Performance Analysis and Optimization of Horizontal Axis Hydrokinetic Turbine Blade

A Dissertation submitted towards the partial fulfilment of
the requirement for the award of degree of

Master of Technology
in
Renew able Energy Technology

Submitted by
GOVIND PATEL

2K14/RET/08

Under the supervision of

Dr. RAJ KUMAR SINGH
(Associate Professor, Department of MECHANICAL Engineering)

Department of MECHANICAL Engineering
Delhi Technological University
(Formerly Delhi College of Engineering)
Delhi-110042
2014-2016

DELHI TECHNOLOGICAL UNIVERSITY

Established by Govt. Of Delhi vide Act 6 of 2009

(Formerly Delhi College of Engineering)

SHAHBAD DAULATPUR, BAWANA ROAD, DELHI-110042

CERTIFICATE

This is to certify that the dissertation title "**Performance Analysis and Optimization of Horizontal Axis Hydrokinetic Turbine Blade**" submitted by **Mr. GOVIND PATEL, Roll. No. 2K14/RET/08**, in partial fulfilment for the award of degree of Master of Technology in "RENEWABLE ENERGY TECHNOLOGY**",** run by Department of MECHANICAL Engineering in Delhi Technological University during the year 2014-2016., is a bonafide record of student's own work carried out by him under my supervision and guidance in the academic session 2014-16. To the best of my belief and knowledge the matter embodied in dissertation has not been submitted for the award of any other degree or certificate in this or any other university or institute.

Dr. RAJ KUMAR SINGH

Supervisor

Associate Professor

Department of MECHANICAL Engineering

Delhi Technological University

Delhi-110042

DECLARATION

I hereby declare that all the information in this document has been obtained and presented in accordance with academic rules and ethical conduct. This report is my own work to the best of my belief and knowledge. I have fully cited all material by others which I have used in my work. It is being submitted for the degree of Master of Technology in Renewable Energy Technology at the Delhi Technological University. To the best of my belief and knowledge it has not been submitted before for any degree or examination in any other university.

GOVIND PATEL

M. Tech. (RET)

2K14/RET/08

Date: JUNE, 2016

Place: Delhi Technological University, Delhi

ACKNOWLEDGEMENT

I owe my gratitude to all the people who have helped me in this dissertation work and who have made my postgraduate college experience one of the most special periods of my life.

Firstly, I would like to express my deepest gratitude to my supervisor **Dr. RAJ KUMAR SINGH** Associate Professor (Mechanical Engineering) for his invaluable support, guidance, motivation and encouragement throughout the period during which this work was carried out. I am deeply grateful to **Dr. R. S. MISHRA,** H.O.D. (Department of Mechanical Engineering) for their support and encouragement in carrying out this project.

I also wish to express my heart full thanks to my classmates as well as staff at Department of Mechanical Engineering of Delhi Technological University for their goodwill and support that helped me a lot in successful completion of this project.

Finally, I want to thank my parents, family and friends for always believing in my abilities and showering their invaluable love and support.

GOVIND PATEL

M. Tech. (RET)

2K14/RET/08

ABSTRACT

Hydrokinetic energy is the kinetic energy of a water mass due to its movement. The faster the water flow, the larger hydrokinetic energy it contains. Hydrokinetic turbines are used to covert this kinetic energy to electricity generation. Since the performance of these turbines played an important role in viability of hydrokinetic power generation projects hence different design parameters are studied to enhance its power generation capacity and performance.

In this project Chord Length and hence Solidity of hydrokinetic turbines at constant blade pitch angle and TSR is analyzed. The values of TSR taken is 3.36 at rotation speed of 90 rpm, the Chord Length taken is 0.12, 0.15 and 0.20 mt at blade pitch angle of 10 degree and flow speed 2.8 m/s. The design of turbine blade is done with the help of Airfoil-Tool, Pro-E software and meshing is done at Hypermesh. The analysis is done with the help of ANSYS FLUENT. After analysis we found that with increase in the chord length (or solidity) of turbine blade the thrust, torque and power generated increases and because of this coefficient of performance is also increases.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

Use of renewable energy is very demanding over the last few years due to environmental issues. Serious changes in the climate are occurring due to the high emissions of greenhouse gases. Widely used renewable energy sources include wind, solar and water energy.

1.1 HYDRO-POWER

Use of hydropower energy can solve the problem of energy crisis in the world. Hydropower energy has been used for many year and over the years water turbines have been developed and become more and more advanced. Now hydropower is the most important energy source available. Hydropower is contributing around 20% of energy world-wide.

Hydro turbines are the fundamental elements in hydro-mechanical equipments and play a significant role for power production.

Hydropower is currently the largest and most easily accessible power source in the renewable energy sources to generate electricity due to its high energy density. The hydroelectric power plant has to be installed with adequate turbine type in order to achieve the maximum output and higher efficiency. There are several different kinds of water turbines and can be divided into impulse and reaction turbines. An impulse turbine is where the water pressure is transformed into kinetic energy before the water reaches the runner of the turbine. The energy hits the runner in a form of a high-speed jet. A turbine, where the water pressure applies a force on the face of the runner blade is called a reaction turbine. Three types of turbines i.e. Pelton, Francis and Kaplan are usually utilized in the modern field of hydropower.

Energy from falling water and from run-of-river can be harnessed to obtain electrical power. Theoretical energy from a site depends on the flow of the water and the height of the water fall at the site. In order to estimate the hydropower potential from a site, it is important that the head and the flow of water over a period of time be measured.

The process of hydrokinetic energy conversion implies utilization of kinetic energy contained in river streams, tidal currents, or other man-made waterways for generation of electricity. Hydrokinetic conversion systems, mostly is at its early stage of development, may appear suitable in harnessing energy from such renewable resources. A number of resource quantization and demonstrations have been conducted throughout the world and it is believed that both in-land water resources and offshore ocean energy sector will benefit from this technology.

Hydro-power Classification

TYPE	STATION CAPACITY
Pico/Watermill	Upton 5 kW and below
Micro	Up to 100 kW
Mini	101 kW to 2000 kW
Small	2001 to 25000 kW

Table 1.1 : hydro-power clssification

1.3 HYDRO-POWER IN INDIA:

India is the 7th largest producer of hydroelectric power. Hydroelectric power potential of 84,000 MW at 60% load factor is one of the largest in the world. The present installed capacity as on 31 March 2016 is 42,783 MW which is 14.35% of total utility electricity generation capacity in India. In addition 4,274 MW small hydro power units are installed as on 31 March 2016. During the year 2014-15, the total hydro electricity generation in India was 129 billion Kwh which works out to 24,500 MW at 60% capacity factor. India also imports surplus hydro power from Bhutan. India has been dominant player in global hydro power development.

India is blessed with immense amount of hydro-electric potential and ranks 5th in terms of exploitable hydro-potential on global scale. India is endowed with economically exploitable and viable hydro potential assessed to be about 148,701 MW. In addition, 6,780 MW from small, mini, and micro Hydel schemes (<25 MW) have been assessed. Also, 56 sites for pumped storage schemes with an aggregate installed capacity of 94,000 MW have been identified. The hydro power potential in central India forming part of Godavari,Mahanadi, Nagavali, Vamsadhara and Narmada river basins has not yet been developed on major scale due to potential opposition from the tribal population.

The Hydro Power turbine was developed to produce a maximum amount of electrical power with the kinetic energy of flowing waters. Because the amount of kinetic energy (velocity) varies from river to river, the capacity of the turbine ranges from a minimum of a few watts to a maximum of 5kW. The new hydropower technology of harnessing hydrokinetic potential in rivers by hydrokinetic turbines could be a possible way to cater to the energy need of the rural communities. Also with growing agitation and reluctance of the government in setting up conventional hydropower plants this technology has several advantages compared to other renewable technologies for water rich areas i.e. no impoundment.

Hydropower offers an advantage over fossil fuels because it uses water as a renewable fuel source. Water is a clean fuel and does not release any particulate into the air. The production of electricity from falling water, due to the gravitational force, accounts for 16% of global electricity generation and is expected to increase about 3.1% each year over the next 25 years. Electrical demands, rising diesel fuel prices, as well as fossil fuel-based energy is limited and in fact is depleting, and subjected to use of renewable technologies. Conventional hydropower utilizes the potential energy of water through the construction of dams or reservoirs. An estimated potential of about 20,000 MW of small hydro power projects exists in India. Ministry of New and Renewable Energy has created a database of potential sites of small hydro and 6,474 potential sites with an aggregate capacity of 19,749.44 MW for projects up to 25 MW capacity have been identified. It is concluded that in India there is a scope to develop small hydro power station at remote area to make better life of people and fulfill their requirement. However, this requires a large capital investment and technology development and can have significant consequences on the local aquatic environment.

1.4 HYDROKINETIC POWER

Among all renewable energy technologies, hydro power generation is considered to be prime choice in terms of contribution to the electricity generation. Lot of hydro potential exists in canals and rivers in India. However, technology to tap this potential has not been successfully developed and readily available. Hydro kinetic turbine is considered to be suitable technology for power generation at such sites which can be used to capture the kinetic energy of flowing water. Hydrokinetic technology is under development and many studies were carried out and available in the literature.

Hydrokinetic technology has the potential to generate a great amount of electricity with a minimum impact to the environment. Hydrokinetic is the kinetic energy of a water mass due to its movement. The faster the water velocity, the larger hydrokinetic energy it contains. Another advantage of hydrokinetic turbines is their modular nature which leads to a scalable energy output and continuous energy production under zero static head. These attributes reduce the need of energy storage capacity, and as a result of this, hydrokinetic turbines find increased usage in remote locations. There are two types of hydrokinetic energy result from two popular types of water movements: current based and wave-based hydrokinetic energy. Current-based hydrokinetic energy can be found in river streams, artificial waterways, irrigation canals, dam head/tailrace, tidal and ocean currents. This movement of water can be utilized to generate electricity by a hydro mechanical conversion device that is turbine. Generally these are called as river current turbine (RCT) or hydro kinetic turbines. Many investigators investigated various types of hydrokinetic turbines and presented their performance under different conditions. The performance of hydrokinetic turbines was analyzed theoretically and experimentally under different studies. As widely known in the literature, the hydrokinetic power theoretically available in a river segment, having velocity V, flowing through a cross section and can be expressed as:

$$P = (1/2)\rho A V^3$$

Where, ρ is the density of water. Fresh water has a density of 1000 kg/m3. Hydrokinetic power resource evaluation has been done in different parts of the world. Although the theoretically available power is high, the technically recoverable power is much lower. The function of hydrokinetic turbines is to capture the kinetic energy of flowing water current and transfer it into a shaft. Hydrokinetic turbines can only capture a fraction of the kinetic energy in the water that pass through its cross section. The fraction is known as power coefficient, Cp.

The power captured by a hydrokinetic turbine can be expressed as:

$$\begin{aligned} Pactual &= Cp \times Ptheory \\ &= (1/2)\, Cp\, \rho\, A\, v^3 \end{aligned}$$

1.5 CLASSIFICATION OF HYDROKINETIC TURBINES:

Turbine Systems are classified as –

Axial (Horizontal): Rotational axis of rotor is parallel to the incoming water stream (employing lift or drag type blades),

Vertical: Rotational axis of rotor is vertical to the water surface and also orthogonal to the incoming water stream (employing lift or drag type blades)

Cross-flow: Rotational axis of rotor is parallel to the water surface but orthogonal to the incoming water stream (employing lift or drag type blades)

At the present state of this technology, both horizontal and vertical axis turbines are key contenders for further research, development, and demonstration (RD&D) initiatives. In addition to aiming for specific applications (such as, tidal currents or river streams), a great number of development efforts are directed toward realizing solutions that may serve both of these areas. Duct augmentation is another area, which apparently did not find much success in the wind energy domain. However, it is perceived as a critical element to hydrokinetic conversion concepts.

1.5.1 Energy conversion

The hydrokinetic method of extracting energy from flows converts the kinetic energy of flow to mechanical shaft power by a propeller-like device which generates electrical current by a dynamo (or generator) attached to the shaft in a manner analogous to a wind turbine. Power generated is proportional to the speed of the current cubed. Thus the available power depends primarily on the speed of the current. The minimum current required to operate a hydrokinetic

Water depth affects site selection, since rotor diameter is dependent on adequate water level above the installed device. For these reasons, hydrokinetic devices work best in locations with relatively steady flow throughout the year and without extended periods of low water level.

The energy flux contained in a fluid stream is directly dependent on the density of the fluid, cross-sectional area, and fluid velocity cubed. In addition, the conversion efficiency of hydrodynamic, mechanical, or electrical processes reduce the overall output. While turbine systems are conceived as prime choices for such conversion, other non-turbine approaches are also being pursued with keen interest.

Two main areas where hydrokinetic devices can be used in power generation purposes are, (a) tidal current, and (b) river stream. Ocean current represent another potential source of ocean energy where the flow is unidirectional, as opposed to bidirectional tidal variations. In addition to these, other resources include, manmade channels, irrigation canals, and industrial outflows. While all hydrokinetic devices operated on the same conversion principles regardless of their areas of application, a set of subtle differences may appear in the forms of design and operational features.

CHAPTER 2

LITERATURE REVIEW

Swiderski J., et al. [1] carried out research worked on the universal design methodology for water turbines. Interactive geometry editor was coupled through data files using the 3D viscous flow analysis software. Researchers presented the general structure of the design algorithm as well as example of the design optimization. It illustrated that authors are using it for design purposes in the small hydropower field assuming the trustworthiness of the CFD results.

Bennett K., and Swiderski, J. [2] worked for the remedy of severe blade cavitations and hub seal failure at Elliott Fall small-hydro generating station. Replacement of the pumps with new small-hydro turbines was considered in the study. As a solution, existing stays vanes, hub and blades were replaced with a custom runner and distributor. These were designed and developed using computational fluid dynamics (CFD) techniques. Their study illustrated the original design work of the power plant, the problems encountered, the solutions adopted, and the repair/upgrade results. It indicates the utilization of CFD for the custom design of small-hydro turbines is not only appropriate but cost-effective tool.

