145	0.037953	7.325935
11	0.62	0.155	0.035822	6.647077
12	0.66	0.165	0.033689	6.040399
13	0.7	0.175	0.031556	5.501798
14	0.74	0.185	0.029421	5.026349
15	0.78	0.195	0.027282	4.608302

16	0.82	0.205	0.025131	4.241085
17	0.86	0.215	0.02296	3.917302
18	0.9	0.225	0.020756	3.628732
19	0.94	0.235	0.018505	3.366334
20	0.98	0.245	0.016189	3.120241

Table 4.2: Blade Chord and twist Distribution results

4.2 Computational Fluid Dynamic Analysis of Turbine

ANSYS workbench 17.0 software was used for the computational fluid dynamic analysis of the turbine rotor. The three dimensional geometric model of the turbine rotor designed in Autodesk Inventor Professional 2014 was converted in an IGES type format file. The IGES format geometry is then imported to ANSYS workbench 17.0 for processing in ANSYS CFX.

The ANSYS CFX is robust, flexible general purpose computational fluid dynamics software packages based on finite volume method, and used to solve wide ranging fluid flow problems of all level. The ANSYS CFX process consists of three primary steps. The first step consists of pre-processing. This step helps in describing the geometry in best possible manner. The fluid domain of interest and mesh can be generated in the pre-processing step. The second steps the solver setting. In this step the fluid material properties, flow physical model and different boundary condition are set to solve the problem. The third step is the post processing step. In the post processing step the results can be obtained. And these results can be analyzed by different methods like contour plots, vector plots and streamlines.

The ANSYSCFX was used for the analysis in this research work. The geometric designed model was imported and then boundary conditions were assigned. Through the model cell ANSYS CFX is launched and the solution is performed.

4.3 Mesh Generation

Mesh generation is the most important step in computational fluid dynamic analysis. It is preferred to generate the fine meshing to increase the number of elements and nodes because accuracy of results increases with the increase of number of elements and nodes however higher number of meshing elements increase the run time of the simulation. ANSYS ICEM CFD is enable to give the solution of these problem as it provides a standard method to generate required meshing for the simulation. Therefore ICEM CFD was used in this research work for high quality meshing to obtain more accurate results. The geometry setup and the number of elements and nodes of each part are given below. The mesh setup of each part is given below.

Name	Mesh Type	No. Nodes	No. Element
Blades	Tetrahedral	87091	173800
Hub	Tetrahedral	12175	23674
Inner_D	Unstructured	13374	25952
Outer_D	Unstructured	10992	20752

Table 4.3: Mesh setup for CFD analysis

The ANSYS ICEM CFD tool was used to generate mesh on the whole geometry blades, hub, circular domain and external main domain. The mesh of each part is shown in figure 4.2 to 4.5.

Fig. 4.2: Grid system of blade

Fig. 4.3 Grid system of Hub

Fig. 4.4 Grid system of Circular Domain

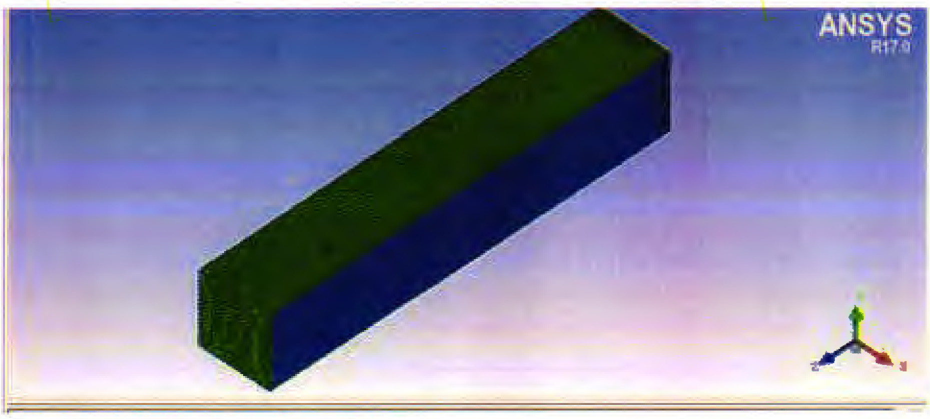


Fig. 4.5 Grid system of External Domain

4.4 Boundary condition

In order to complete CFD simulations, it is necessary to specify the boundary conditions to CFD domain. The results can be obtained from the CFD post on the basis of applied boundary conditions. The boundary conditions used in this thesis is based on the circulating water channel (CWC) specifications. The specifications of the circulating water channel are shown in below table 4.4.

	Measuring section	Main particular
Length (m)	2.3	6.0
Height (m)	0.9	3.0
Breadth (m)	1.0	1.0
Max.velocity (m/s)	-	1.2

Table 4.4: Specification of circulating water channel

The circulating water channel is used to experimentally predict the performance of tidal current turbine. They can obtain more reliable and accurate results, but is expensive and required a great deal of experiment and time. On the other hand, numerical analysis can obtain various results at low cost and is used in a variety of fields. Therefore in this research work the performance of tidal current turbine was predicted numerically using ANSYS CFX.

4.4.1 Domain Specification

The external domain is modeled in a rectangular shape similar to the measuring section of circulating water channel of length 5.0 meter, width 1.0 meter and height 0.8 meter. And the internal rotation area where the turbine rotates is a cylinder of diameter 0.6 meter and height of 0.11meter as shown in figure 4.6 below.

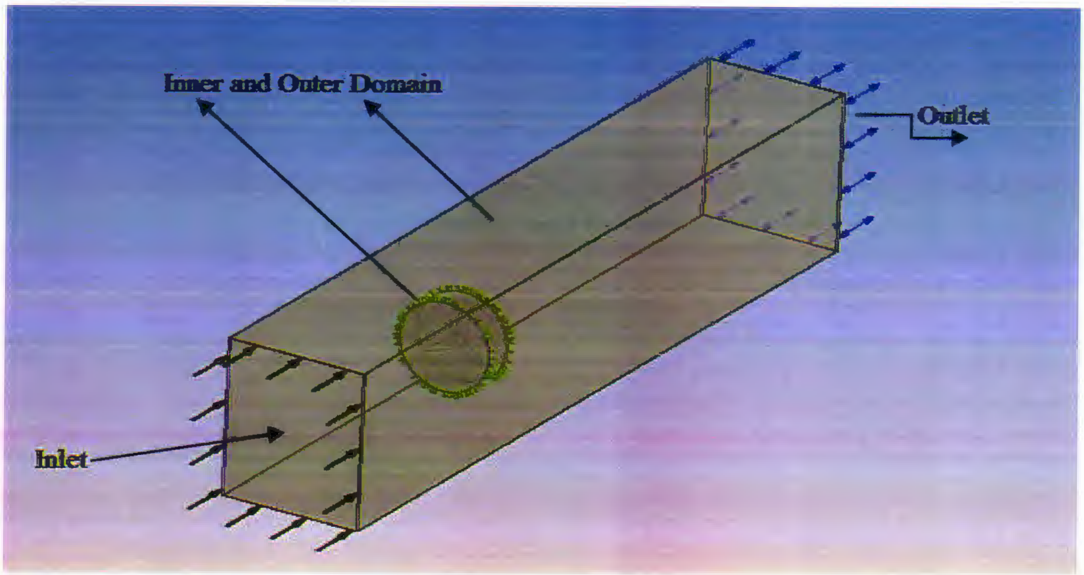


Fig 4.6: Specification of external and internal domain

4.4.2 Inlet

The inlet to the domain was set as a velocity inlet with the velocity specification method as magnitude, normal to boundary, which resulted in the flow being aligned with the rotational axis of the turbine. The magnitude of design velocity for all model presented in this thesis is 1 m/s, and the turbine diameter is 0.5 m by taking the specification of the circulating water channel given in table 4.4.

4.4.3 Outlet

The external domain outlet area is on the basis of an opening condition and it is calculated from the change of flux because of the turbine. The mass and momentum option were selected to opening pressure and direction the relative pressure were set to "0" Pascal. The flow direction specification method was set as normal to boundary.

4.4.4 Walls

The sides and floors of the external domain used the wall condition. The mass and momentum option were chosen to no slip wall condition and the wall roughness option were chosen to smooth wall condition. The top of the external domain used the wall condition and in boundary detail tab the mass and momentum option were selected to free slip wall condition

4.4.5 Turbine Blades

All the blades of the turbine were used the wall condition. The mass and momentum option were chosen to no slip wall condition and the wall roughness option were chosen to smooth wall condition.

4.4.6 Interfaces

The three types of interfaces were defined in both circular and external main domain. All the three interfaces include the interface_Circular, Interface_Inlet and interface_oulet. The interface type was selected to fluid-fluid interface in all interfaces. The interface model option was set to general connection, and the frame change or mixing model option were set to frozen rotor. The frozen rotors were selected because in rotating analysis it reduces the simulation time and produce more appropriate solution. The pitch change option was set to automatic, and the mesh connection method was set to general grid interface (GGI) method. The GGI is a coupling interface used for joining two faces of non-conformal meshes, in which the mesh points on either side do not match with each other. So instead of remeshing the GGI connection is used to match and connect the meshes of both sides for the simulation.

4.4.7 Rotation of Turbine Using MRF

In situation where the rotating body moves through the fluid, a moving reference frame (MRF) or dynamic mesh model is required. The dynamic mesh models are very complex and have very high computational cost. Therefore instead of dynamic mesh, the MRF model was applied to internal domain. The rotation of the turbine was simulated by selecting frame motion in the cell zone conditions for the circular domain named turbine. The rotation-axis origin was set as 0, 0, 0 and the rotation-axis direction was along the z axis. The rotational velocity was varied to cover the operational range of the turbine while the translational velocity was set to 0 ms^{-1} in all 3 directions.

4.5 Solver control setting

ANSYS-CFX solver solves all the solution variables for the simulation for problem specification which is generated in CFX-Pre. One of the most important features of ANSYS-CFX solver is the use of coupled solver as a single system. The coupled solver is faster and required fewer iterations to converge the flow solution. In solver control setting, the minimum iteration was set to 1 while the maximum iteration was set to 200 with auto time scale. The residuals, convergence criteria was set to 1.E^-4.

