CFD-BASED PERFORMANCE ANALYSIS OF KAPLAN TURBINE FOR MICRO HYDRO RANGE

A DISSERTATION

Submitted in partial fulfillment of the requirements for the award of the degree
of

MASTER OF TECHNOLOGY
in
ALTERNATE HYDRO ENERGY SYSTEMS

By
ALOK MISHRA

ALTERNATE HYDRO ENERGY CENTRE
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE -247 667 (INDIA)
JUNE, 2012

CANDIDATE'S DECLARATION

I hereby declare that the work which is presented in this dissertation, entitled, "**CFD-BASED PERFORMANCE ANALYSIS OF KAPLAN TURBINE FOR MICRO HYDRO RANGE**", submitted in partial fulfillment of the requirement for the award of the degree of Master of Technology in "**Alternate Hydro Energy Systems**" in **Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee,** is an authentic record of my own work carried out during the period from July 2011 to June 2012 under the supervision and guidance of **Shri. M.K. Singhal,** Senior Scientific Officer and **Dr. R. P. Saini,** Head & Associate Professor, Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee, Roorkee (India).

I also declare that I have not submitted the matter embodied in this dissertation for award of any other degree.

Date: June 11, 2012

Place: Roorkee

(ALOK MISHRA)

CERTIFICATE

This is to certify that the above statement made by the candidate is correct to the best of my knowledge.

(Shri. M.K. Singal)
Senior Scientific Officer,
Alternate Hydro Energy Centre,
Indian Institute of Technology,
Roorkee – 247 677.

(Dr. R. P. Saini)
Head & Associate Professor,
Alternate Hydro Energy Centre,
Indian Institute of Technology,
Roorkee – 247 677.

ACKNOWLEDGEMENT

I feel much honored in presenting this dissertation report in such an authenticable form of sheer endurance and continual efforts of inspiring excellence from various coordinating factor of cooperation and sincere efforts drawn from all sources of knowledge. I express my sincere gratitude to **Shri. M.K. Singhal,** Senior Scientific Officer and **Dr. R. P. Saini,** Head & Associate Professor, Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee for their valuable guidance and infilling support for the completion of dissertation work.

I am also grateful to all faculty members and staff of Alternate Hydro Energy Centre, Indian Institute of Technology, Roorkee. I extend my thanks to all classmates who have given their full cooperation and valuable suggestion for my dissertation work. I would like to express my humble respect and special thanks to my parents & others who directly or indirectly helped me during completion of this dissertation work. Last but not the least; I express my sincere gratitude to the God to achieve me the success in desired goal.

Date: June 11, 2012

Place: Roorkee

Alok Mishra

(ALOK MISHRA)

ABSTRACT

Electricity generation through conventional energy resources causes pollution, global warming & further power deficiency in the world due to increase in load demand causes the power sector to shift towards the renewable energy resources. Large part of renewable energy is covered by the hydropower, which is most promising and economical resource of energy. India is endowed with rich hydropower potential; it ranks fifth in the world in terms of usable potential. Micro hydro power plant comes under the renewable energy of hydro power. The micro hydro plant plays very important role in extraction of power from ultra low head available in stream and it helps electrification of rural which are not connected from grid.

Turbine is most critical component in hydro power plant because it affects the cost as well as overall performance of the plant. The cost effective design for any power plant directly depend upon the hydro turbine performance. Performance analysis of hydro turbine gives high quality and reliability of hydro power project and also ensure the project is economical viable or not. The turbines associated with micro hydro range have poor part load efficiency. There is need of turbine which provide good part load efficiency to extract full potential only Kaplan turbine provide good part load efficiency. To determine the part load efficiency of Kaplan turbine it is essential to do the performance analysis of turbine for different operating conditions.

The conventional approach to assess turbine performance is its model testing which becomes costly and time consuming. For micro hydro range cost is dominating factor so the model testing can't use. Computational fluid dynamics(CFD) has becomes a cost effective tool for predicting the performance of turbine and also for predicting detailed flow information in turbine space to enable the selection of best operating condition.

In the present study, the turbine of 100 kW of capacity has been considered for the CFD based performance analysis. The 3D model of the parts has been created in PRO-E wildfire-2 modeling software. Meshing of the models has been done with help of ANSYS-14 software and simulation has been done with help of FLUENT software which is one of the modules of ANSYS-14 software. The simulation has been carried out for design and off-design conditions at four different operating points. For different operating condition, at each operating condition four simulations has been taken place to optimize the runner blade angle. The optimized runner blade angle has been analyzed in order to investigate the part load efficiency of Kaplan turbine.

CONTENTS

Particulars	Page. No.

LIST OF TABLES

LIST OF FIGURES

NOMENCLATURE

LIST OF SYMBOLS

SYMBOL	DESCRIPTION
A	Area (m^2)
B	Clear width of the draft tube at exit end (m)
B_g	Height of wicket gate (m)
c	Absolute velocity (m/s)
D	Diameter of runner (m)
D_g	Diameter of wicket gate (m)
d	Diameter of hub (m)
g	Acceleration of gravity (m/s^2)
H	Height of draft tube (m)
h	Depth of draft tube (m)
Hn	Rated head (m)
H_1	Head at leading edge of turbine blade (m)
H_2	Head at tailing edge of turbine blade (m)
K_{ug}	Speed ratio
L	Length of draft tube (m)
L_g	Length of the wicket gate (m)
Ns	Specific speed
N	Rotation speed (rpm)
P	Power (kW)
P_{t1}	Pressure at the inlet of turbine (Pa)
P_{t2}	pressure at outlet of turbine (Pa)
Q	Discharge (m^3/s)
T	Torque acting on the runner (N-m)
ρ	Water density (kg/m^3)

CHAPTER 1
INTRODUCTION AND LITERATURE REVIEW

1.1 GENERAL

Demand of power increases day by day and supply of power still not matches the demand so power sector is reliable business. There is two type of source of power generation conventional and non-convectional. Fossil fuel used for convectional type of power generation which causes of pollution and global warming so government more concern about non conventional source. The demand for increasing the use of renewable energy has risen over the last few years due to environmental issues. The high emissions of greenhouse gases have led to serious changes in the climate. Although the higher usage of renewable energy would not solve the problems over night, it is an important move in the right direction. The most important renewable sources are hydropower, biomass, geothermal, solar and wind but technical, economic and environmental benefits of hydroelectric power make it an important contributor to the future world energy mix. Hydropower contributes one-fifth of the world's power generation. In fact, it provides the majority of supply in 55 countries. For several countries, hydropower is the only domestic energy resource. Its present role in electricity generation is therefore substantially greater than any other renewable energy technology [1].