Mukherji, S.S., et al.[3] Discussed the hydrodynamic performance of horizontal axis hydrokinetic turbines (HAHkTs) under different turbine geometries and flow conditions. Hydrokinetic turbines are a class of zero-head hydropower systems which utilize kinetic energy of flowing water to drive a generator. A detailed computational fluid dynamics study was performed using the k- shear stress transport turbulence model to examine the effect of various parameters like tip-speed ratio, solidity, angle of attack, and number of blades on the performance HAHkTs having power capacities of 12 kW. For this purpose, a three-dimensional numerical model was developed and validated with experimental data. The numerical studies estimate optimum turbine solidity and blade numbers that produce maximum power coefficient at a given tip speed ratio. Simulations were also performed to observe the axial velocity ratios at the turbine rotor downstream for different tip speed ratios which provide quantitative details of energy loss suffered by each turbine at an ambient flow condition. The velocity distribution provides confirmation of the stall-delay phenomenon due to the effect of rotation of the turbine and a further verification of optimum tip speed ratio corresponding to maximum power coefficient obtained from the solidity.

Nitin kolekar, et al.[4]. Done research work on the performance of hydrokinetic turbines turbines which depends on various parameters like number of blades, tip speed ratio, type of airfoil, blade pitch, chord length & twist and its distribution along the blade span. Maximization the performance of a horizontal axis hydrokinetic turbine through a coupled computational fluid dynamics-blade element momentum (hydrodynamic) analysis is done. Optimization is carried out using both robust and deterministic-design optimization schemes and compared. Analysis is focused on constant chord blades due to low cost and ease of fabrication. Finally, CFD analysis results are presented for optimized geometry.

Udit Tewari, Karl Kolmsee et al.[5]. Gives the experimental datas which we are using in hydrokinetic power generation projects in uttarakhand. Uttarakhand is a northern state of India located in the foothills of the Himalayan mountain ranges. It is rich in water resources with many perennial rivers originating in the state and has enormous potential for hydropower generation. Despite of this fact the recent floods and agitation on setting up large and small hydropower plants has put a question mark on the future hydropower development and thus future of rural electrification in the state. In this context, the paper presents an idea of using hydrokinetic energy as a viable solution for securing low impact and viable power for rural communities in Uttarakhand.

The power output of the turbine depends on the water velocity of a river or canal. The cut-in speed of the turbine i.e. the speed at which the turbine starts supplying useful power is 0.7 m/s. But the threshold velocity is usually 1.5 m/s were the turbine output is 1 kW output and is considered economically rational. The turbine produces maximum power output of 5 kW at river velocity of 2.8 m/s. Rotational speed ranges from 90rpm to 230 rpm.

Zhou, L, et al. [6] studied that dynamic stresses in blades are related to hydraulic instability as shown by the flow calculations and measured stresses. Studies were also carried out for calculating the dynamic stresses caused by the unsteady hydraulic loadings. Thereafter dynamic interaction problem in blades was analyzed. It was done from solution of 3D unsteady flow through its flow passage. Unsteady RANS equations were solved for modelling the flow within the complete flow path of the turbine and so the hydraulic forces were used as the boundary condition. Dynamic stresses in the blade were found low under the optimum operating conditions and were high under low-output conditions. It indicated that prediction of dynamic stresses is possible during design stage.

Chica E, Perez F. at el.[7] The finite element model (FEM) was successfully used for stress and displacement calculation on the turbine blade under the influence of the gravitational, centrifugal and hydrodynamic loading. The influence of the material and the blade structure shape on its structural behavior were presented and discussed. Safe working stresses and strains were identified and checked. It was found that the stresses produced in all of the analyzed models do not exceed the safe working stresses. Based on these results, the blade models with balsa wood core and layups of glass fiber (type E), stainless steel or aluminum can be selected for manufacturing the blade due to their high stiffness, hydrodynamic loading resistance and low weight compared with the other analyzed models.

Deschenes, C., and Fraser, R. [8] presented a review of current tendencies regarding measurements on models of hydraulic turbines and discussed how such tests can be used to improve the overall performance of hydraulic turbines, taking into account overall efficiency, lifetime expectancy and environmental issues. New strategies for funding major R&D projects of current concern were also discussed. Present hydraulic market experience encounters strong pressures, both of a regulatory and a public driven nature, to mitigate environmental problems associated with hydraulic turbines, including reducing fish mortality by rendering existing or new units more “fish friendly” and compensating for dissolved oxygen depletion by turbine aeration. In addition, many powerhouses are ageing and in need of major overhauls. This means that manufacturers have to develop machines that can be installed within existing civil constraints, whilst still providing proven efficiency or power output, longer lifetime expectancy, and hopefully fulfilling new environmental requirements.

Dragu, C., and Soens, J. [9] worked on the literature survey of the advantages and drawbacks of small and micro hydro power plants. Earlier special attention was given to the wind and solar power. Comparison with other renewable becomes very important. High initial and low operating costs are the main economic characteristics of hydropower. From the studied literature, an easy algorithm can be proposed to assure the maximum efficiency for minimum costs of a hydro potential site. Last decade has shown increase of demand for energy resources in all the economical sectors, where demand for electricity stands at the highest rate of growing. Position of the renewable technologies is expected to become more and more important in the global environment. . Small-scale renewable generation being the cost-effective can bring electricity to remote areas which are far away from transmission lines.

Busea, C, et al. [10] carried research work on studying the axial hydraulic turbine runner optimization using the finite element analysis software. Using the CFD simulation, with less response time, modification can be investigated in a short time. Latest CFD techniques can predict energetic characteristics with certain accuracy level, as fluid simulation techniques do not replace experimental tests in precise.

Ferrando, L, et al. [11] worked on the application of a surface parameterization to a blade of Francis runner turbine. This geometric representation should be used as a practical tool in the process of design optimization. Most parameterizations are based on blade section approaches. The parameters are typically angles, lengths that have a clear meaning to the hydraulic designer. Span-wise functions are sometimes used to ensure coherence between the sections and the smoothness of the constructed blade surfaces. In this case, the section-to-section approach was replaced by a purely surface method. The blade is modelled using a camber surface and thickness distributions, and the design parameters are kept as close as possible to their original physical meaning. Smooth blade surfaces are ensured, and a reduced number of variables is sufficient to describe realistic designs. Their line of research aimed to introduce a surface parameterization approach which provides a representation of the blade. One of the benefits of this methodology is the reduction of design parameters involved as this approach is no longer section dependent. Other advantages reside in the easiness to obtain smooth geometries. Finally, it is also important to point out that data exchange between programs (i.e. CAD, Mesh Generator) may now use surface representation. This entails that subjective reconstruction of this surface is no longer necessary. As a conclusion, with their approach and the reduction of design parameters, the design optimization process becomes shorter in terms of time and effort.

Williams, A.A., and Simpson, R.G., [12] worked on research project for developing standard design procedure for Pico propeller turbines for local manufacturing in the developing countries. A 5 kW demonstration turbine was set up at a test site in Peru. Computational Fluid Dynamics (CFD) has been used to obtain overall performance data for the turbine and to assist in the design of a new rotor. It was found that an incorrect matching between the rotor design and the available flow rate at the site significantly affected the turbine operation and in order to provide an acceptable performance it was possible to adjust the runner design and operating speed of the turbine. Authors concluded that the computational fluid modelling can be used as an appropriate design tool.

Cruz, et al. [13] carried research on the application of minimum pressure coefficient criteria for the axial-flow hydraulic turbines cascade geometry design. In recent works, the criteria was tested for the axial fan, showing that it is suitable to define the initial geometry for machine design. The global parameters that supply the principal dimensions of the turbine were obtained from the literature as based upon statistical data of installed power plants. The simulation domain grid was generated with CFX-TURBO. Results were obtained to analyze the fluid flow through blade runner. In this way, a study was carried out on a small axial-flow turbine specifically designed for operating in small rivers.

Keck, H., and Michler, W., [14] studied different methods for life cycle analysis based on the dynamic loading. Validated pressure pulsations resulting from CFD have shown prediction of pressure load as input to the structural analysis of the runner.

ShuHong, L., and Jie, S., [15] treated the unsteady turbulent flow computation based on the modified turbulence model through the flow passage to simulate the pressure fluctuations. The conventional method to assess turbine performance is its model testing which becomes costly and time consuming for several design alternatives in design optimization. Computational fluid dynamics (CFD) has become a cost effective tool for predicting detailed flow information in turbine space to enable the selection of best design.

Prasad, V., and Gahlot, V.K., [16] carried out the 3D real flow analysis in an experimentally tested axial flow turbine and different flow parameters were computed at three operating regimes to find the best operating regime. The computed efficiencies were critically compared with experimental values and found to bear close comparison. In axial flow turbine, water passes through the series of blade rows and changes its direction from radial to axial. Runner is the most important component of the turbine and its blade profile is designed at different sections from hub to casing to get the best performance. The rotation of runner and operation of turbine either below or above the rated conditions cause variation of flow parameters from hub to tip.The combination of advanced numerical techniques and computational power has lead to computational fluid dynamics (CFD). It is an efficient and inexpensive tool to make internal flow predictions to good accuracy and, any sort of flow problems can be detected and further improvements can be made on the geometry of turbine components. It has made possible to obtain a significant enhancement in efficiency. CFD can be used to check efficacy of alternate designs of turbines for optimization before final experimental testing of selected designs is resorted.

Khare, R., et al. [17] carried simulation of three dimensional flow in a mixed flow (Francis) turbinee using ANSYS CFX 10 to study the flow pattern in the turbine. Futher work was done on the computation of various losses and efficiency at different operating conditions.

Helena, M.R., and Mariana. S., [18] conducted research on hydro turbines for optimization and the selection of adequate hydro turbines. The hydrodynamic fluid mechanical analysis require the use of complex advanced models which apply the equations of Navier-Stokes by using mathematical models of conservation laws, for the study of the turbulent flow behaviour. To determine the correlation between the flow velocity and pressure fields, they used k-ε model. They aimed to search for new solutions regarding the energy production of available low power in water systems. The hydropower equipment known as micro-hydro, represents an advantageous economic alternative in terms of hydroelectric exploitation in water systems, when compared with dissipative structures. This type of energy associated to low power production and the financial search have been very conservative in what concerns the development of micro turbines, in alternative to major powers. With the purpose of providing and developing future areas with energy potential, especially in developing countries, rural or isolated areas, the use of micro-turbines can promote the economical development and the creativity for designing new solutions. With the objective of promoting the use of energy associated with the installation of low power systems (i.e., low heads and/or discharges), for example in water systems supply or irrigation, the hydrodynamic flow through the selected turbines to these conditions and the estimation of the best efficiency point, with the aim of finding new possible applications in systems already existent, can be developed.

Sutikno, P., and Adam, I.K., [19] carried out research in order to develop a hydro turbine to be used for specific site of lower head of less than 1.2m. Development of very low head turbine has been done in this research using the simple civil construction and economically viable. The recent development of computational techniques has allowed a substantial improvement in hydraulic turbine design. Initial geometry capable to assist certain characteristics of turbine performance is step for useful numerical turbine analysis. results were obtained using the 3-D FLUENT flow to analyze the fluid flow through blade runner. Study was carried out on a small axial-flow turbine, specifically designed to operate in a very low head and finally evaluating the results for hydraulic efficiency prediction of the turbines. The prototype of turbine system was tested by using small channel system. Tested result was obtained for maximum efficiency of 90% and the power output simulation and experimental has the differential less than 5%. Hydraulic turbine design involves many stages of iterative calculations.

Thapa, B.S., and Panthee, A., [20] carried out study to simulate the design and operational problems of Francis and Pelton turbines by the application of computational tools. A new program was developed to optimize designs of Francis runners for sediment erosion problems and CFD analysis of some new design conditions was done. True size Pelton runner model was developed by using Solid Works for stress and fatigue analysis by using Cosmos Works in its operating conditions. It was found that numerical models are capable to simulate the design and operational problems in hydro turbines. The computational tools can also suggest the design optimization needed to minimize these problems. It was observed that sediment erosion in Francis turbines can be reduced up to 50% by optimizing the hydraulic design. After detailed literature review and survey on the hydro-power development, it was concluded to focus to carry research work on the development of low head hydro turbine runner blades to acquire maximum power output through the geometrical changes in the profile structure using CFD.

2.1 GAP IDENTIFICATION:

On studying different literature review, it is observed that there are different design parameters and methodologies of analysis on which work should be done for the performance improvement of Hydrokinetic turbines. Hydrokinetic turbines very often suffer from low-efficiency which is primarily due to its operation in a low tip-speed ratio regime. This makes the design of a Hydrokinetic turbine a challenging task. The three-dimensional results for optimum design have suggested a strong dependence of maximum Cp on TSR when different turbine geometries (i.e., solidity, angle of attack, and number of blades) are being considered. Increase in turbine solidity and blade numbers results in increased Cp under the entire operating range of TSR studied with maximum Cp observed in lower TSR. The studies also indicates that lower solidity turbine is preferable for Hydrokinetic turbines design where low starting torque and high rotating speed are required.

TSR, CHORD LENGTH, and Blade Pitch angle are the three main parameters for the analysis and optimization of performance of hydrokinetic turbine blade where further research work is need to be done. As these parameters affect the performance of turbine and hence the overall viability of power project. Our objective is to maximize the thrust force generated by the system, power generated and coefficient of performance Cp. The computational models used to evaluate the design, such as Computational Fluid Dynamics to study performance, as well as a Finite Element Analysis to check the structural integrity of the turbine in preparation for manufacturing. Results of the flow field analysis and a static structural analysis can be used for optimization of performance of hydrokinetic turbines.

CHAPTER 3

PROBLEM IDENTIFICATION, METHODOLOGY AND FORMULATION

An efficient and cost-effective energy production is directly related to the proper selection of design parameters, an optimum turbine design is required for effective use under variable flow conditions.

3.1 PROBLEM IDENTIFICATION:

The primary barrier to commercialization of hydro-kinetic power generation technology lies in its low efficiency which is primarily due to its operation in a low tip-speed ratio (TSR) regime.