4.6 CFD Post processing

4.6.1 Performance Curve of TCT for CFD validation

The full rotor (three blades and hub) were analyzed through BEM and CFD method in this research work to predict the torque, pressure and velocity streamlines occurring on the blade profile. The performance curve of tidal current turbine on a 1.0 m/s design velocity is shown in figure 4.7. The coefficient of power values were calculated from BEM analysis and from CFD analysis as discussed in section 3.4.4. The torque values were calculated from the function calculator of ANSYS-CFX for seven repetitive analysis. These seven analysis were performed for a tip speed ratio (TSR) 2 to 8.

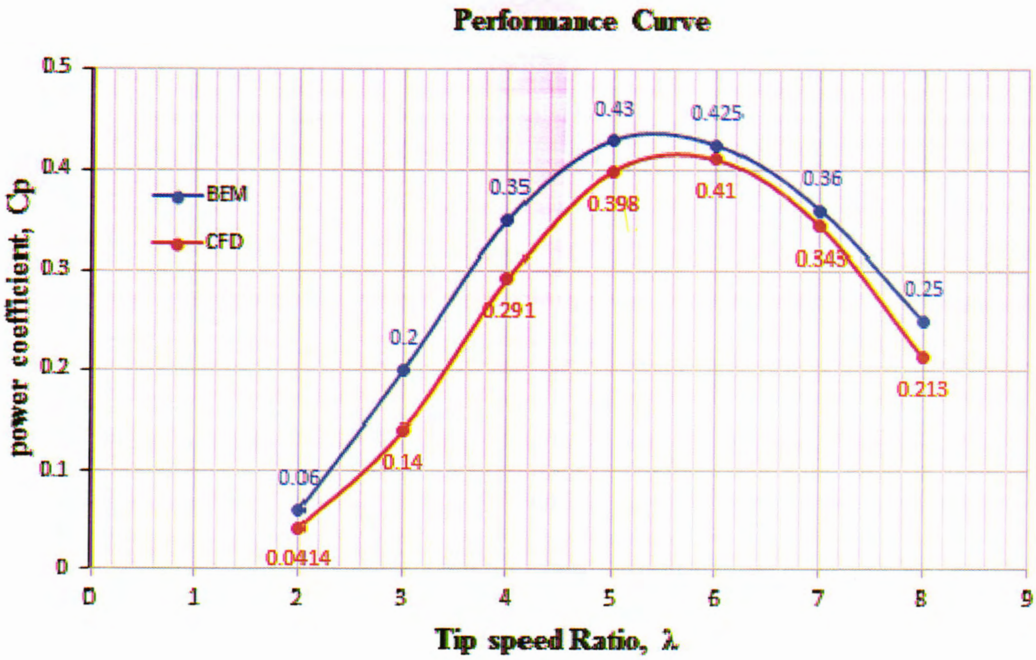


Fig. 4.7: Performance Curve of TCT

The figure shows the coefficient of power values increasing linearly both for BEM and CFD analysis at tip speed ratio (TSR) 2, 3 and 4 respectively. And the maximum power coefficient (Cp) value 0.43 occurs at a TSR value 5 in BEM analysis, while maximum Cp value 0.41 occurs at a tip speed ratio of 6 in CFD analysis. Passing through the TSR 5 and 6, the power coefficient (Cp) value starts decreasing by increasing the TSR in both cases. Hence the BEM and CFD analysis have the same trend and creates equal power, so the BEM result validate the CFD result of this research work. Also According to Betz's law no turbine can capture more than 59.3 % of kinetic energy (KE). And the peak efficiency value of the tidal current turbine is in range of 39 to 48%. It is clear from the figure and above discussion that, the BEM and CFD have the same result for tidal current turbine designed in this research work and creates reasonable estimates of power output.

4.6.2 Pressure at Front Side of the Turbine

The pressure at front side of the tidal current turbine is shown in below figure 4.8 for a tip speed ratio (TSR) 2 to 7. The figure shows that, at a lower TSR the fluid is rapidly flowing on the leading edge of the rotor which produces high pressure at the tip area. And the pressure values increases from root to tip at leading edge at TSR value of 2, 3, 4 and respectively. By increasing the TSR value further reduces the pressure area which causes to negative pressure. The negative pressure occurs at TSR value of 6 and 7 respectively. The negative pressure causes to produce less torque in tidal current turbine, which reduces the power coefficient (Cp) of the tidal current turbine. So at higher value of tip speed ratio (TSR) the efficiency of the tidal current turbine is minimum.

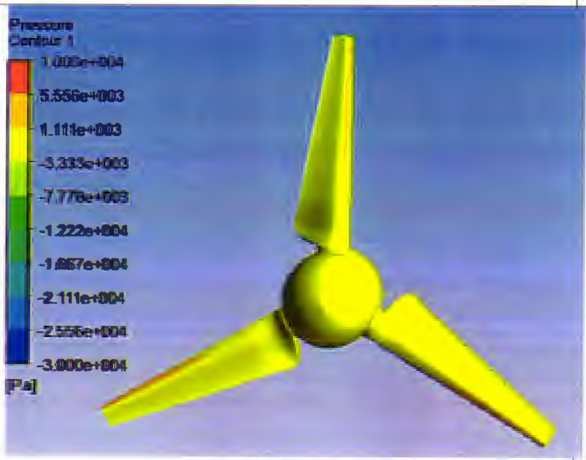
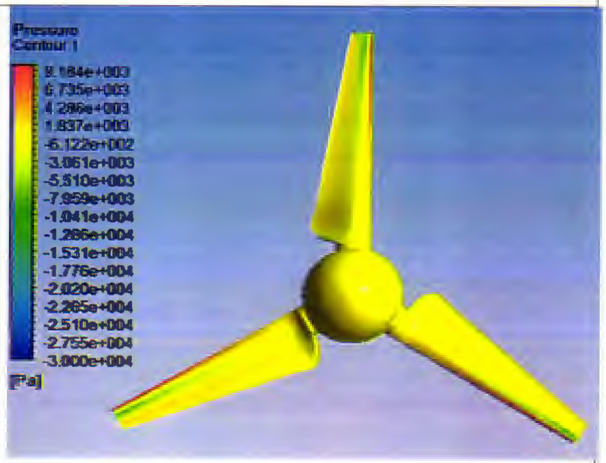
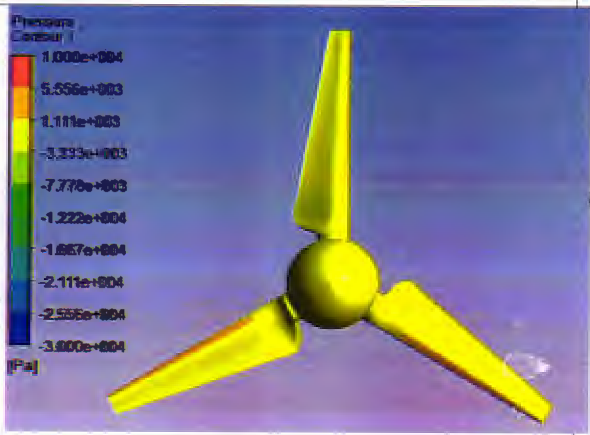
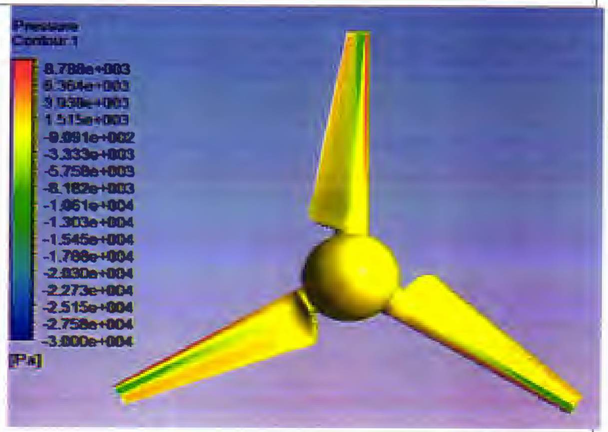
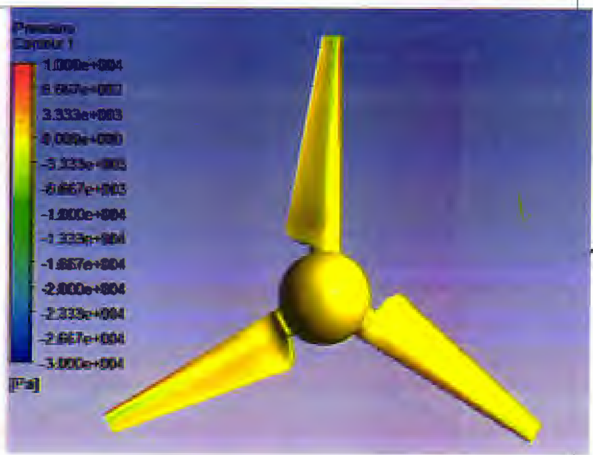
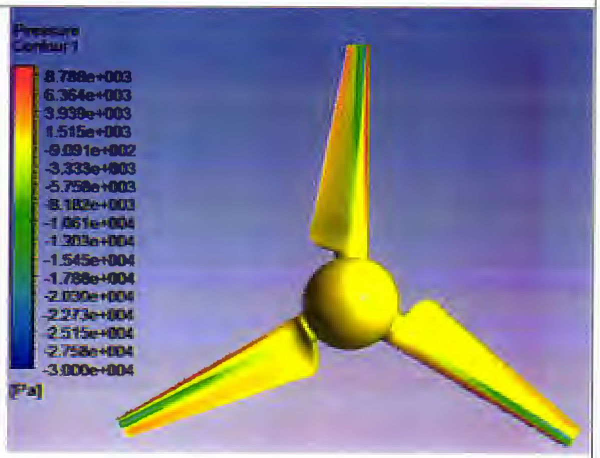
λ	Pressure	λ	Pressure
2		5	
3		6	
4		7	

Fig. 4.8: Rotor pressure distributions (Pressure side).