1.2 ENERGY SCENARIO OF INDIA POLICY OF INDIA

The National Electricity Policy (NEP) stipulates 'POWER FOR ALL' and annual per capita consumption of electricity to rise to1000 units by 2012. Electricity is in the concurrent list in the contribution and the primary responsibility of structuring its availability and distribution is that of the states. However, both the centre and the state have to play a decisive and positive role. The India installed power generation capacity in India as on 31st March 2012 was 1,99,627.03 MW comprising of 1,31,353.18 MW thermal (65.80%), 38,990.40 MW (19.53%) hydro 4,780.00 MW nuclear (2.39%) and 24,503.45 MW R.E.S(12.27%)[2]. On the other hand the share of the state sector has declined from 82.5% to 47.28% while the share of private sector has gone up from 5.2% to 21.5% during the same period. To fulfil the objective of the NEP, a capacity addition of 78,700 MW has been proposed for the 11 plan. This capacity addition is expected to provide a growth of 9.5% to the power sector. During 12th Plan (2012-2017) constitutes 40,000 MW (Thermal), 30,000 MW (Hydro) and 12,200 MW (Nuclear) [3].

Total estimate cost of the electricity sold is Rs. 2,05,000 crore in 2008-2009 and gross subsidy on electricity is in the range of 4600 crore. The total commercial loss of the state power sector is around Rs. 26,000 crore. The billing and collection efficiencies are in the range of 71% and 94% respectively (national average). The aggregate technical and (AT&C) losses are in the range of 18%-62% with an average national level figure estimated at around 33.07% primarily on account of power theft and inefficient billing and collection. Figure 1.1 present contribution of various energy sources in India power sector in percentage

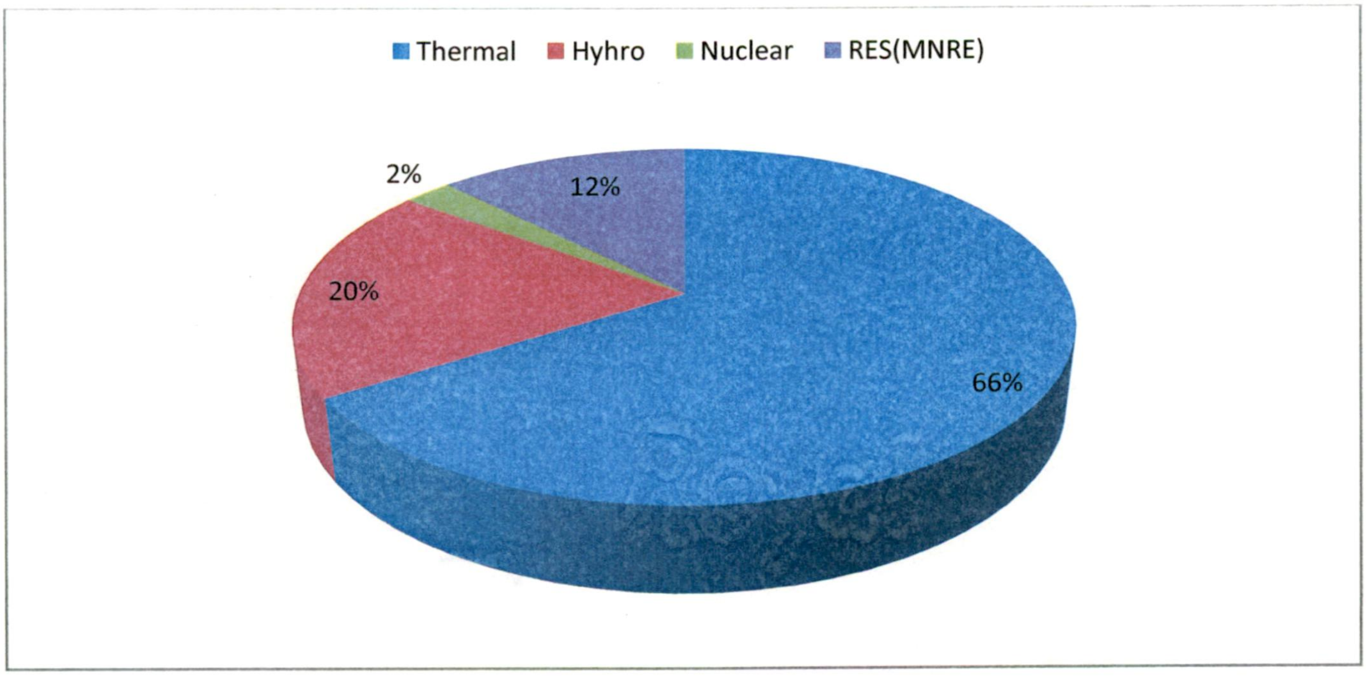


Figure 1.1: Contribution of energy sources in Indian power sector [2]

Hydropower is the most promising among all the renewal energy sources. It is a clean source of power produced when the water turns a hydraulic turbine. It provides the electricity essential for the economic and social development of society. The most important fact to be noticed is that the economic and social development of the society. The most important fact to be noticed is that the maximum potential of the hydro power is yet to be harness is many countries. Through hydropower development started with small units in the beginning, harnessing medium and small hydro because of their comparative economics. India has a large potential in the medium and large projects. The inherent drawbacks associates with large hydro are the large period, large area along with the vegetation has to be submerged which results in relocation as recently happened in the Tehri Hydro Power Plant. Political and environmental implications have compelled the planners to think for some other alternative of this large hydro resulting in the emergence of small hydro resulting in the emergence of small hydro [4].

According to world energy outlook 2009[5] in year 2030 hydropower generation in India is more the coal thermal power plant; India is endowed with rich hydropower potential; it ranks fifth in the world in terms of usable potential. This is distributed across six major river systems (49 basins), namely, the Indus, Brahmaputra, Ganga, the central Indian river systems, and the east and west flowing river systems of south India. From these rivers India has 15,000 MW Power potential.

Hydropower plant is more reliable and boosting area of business in India because most of the potential still remain to harness. Hydro power projects are generally categorized in two segments i.e. small and large hydro. In India, hydro projects up to 25 MW station capacity have been categorized as Small Hydro Power (SHP) projects [6].

1.3 SMALL HYDROPOWER PLANT (SHP)

Small hydropower plant comes under the renewable energy, for encouraging this area government provide subsidy for developing small hydropower plant. The Ministry of Non-conventional Energy Sources (MNES), Government of India, Increase in the share of clean power: Renewable (bio, wind, hydro, solar, geothermal & tidal) electricity to supplement fossil fuel based electricity generation. Small Hydro Power (SHP) Programme is one of the thrust areas of power generation from renewable in the Ministry of New and Renewable Energy. It has been recognized that small hydropower projects can play a critical role in improving the overall energy scenario of the country and in particular for remote and inaccessible areas. The Ministry of Non-conventional Energy Sources (MNES) is encouraging development of small hydro projects both in the public as well as private sector. Equal attention is being paid to grid-interactive and decentralized projects [6]. Classification of hydro power plant is shown in Table 1.1

Table 1.1 Classification of SHP according to unit size [6]

S. No.	Size	Unit Size
1	Micro	Up to 100 kW
2	Mini	101 to 2000 kW
3	Small	2001 to 5000 kW

1.4 TYPES OF SMALL HYDROPOWER PLANT SCHEMES

SHP Project in India can be broadly categorized in two categorises as; SHP project in the hills, where small stream are available. These are mostly of medium/high head utilizing small discharges. The water is diverted by the weir and intake, is conveyed to the fore bay, at the entrance to the penstock. The penstock conveys to the turbine in the power house to generate electricity. These projects may further categorise as run-of-river schemes and dam based schemes [7].