The performance of these turbines is governed by the three non-dimensional parameters:

(a) TIP-SPEED RATIO (TSR): It is defined as the ratio of blade tip speed to fluid speed.

$$TSR = \frac{R\Omega}{U}$$

The relation between TSR and power coefficient Cp can be understood intuitively. If the turbine's blades spin too slowly, then most of the water will pass through the rotor without being captured by the blades. However, if the turbine spins too fast, then the blades will always travel through used, turbulent water. There must be enough time lapses between two blades travelling through the same location so that new water can enter and the next blade can harness the power from that new water, not the used, turbulent water.

(b) SOLIDITY(σ): It is defined as the ratio of blade chord length times the number of blades to turbine circumference.

$$\sigma = \frac{Bc}{2\pi R}$$

We selected three main parameters for the analysis and optimization of performance of hydrokinetic turbine blade. These are CHORD LENGTH (varying), TSR(constant) and Blade Pitch angle (constant). Our objective is to maximize the thrust force generated by the system, power generated and coefficient of performance Cp.

3.2 NUMERICAL METHODOLOGY

The computational models used to evaluate the design, such as Computational Fluid Dynamics to study performance, as well as a Finite Element Analysis to check the structural integrity of the turbine in preparation for manufacturing.

3.2.1 COMPUTATIONAL FLUID DYNAMICS (CFD):

Computational Fluid Dynamics is one of the branches of fluid mechanics that uses numerical methods to solve and analyse problems of fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic or turbulent flows are an ongoing area of research. CFD provides a qualitative as well as quantitative prediction of fluid flows by means of mathematical modelling (partial differential equations) and numerical methods (discretization and solution techniques). Computational Fluid Dynamics (CFD) is computer based simulation to analyse systems involving fluid flow, heat transfer and associated phenomena. A numerical model is first constructed using a set of mathematical equations that describe the flow. These equations are then solved using a computer programme in order to obtain the flow variables throughout the flow domain.

3.2.2 METHODOLOGY

In all of these approaches the same basic procedure is followed.

3.2.3 CFD OF TURBINE BLADE:

The experimental approach of evaluating the performance of turbine is costly as well as time consuming. Conversely CFD approach is faster and large amount of results can be produced at virtually no added cost.

A growing availability of computer power and a progress in accuracy of numerical methods, brought turbo machinery CFD methods from pure research work into the competitive industrial markets. Many soft wares are available in the market for numerical analysis of turbo-machines viz. Fluent (UK and US), CFX (UK and Canada), Fidap (US), Polyflow (Belgium), Phoenix (UK), Star CD (UK), Flow 3d (US), ESI/CFDRC (US), SCRYU (Japan) and more. Computational fluid dynamics study has been carried out using ANSYS on the original geometry and results obtained were compared to validate with the operational / experimental data.

3.2.4 MESH GENERATION:

For mesh generation, flow domain has to be divided in small cells. Its distribution locates the flow variables. These variable gradients are accurately calculated on the fine mesh. A fine mesh has particular importance in the regions where large variations are expected in the flow are expected. It requires more computational power and time and mesh size is optimised by performing mesh-independence study. It starts from coarse mesh, until it refines by the time simulation results are not affected by doing any further refinement.

The geometry was meshed with tetrahedral cells of defined global size. Periodic surfaces were generated for the estimation of the effect of other three blades. For CFD analysis, turbulence model k- was used, as it is high Reynolds number model.

3-D MESHING OF WHOLE SYSTEM INCLUDING OUTER ROTATING WALL. INNER ROTATING WALL, TUBINE BLADE AND HUB.

This meshing is done by the software HYPERMESH.

CASE 1: Blade of CHORD LENGTH 0.12 mt.

Number of NODES: 4.1×10^4

Number of ELEMENTS: 1.95×10^5

CASE 2: Blade of CHORD LENGTH 0.15 mt.

Number of NODES: 7.1×10^4

Number of ELEMENTS: 3.46×10^5

CASE 3: Blade of CHORD LENGTH 0.2 mt.

Number of NODES: 4.5×10^4

Number of ELEMENTS: 2.1×10^5

3.2.5 BOUNDARY CONDITIONS:

During preprocessing The geometry (physical bounds) of the problem is defined. The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform. The physical modeling is defined or example, the equations of motion. Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined. The simulation is started and the equations are solved iteratively as a steady-state or transient. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

Stationary wall boundary condition was given to the shroud i.e. outer casing. Blades and hub were given the rotating boundary conditions, while other surfaces were taken as periodic boundary conditions. CFD analysis have been carried out on the original blade geometry at different flow rate conditions. Experimental / operational data was taken from the site of the hydro-power turbine unit at uttarakhand power project.

1. Inlet velocity of flow: 2.8 m/s
2. Number of Blades: 3
3. Hub: 0.1 mt.
4. Inner Rotating
   Wall: Diameter: 2.1
   mt. Length: 0.6 mt.
5. Outer Rotating
   Wall: Diameter: 4
   mt. Length: 0.8 mt.
6. Rotational Speed of Blade: 90rpm

.

3.2.6 ANSYS FLUENT

The use of advanced computational fluid dynamics techniques is very helpful in the analysis of complex flow patterns. In this work ANSYS FLUENT was used for such a purpose. FLUENT is a general purpose computational program for modeling fluid flow, heat transfer, and chemical reactions. FLUENT models this wide range of phenomena by solving the conservation equations for mass, momentum, energy using a control volume based finite-difference method.

Steps to solve problem:

Once the important features of the problem are determined, the basic procedural steps are those shown below:

1. Create or import the model geometry and grid.
2. Choose the basic equations to be solved (turbulence transport).
3. Identify additional models needed.
4. Specify the boundary conditions.
5. Specify the fluid properties.
6. Adjust the solution control parameters (optional).
7. Calculate a solution (fluid phase and/or dispersed phase).
8. Examine the results.

3.2.7 Governing equations

The numerical modeling of hydro-kinetic turbines is complicated due to the rotation of the turbine coupled with turbulence and stall effects. A moving reference frame was incorporated to take this into account and transform an unsteady flow in an inertial (stationary) frame to a steady flow in a non-inertial (moving) frame. A constant rotational speed ( ) is provided on a steadily rotating flow geometry, and equations of fluid motion have been transformed to a rotating frame as shown below

$$\nabla \cdot \vec{U}_r = 0,$$

$$\rho\left[\frac{\partial}{\partial t}(\vec{U}_r) + \nabla \cdot (\vec{U}_r\vec{U}_r) + (2\vec{\Omega} \times \vec{U}_r + \vec{\Omega} \times \vec{\Omega} \times \vec{r})\right] = -\nabla p + \nabla \cdot \tau_r,$$

Where Ur is the relative velocity viewed from rotating reference frame, is the rotational speed of the turbine,

the Coriolis force = $(2U\ )$

the centrifugal force = (r)

the pressure gradient across the turbine ∇p

The viscous stress tensor is defined as:

$$\tau_r = \mu_{eff}[(\nabla\vec{U} + \nabla\vec{U}^T) - \frac{2}{3}\nabla \cdot \vec{U}I],$$

where U is the absolute fluid velocity and I is the identity tensor. The molecular viscosity is the sum of the dynamic viscosity and the turbulent eddy viscosity. lt being calculated from a representative turbulence model. Amongst different turbulence models that exist in literature, the k- shear stress transport (SST) model was chosen for the analysis due to its capability of providing accurate flow-field predictions under adverse pressure gradient and separated flow conditions both of which are prevalent in hydro-kinetic turbines. The k- SST model is based on the robust and accurate combination, which uses k- model in near wall region and k- model in far field region. For flows having adverse pressure gradients, the level of eddy viscosity primarily determines the accuracy of the turbulence model in predicting flow separation. Since the standard k-model fails to predict pressure induced separation, the model was reconstructed enforcing Bradshaw's observation in which turbulent shear stress is proportional to the turbulent kinetic energy in the wake region of the boundary layer. Therefore, using the k-formulation, the model solves for the transport of turbulent shear stress which controls the level of eddy viscosity in the outer part of boundary layer. However, since the k- model has strong sensitivity to the free-stream value outside the boundary layer, a transformed k- model is applied on the far wall region due to its insensitive nature to free stream turbulence. The governing equations for k- SST model is given by

$$\frac{\partial}{\partial t}(\rho k) + \nabla\cdot(\rho k\vec{U}) = \tau_{ij}\nabla\vec{U} - \beta^*\rho\omega k + \nabla\cdot[(\mu+\sigma_k\mu_t)\nabla k],$$

$$\frac{\partial}{\partial t}(\rho\omega) + \nabla\cdot(\rho\omega\vec{U}) = \frac{\gamma}{\nu_t}\tau_{ij}\nabla\vec{U} - \beta\rho\omega^2 + \nabla\cdot[(\mu+\sigma_\omega\mu_t)\nabla\omega] + 2(1-F_1)\rho\sigma_{\omega 2}\frac{1}{\omega}\nabla k\nabla\omega$$

where F1 denotes the blending function which is designed in such a manner that it assumes the value of unity inside the viscous sub-layer where original k- model is activated and it gradually switches to zero in the wake region where transformed k- model is activated. The model constants are the same as provided in the original work and are not repeated for the purpose of brevity.

The present study assumes steady, incompressible flow where the numerical solution is carried out by solving conservation equations for mass and momentum by deploying an unstructured grid finite volume methodology using commercial CFD software (Fluent 13.0). The geometrical model was created using PRO-E and meshed in HYPERMESH. Numerical simulations were performed to obtain flow hydrodynamics for three-dimensional rotating boundary conditions. The choice of hydrofoil for hydro-kinetic turbines is primarily governed by the geometry that produces maximum lift coefficient (CL) under the operating range of Re. Previous studies used SG-6043 airfoil for the design of small wind turbines due to its capability of producing large CL in the Re range of 10^5– 10^6. Since the Re for our case also lies within this range, the SG-6043 airfoil was selected as a hydrofoil for the HAHkT blades. SG-6043 hydrofoil is a part of a family of untwisted, constant pitch blades. A turbine of radius 1 m was chosen for the three-dimensional rotational analysis. The computational domain consists of two cylinders; the inner cylinder and outer cylinder. The turbine is placed inside the inner cylinder. Multiple reference frames have been adapted with a stationary outer cylinder and rotating inner cylinder and an interior boundary between the two.

3.3 FORMULATION AND DESIGN

Type of airfoil used is SG6043 which is used by earlier studies in this field. Similarly number of blades, Blade pitch angle, and Blade Radius are taken on the basis of earlier work. The inlet velocity of flow taken according to the flow potential of rivers of uttarakhand. The average range of river flow velocity are 1.5-3.5 m/s which are more than the flow speed used by earlier studies, which used normally 2 m/s average flow speed.

Rotational speed is taken according to the rotational speed using in uttarakhand hydro-kinetic power generation projects. Our value of TSR is according to our rotational speed and flow velocity. The Chord length varies in the range which are optimize by many research work but with different values.

The various parameters are:

7. Blade Radius: 1 mt.
8. Blade Pitch(degree): 10
9. Type of Airfoil : SG6043
10. Inlet velocity of flow: 2.8 m/s
11. Number of Blades: 3
12. Hub: 0.1 mt.
13. Inner Rotating Wall: Diameter: 2.1 mt. Length: 0.6 mt.
14. Outer Rotating Wall: Diameter: 4 mt. Length: 0.8 mt.
15. Rotational Speed of Blade: 90rpm

10\. Tip Speed Ratio: 3.36

11\. Chord Length(mt): 0.12, 0.15, 0.20

3.3.1 Design of Blade.

We chose SG6043 airfoil based on earlier work from airfoil tool and change it according to our required chord length. We get the CSV file and export it to the Pro-E and create a 3-D blade of hydrokinetic turbine of radius 1 mt. A hub of radius 0.1 is created and we create two more blade on it so our total number of blade become 3 at an angle of 120 degree.

The Chord Length used to generate airfoil and hence blade are 0.12.0.15,0.2 meters.

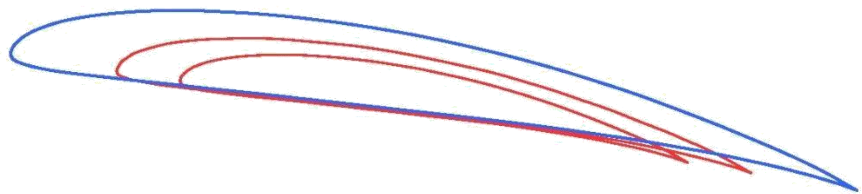


Fig.3.7 Comparison of all the three Airfoil Used for analysis.

3.3.2: DESIGN OF INNER AND OUTER ROTATING WALL and DEFINING DOMAIN.

Dimensions given:

1. Inner Rotating
   Wall: Diameter: 2.1
   mt. Length : 0.6 mt.
2. Outer Rotating
   Wall: Diameter: 4
   mt. Length: 0.8 mt.

GRID INDEPENDENT SOLUTION:

A grid independence study was carried out to study effect of number of elements on the CFD analysis. Mesh size was varied from a coarser mesh to finer mesh and flow variable were monitored. The numerical computations were initially carried out with various levels of refinement of mesh in order to obtain a grid independent solution. For the present case, the grid independence study was performed for three dimensional case by calculating the total thrust generated at the blades using three different grid sizes. The fractional change in the magnitude of the thrust was calculated based on the formula,

$$\text{ERROR (\%)} = (T-T_o)/T_o \times 100$$

Where T denotes thrust at different grid sizes and To denotes thrust corresponding to grid independent geometry.

CHAPTER 4

CALCULATIONS, RESULTS AND DISCUSSION.

4.1 CALCULATIONS AND ANALYSIS:

Analysis is done using ANSYS software, in which fluent is used. Solidity of turbine is 0.057,0.072 and 0.095 for the chord length of 0.12, 0.15, 0.20 mt of blade respectively.

4.1.1 ANALYSIS 1:

CHORD LENGTH= 0.12 mt.

Rotation speed= 90 rpm or 9.42 rad/sec

Flow speed= 2.8 m/s

Hence TSR = (1x9.42)/2.8

= 3.36

Total Thrust generated on single blade,

F = 4412 N

Hence total thrust generated on three blades

F (total) = 4412x3

= 13236 N

Torque generated on one blade= 2799 N-mt.

Hence Torque generated on three Blades= 2799x3

= 8397 N-mt.

But this thrust and Torque generated include thrust and torque generated due to both rotation of blades and flow of water. We need the thrust and torque generated only due to flow of water.