4.6.3 Pressure at Back Side of the Turbine

The pressure at back side or suction side of the rotor is shown in figure 4.9. The pressure at back side or suction side is also plotted for several values of tip speed ratio (TSR) 2 to 7. The figure shows the increase of negative pressure up to the tip speed ratio of 5 and then steadily slow down.

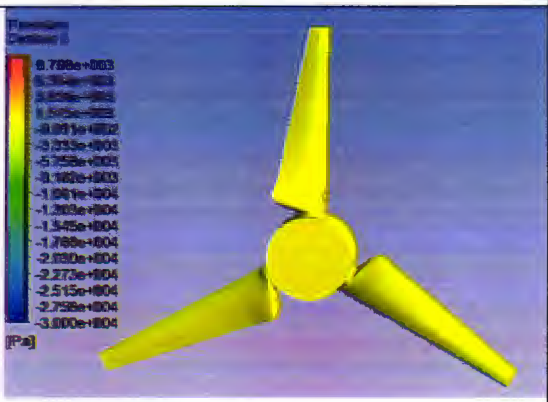
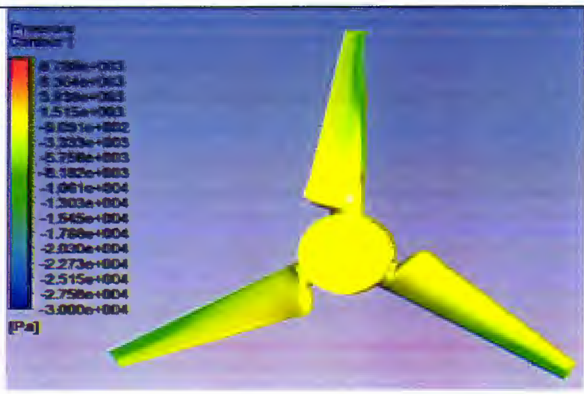
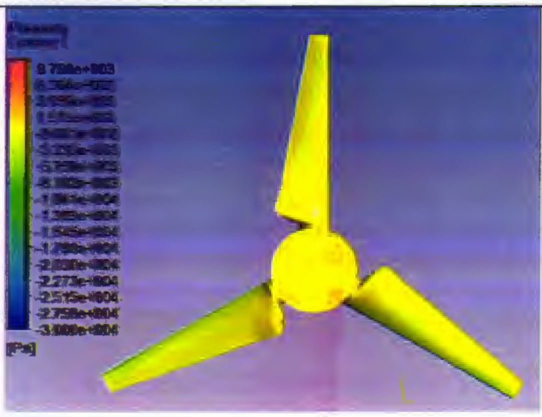
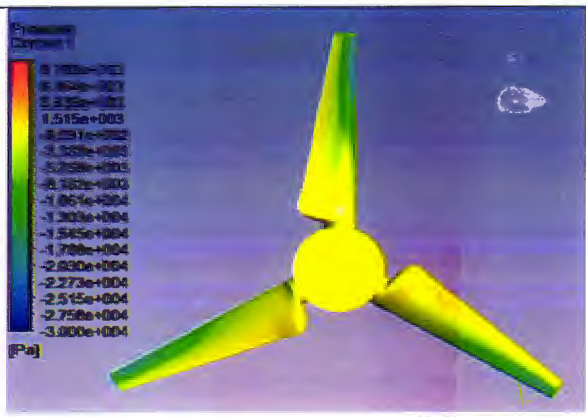
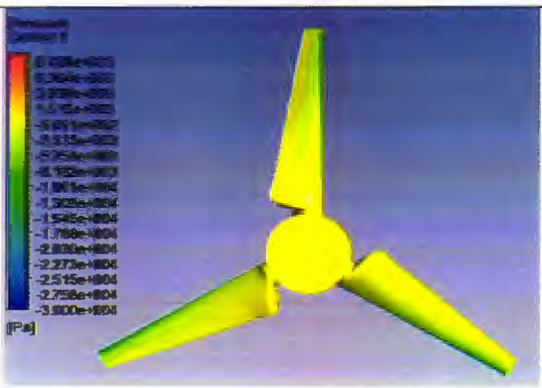
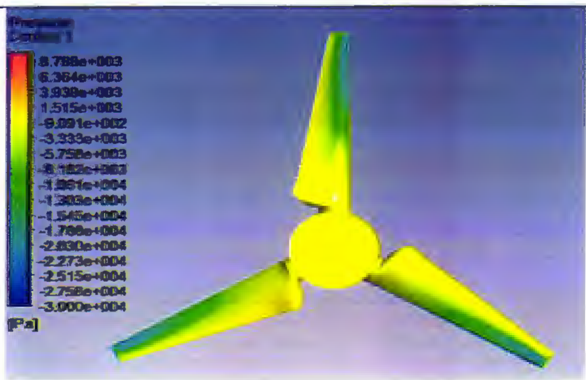
λ	Pressure	λ	Pressure
2		5	
3		6	
4		7	

Fig.4.9: Rotor pressure distributions (Suction side).

4.6.4 Blade Pressure Distribution

The pressure around the blade profile in a stationary frame is shown in figure 4.10. The total pressure in stationary frame was plotted on the plane. The plane was created at the tip of the blade in ANSYS-CFX post. As shown in figure by increasing the tip speed ratio (TSR) the negative pressure at the suction and the pressure on the front portion nearby the leading edge gradually increases. And when it reaches the TSR value of 5, negative pressure on the blade front portion as well as the pressure difference between the suction and front side decreases. Due to the decrease in pressure difference, the lift force also decreases and causes the reduction in torque and the minimum torque occur at TSR value of 6 and 7 respectively.

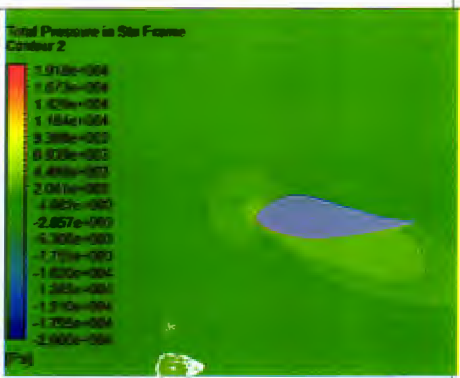
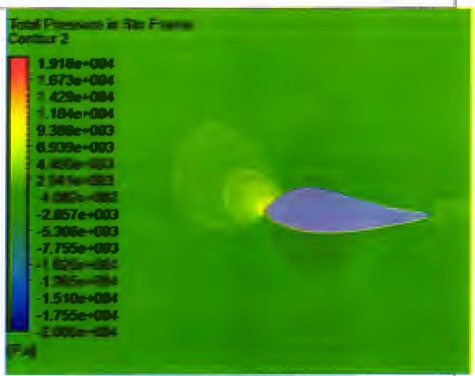
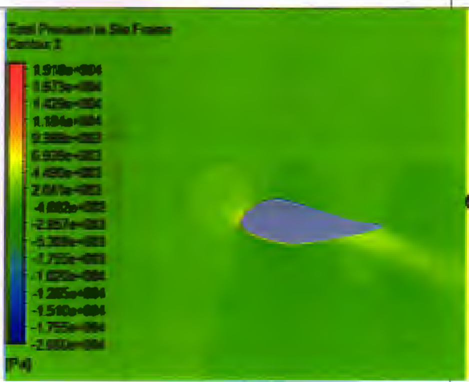
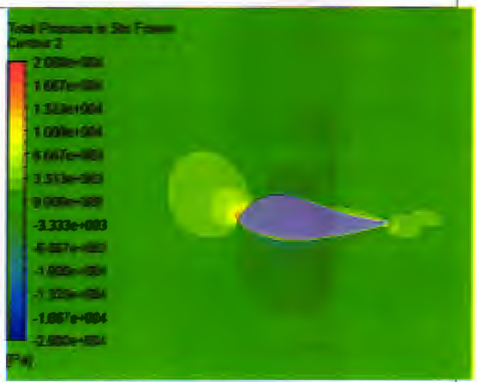
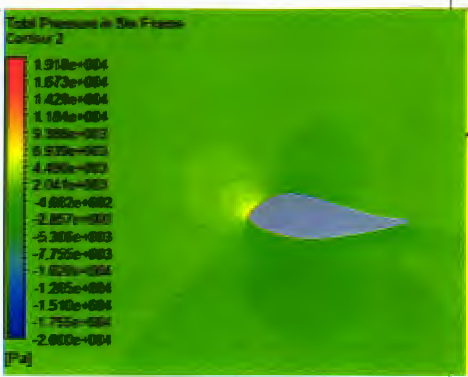
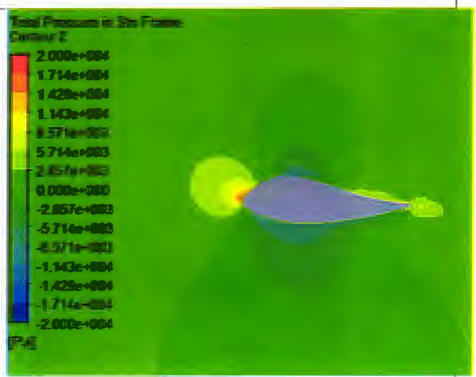
λ	Pressure	λ	Pressure
2		5	
3		6	
4		7	

Fig.4.10: Blade Pressure Distribution.

4.6.5 Turbulence Kinetic Energy

The turbulence kinetic energy of the turbine blade profile is shown in figure 4.11. As shown in figure at TSR value of 2 the fluid inlet angle flowing into the blade is higher because of low TSR value. The lift force is less because of large angle of attack and stall occurs. As the TSR value of 5 is achieved, the angle of attack value also approaches to the design value, and until the maximum value of lift force achieves. After passing the TSR value of 5, as it moves towards the trailing edge of the turbine blade, the turbulence occurs at this position because the fluid does not flow along the surface of the blade and falls off and the drag caused by the turbulence reduces the torque of the tidal turbine.