SHP project in the plains and other region of the country, which utilize water regulated for other purposes like irrigation/drinking water canals etc. are usually of the low head utilizing large discharge. These purposes are called canal based schemes.

1.4.1 Dam Toe Based Scheme

It is the most common type of power plant. This plant has storage reservoir provided by constructing a dam across river. The storage of water takes care of fluctuation of supply during flood and lean period. This plant is suitable for base and peak load.

1.4.2 Run of River Scheme

In this type of power plant, use the natural flow and elevation drop of a river for generation of electricity. Such projects divert some or most of water through a pipe or tunnel leading to electricity generation turbine then returns to the river downstream. Run of river plant may be with pondage or without pondage. The plant with pondage provided with a pandage to storage to store the water, to take care of duty variation.

1.4.3 Canal Based Scheme

Canal based small hydropower scheme is planned to generate power by utilizing the fall in canal. These schemes may be planned in the canal itself or in the bypass channel. These are low head and high discharge scheme. These schemes are associated with advantage such as low gestation period, simple layout, no submergence and rehabilitation problems and practically no environmental problems.

1.5 IMPORTANCE OF MICRO HYDROPOWER

The current concern on the global environment has imposed a new sustain on the production of electricity. The emphasis is put on development of environmental friendly energies to promote the sustainable social development. It is in these circumstances, the micro hydro power is drawing more attention. A rural population is scattered, poor and unaware of technological progress. For such area, the isolation micro-hydro power plants are usually the least-cost option. This is mainly because other options for supply of energy such as extension, diesel power, etc are more expensive and difficult to install or operate in long run [8].

Since small water stream are usually available in the most of the region, micro-hydro power plant can easily meet the energy needs of small village or cluster of settlements. The needs may be in the form of electricity or motive power to be used for agro-processing, wood working and for other small scale industries. This use of electricity in form of heat can be contribute significantly towards reducing the burning of wood and other biomass , which has many derogatory implication in terms of environmental and health. In addition to meet the needs of an area, a properly designed, install and managed micro-hydro power plant can also contribute significantly towards employment generation, improved living condition and improved education facilities.

1.5.1 Market for Micro-Hydro Power

Supplying improved energy services to people for the first time is difficult; but supplying such services profitably to very poor people who live far away from roads and the electricity grid pose a particularly difficult challenge. However micro hydro compares well with other energy supply technologies in these difficult markets. But despite this, micro hydro appears to have been relatively neglected by donors, the private sector and governments in the allocation of resources and attention. In the past rural electrification by means of grid extension was the option favoured by donors.

But the relative neglect of micro hydro has also been in part due to the fact that the circumstances under which it is financially profitable, has not been systematically established, particularly in ways that investors find credible. In addition, while it is known that the growth and sustainability of the micro-hydro sub-sector depends on certain types of infrastructure and institutional investments, it was often not clear which elements of this "enabling environment" are essential nor how it is best financed [9].

1.6 COMPONENTS OF SHP

The components of small hydro power plant can be divided into two parts

i. Civil component
ii. Electro-Mechanical components

1.6.1 Civil Components

Diversion works, power channel, desilting tank, spillway, fore bay, penstock, power houses, tail race etc comes under civil components. Figure 1.2 shows different civil components of SHP.

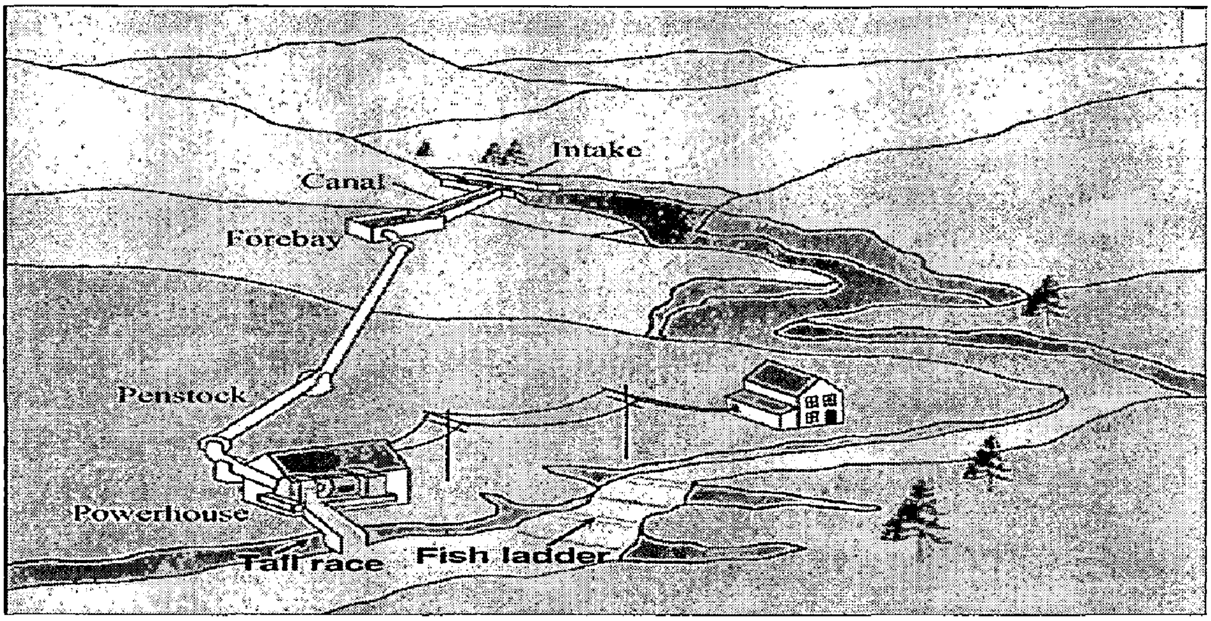


Figure 1.2 Scheme of Run of river Plant

1.6.2 Electromechanical Components

Hydro turbine, gate, valve, generator, transmission, controls, distribution etc comes under the electromechanical components.

1.7 HYDRO TURBINES

Hydraulic turbines are the machines which use the energy of water and convert it into mechanical energy. As such these may be considered as hydraulic motors or prime movers. The mechanical energy developed by a turbine is used in running an electrical generator which is directly coupled to the shaft of the turbine. The electric generator thus develops electric power, which is known as hydro-electric power.