Hence again analyzing the system with zero flow speed, which gives us thrust and torque generated due to rotation and on subtracting this from total torque and thrust, we get the thrust and torque generated due to flow of water.

Thrust due to Rotation of blade= 3668 N

Hence total thrust generated due to rotation of three blade= 3668x3

= 11004 N

Hence THRUST generated due to Flow of Water= 13236-11004

= 2232 N

Torque generated due to rotation of blade=2448 N-mt

Hence torque generated due to rotation of three blades= 2448x3

= 7344 N-mt

Hence total torque generated due to flow of water= 8397-7344

= 1053 N-m

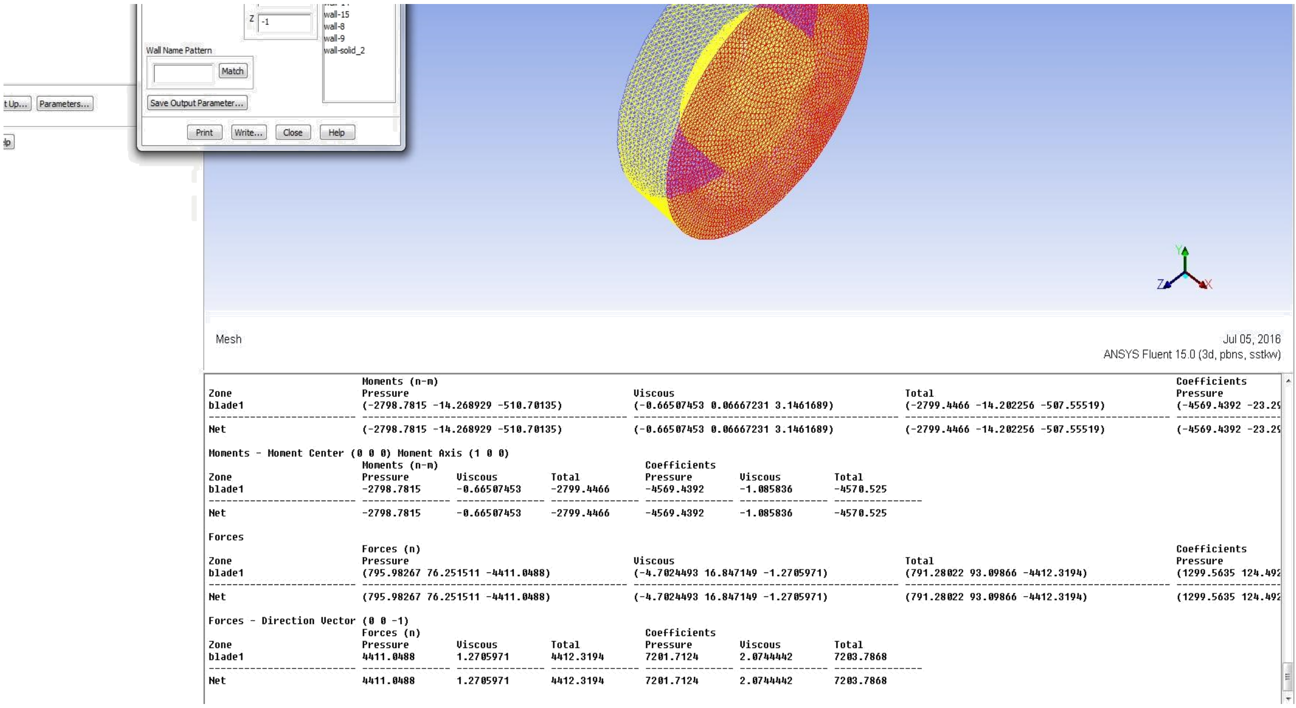


Fig.4.1 total THRUST and TORQUE generated.(for chord length 0.12 mt)

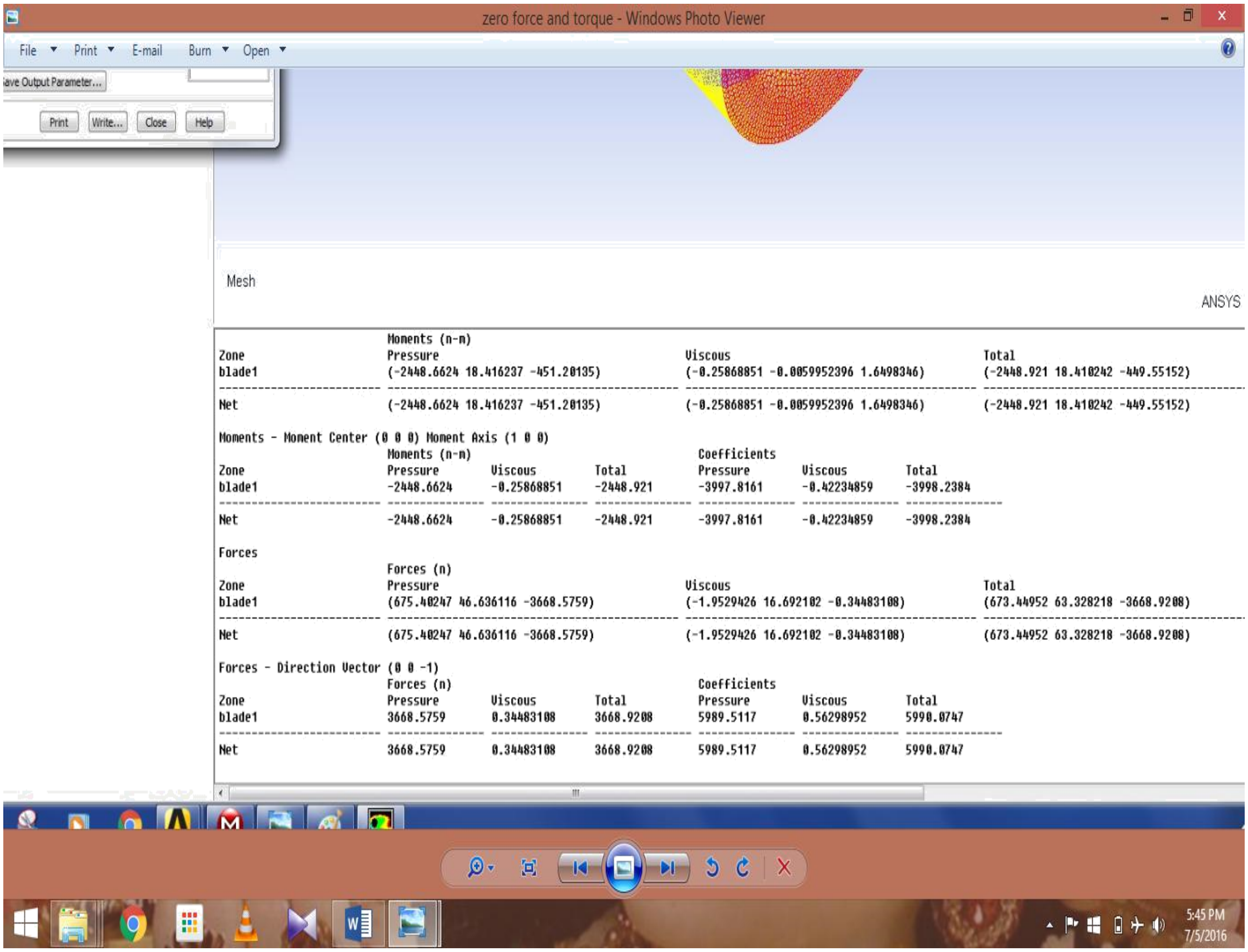


Fig.4.2 total THRUST and TORQUE generated due to flow.(for chord length 0.12 mt)

Power generated by the turbine due to flow, P= T x

w w is rotation speed in rad/sec.

P= 1053x9.42

P= 9919 w

Hydropower supplied = 0.5 x x A x V^3

= 0.5x1000x3.14x$(2.8)^3$

= 34482 W

The performance of HAHkTs is primarily determined by the power coefficient (Cp) defined as

$$C_P = \frac{P_{out}}{\frac{1}{2}\rho U^3 A}$$

Hence Coefficient of Performance Cp= 9919/34482

Cp = 0.29

RESULTS FOR ANALYSIS 1:

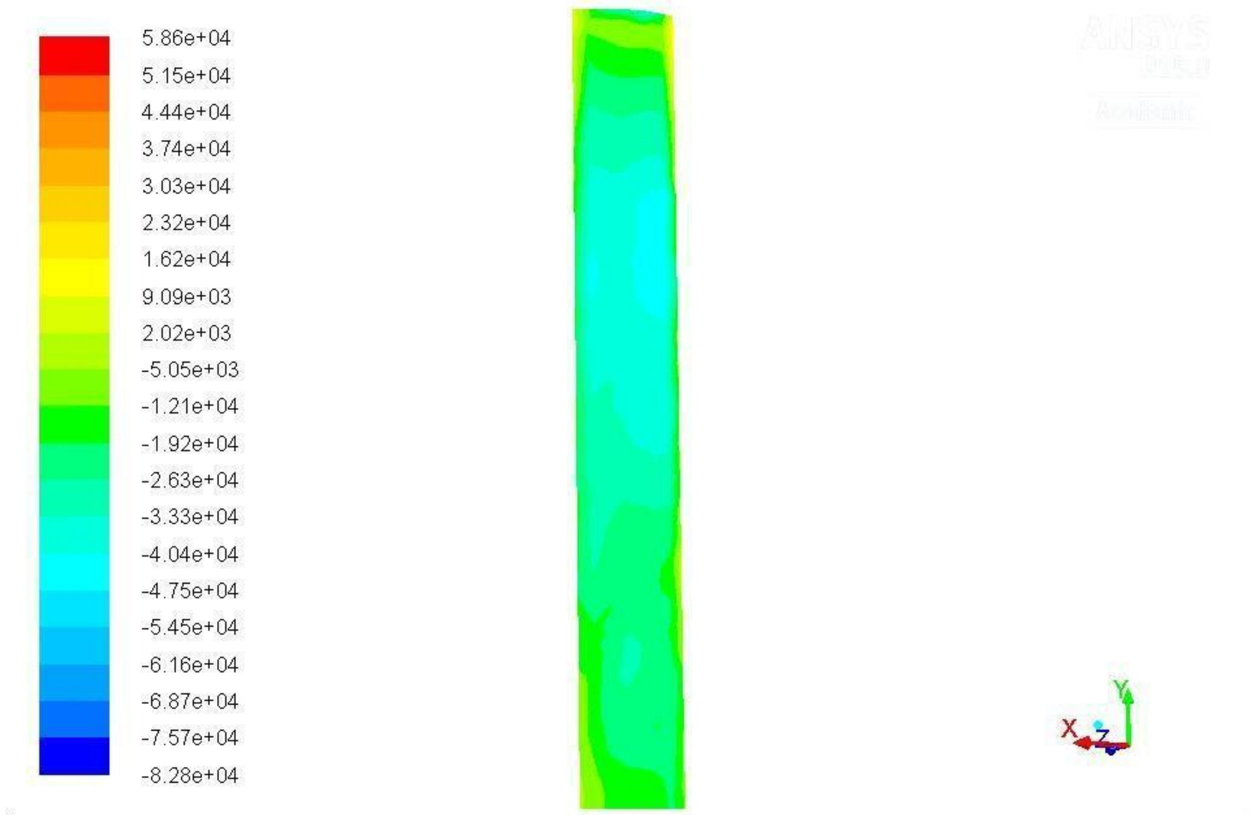


Fig.4.3 Suction side of blade.

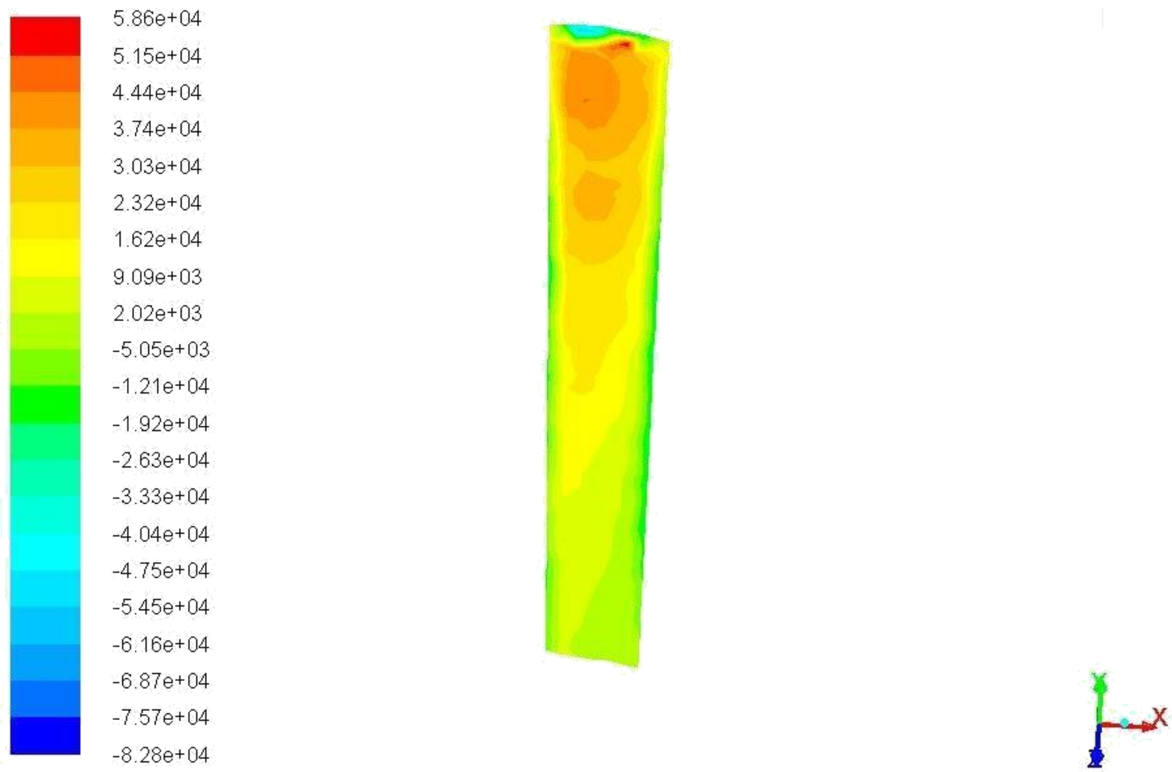


Fig.4.4 Pressure side of blade

4.1.2 ANALYSIS 2:

CHORD LENGTH= 0.15 mt.