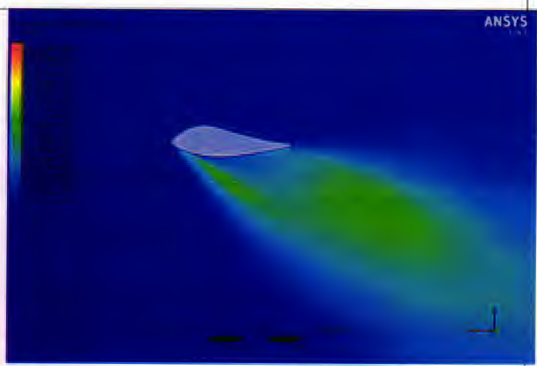
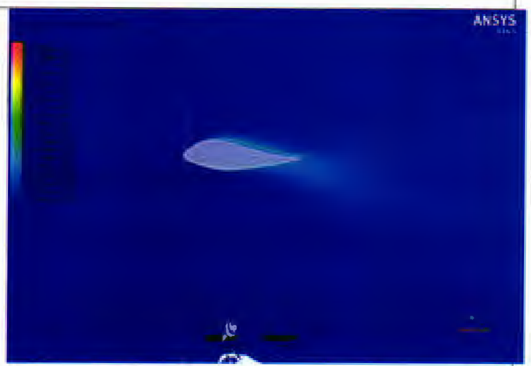
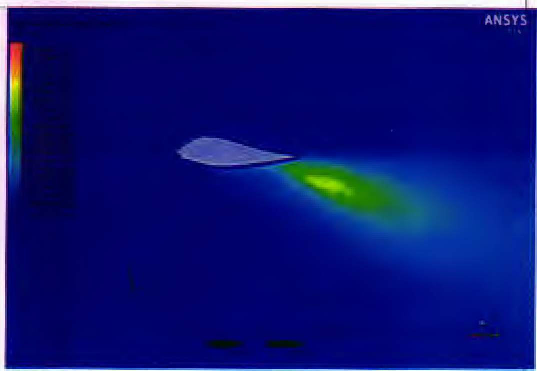
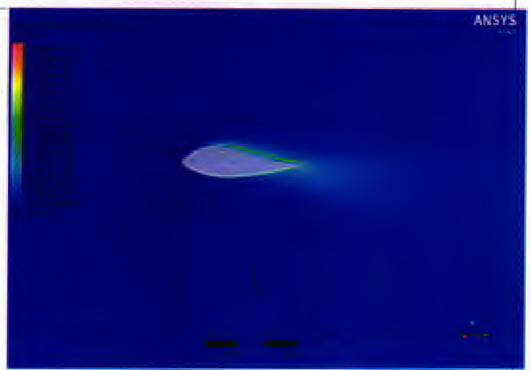
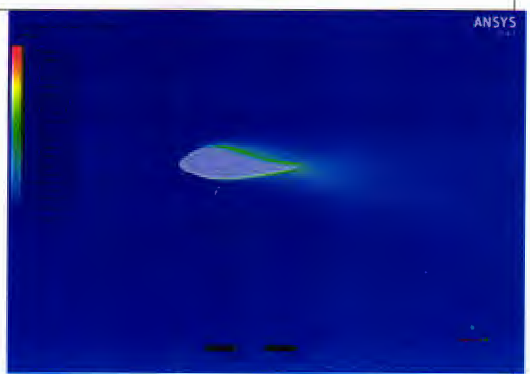
λ	Turbulence Kinetic Energy	λ	Turbulence Kinetic Energy
2		5	
3		6	
4		7	

Fig.4.11: Blade Turbulence Kinetic Energy Distribution.

4.6.6 Blade Streamline and Velocity

The blade streamlines and velocity are shown in figure 4.12. It is clear from the figure that at TSR value of 2 the stall is maximum. But as the TSR increases the angle of attack becomes closer to the design angle of attack, and the fluid inlet angle reduces. In case of less than 5 value of TSR, the highest lift to drag (L/D) ratio is achieved. But when the value of TSR becomes more than the decrease in lift reduces the torque due of the lower angle of attack than the maximum L/D ratio.

It was also found from the CFD analysis that near the 5 value of TSR the angle of attack reaches almost nearer to the design angle of attack which is the angle at highest glide ratio. However after the achievement of value 5 of TSR blade experiences higher angle of attack and turbulence occurs and torque reduces. Going further ahead the angle of attack increases further due to which the torque reduces further more.

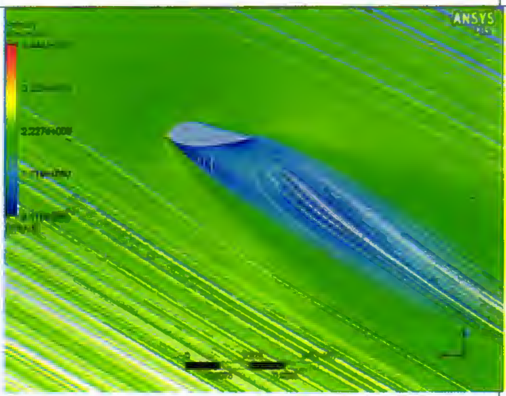
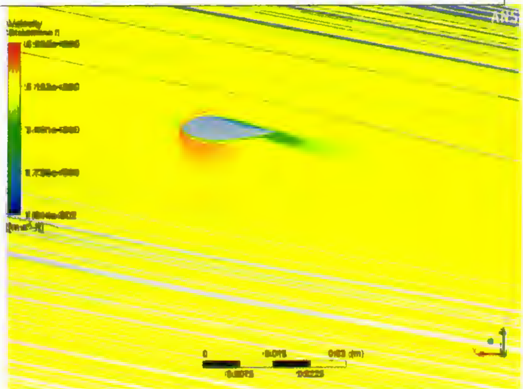
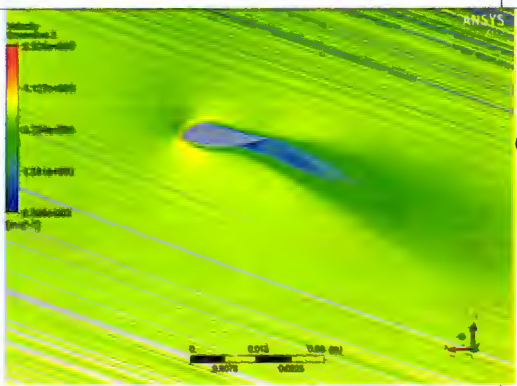
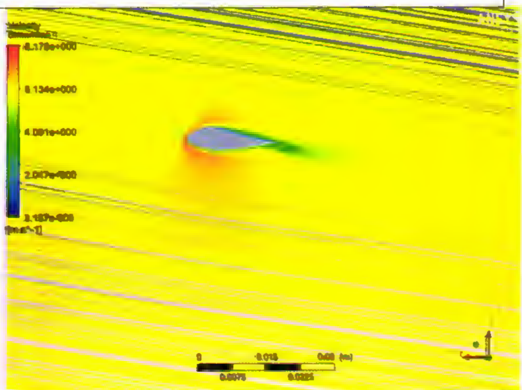
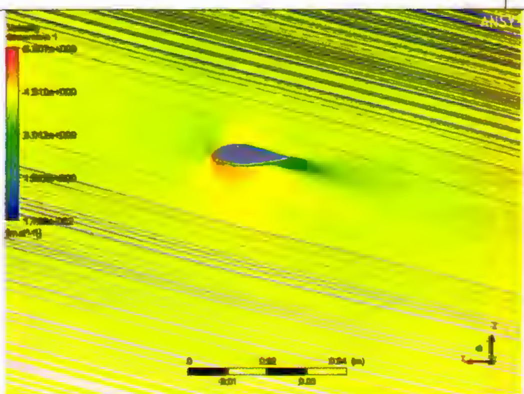
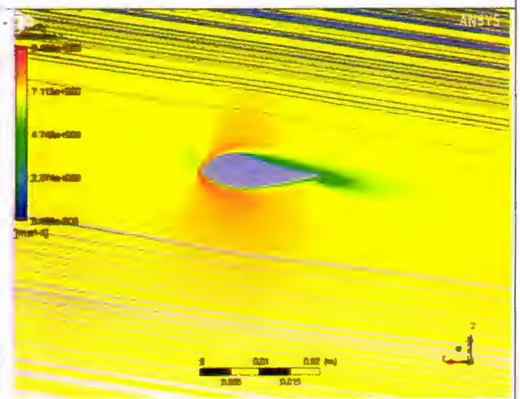
λ	Blade Streamline and Velocity	λ	Blade Streamline and Velocity
2		5	
3		6	
4		7	

Fig.4.12: Blade Streamline and Velocity Distribution.

4.6.7 Rotor Streamlines

The rotor streamlines are shown in figure 4.13. During passing through the turbine, swirling effect is higher as shown in figure. The rotor streamline is expanded maximum at the TSR value of 5 due to the increase of speed ratio and maximum efficiency is achieved. The wake is developed due to the cause of turbine and becomes more complex as the TSR increases.