1.8 CLASSIFICATION OF HYDRO TURBINES

According to action of the water flowing through the turbine runner the turbine may be classified as impulse turbine and reaction turbines. In impulse turbine Energy of water converts into kinetic energy or velocity head by passing it through a nozzle provided at the end of pen stock. The water coming out from nozzle is formed into a jet which impinging on a series of buckets of the runner thus causing it to revolve. Some of the impulse turbines are Pelton wheel, Turgo-impulse wheel, Banki turbine etc. In reaction turbine utilises both velocity and pressure of water on its moving blades. Francis and Kaplan are the examples of reaction turbines.

According the direction of flow of water turbine can be classified (i) tangential flow turbine like pelton turbine, (ii) radial flow turbine, (iii) axial flow turbine like Kaplan turbine, and (iv) combination of radial and axial flow like Francis turbine. In tangential flow, the water flows tangential to the rotation of the runner. In axial flow, water flows parallel to the axis of the turbine shaft. In radial flow, the water flows inwardly or outwardly along the radius of the wheel. In mixed flow, water enters the blades radially and exits axially which is parallel to the turbine shaft.

According to the head and quantity of water available turbine can be classified as (i) High head turbine, generally impulse turbine come under high head, (ii) Medium head turbine, generally Francis turbine comes in this range,, (iii) Low head turbine, generally Kaplan are the low head turbine.

1.8.1 Pelton Turbine

This is the only hydraulic turbine of the impulse type now in common use. It is named after Lester A. Pelton, the American engineer who contributes much to its development in about 1880. It is an efficient machine and it is particular suited to high head applications. The rotor consists of a circular disc with a number of blades (usually called buckets) spaced around the periphery. One or more nozzles are mounted in such a way that each nozzle directs its jet along a tangent to the circle through the centres of the buckets. A splitter or ridge splits the oncoming jet into two equal streams so that the following round the inner surface of the bucket, the two steams depart from the bucket in a direction nearly opposite to that of the incoming jet. Figure 1.3 shows Pelton turbine and its components.

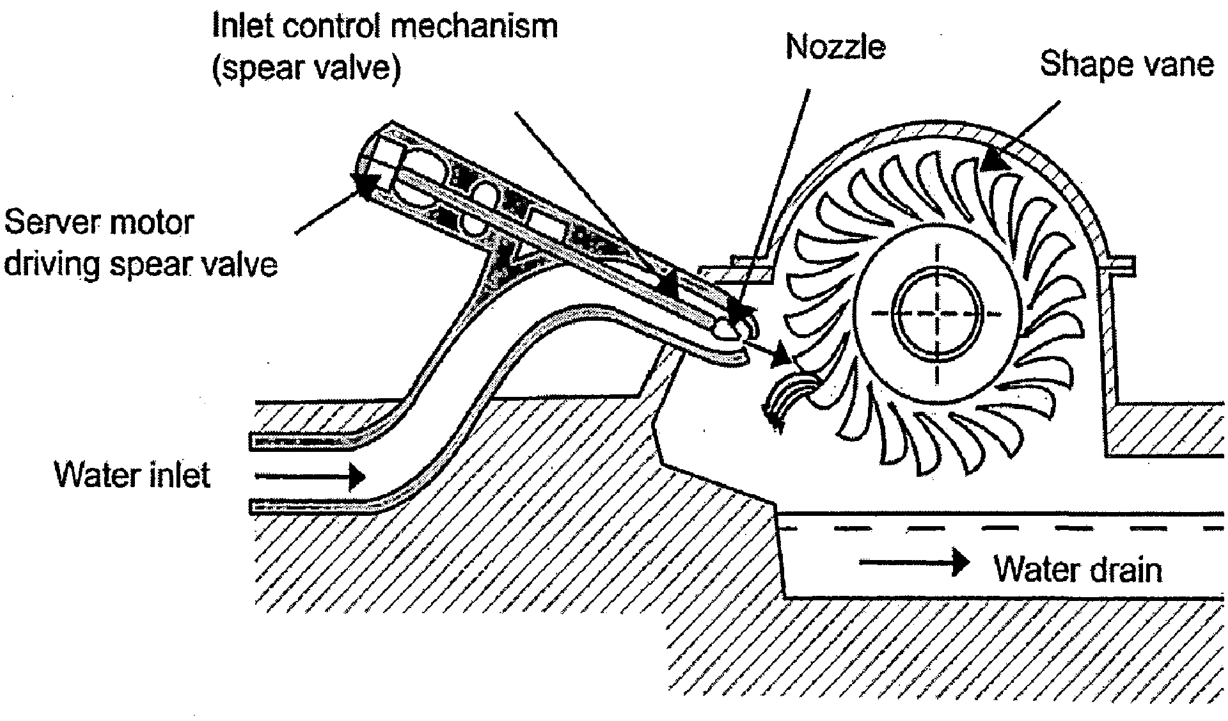


Figure 1.3 Scheme of a Pelton turbine [10]

1.8.2 Francis Turbine

The Francis turbine was developed by James Bichens Francis around 1849. Its key characteristic is the fact that water changes direction as it passes through the turbine. The flow enters the turbine in radial direction, flowing towards the axis, but it exits along the direction of that axis. It is for this reason that the Francis is sometimes called a mixed flow turbine.

The blades of a Francis turbine are carefully shaped to extract the maximum amount of energy from the water flowing through it. Water should flow smoothly through the turbine for the best efficiency. The force exerted by the water on the blades causes the turbine to spin and the rotation is converted into electricity by the generator. Francis turbine can capture 90-95% of the energy in the water. Figure 1.4 shows Francis turbine and its components.