Rotation speed= 90 rpm or 9.42 rad/sec

Flow speed= 2.8 m/s

Hence TSR = (1x9.42)/2.8

= 3.36

Total Thrust generated on single blade,

F = 5913 N

Hence total thrust generated on three blades

F (total) = 5913x3

= 17739 N

Torque generated on one blade= 3637 N-mt. Hence

Torque generated on three Blades= 3637x3

=10911 N-mt.

Thrust due to Rotation of blade= 4743 N

Hence total thrust generated due to rotation of three blade= 4743x3 =

14229 N

Hence THRUST generated due to Flow of Water=17739-14229 N

= 3510N

Torque generated due to rotation of blade= 3099 N-mt

Hence torque generated due to rotation of three blades= 3099x3

= 9297 N-mt

Hence total torque generated due to flow of water= 10911-9297

=1614 N-mt

Power generated by the turbine due to flow, P= Tx w

w is rotation speed in rad/sec.

P= 1614x9.42

P= 15.2 Kw

Hydropower supplied= 0.5 x x A x V^3

= $0.5x1000x3.14x(2.8)^3$

= 34482 W

The performance of HAHkTs is primarily determined by the power coefficient (Cp) defined as

$$C_P = \frac{P_{out}}{\frac{1}{2}\rho U^3 A}$$

Hence Coefficient of Performance Cp= 15200/34482

Cp = 0.44

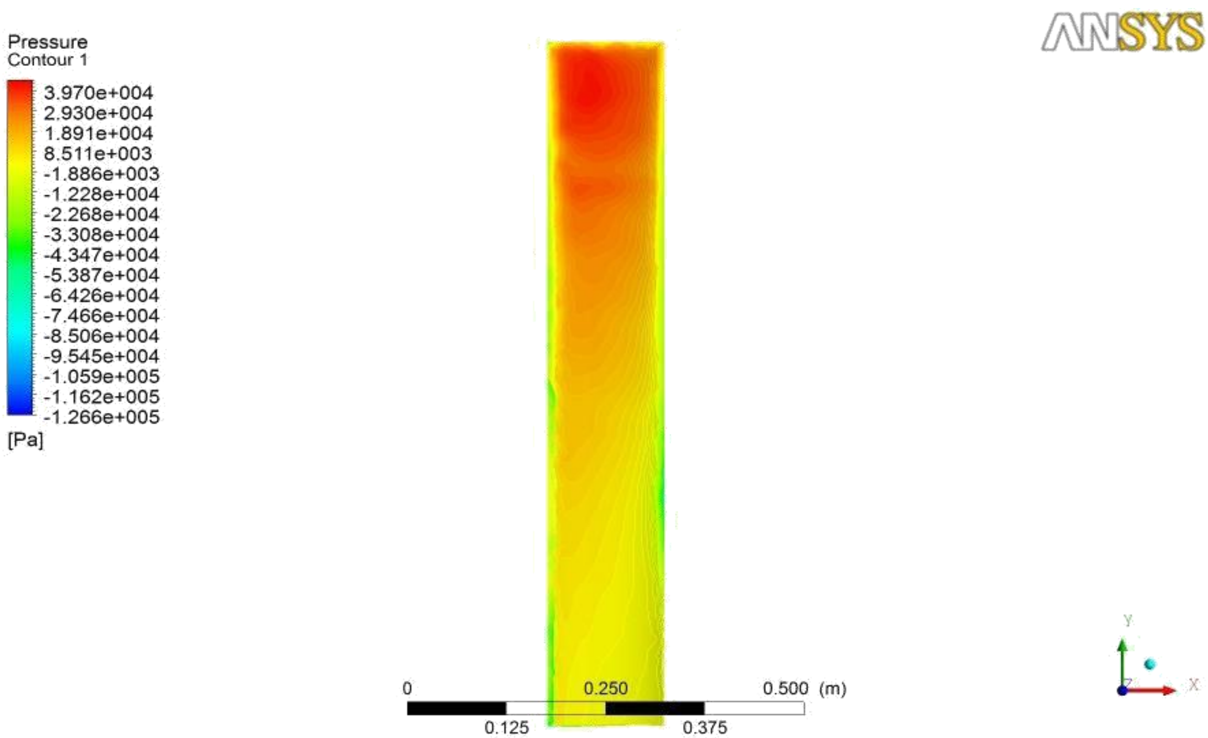


Fig.4.9 Pressure side of blade.

Fig.4.10 side view of flow.

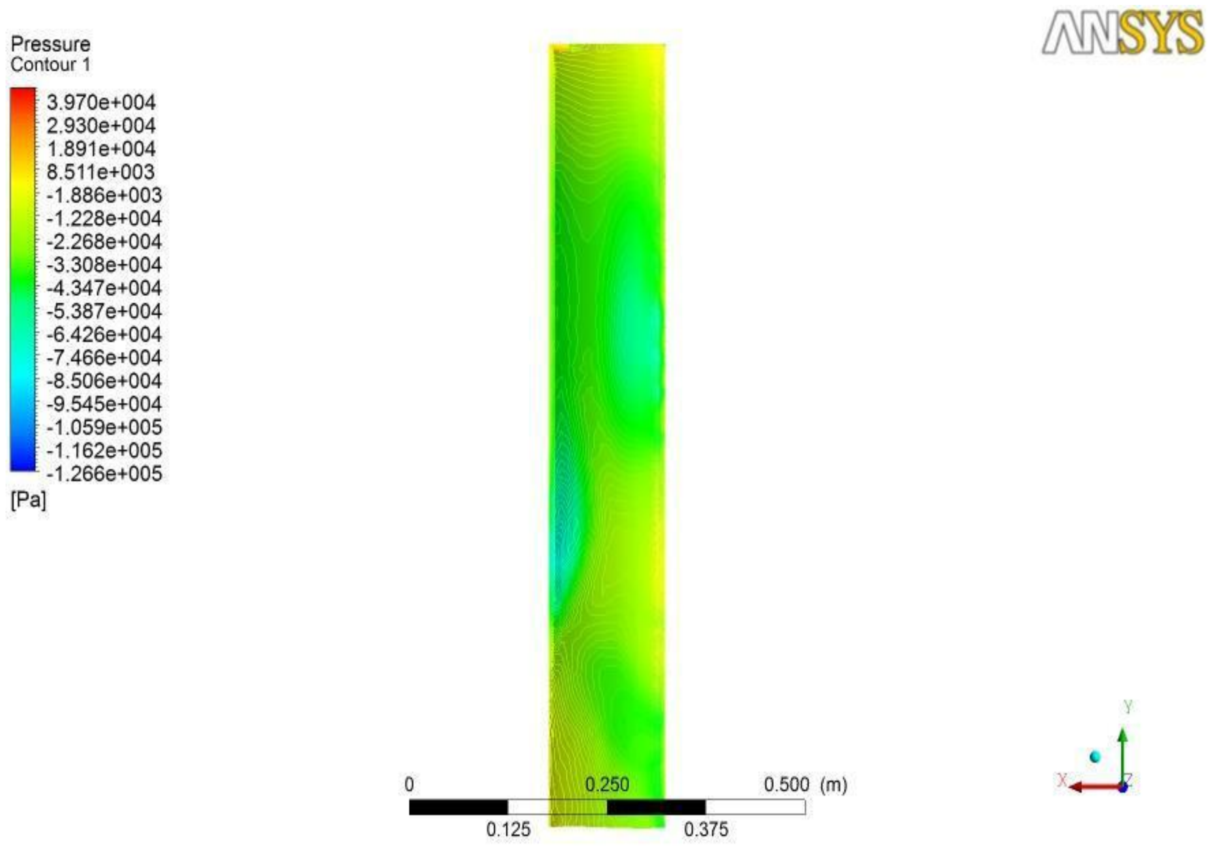


Fig.4.11 suction side of blade

4.1.3 ANALYSIS 3:

CHORD LENGTH= 0.20 mt.

Rotation speed= 90 rpm or 9.42 rad/sec

Flow speed= 2.8 m/s

Hence TSR = (1x9.42)/2.8

= 3.36

Total Thrust generated on single blade,

F = 8143 N

Hence total thrust generated on three blades

F (total) = 8143x3

= 24429 N

Torque generated on one blade= 4987 N-mt. Hence

Torque generated on three Blades= 4987x3

=14961 N-mt.

Thrust due to Rotation of blade= 6888 N

Hence total thrust generated due to rotation of three blade= 6888x3 = 20664 N

Hence THRUST generated due to Flow of Water=24429-20664 N

= 3765 N

Torque generated due to rotation of blade=4335 N-mt

Hence torque generated due to rotation of three blades= 4335x3

= 13005 N-mt

Hence total torque generated due to flow of water= 14961-13005

=1956 N-mt

Power generated by the turbine due to flow, P= T x

w w is rotation speed in rad/sec.

P= 1956x9.42

P= 18.42 Kw

Hydropower supplied = 0.5 x x A x V^3

= 0.5 x 1000 x 3.14 x $(2.8)^3$

= 34482 W

The performance of HAHkTs is primarily determined by the power coefficient (Cp) defined as