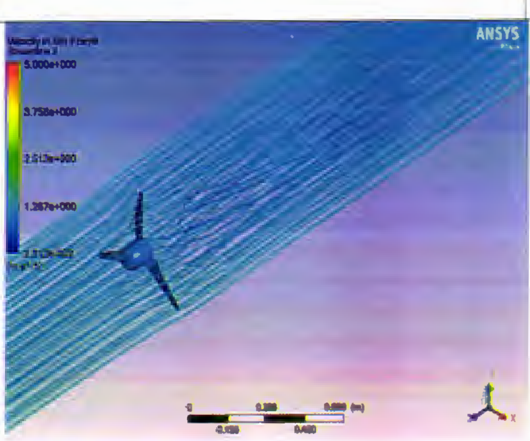
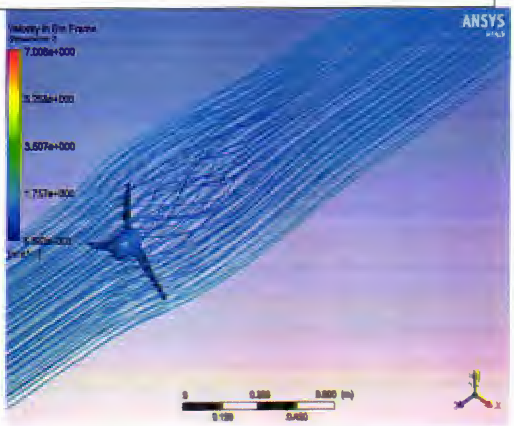
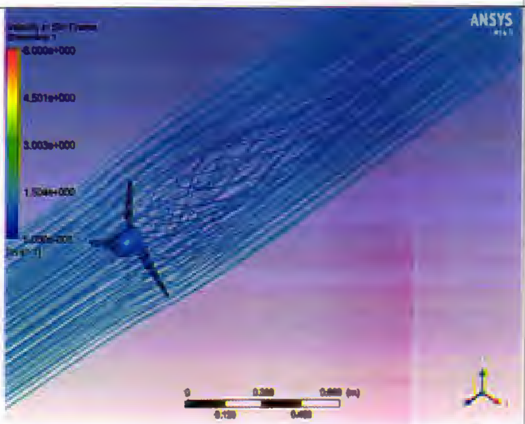
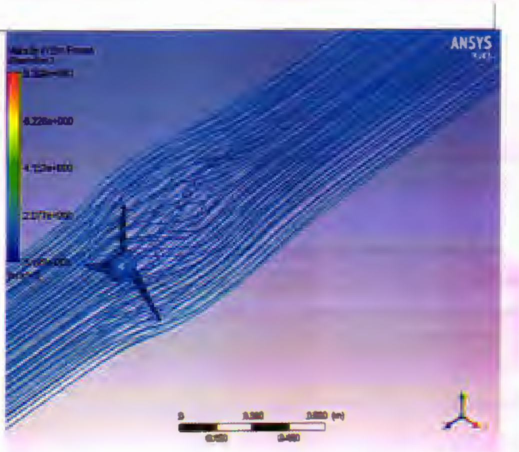
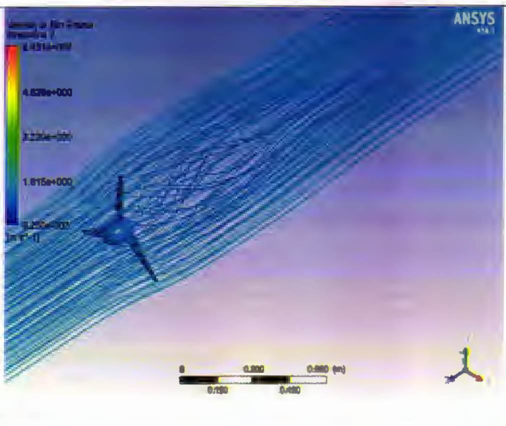
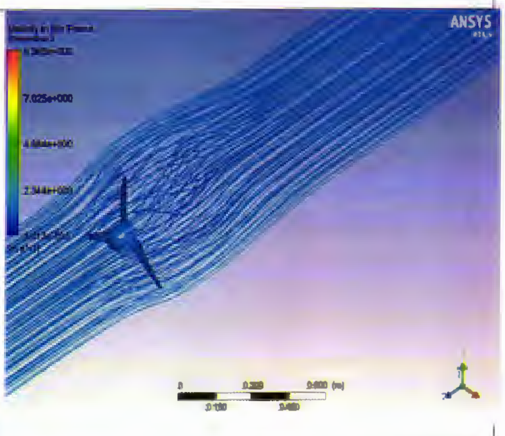
λ	Streamlines		Streamlines
2		5	
3		6	
4		7	

Fig.4.13: Rotor Streamlines Distribution.

4.7 Dynamic Analysis of Tidal current turbine

The dynamic analysis normally refers to transient vibration analysis of the structures. The vibration in tidal current turbine is a complex phenomenon. It is produced due to hydrodynamic loads. The vibration can cause resonance which leads to damage of the structures. TCT should be designed to avoid resonance in structure and prevent destruction of related components. The purpose of this research work is to investigate integrity of TCT against vibration caused by the fluid forces. Three principal aspects of dynamics of TCT behavior will be presented in this thesis, (1) modal analysis, (2) Pre-stress analysis, (3) forced vibration analysis. Finite element analysis method is used to simulate dynamic behavior of tidal current turbine.

4.8 Finite Element Analysis (FEA)

FEA is a numerical approach based on the principle of variation calculus which is used for solution of complicated problems. The accuracy of response of physical system is based on discretization and boundary conditions. Commercially available FEA software packages like ELFINI, NASTRAN, ABACUS, ANSYS, ADINA, STAAD and NISA, provide solution to investigate the number of problems i.e., monotonic and cyclic analyses, thermal analysis etc. In this work, ANSYS workbench is used for the analysis of Finite Element (FE) model of a TCT to investigate the structure response through fluid structure interaction (FSI).

4.8.1 Advantages of Finite Element Method (FEM)

As previously stated, FEM is applicable for both types of problem such as structural and non-structural. Because of the large number of advantages this method is very popular. This method has ability to

1. Handle the irregular geometries simply.
2. Handle static and transient loading conditions easily.
3. Geometries composed of several materials can be examined individually.
4. Handle large number of boundary conditions.
5. Mesh sizing is an effective tool for controlling mesh sensitivity.
6. It can update mesh model easily for iterating same parameters on new geometries.
7. Include mass scaling and concentrated masses for dynamic effects.
8. It can be incorporated by nonlinear behavior due to large deformations and materials nonlinearity.

Structural analysis facilitates designer to calculate the vibration, stress and thermal problems to evaluate design changes before construction of prototype with least number of iterations before product manufacturing.

4.9 FSI Models

When a deformable solid structure interacts with internal fluid pressure is exerted because of fluid that causes the deformation in structure and fluid flow alters then Fluid-structure interactions (FSI) occurs. A TCT operating in tidal currents can be regarded as typical fluid structure problems. Recent advances in FSI have enabled high fidelity solutions of complex fluid and structural problems. FSI modeling approach has been extensively used in the wind turbine industries for the determination of the structural response due to flow.

Fluid forces acting on blades, whilst resulting in torque which causes turbine to rotate. It also deforming blade due to pressure acting on it. Deformation of blades will change flow field around turbine which change flow field around blade. CFD models were coupled with FEA models to predict deformation of the blades and the resulting change in hydrodynamic forces. In this thesis hydrodynamic forces were calculated by CFD and it was transferred to FEA model where TCT deformations were then calculated.

4.10 Modal Analysis of TCT

The phenomenon of resonance due to the excessive vibratory motion occurs in many actual processes. It is important to determine the frequency quality and quantity to analyze the vibration problems. Modal analysis is mainly used to investigate response of structure for applied boundary conditions. It is used to determine the mode shapes and natural frequencies of the vibration of any structure.

Modal analysis of TCT has been conducted taking into account constraining effect on the back side of rotor. The constraints are applied in all degrees of freedom on rotor. And rotor or TCT is analyzed in static conditions. First three natural frequencies and mode shapes are calculated by using ANSYS, which are given in table in table 4.5 and figure 4.14 to 4.16.

Natural frequencies obtained by FEM and modal analysis for an TCT	
Modes	Damped Frequency (Hz)
1	126.04
2	171.93
3	177.02