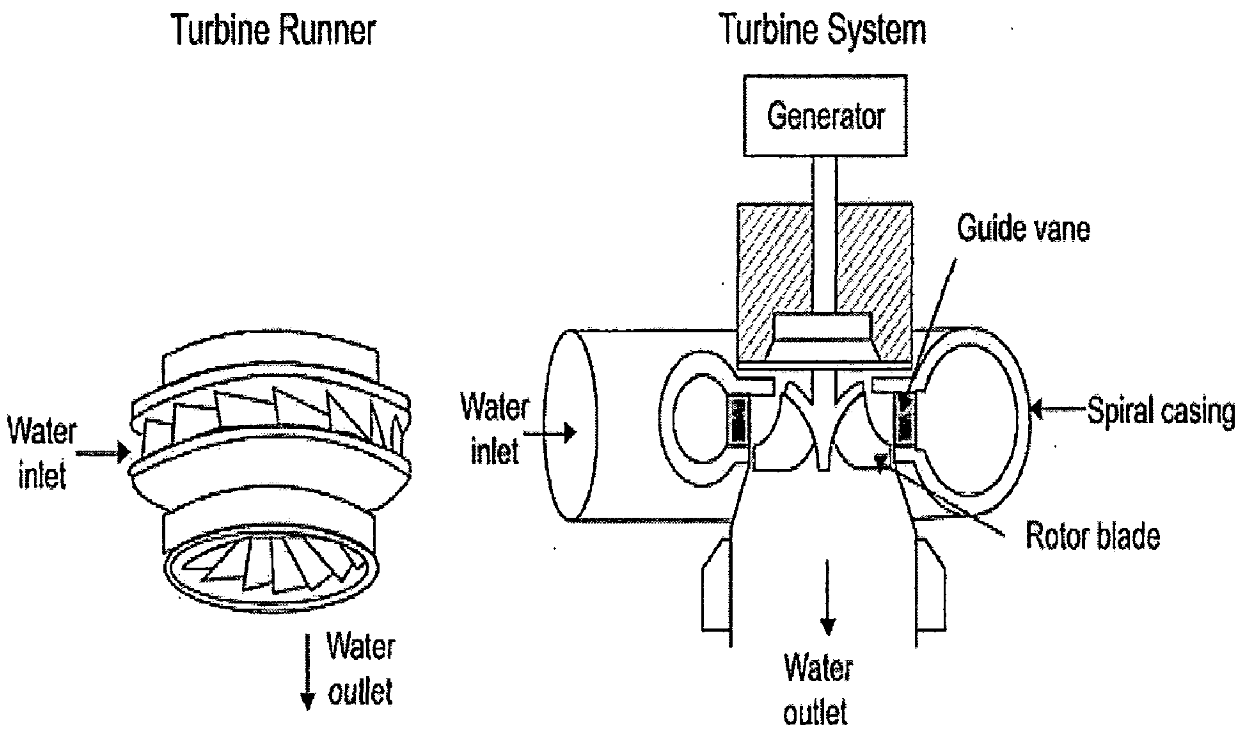


Figure 1.4 Scheme of a Francis turbine [10]

1.8.3 Kaplan Turbine

Kaplan turbine is a special propeller turbine in which the individual runner blades are pivoted to the hub, so that their inclination may be adjusted during operation responding to change in the load. The blades are adjusted automatically rotating about the pivots with the help of a governor servo-mechanism. The blades are attached to a hollow boss and hollow shaft. The servo-mechanism to rotate the blades is housed inside the hollow boss and shaft. In this turbine there are two controlling parameter wicket gate and adjustable blades, by these parameters Kaplan sustain the load fluctuation and have large part load efficiency. This turbine developed by the Austrian engineer V. Kaplan. Figure 1.5 shows the Kaplan turbine and its components.

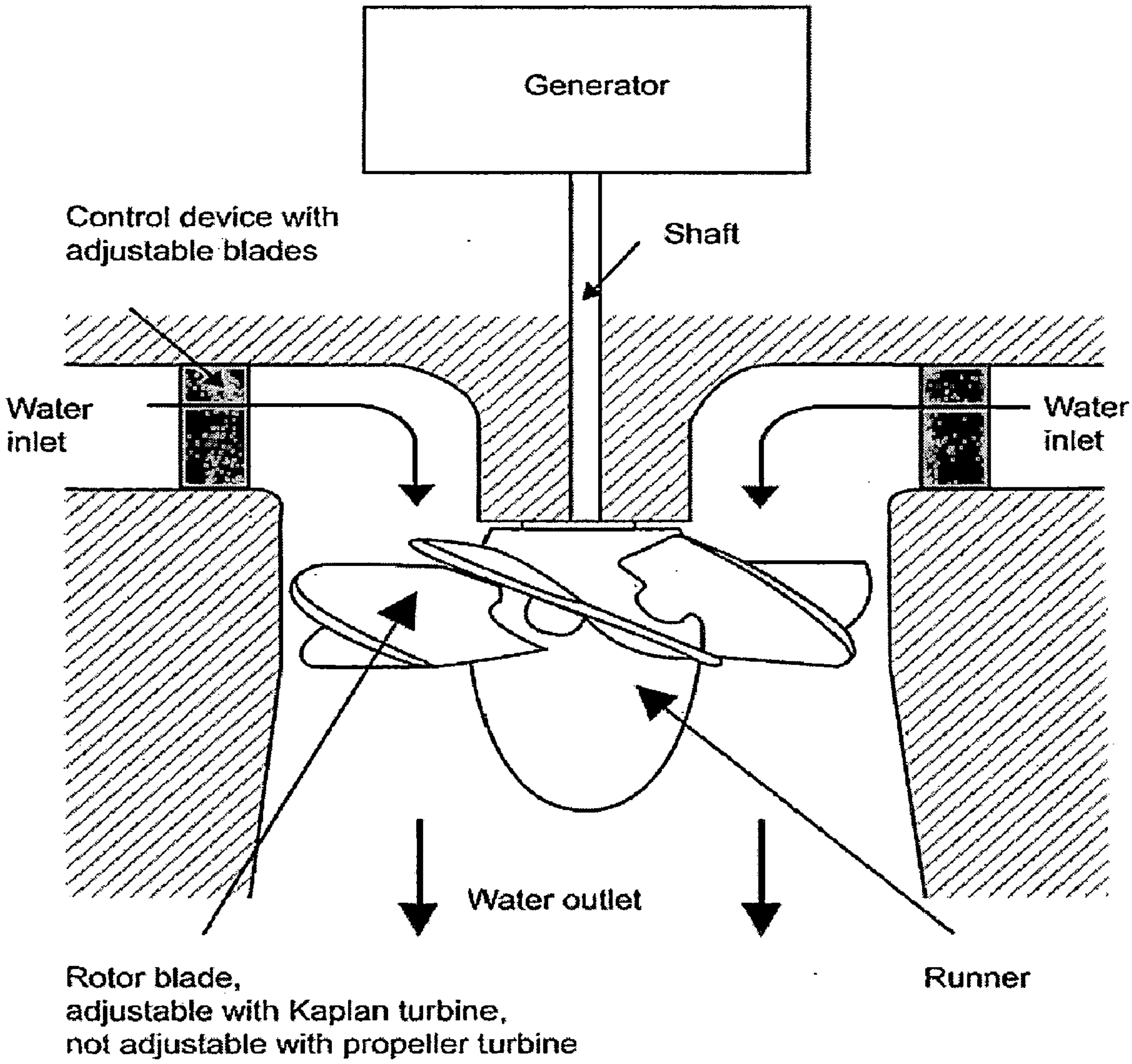


Fig 1.5 Scheme of a Kaplan turbine [10]

1.9 PERFORMANCE ANALYSIS OF HYDRO TURBINES

Turbines are often required to work under varying conditions of head, speed output and gate opening. As such, in order to predict their behaviour, it is essential to study the performance of turbine under varying condition. The variation in the condition for working may be as follows

i. The head and hence the output of turbine may change, speed being correspondingly adjusted so that no appreciable change in efficiency occurs, the gate opening remaining constant.
ii. The output may be varied by the movements of the gates or the spear, the head and speed remaining constant. These are normal operating conditions for most of the turbines.
iii. The head and speed may vary. Such variations are common particularly in low head units. It may, however, be stated that although the speed is permitted to very narrow limit, the head may vary even 50% or move.
iv. The speed may be allowed to vary by adjusting the load on the turbine, the head and gate opening remaining constant. These conditions can be developed only for laboratory turbine or those in the test plant and are otherwise uncommon [11].