$$C_P = \frac{P_{out}}{\frac{1}{2}\rho U^3 A}$$

Hence Coefficient of Performance Cp= 18425/34482

Cp = 0.53

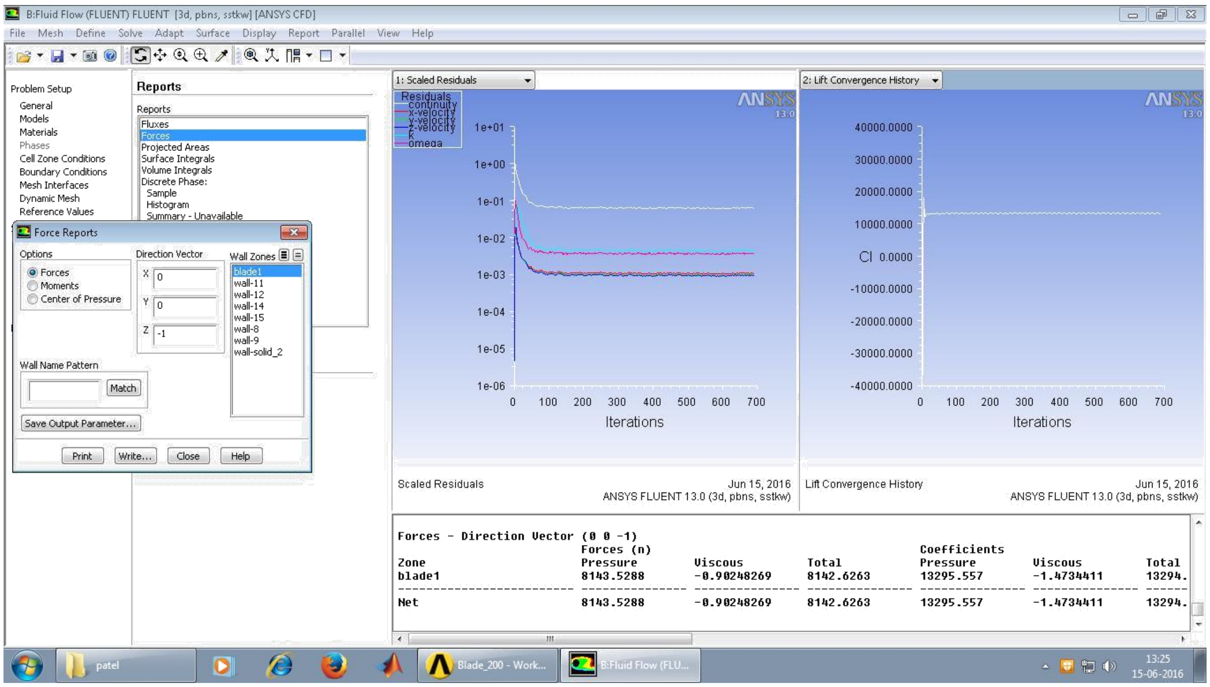


Fig.4.12 total THRUST generated

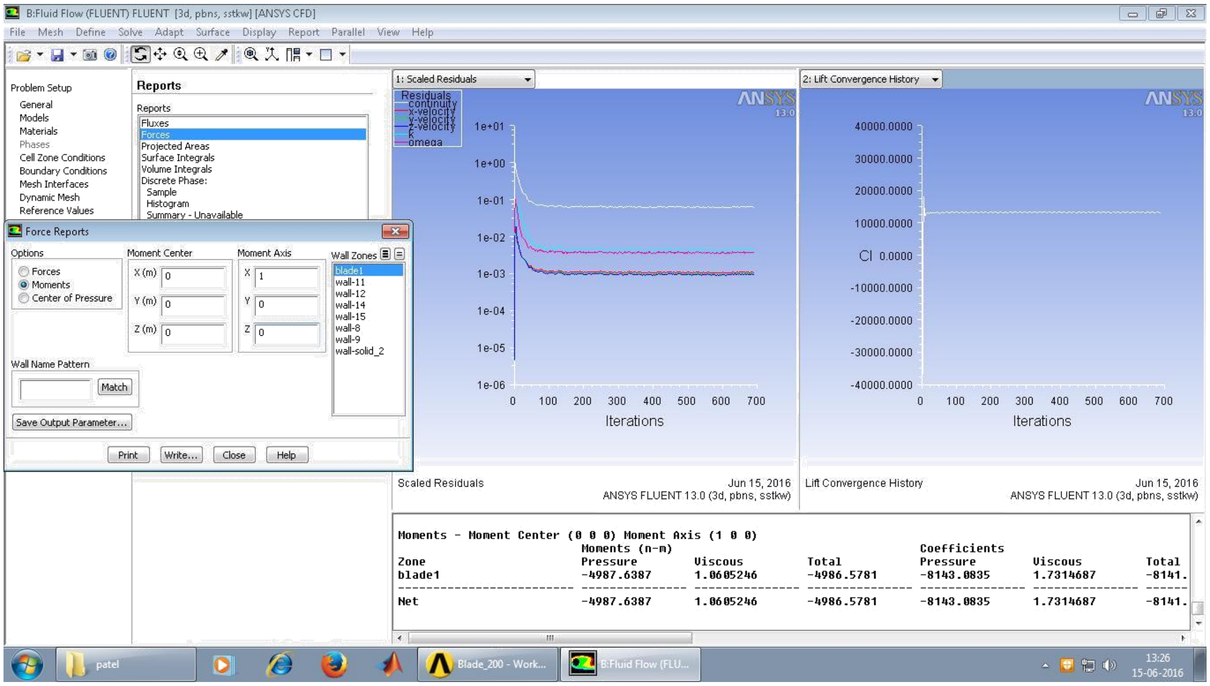


Fig.4.13 Total TORQUE generated

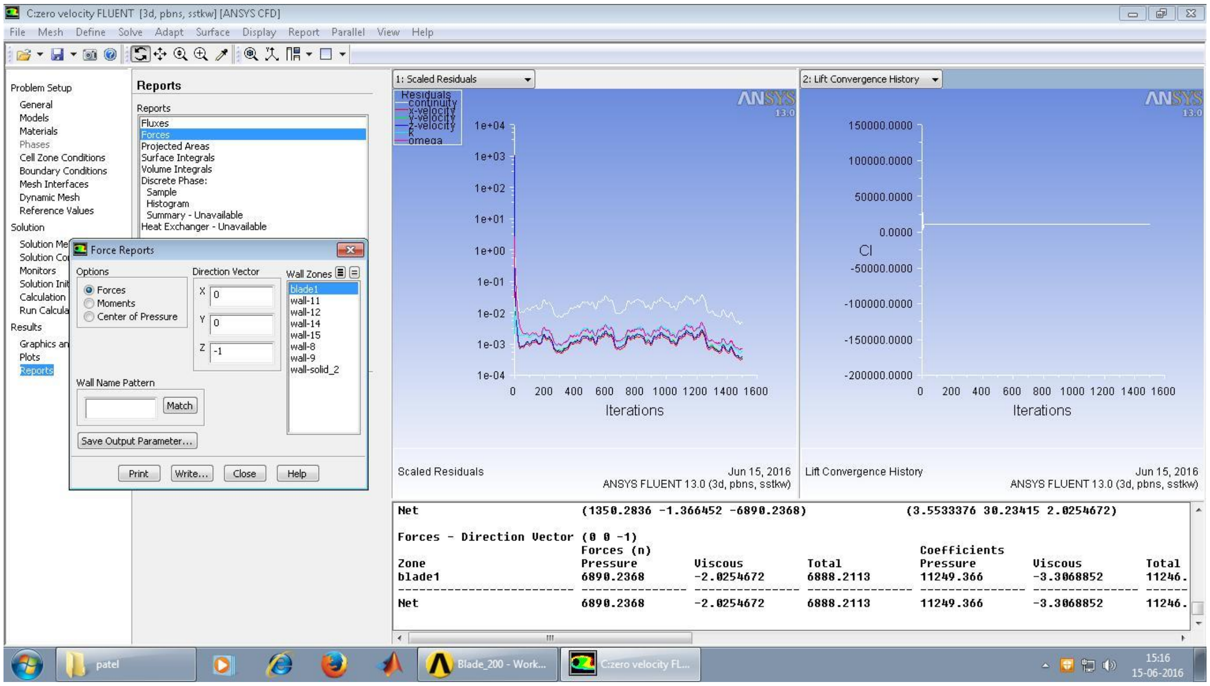


Fig.4.14 Thrust generated due to flow.

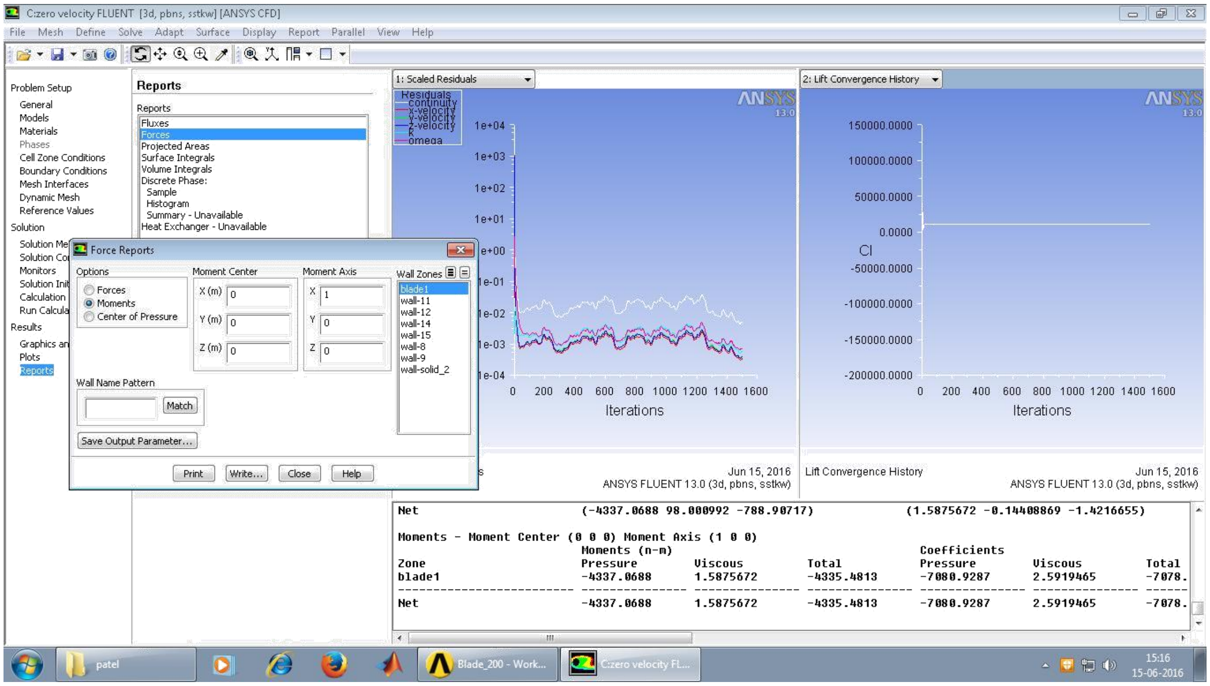


Fig. 4.15 Torque generated due to flow

RESULT OF ANALYSIS 3:

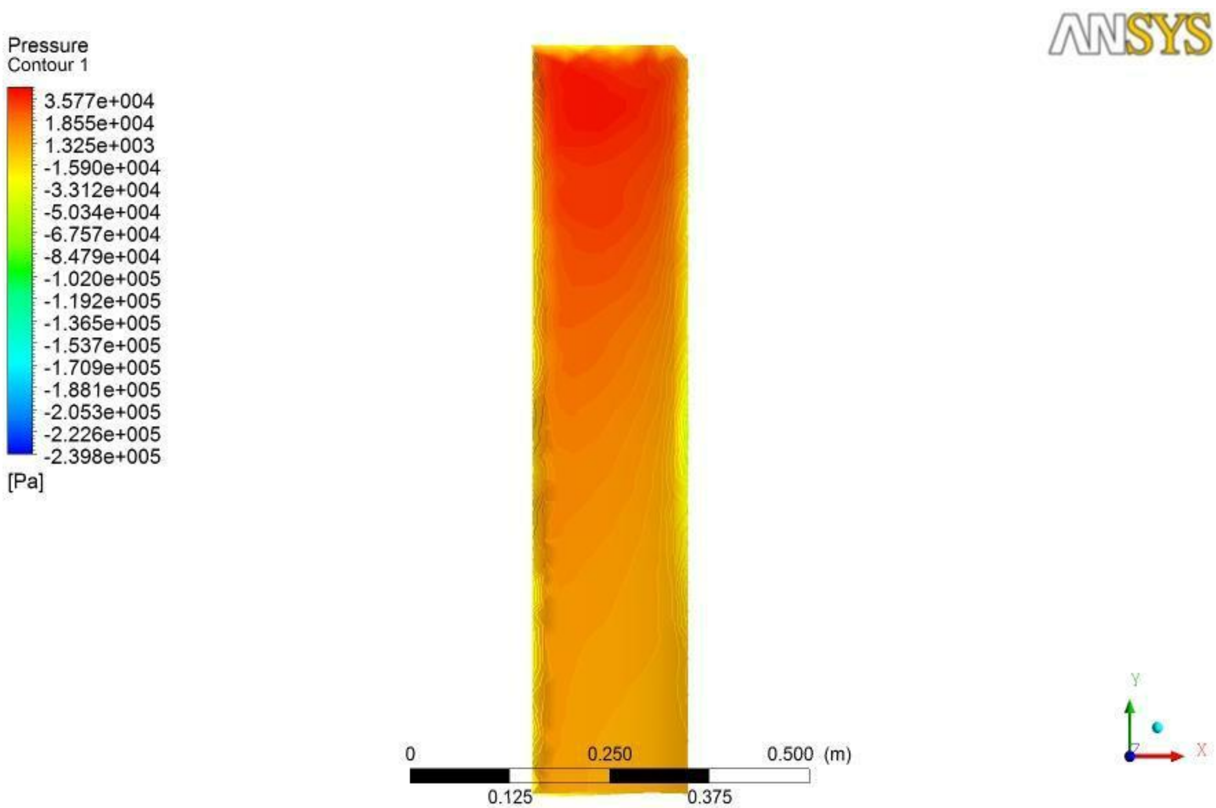


Fig.4.16 Pressure side of Blade

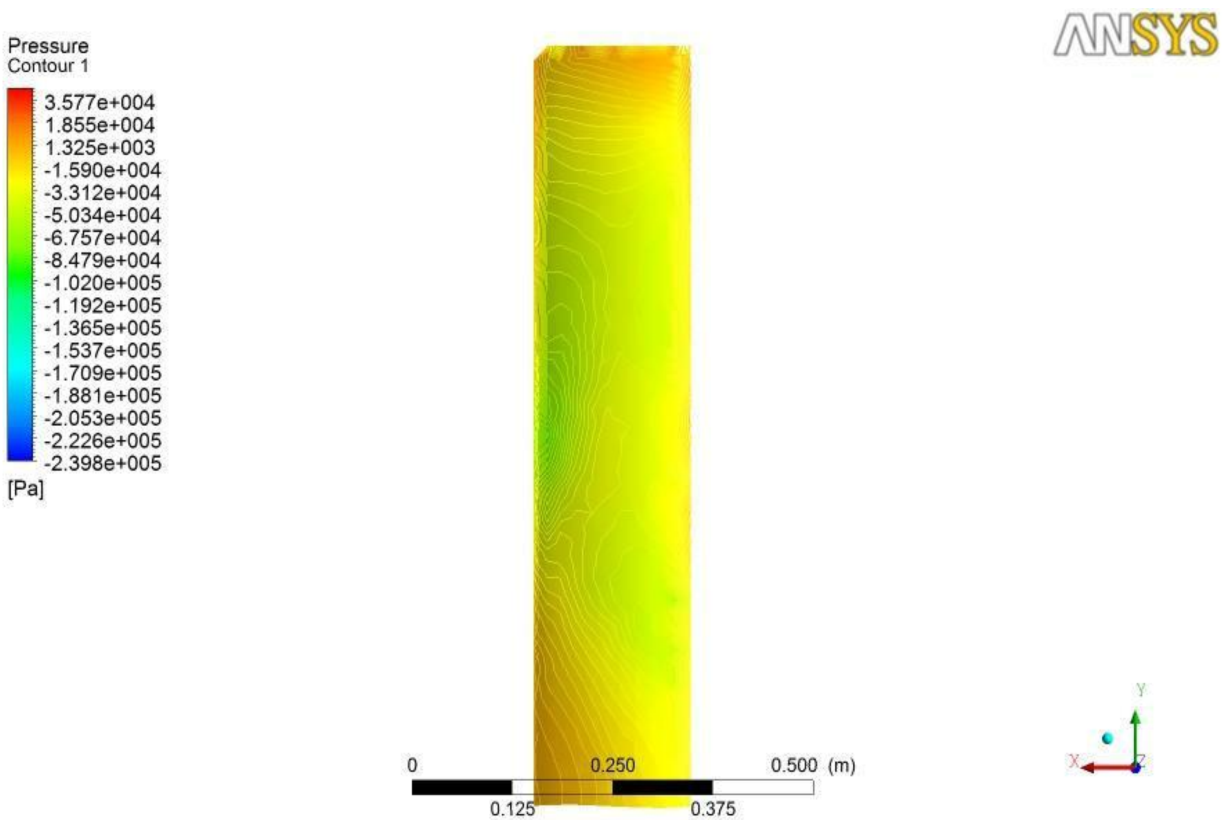


Fig.4.17 Suction side of Blade

Fig.4.18 front view of flow

Fig. 4.19 side view of flow

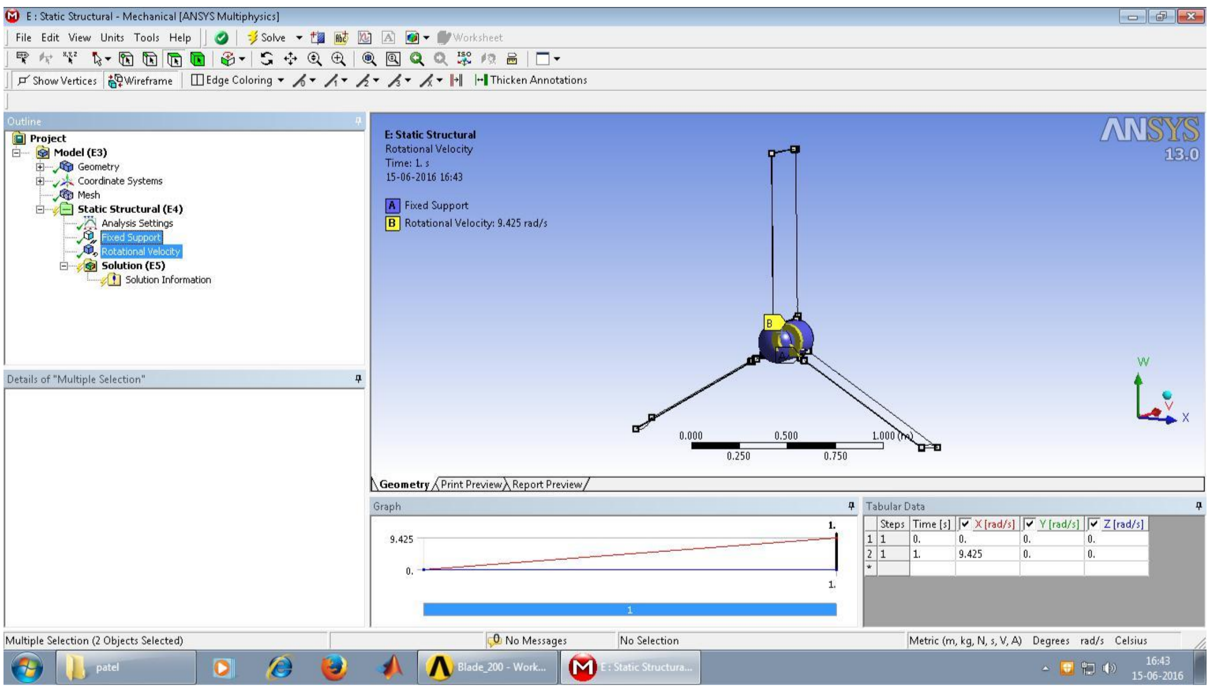
4.2 STRUCTURAL STRESS ANALYSIS:

Structural stability of turbine blades is very important during operations in the hydro-power plants. As such it is very necessary that all the mechanical equipment especially turbine blades should conform to the stability criteria from strength point of view.