Table 4.5: First three Natural frequencies of modal analysis

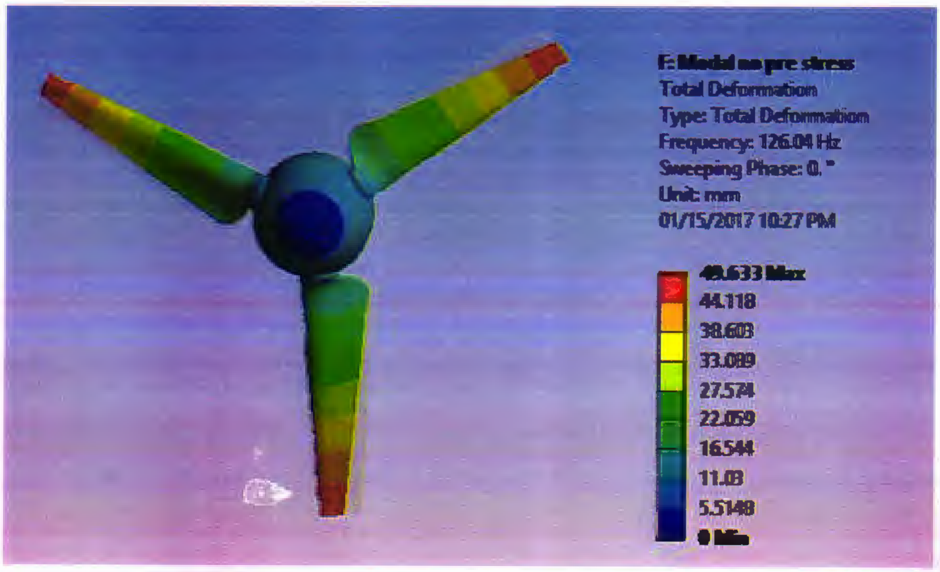


Figure 4.14: Modal Analysis Mode Shape 1 for a frequency of 126.04 Hz

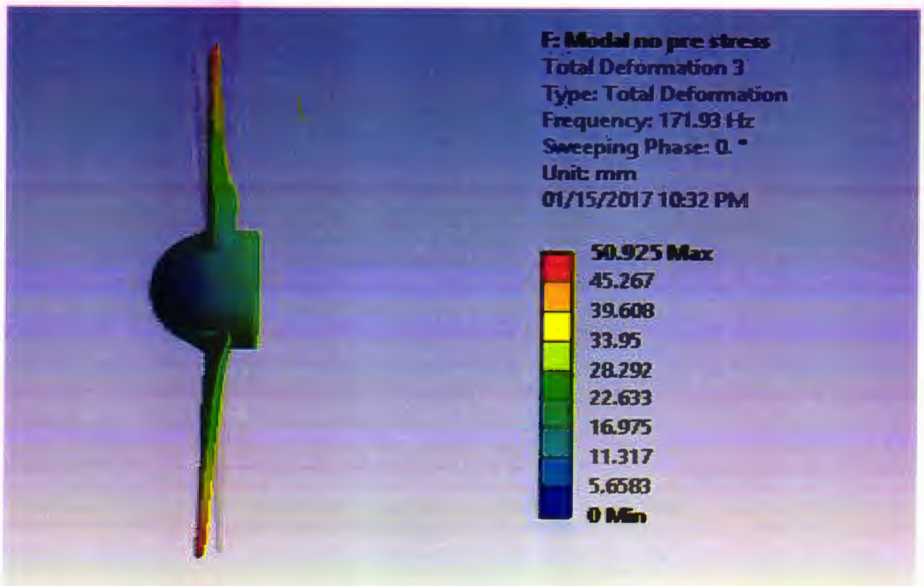


Figure 4.15: Modal analysis mode shape 2 for a frequency of 171.93 Hz

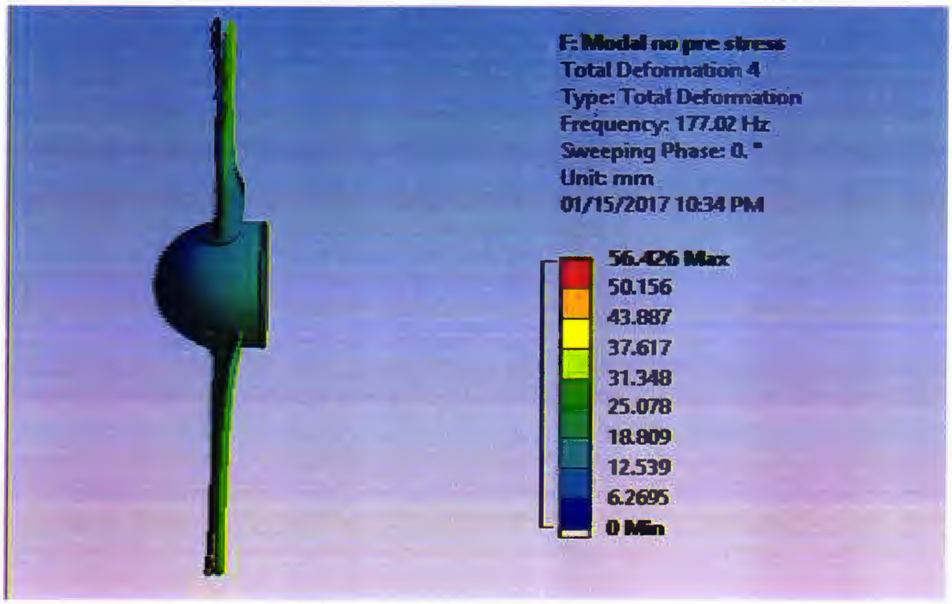


Figure 4.16: Modal analysis mode shape 3 for a frequency of 177.02 Hz

From the BEM analysis it is clear that tidal current turbine rotates at 191 revolutions per minutes. If we divide 191 RPM by 60 seconds we get forcing frequency of rotor due to rotation is 3.183 Hz. From table 4.5, it is clear that there is no natural frequency in range of calculated forcing frequency, Hence rotor is not going to resonate. And there is no potential failures observed in rotor structure.

4.11 Pre-Stressed Vibration Analysis

Dynamic analysis of pre-stressed vibration was also analyzed by using ANSYS software. In pre-stressed modal analysis a rotational speed of 20 radians per seconds were applied to tidal current turbine. The rotational speed produces centrifugal forces. These forces may effects the resonant frequencies of blade. The internal forces due to the rotation influence the deformation of rotating object subjected to dynamic load. This increases the natural frequencies with rotor speed and this phenomenon is known as centrifugal stiffening. This is significant parameter to consider in design of high speed tidal turbines.

Pre-stressed modal analysis was conducted in ANSYS workbench. The results of the pre-stressed (Natural frequencies and mode shapes) are presented in table 5. The natural frequencies of the unstressed blade are also included for comparison. The mode shapes for the pre-stressed modal analysis are shown in figure 4.17 to 4.19.

Natural frequencies obtained by FEM , Pre-stressed modal analysis and unstressed modal analysis for TCT		
Mode	**Pre-stressed Damped Frequencies (Hz)**	**Modal analysis Damped Frequencies (Hz)**
1	14.496	126.04
2	388.53	171.93
3	455.11	177.02

Table 4.6: First three Natural frequencies of pre-stressed modal analysis

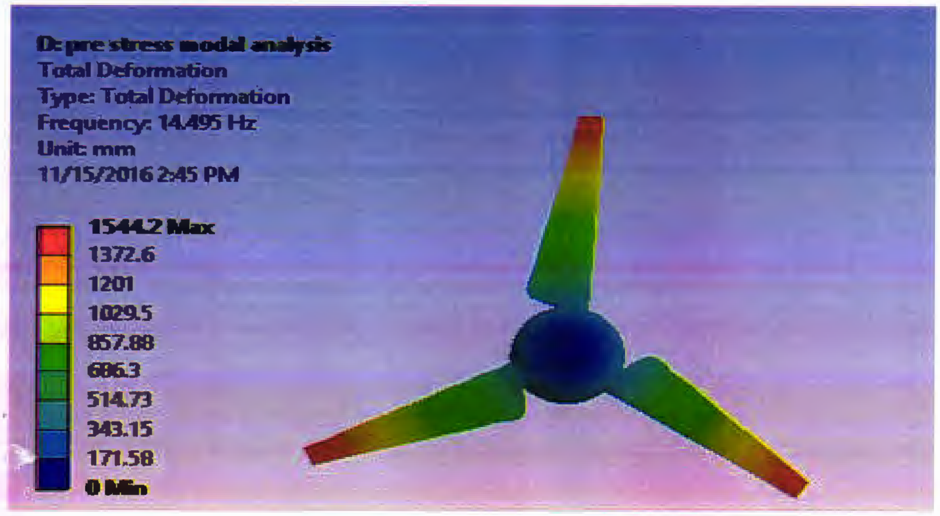


Figure 4.17: pre-stressed modal Analysis Mode Shape 1 for a frequency of 14.4 Hz

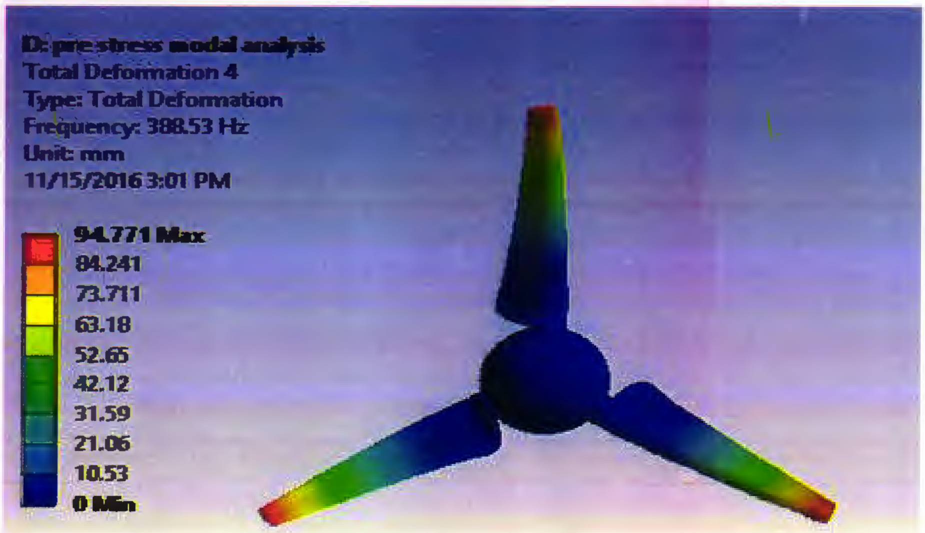


Figure 4.18: pre-stressed modal Analysis Mode Shape 2 for a frequency of 388.5 Hz

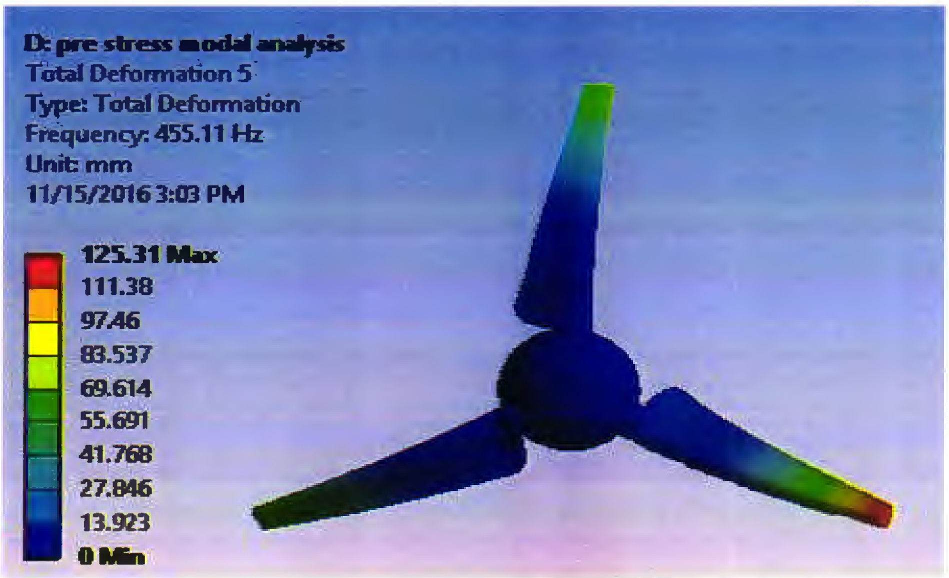


Figure 4.19: pre-stressed modal Analysis Mode Shape 3 for a frequency of 455.1 Hz

4.12 Comparison of non-stressed and pre-stressed Modal analysis

Two types of modal analysis for blade and hub assembly is used for making comparison of non stressed modal analysis and pre stress modal analysis. The natural frequencies and mode shapes of no stressed modal analysis were compared to the pre stressed modal analysis. The results show that Natural frequencies of pre-stressed modal analysis are almost 2 times higher than no stressed modal analysis. And it is clear from analysis that water pressure fluctuation during tidal current turbine operation can cause excite natural mode easily. Natural mode is further elaborated using Eigen buckling analysis

The mode shapes of tidal current turbine shows the deflection at several natural frequencies. The red coloured area in hub and blade assembly shows the maximum chances of failure due to the higher deflection. In order to minimize the chance of failure the external excitation corresponding to each natural frequency must be prevented. In general the maximum deflection is observed at the tip of the blade of TCT because this area is away from the hub, and the support of the hub become very small. The light blue coded area shows the minimum deflection area of the tidal turbine, and this area is safe from the fluid loads.