1.9.1 Advantage of Computational Fluid Dynamics(CFD) Based Performance Analysis of Hydro Turbine

The conventional methods to analyze turbine performance is its model testing which becomes costly and time consuming for several design alternative in design optimization. Computational fluid dynamics (CFD) has become a cost effective tool for predicting performance of turbine. The combination of advance numerical techniques and computational power has lead to computational fluid dynamics (CFD). It is efficient and inexpensive tool to make to make internal flow predictions to good accuracy and, short of flow problem can be detected and further improvements can made on geometry of turbine components. It has made possible to significant enhancement in efficiency of hydro turbine. CFD can be used to check efficacy of alternate design of turbine for optimization before final experimental testing of selected design is resorted. However, in order to prove reliability of those tools for application to turbine, validation with known experimental result is required [12].

1.10 NEED OF COMPUTATION FLUID DYNAMICS

Computational fluid dynamics (CFD) is concerned with numerical solution of differential equations governing transport of mass, momentum, and energy in moving fluids.

CFD activity emerged and gained prominence with availability of computers in the early 1960s. Today, CFD finds extensive usage in basic and applied research, in design of engineering equipment, and in calculation of environmental and geophysical phenomena. Since the early 1970s, commercial software packages (or computer codes) became available, making CFD an important component of engineering practise in industrial, defence, and environmental organizations.

For a long time, design (as it relates to sizing, economic operation, and safety) of engineering equipment such as heat exchangers, furnaces, cooling towers, internal combustion engines, gas turbine engines, hydraulic pumps and turbines, aircraft bodies, sea-going vessels, and rockets depended on painstakingly generated empirical information. The same was the case with numerous industrial processes such as casting, welding, alloying, mixing, drying, air-conditioning, spraying, environmental discharging of pollutants, and so on. The empirical information is typically displayed in the form of correlations or tables and nomograms among the main influencing variables. Such information is extensively availed by designers and consultants from handbooks.

The main difficulty with empirical information is that it is applicable only to the limited range of scales of fluid velocity, temperature, time, or length for which it is generated. Thus, to take advantage of economies of scale, for example, when engineers were called upon to design a higher capacity power plant, boiler furnaces, condensers, and turbines of ever higher dimensions had to be designed for which new empirical information had to be generated all over again. The generation of this new information was by no means an easy task. This was because the information applicable to bigger scales had to be, after all, generated via laboratory-scale models. This required establishment of scaling laws to ensure geometric, kinematic, and dynamic similarities between models and the full-scale equipment. This activity required considerable experience as well as ingenuity, for it is not an easy matter to simultaneously maintain the three aforementioned similarities. The activity had to, therefore, be supported by flow-visualization studies and by simple (typically, one-dimensional) analytical solutions to equations governing the phenomenon under consideration. Ultimately, experience permitted judicious compromises. Being very expensive to generate, such information is often of a proprietary kind. In more recent times, of course, scaling difficulties are encountered in the opposite direction..

Clearly, designers need a design tool that is scale neutral. The tool must be scientific and must also be economical to use. An individual designer can rarely, if at all, acquire or assimilate this scale neutrality. Fortunately, the fundamental laws of mass, momentum, and energy, in fact, do embody such scale-neutral information. The key is to solve the differential equations describing these laws and then to interpret the solutions for practical design. [13]

1.10.1 Definition of Computational Fluid Dynamics

CFD is the analysis of the system involving fluid flow, heat transfer and associated phenomena such as chemical reaction by means of computer based simulation. The physical aspect of fluid flow in hydro turbine governed by two fundamental lows:

i. **Conservation of mass:**

In all the real life condition mass is always conserved on macro as well as micro levels. The generalized mass conservation equation is

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \left(\rho \vec{V}\right) = 0 \tag{1.1}$$

ii. **Conservation of momentum**

The external forces acting on a volume element in a flow field are considered to be consisting of surface forces and body forces. The surface forces results from the stresses acting on the surface of the volume element such as shear stresses, pressure forces and surface tension. And the body forces may result from the effects such as the gravitational, electric and magnetic fields acting on a body of fluid. The generalized momentum conservation equation in differential form is given below [11]:

$$\rho\left[\frac{\partial \vec{V}}{\partial t} + \vec{V} \bullet \nabla \vec{V}\right] = F_b - \nabla p + \mu \nabla^2 \vec{V} + \frac{\mu}{3} \nabla\left(\nabla \bullet \vec{V}\right) \tag{1.2}$$

1.10.2 CFD Analysis

Following are the basic concepts of CFD analysis.

a. Problem identification and pre-processing
   i. Define modelling goals.

ii. Identify the domain which will be model.
iii. Design and create the grid.

b. Solver execution

iv. Set up numerical model.
v. Compute and monitor the solution.

c. Post processing

vi. Examine the results.
vii. Consider revisions to the model.

1.10.3 Steps to Solve the Problem Using CFD Approach

The various steps required to solve the problem using CFD are described below [14].

i. Creation of Mathematical Model
ii. Choose a Discretization Method

There are many methods, but the most important ones are:

a. Finite Difference Method (FDM)
b. Finite Volume Method (FVM) and
c. Finite Element Method (FEM)

iii. Numerical Grid Generation
iv. Finite Approximation
v. Solution of Algebraic Equations
vi. Convergence Criteria

1.10.4 Numerical Simulation Methodology

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms to known quantities [15, 16].

FLUENT provides the following choices of turbulence models:

i. K-ε Models
   - Standard k-ε model
   - Renormalization-group (RNG) $k-\varepsilon$ model
   - Realizable k-ε model

ii. k-w Models
   - Standard $k-\omega$ model
   - Shear-stress transport (SST) $k-\omega$ model

iii. v^2-f Model

iv. Reynolds Stress Model (RSM)

v. Detached eddy simulation(DES) model

vi. Large eddy simulation (LES) model

1.11 LITERATURE REVIEW

Prasad et al [17] emphasis that the conventional methods for turbine performance evaluation is its model testing which becomes costly and time consuming for several design alternative in design optimization. Computational fluid dynamics (CFD) has become a cost effective tool for predicting detailed flow information in space to enable the section of the best design. In this paper, they compare the efficiency comes though experiment and CFD analysis with three different the guide vane angles obtain the efficiency come approximately same.