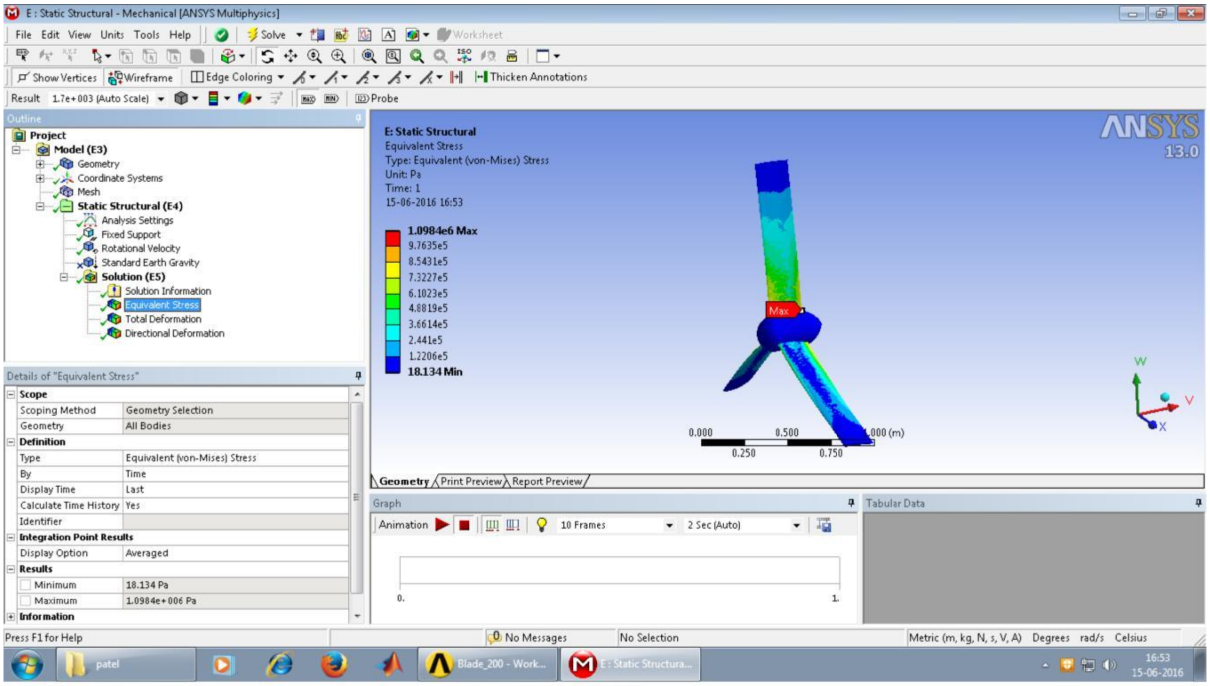
Material used: Stainless steel AISI 304

This analysis takes place on blade only. Structural stress includes stress generated due to rotation, gravity and water flow mainly. The optimized result i.e. at 0.2 mt chord length we used for the stress analysis.

STEP 1: SELECTION OF BLADE



STEP 2: CENTRIFUGAL STRESS DISTRIBUTION IS



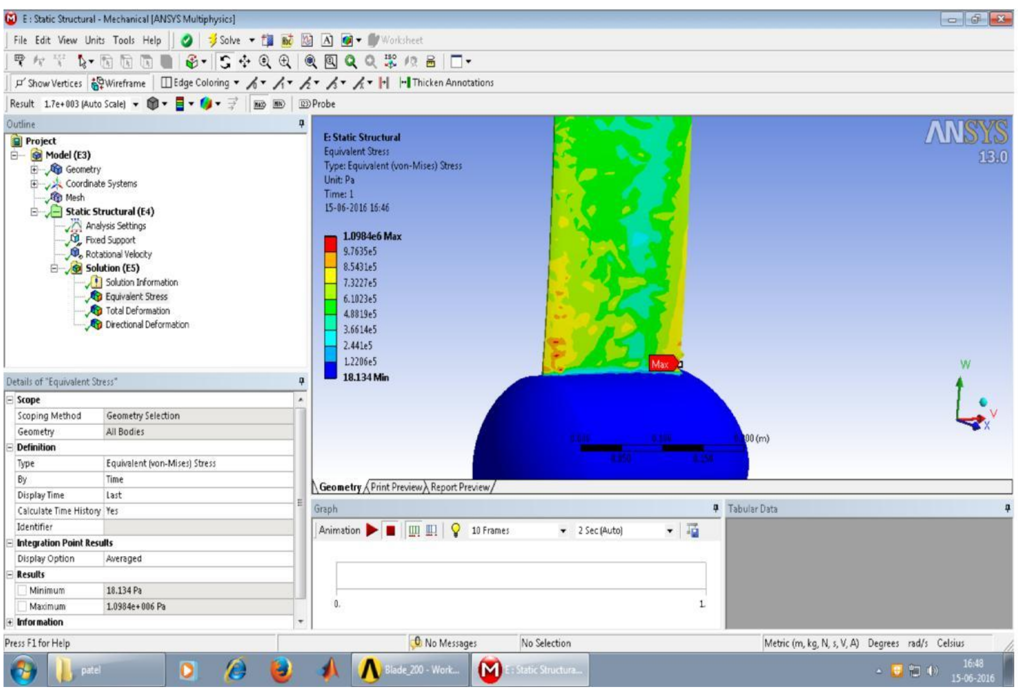
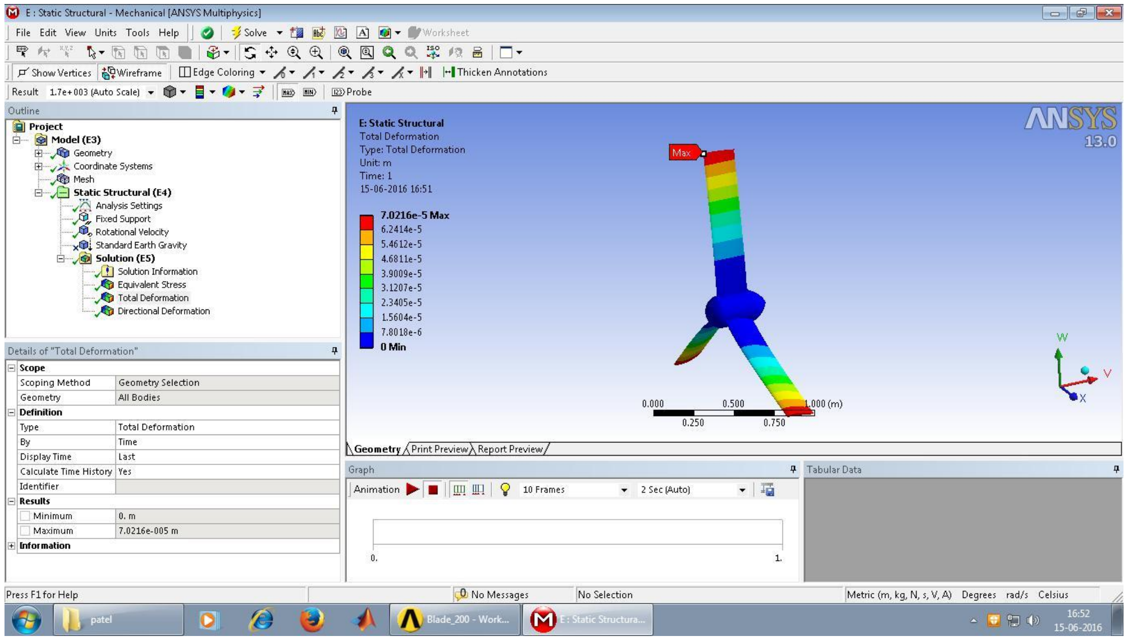
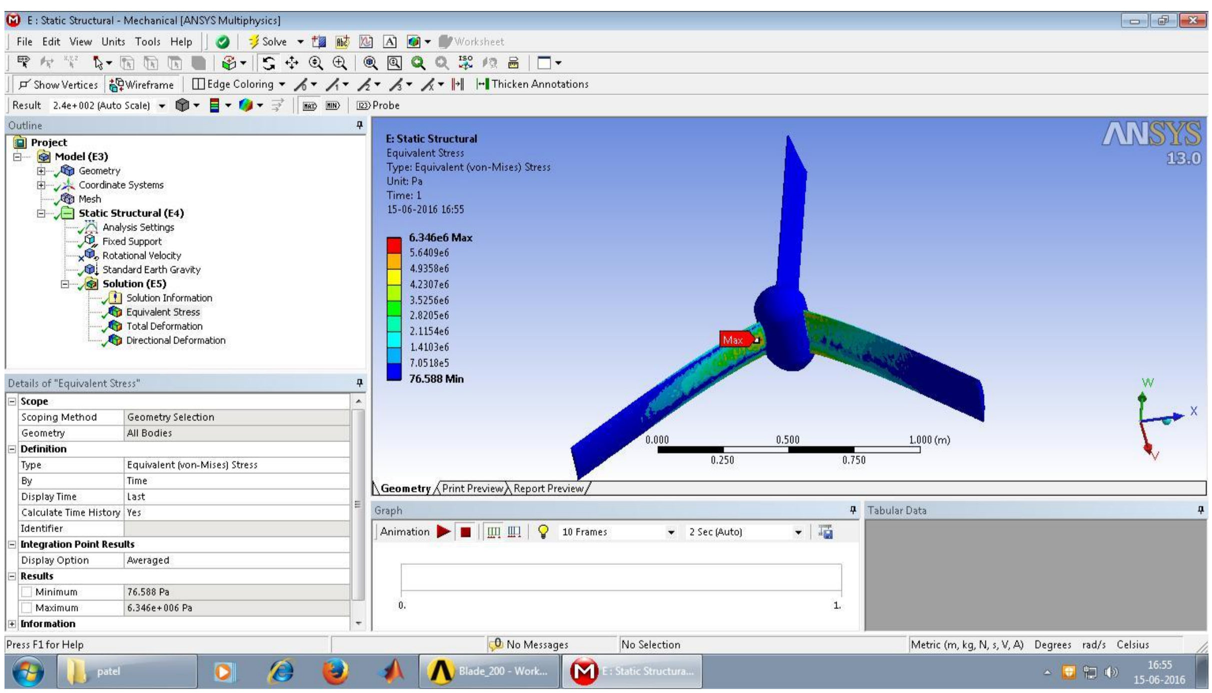


Fig.4.20 Maximum stress is near the hub.

Fig4.21Deformation due to centrifugal stress, maximum at the tip of the blade.



STEP 3: STRESS DUE TO GRAVITY.