4.13 Forced vibration Analysis

The oscillation due to the external force is known as forced vibration. The oscillation that arises in moving objects i.e. turbine and airplanes wings are the example of forced vibration. The vibration produced in tidal current turbine due to the repeated hydrodynamic loads. The repeated hydrodynamic load causes resonance in the turbine, due to which the turbine failure occurs. In order to avoid resonance in tidal current turbine (TCT) the transient structural analysis were carried out in ANSYS workbench to find out the structural response of the TCT against the influence of the fluid loads. The transient analysis was carried out for time duration of zero to three seconds, and directional deformation and total deformation of TCT were visualized. The deformation becomes constant as steady state CFD analysis and random vibration spectrum is more suitable for simulation on actual spectrum of vibration. The results of the transient analysis for forced vibration are shown in table 4.7.

Directional deformation obtained by FEM, from transient structural Analysis		
Time (S)	**Minimum(mm)**	**Maximum(mm)**
0.2	-7.28E-02	2.91E-02
0.4	-7.30E-02	2.92E-02
0.6	-7.30E-02	2.92E-02
0.8	-7.30E-02	2.92E-02
1	-7.30E-02	2.92E-02
1.2	-7.30E-02	2.92E-02
1.4	-7.30E-02	2.92E-02
1.6	-7.30E-02	2.92E-02
1.8	-7.30E-02	2.92E-02
2	-7.30E-02	2.92E-02
2.2	-7.30E-02	2.92E-02
2.4	-7.30E-02	2.92E-02
2.6	-7.30E-02	2.92E-02
2.8	-7.30E-02	2.92E-02

Table.4.7: Directional deformation of TCT

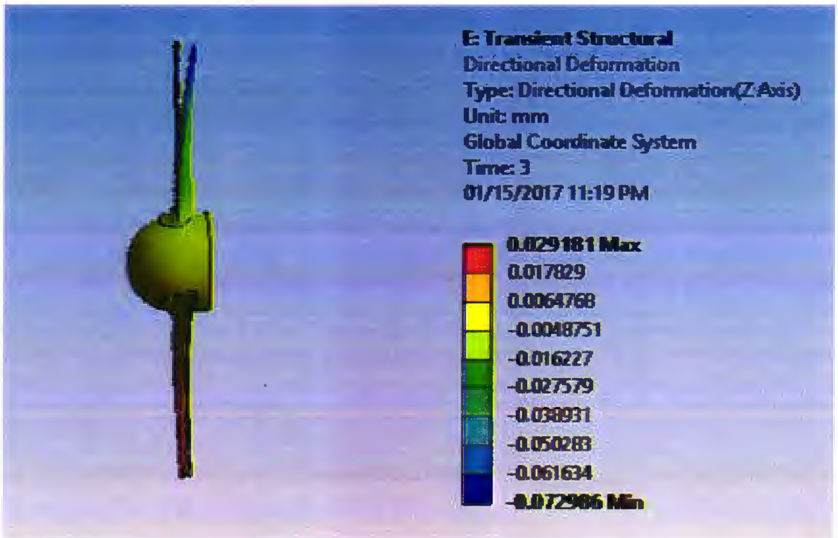


Fig.4.20: Directional deformation of TCT

The total deformation of the TCT was also analysed. The total deformation of the TCT is shown in table 4.8.

Total deformation obtained by FEM, from transient structural Analysis		
Time (S)	**Minimum(mm)**	**Maximum(mm)**
0.2	0	7.48E-02
0.4	0	7.50E-02
0.6	0	7.50E-02
1	0	7.50E-02
1.2	0	7.50E-02
1.4	0	7.50E-02
1.6	0	7.50E-02
2	0	7.50E-02
2.2	0	7.50E-02
2.4	0	7.50E-02
2.6	0	7.50E-02
2.8	0	7.50E-02
3	0	7.50E-02

Table 4.8: Total deformation of TCT

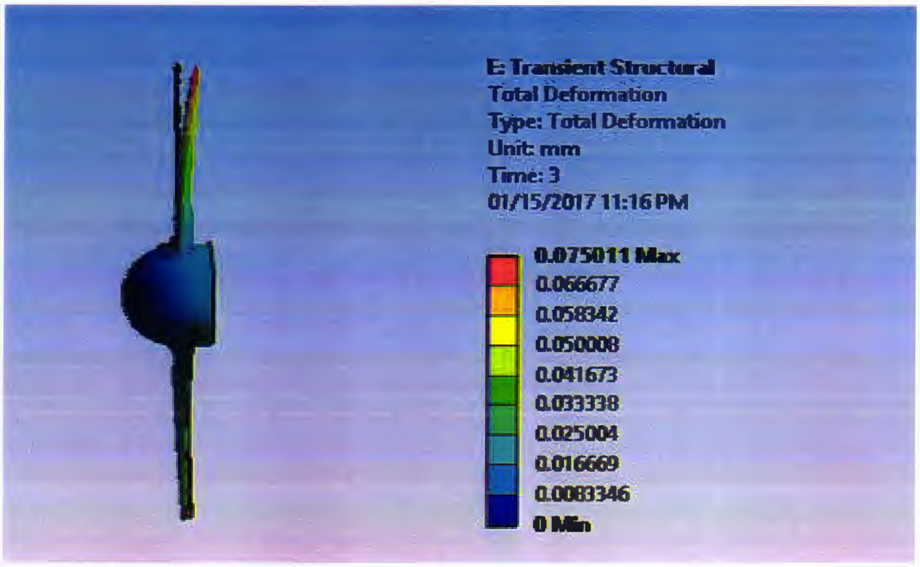


Figure 4.21: Total deformation of TCT

It is clear from above performed analysis that tidal turbine blade has low natural frequency for exciting natural modes. It should be kept for designer perception for defining joining and connecting blades with hub. Flexible joints are recommended to suppress vibrations damped quickly. And prevent the tidal current turbine (TCT) from the failure. It must have damped vibrations earlier before any natural modes can activate.

CHAPTER 5

Conclusion and Future Recommendation

5.1 Conclusion

In this research work horizontal axis tidal current turbine (HATCT) has been modelled on the basis of blade element momentum theory (BEMT). The TCT is then analyzed by using ANSYS software. The ANSYS CFX is used for the performance assessment of tidal current turbine. And Finite Element Analysis is performed in ANSYS workbench for the vibration analysis of the TCT. The main conclusions drawn from this research are listed below:

1. Mostly research work is related to the horizontal axis turbine (HAT) in Tidal current turbine technology (HATCT) and it is recommended to focus more on these devices in future because of high performance and stability.
2. There is so much potential of tidal current energy in whole world and Pakistan which can be economically extracted by horizontal axis turbines.
3. Comprehensive research work and facilities are required to investigate more technical and feasible extraction of this energy.
4. The blade element momentum theory (BEMT) may be used to estimate the performance of horizontal axis tidal current turbines.
5. Autodesk Inventor is suitable for the modelling of complex geometric design of tidal turbine as it reduces the complication.
6. The bolted hub to blade joints in tidal current turbine provides ease of assembly with economic edge and free of galvanic corrosion.
7. The performance of tidal current turbine through ANSYS CFX can be predicted accurately and easily.
8. For the performance of tidal current turbine the optimum tip speed ratio (TSR) is 5 for the design velocity of 1 m/s. The higher values of TSR are not recommended due to lower power co-efficient.

9. The accurate results of dynamic analysis of tidal current turbine may be investigated through Finite element package ANSYS.
10. The transient analysis was performed to accurately predict the forced vibration response of tidal current turbine.
11. Analysis of the tidal current turbine blade and hub assembly through ANSYS 17.0 saves analysis run time and provides accurate results.

5.2 Recommendation for future work

1. A detail investigation of velocity along the sea depth can be investigated to properly design the tidal current turbine to extract maximum energy from the flowing water.
2. Investigation of the economic and Technical feasibility for the actual installation of tidal current turbine at the defined site.
3. On the basis of available information the sea model may be developed to define the boundaries of the available sites for the installation of array of tidal current turbines.
4. The survey for the identification of new site is required for the estimation of the total potential of tidal energy in Pakistan.
5. Investigations of different type's materials are required to increase the life time of tidal current turbines.
6. Identification of the protection method of turbine infrastructure from corrosion and other environmental effects.
7. Development of mathematical and numerical model to investigate the behaviour of impact load that is caused by the sediments and habitants of marine animals.
8. Development of computational fluid dynamics (CFD) algorithms and software based programs for the analysis and performance estimation of specific turbine models.
9. Two way FSI analysis are required to validate the performance of tidal current turbine
10. Detail fluid structure interaction research is required for the effective development and continuous improvement of tidal turbine technology.
11. Two way FSI analysis are required to investigate the structural response of tidal current turbine against repeating fluid loads.

References

[1] J. Horton, "How Ocean Currents Work," Retrieved October, vol. 15, p. 2010, 2008.