Drtina and Sallberger [18] investigate the use of CFD for predicting the flow in hydraulic turbine has brought further substantial improvements in their design, and result is a more complete understanding of flow processes and their influences on turbine performance. Detail of flow separation, loss source and distributions in components both design and off-design as well as detecting low pressure level associated with the risk of cavitation are now amenable to analysis with aid of CFD.

Jain et al [12] predicted the performance and the efficiency of the Francis turbine using the CFD approach and to validate the same with model testing result. The result emphases that CFD approach is faster and very large amount of result can be produced at virtually no add cost. CFD approach may be helpful in improvement of existing efficiency measuring techniques and evaluation of the performance of hydro turbines to enhance the viability of hydropower development.

Wu et al [19] studied the runner and guide vanes are optimized to the greatest extent, and the stay vanes are locally modified with a possible minimum cost under the geometrical constraints of the existing machine. The performance of the new design is verified by model tests, and exceeds required improvements. A computational fluid dynamics-based design system with the integration of three blade design approaches, automatic mesh generator and CFD codes enables a quick and efficient design optimization of turbine components. The highly successful combination of the CFD-based design optimization with model testing has finally resulted in a new model which can provide about 23% increase in power and over 3% upgrade in peak efficiency. In addition the model provides for a thoroughly improved cavitation characteristic with extremely smooth performance over a much wider range of operations compared to the existing design.

Thakker et al [20] compares Computation Fluid Dynamics (CFD) analysis with Experimental analysis of 0.6 m Impulse turbine with fixed guide vanes used for wave energy power conversion. The 2D CFD result for 0.6m Impulse turbine with fixed guide vanes were extrapolated to predict 3D performance and are giving similar trends as what we get from experimental analysis, which is good sign.

Lipej [21] investigates optimization methods for the design of axial flow turbine by using the CFD. This paper describes the complete optimization procedure for the design of axial runners. The procedure consists of a design program, numerical flow analysis and a multiobjective genetic algorithm. The benefits of this approach in terms of reduced design time are very important. Without the optimization method such a design procedure can take a few weeks, but using the proposed method the time can be reduced to a few days. The results of the optimization procedure help show how each design parameter influences the energetic and cavitation characteristics of axial runners. Using the genetic algorithm in combination with three dimensional numerical flow analysis was found to be useful for the design of hydraulic machines, because the initial geometry from the design program has a relatively high level of efficiency.

Thum [22] studied The numerical design optimization for complex hydraulic machinery bladings requires a high number of design parameters and the use of a precise CFD solver yielding high computational costs. To reduce the CPU time needed, a multilevel CFD method has been developed. To guarantee a sufficiently accurate result, the code is calibrated by a Navier-Stokes recalculation of the initial design and can be recalibrated after a number of optimization steps by another Navier-Stokes computation. After having got a convergent solution, the optimization process is repeated on the second level using a full 3D Euler code yielding a more accurate flow prediction. The optimization was performed on the lowest CFD

level, with the EQ3D code, which yields a blade geometry withmuch better flow behaviour. Only on the final optimization level the most powerful but very time-consuming NS3D code is applied for the fine-tuning of the blade shape with respect to minimum losses. Thus, the multilevel CFD technique has proven to be an efficient engineering tool for the optimum design of hydraulic machinery bladings.

Viscanti et al [23] studied Fluid dynamics optimization of new blade profiles and meridional channel is achieved using CFD. A trial and error approach is used to modify geometric shapes in order to increase blade efficiency and to reduce cavitation phenomena in the new runner. The most regular trend and the higher values for relative efficiency remark how CFD is a powerful tool to improve fluid dynamics in Francis runners. Optimization required a lot of CFD runs but it helps to understand the very complicated 3D flow field inside the Francis turbine and to detect improvement areas in future runner designs.

Liu et al [24] evaluated hydraulic performances of a prototype Kaplan turbine having a runner diameter of 8 m, which are predicted by using the steady turbulent flow analysis with standard k-s turbulence model for the entire flow passage. They find Prediction of the performances is robustly conducted by the computational method, where the working head is used as the given condition.

Kim et al [25] investigates influences of pressure, tangential and axial velocity distributions on turbine performance by using commercial CFD codes. In order to acquire basic design data of tubular type hydro turbine, output power, head, and efficiency characteristics due to the guide vane opening angle are examined in detail. After CFD analysis they found the influence of passageway on the velocity distributions is negligible and the tangential velocity component has its maximum value at runner vane inlet region, on the contrary, axial velocity component has its maximum value at runner vane exit.

Choi et al [26] investigate the performance of a newly developed direct drive hydro turbine (DDT), which will be built in a caisson for a wave power plant. Experiment and CFD analysis are conducted to clarify the turbine performance and internal flow characteristics. The results show that the DDT obtains fairly good turbine efficiency in cases with and without wave conditions. They predict direct drive hydro turbine system shows fairly good turbine efficiencies of 51.7% at the best point by unidirectional steady flow without input wave condition and 48.6% by reciprocating flow with an input wave condition in a 2-D wave channel.

Prasad et al [27] studied 3D simulation for flow in mixed flow (Francis) turbine passage using CFD software ANSYS CFX 10 software for study of flow pattern within turbine space and computation of various losses and efficiency at different operating regimes. They found computed efficiency values at maximum efficiency regimes for three guide vane openings are close to that of experimental values and hence validate CFD analysis in turbine flow passage. The efficiency characteristics from CFD simulations are very similar to that obtained in experimental testing of Francis turbine models.

Carija and Mrsa [28] Simulated the steady fluid flow with moving reference frame in the entire Francis turbine consisting of the spiral casing, draft tube, 10 stay vanes, 20 guide vanes and 15 runner blades. Calculation was performed using commercial fluid flow solver FLUENT on a Linux cluster.

Nicolle et al [29] analysed the CFD based performance of existing turbines for many years. They investigated the impact of such differences by presenting the CFD analysis of various blades measured on the same runner. This paper shows that obtaining a representative geometry is not always a straightforward process because of the small geometrical difference between the blades. Besides the blade geometry itself, one must also take into account the influence of the blade tip clearance for the propeller turbine.

Kyriacou et al [30] proposed CFD Based design-optimization method for hydraulic machinery. They extended the optimization method of the previous paper by making it capable to accommodate and exploit pieces of useful information archived during previous relevant successful designs. An optimization platform that combines evolutionary algorithms with the ability to exploit information contained in previous successful designs, in order to reduce the computational burden, is used to design a Francis runner. New designs are defined as combinations of the previous successful designs, after clustering similar design variables into groups and associating one weight per group of variables and archived design.

Choi et al [31] studied CFD analysis on the performance and internal flow of the turbine is conducted by an unsteady state calculation using a two-phase flow model in order to embody the air layer effect on the turbine performance effectively. The result shows that air layer effect on the performance of the turbine is considerable.