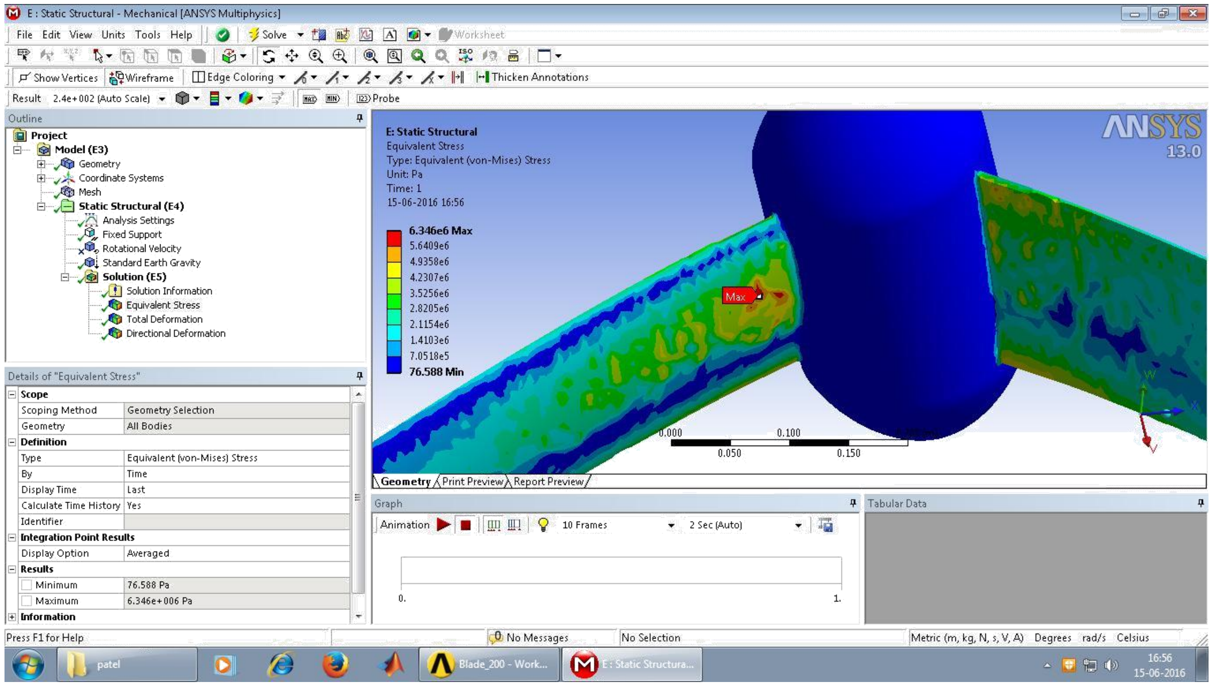
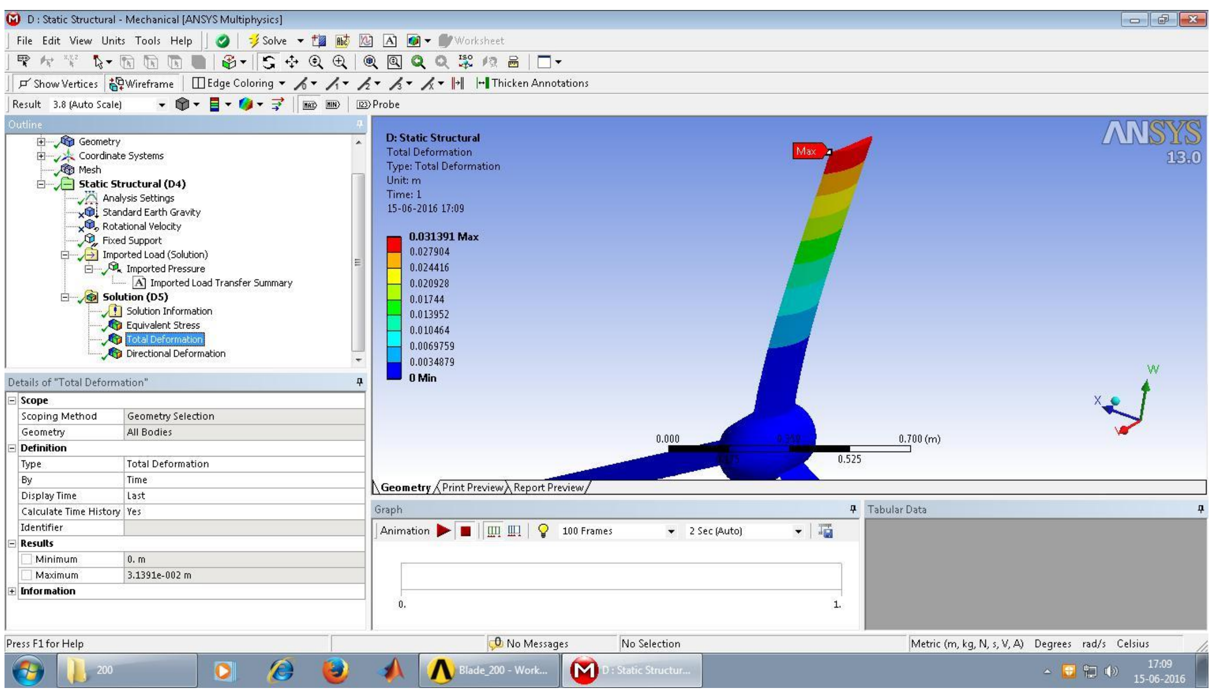


Fig 4.22DEFORMATION is maximum at the tip of the blade as thrust is maximum.



4.3 DISCUSSION:

Different values of thrust, torque, power generated and coefficient of performance obtain are summarizes in the table below.

BLADE PITCH ANGLE	**CHORD LENGTH (METER)**	**SOLIDITY**	**TSR**	**THRUST (N)**	**TORQUE (N-mt)**	**POWER (KW)**	**COEFFICIENT OF PERFORMANCE (Cp)**
10	**0.12**	**0.057**	**3.36**	**2232**	**1053**	**9.919**	**0.29**
10	**0.15**	**0.071**	**3.36**	**3510**	**1614**	**15.2**	**0.44**
10	**0.20**	**0.095**	**3.36**	**3765**	**1956**	**18.42**	**0.53**

Table:4.1 different values obtain according to different input parameters

In these results we found that there is increase in Torque, Thrust and Power generated by hydrokinetic turbine system as we increases the Chord Length of blade. Coefficient of performance is increases as 0.29, 0.44, 0.53 for the chord length of 0.12, 0.15, 0.2 mt respectively. For the validation purpose earlier research work is used here which shows that the coefficient of performance increases with increasing chord length [3]
[21]. A corresponding range of thrust generated is also shown in the table for different chord length, TSR and blade pitch angle [4].

Variables			CFD		BEM	
Blade Pitch (°)	Chord (m)	TSR	Thrust (N)	C_p	Thrust (N)	C_p
10	0.12	4.0	4753	0.45	4832	0.42
12	0.12	4.0	4241	0.42	4403	0.43
13.5	0.14	3.1	3627	0.35	3881	0.36
14.4	0.14	3.5	3817	0.38	4121	0.39
6.9	0.1	6.9	8134	0.33	6577	0.35

Fig.4.26 Thrust and Cp values obtain in earlier work [4]

PERFORMANCE ANALYSIS OF BLADE ON DIFFERENT TSR:

Blade taken is of 150 mm for this analysis.

ANALYSIS 1: TSR = 2.5; rotational speed is 7 rad/sec of 67 rpm.

Total Torque generated on one blade = 2103 N-mt.

Hence Torque generated on three Blades = 2103x3

=6309 Nm

Torque generated due to rotation of blade= 1743 N-mt

Hence torque generated due to rotation of three blades= 1743x3

= 5229 N-mt

Hence total torque generated due to flow of water= 6309-5229

=1080 N-mt

Power generated by the turbine due to flow.

P= 1080x7

P= 7560 W

Hence Coefficient of Performance Cp= 7560/34482

Cp = 0.22

ANALYSIS 2: TSR = 3; rotational speed is 8.4 rad/sec of 80 rpm.

Total Torque generated on one blade = 2910 N-mt.

Hence Torque generated on three Blades = 2910x3

=8730 Nm

Torque generated due to rotation of blade= 2508 N-mt

Hence torque generated due to rotation of three blades= 2508x3

= 7524 N-mt

Hence total torque generated due to flow of water= 8730-7524

=1206 N-mt

Power generated by the turbine due to flow.

P= 1206x8.4

P= 10130 W

Hence Coefficient of Performance Cp= 10130/34482

Cp = 0.29

ANALYSIS 3: TSR = 3.36; rotational speed is 9.4 rad/sec of 90 rpm.

Coefficient of Performance **Cp= 0.44**

ANALYSIS 4: TSR = 3.75; rotational speed is 10.5 rad/sec of 100 rpm.

Total Torque generated on one blade = 4421 N-mt.

Hence Torque generated on three Blades = 4421x3

=13263 Nm

Torque generated due to rotation of blade= 3986 N-mt

Hence torque generated due to rotation of three blades= 3986x3

= 11958 N-mt

Hence total torque generated due to flow of water= 13263-11958

= 1305 N-mt

Power generated by the turbine due to flow.

P= 1305x10.5

P= 13702 W

Hence Coefficient of Performance Cp= 13702/34482

Cp = 0.39

TSR	**ROTATION SPEED (rpm)**	**Coefficient of performance Cp**
2.5	67	0.22
3.0	80	0.29
3.36	90	0.44
3.75	100	0.39

Table 3: variation of coefficient of performance with TSR .

CHAPTER 5

CONCLUSION AND FUTURE SCOPE

5.1 CONCLUSION

1. As we increases the chord length of turbine blade as 0.12, 0.15, 0.20 mt, the thrust force, torque and hence power generated by the turbine is increases and because of this the coefficient of performance of turbine increases as 0.29, 0.44, 0.53 respectively. Analysis of blade with different TSR shows that coefficient of performance initially increases with TSR but after a point it start decreases.

2. Material chosen for blade is STAINLESS STELL AISI 304, in which maximum stress due to gravity is 6 MPa. Centrifugal stress is 1.1 MPa and deformation due to Centrifugal stress is 0.07 mm which is negligible. Total maximum stress generated is 346 MPa which is at negligible area so can be neglected.

Hence our material is safe.

5.2 FUTURE SCOPE:

We can use different values of CHORD LENGTH, TSR, BLADE PITCH ANGLE, to optimize further as this project use three combination as such. Apart from this we can change material of blades in order to have stress analysis. Such materials like GLASS FIBRE, COMPOSITE CARBON etc. but as motive is to generate power at minimum cost and these materials are costly, so such material should be used according to power generation, potential of sites. Further experimental datas as used in this project can be used for analysis so that result can be optimized according to Indian perspective and potential.

Recommendations:

1. Study on the use of composite material as replacement to the conventionally used materials on blades for the cost effectiveness and better and better performance. •
2. To carry research for improvement in the turbine efficiencies depending on the blade tip clearance criteria using CFD techniques.
3. To carry research work on the turbine balancing techniques using latest computational techniques.
4. Research work in the area of hydraulic structures need more attention. Use of CFD can lead to improved and optimum design of hydraulic structures.
5. To investigate the criteria for optimum design and development of other parts of power generation systems through dynamic analysis.
6. The 3D scale modeling of the given obtained results can be done for the practical analysis.

REFERENCES

1. Swiderski J., J. Martin, "Practical Implementations of Computational Fluid Dynamics in the Design Practice - Virtual Hydraulic Laboratory", Conference materials, Hydroforum, Poland, 2000.

2. Bennett K., and Swiderski, J., "Application of CFD Turbine Design for Small Hydro Elliott Falls, A Case Study", Waterpower XII, Salt lake City, 2001.

3. Mukherji, S.S., et al., Numerical Investigation Optimum and Evaluation of HydrodynamicPerformance of a Horizontal Axis Hydrokinetic Turbine. Journal of Renewable and Sustainable Energy, 2011. 3:p 063105.

4 . Nitin Kolekar at el. Hydrodynamic Design and Optimization of Hydrokinetic Turbine using a robust design method. Proceedings of the 1st marine energy technology symposium METS13 April 10-11, 2013.

5. Udit Tewari, Karl Kolmsee et al. HYDROKINETIC ENERGY FOR ENLIGHTENING THE FUTURE OF RURAL COMMUNITIES IN UTTARAKHAND International Conference on Hydropower for Sustainable Developmen Feb 05-07, 2015, Dehradun.

6. Zhou, L., Wang, Z., Xiao, R., and Luo, Y., "Analysis of dynamic stresses in Kaplan turbine blades", International Journal for Computer-Aided Engineering and Software Vol. 24 No. 8, pp. 753-762, 2007.

7. Chica E, Perez F. at el "Rotor Structural Design of A Hydrokinetic Turbine" International Journal of Applied Engineering Research ISSN 0973-4562 Volume 11, Number 4 (2016) pp 2890-2897.

8. Deschenes, C., Fraser, R., and Fau, J.P., "New Trends in Turbine Modelling and New Ways of Partnership", Laval University Hydraulic Machinery Laboratory (LAMH), Canada, 2002.

9. Dragu, C., Soens, J., and Belmans, R., "Small-Scale Renewable Energy in The Next Century Market Hydro Plants - State of The Art and Applications", ELECTA, Energy Institute, Belgium, 2003.

10. Busea, C., and Jianu, B., "Optimization of axial hydraulic turbines runner blades using Hydrodynamic simulation techniques", 6th International Conference on Hydraulic Machinery and Hydrodynamics, Timisoara, Romania, October 21 - 22, 2004

11. Ferrando, L., Kueny, J.L., Avellan. F, Pedretti, C, and Tomas, L., "Surface Parameterization of a Francis Runner Turbine for Optimum Design", 22nd IAHR Symposium on Hydraulic Machinery and Systems, June 29 – July 2, Sweden, 2004

12. Williams, A.A., and Simpson, R.G., "Application of computational fluid dynamics to the design of pico propeller turbines", Proceedings of the International Conference on Renewable Energy for Developing Countries – 2006

13. Cruz, A.G.B., Mesquita, A.L.A, and Blanco, C.J.C., "Minimum Pressure Coefficient Criterion Applied in Axial-Flow Hydraulic Turbines", Journal of the Brazil Society of Mechanical Science & Engineering, Vol. XXX, No. 1, pp.30-38, January-March 2008

14. Keck, H. Michler, W. Weiss, T. and Sick, M., "Recent Developments in the Dynamic Analysis of Water Turbines", Proceedings of the 2nd IAHR International Meeting of the Workgroup on Cavitations and Dynamic Problems in Hydraulic Machinery and Systems, pp.9-20, Timisoara, Romania, October, 2007

15. ShuHong, L., Jie, S., ShangFeng, W., and YuLin, W., "Numerical simulation of pressure fluctuation in Kaplan turbine", Science in China Press, vol. 51, No. 8, pp.1137-1148, August, 2008

16. Prasad, V., Gahlot, V.K., and Krishnamachar., "CFD approach for design optimization and validation for axial flow hydraulic turbine", Indian Journal of Engineering & Materials Sciences Vol. 16, pp. 229-236, August 2009

17. Khare, R., Prasad, V., and Kumar, S., "Derivation of Global Parametric Performance of Mixed Flow Hydraulic Turbine Using CFD" Hydro Nepal: Journal of Water, Energy and Environment, Vol-7, pp 60-64, 2010.

18. Helena, M.R., Mariana. S., "Hydrodynamic and performance of low power turbines: conception, modelling and experimental tests", International Journal of energy and environment, Volume 1, No. 3, pp.431-444, Lisbon, Portugal, 2010

19. Sutikno, P., and Adam, I.K., "Design, Simulation and Experimental of the Very Low Head Turbine with Minimum Pressure and Free Vortex Criterions", International Journal of Mechanical & Mechatronics

20. Thapa, B.S., Panthee, A., and Thapa, B., "Computational Methods in Research of Hydraulic Turbines", International Journal of Advanced Renewable Energy Research, Vol. 1, No. 2, pp.1-4, 2012 Engineering IJMME-IJENS Vol. 11, No. 01, pp.9-15, 2011

21. Suchi Subhra mukherji, Design and critical performance evaluation of horizontal axis hydrokinetic turbines,2010 master theses paper 5430.