[2] R. F. Nicholls-Lee and S. R. Turnock, "Enhancing performance of a horizontal axis tidal turbine using adaptive blades," in OCEANS 2007-Europe, 2007, pp. 1-6.

[3] N. Khan, "Marine resources in Pakistan: A tentative inventory," Karachi: Pakistan Business Review, 2011.

[4] H. A. Khan, M. Qurashi, E. T. Hussain, and M. I. Hayee, "Physics in Developing Countries: Past, Present and Future," ed: Citeseer, 2006.

[5] H. Samad, M. A. Hussain, F. Mumtaz, M. Badshah, and M. Khattak, "Design and experimental validation of horizontal axis ocean current turbine," in Power Generation System and Renewable Energy Technologies (PGSRET), 2015, 2015, pp. 1-6.

[6] C. Frost, P. Evans, C. Morris, A. Mason-Jones, T. O'Doherty, and D. O'Doherty, "The effect of axial flow misalignment on tidal turbine performance," in Proc. 1st International Conference on Renewable Energies Offshore, Lisbon, Portugal, 2014.

[7] P. Fraenkel, "Marine current turbines: pioneering the development of marine kinetic energy converters," Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, vol. 221, pp. 159-169, 2007.

[8] N. D. Laws and B. P. Epps, "Hydrokinetic energy conversion: Technology, research, and outlook," Renewable and Sustainable Energy Reviews, vol. 57, pp. 1245-1259, 2016.

[9] S. B. Elghali, M. Benbouzid, and J. F. Charpentier, "Marine tidal current electric power generation technology: State of the art and current status," in Electric Machines & Drives Conference, 2007. IEMDC'07. IEEE International, 2007, pp. 1407-1412.

[10] W. Collier and S. Way, "Full-scale validation study of a numerical tool for the prediction of the loading and hydrodynamic performance of axial flow tidal turbines," in 10th European Wave and Tidal Energy Conference Series, 2013, pp. 2-5.

[11] L. Verdant Power, "4640 13th Street, North, Arlington, VA 22207, USA; October 2008," URL< http://www. verdantpower. com/category/newsroom.

[12] C. B. B. Byrne, W. Cherrett, R. I. Grosvenor, A. Mason-Jones, C. E. Morris, D. M. O'Doherty, et al., "An investigation into the possible effects of cavitation on a horizontal axis tidal turbine," 2011.

[13] S. Bhushan, M. Alam, and D. Walters, "Evaluation of hybrid RANS/LES models for prediction of flow around surface combatant and Suboff geometries," Computers & Fluids, vol. 88, pp. 834-849, 2013.

[14] M. Khan, G. Bhuyan, M. Iqbal, and J. Quaicoe, "Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review," Applied energy, vol. 86, pp. 1823-1835, 2009.

[15] C. Consul, R. Willden, E. Ferrer, and M. McCulloch, "Influence of solidity on the performance of a cross-flow turbine," in Proceedings of the 8th European Wave and Tidal Energy Conference., Uppsala, Sweden, 2009.

[16] G. Keenan, C. Sparling, H. Williams, and F. Fortune, "SeaGen environmental monitoring programme final report," Royal Haskoning: Edinburgh, UK, January, 2011.

[17] C. Frid, E. Andonegi, J. Depestele, A. Judd, D. Rihan, S. I. Rogers, et al., "The environmental interactions of tidal and wave energy generation devices," Environmental Impact Assessment Review, vol. 32, pp. 133-139, 2012.

[18] T. Switzer and D. Meggitt, "Review of literature and studies on electro magnetic fields (EMF) generated by undersea power cables and associated influence on marine organisms," in OCEANS 2010, 2010, pp. 1-5.

[19] S. D. Commission, "Turning the tide: Tidal power in the UK," 2007.

[20] R. G. Miller, Z. L. Hutchison, A. K. Macleod, M. T. Burrows, E. J. Cook, K. S. Last, et al., "Marine renewable energy development: assessing the Benthic Footprint at multiple scales," Frontiers in Ecology and the Environment, vol. 11, pp. 433-440, 2013.

[21] C. Douglas, G. Harrison, and J. Chick, "Life cycle assessment of the Seagen marine current turbine," Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment, vol. 222, pp. 1-12, 2008.

[22] A. Bahaj, W. Batten, and G. McCann, "Experimental verifications of numerical predictions for the hydrodynamic performance of horizontal axis marine current turbines," Renewable Energy, vol. 32, pp. 2479-2490, 2007.

[23] C. Jo, K. Kim, K. Min, T. Yang, and H. Lee, "Study on HAT current generation rotor," Journal of Ocean Engineering and Technology, The Korean society of Ocean Engineers, vol. 16, p. 1, 2002.

[24] L. Hongwei, L. Wei, L. Yonggang, and L. Shun, "MODEL ANALYSIS AND TEST RESEARCH OF HORIZONTAL-AXIAL MARINE CURRENT TURBINE [J]," Acta Energiae Solaris Sinica, vol. 5, p. 016, 2009.

[25] C.-H. Jo, K.-H. Lee, J.-H. Lee, and C. Nichita, "Wake effect on HAT tidal current power device performance," International Journal of Ocean System Engineering, vol. 1, pp. 144-147, 2011.

[26] J. H. VanZwieten, N. Vanrietvelde, and B. L. Hacker, "Numerical simulation of an experimental ocean current turbine," IEEE Journal of Oceanic Engineering, vol. 38, pp. 131-143, 2013.

[27] B. Guo, D. Wang, and W. Shi, "POWER OUTPUT PERFORMANCE CHARACTERISTICS OF BIDIRECTIONAL TIDAL CURRENT TURBINE IN A CIRCULATING WATER CHANNEL," resource, vol. 1, p. 3.

[28] J. H. Lee, S. Park, D. H. Kim, S. H. Rhee, and M.-C. Kim, "Computational methods for performance analysis of horizontal axis tidal stream turbines," Applied Energy, vol. 98, pp. 512-523, 2012.

[29] T. Desmukh and A. S. Gawas, "TIDAL CURRENT ENERGY AN OVERVIEW."

[30] W. Batten, A. Bahaj, A. Molland, and J. Chaplin, "Hydrodynamics of marine current turbines," Renewable energy, vol. 31, pp. 249-256, 2006.

[31] J. Baltazar and J. F. De Campos, "Hydrodynamic analysis of a horizontal axis marine current turbine with a boundary element method," Journal of Offshore Mechanics and Arctic Engineering, vol. 133, p. 041304, 2011.

[32] J. Song, L. Baoshan, K. Kurokawa, A. Sayigh, B. Thapa, B. Fungtammasan, et al., "3» Forum Committee."

[33] B. Kim, S. Bae, W. Kim, S. Lee, and M. Kim, "A study on the design assessment of 50kW ocean current turbine using fluid structure interaction analysis," in IOP Conference Series: Earth and Environmental Science, 2012, p. 042037.

[34] R. McSherry, J. Grimwade, I. Jones, S. Mathias, A. Wells, and A. Mateus, "3D CFD modelling of tidal turbine performance with validation against laboratory experiments," in 9th European Wave and Tidal Energy Conference, 2011.

[35] C. hee Jo, J. young Yim, K. hee Lee, and Y. ho Rho, "Performance of horizontal axis tidal current turbine by blade configuration," Renewable Energy, vol. 42, pp. 195-206, 2012.

[36] M. Harrison, W. Batten, L. Myers, and A. Bahaj, "Comparison between CFD simulations and experiments for predicting the far wake of horizontal axis tidal turbines," IET Renewable Power Generation, vol. 4, pp. 613-627, 2010.

[37] S. Yang, D. Tcherniak, and M. S. Allen, "Modal analysis of rotating wind turbine using multiblade coordinate transformation and harmonic power spectrum," in Topics in Modal Analysis I, Volume 7, ed: Springer, 2014, pp. 77-92.

[38] O. N. Alshroof, G. L. Forbes, N. Sawalhi, R. B. Randall, and G. H. Yeoh, "Computational fluid dynamic analysis of a vibrating turbine blade," International Journal of Rotating Machinery, vol. 2012, 2012.

[39] G. Bir and J. Jonkman, "Modal dynamics of large wind turbines with different support structures," in ASME 2008 27th International Conference on Offshore Mechanics and Arctic Engineering, 2008, pp. 669-679.

[40] L. Tartibu, M. Kilfoil, and A. Van Der Merwe, "Vibration analysis of a variable length blade wind turbine," 2012.

[41] A. Kumar, A. Dwivedi, V. Paliwal, and P. P. Patil, "Free vibration analysis of Al 2024 wind turbine blade designed for Uttarakhand region based on FEA," Procedia Technology, vol. 14, pp. 336-347, 2014.

[42] J.-H. Park, H.-Y. Park, S.-Y. Jeong, S.-I. Lee, Y.-H. Shin, and J.-P. Park, "Linear vibration analysis of rotating wind-turbine blade," Current Applied Physics, vol. 10, pp. S332-S334, 2010.

[43] C. Faudot, O. G. Dahlhaug, and M. A. Holst, "Tidal turbine blades in runaway situation: experimental and numerical approaches," in Proceedings of the 10th European Wave and Tidal Energy Conference (EWTEC13), Aalborg, Denmark, 2013.

[44] A. Mason-Jones, D. O'doherty, C. Morris, T. O'doherty, C. Byrne, P. Prickett, et al., "Non-dimensional scaling of tidal stream turbines," Energy, vol. 44, pp. 820-829, 2012.

[45] J. Orme, I. Masters, and C. MATH, "Design and testing of a direct drive tidal stream generator," in Proceedings of the Institute of Marine Engineering, Science and Technology. Part B, Journal of marine design and operations, 2005, pp. 31-36.

[46] R. Nicholls-Lee, S. Turnock, and S. Boyd, "A method for analysing fluid structure interactions on a horizontal axis tidal turbine," 2011.

[47] M. Ragheb and A. M. Ragheb, Wind turbines theory-the betz equation and optimal rotor tip speed ratio: INTECH Open Access Publisher, 2011.