Cherny et al [32] investigates the method of numerically solving the problems of two hydro turbine fluid flows. They developed Euler model for solving both problems. The first problem considered is a simulation of processing vortex rope downstream the turbine runner. The second problem is an automatic design of runner blade shape.

Khare et al [33] studied 3D turbulent real flow analyses in hydraulic Francis turbine have been carried out at three guide vane opening and different rotation speed using Ansys CFX computational fluid dynamics (CFD) software. The average values of flow parameters like velocities and flow angles at the inlet and outlet of runner, guide vane and stay vane of turbine are computed to derive flow characteristics. The close comparison between the computed and experimental efficiencies values and nearly same best operating point in both cases validates the computational results.

Ying et al [34] studied 3D numerical simulation of unsteady turbulent flow in the entire flow passage of a Francis turbine with computational fluid dynamics (CFD) technology. The 3D unsteady turbulent flow in an entire Francis turbine model is calculated successfully using the CFX-TASC flow software and RNG k-ε turbulence model. Transient flow fields are simulated in the spiral case, the distributor, the runner, and the draft tube for the.optimum operating condition.

Lain et al [35] studied the numerical simulations as a tool for design and optimization of hydraulic turbines. The first part has presented the proper methodology to follow in performing the numerical simulations, which depends on the type of phenomena sought, steady or unsteady. Particularly, it has been performed the numerical simulation of the flow inside a Francis turbine currently operating in Colombia. The obtained results are in good agreement with the in site experiments, especially for the characteristic curve.

Choi et al [36] investigated the effect of turbine's structural configuration on the performance and internal flow characteristics of the cross flow turbine model using CFD analysis. The results show that nozzle shape, runner blade angle and runner blade number are closely related to the performance and internal flow of the turbine. Air layer in the turbine runner plays very important role of improving the turbine performance.

Xiao et al [37] studied the three-dimensional viscous flow is simulated through the whole flow passage of the model Francis turbine. The whole flow passage includes spiral case, distributor, runner and draft tube. The simulation is based on Navier-Stokes equations, the standard $k - \varepsilon$ turbulence model and the SIMPLEC algorithm, which is applied for the solution of the discrete governing equations. Viscous flow analysis is a powerful analytical design tool. It allows designers to evaluate the hydraulic performance of alternative designs before proceeding with testing or to optimize the hydraulic design.

Haurissa et al [38] discussed the effect of nozzle on the second level of turbine blade cross flow turbine. By installing nozzle in the second level of the turbine blades would cause the turbine performance cross flow became more compressed at the blade into the second level. This study concluded in the best of flow turbine blades latitude, using nozzle on the second level entry was better than without using the blade entry nozzle on the second level.

Choi et al [39] studied to optimize the structure of cross-flow turbine and to improve the turbine performance. Optimization of the turbine structure has been made by the analysis of the turbine performance with the variation of the blade angle using a commercial CFD code.. Recirculating flow in the runner passage causes considerable hydraulic loss by which efficiency of the turbine decreases very much. There is optimum configuration of blade angle and relatively small angle of blade inlet at the Stage 1 has higher efficiency. The air layer improves efficiency of the turbine by preventing collision loss at the runner shaft, and by eliminating the recirculating flow in the passage.

Vu et al [40] studied flow simulations of a low head Propeller turbine at various design and off-design conditions. They presented the results on the efficiency hill chart prediction of the Propeller turbine and discuss the consequences of using non-homologous blade geometries for the CFD simulation. The flow characteristics of the entire turbine will be also investigated and compared with experimental data at different measurement planes.

Keck et al [41] studied the application of computational fluid dynamics (CFD) in the design of water turbines and pumps started about 30 years ago. They reviewed This paper reviews the main steps and breakthroughs in the methods that were made during this period, through the eyes of one particular water turbine company which spear-headed some of the first developments for practical applications. CFD has become a powerful tool which requires validation versus smartly designed and executed experiments as well as profound knowledge in fluid mechanics in order to identify numerical problems on one side and to most effectively interpret proper CFD predictions on the other hand.

Daneshkah et al [42] investigated 3D inverse design method for Francis turbine design. Effect of inverse design parameter (stacking condition and blade loading) on the flow field inside the runner was studied in a parametric way. The flow field and suction performance obtained by CFD with single-phase and two-phase flow models were compared between different designs. They shown that parameterization of blade geometry using the inverse design flow related parameters can provide the designer with control over the pressure field inside the runner, which can be used effectively to suppress cavitation phenomena without deteriorating the hydraulic efficiency. The design guidelines they presented can be applied easily to the optimization of other Francis turbine runners. The 3D inverse method is an extremely powerful and practical design tool for designing hydraulic turbine runners.

Lain et al [43] proposed improved k–ω turbulence model, which brings the nonlinear term of the mean fluid flow transition to the ω equation in the original k–ω model of Wilcox. Based on the improved k–ω turbulence model, three dimensional turbulent flow computations is carried out through the whole flow passage including the spiral casing, stay vanes, guide vanes, runner and draft tube of a model Francis turbine. The 3D steady turbulent flow simulation, developed with the improved k-ω model, is used to predict its energy performances of one type of rotating machinery-Francis turbine, including, the turbine hydraulic torque (output) and its efficiency. The predicted results are more reasonable and closer to test data than the results from the RNG k–ε model, the original k–ω model and even the SST k–ω model.

Lipej [44] described how the multiobjective genetic algorithm aids the human decision of the best design solution, based on objective functions obtained by numerical flow analysis. Theoretical and empirical knowledge of the design procedure enables to start the optimization procedure with good initial geometry. A very efficient genetic algorithm in connection with numerical flow analysis was found to be very useful for designing axial runners. The next steps in terms of design procedure will be taking into account more parameters or nonlinear distribution for the variables along the blades for runner design and using previous results to changes distribution of design parameters.

Shuhong et al [45] studied unsteady turbulent flow computation based on the modified RNG k- ε turbulence model through the whole flow passage to simulate the pressure fluctuation in a model turbine. From the comparison of them with the model turbine results, it is seen that their qualitative trend of pressure fluctuations are similar, but an appreciable difference is observed between the amplitudes of pressure fluctuation of the prototype turbine and that of the model turbine. Kaplan turbine was treated in this study, and the following concluding remarks were obtained. (1) The present numerical simulation based on the modified RNG k-ε turbulence model through the whole flow passage reasonably reproduced the pressure fluctuations in a model Kaplan turbine, which were validated by the corresponding model test. (2) Dominant frequencies observed in the pressure fluctuation are rotating frequency and its harmonics, i.e., two times, four times and six times for the present Kaplan turbine with six runner blades. The low frequency also appeared in the pressure fluctuation. (3) Regarding the amplitude of pressure fluctuation, appreciable differences were observed between the model and the prototype. They may be attributed to the effect of Reynolds number, for the number of the prototype is fifty times greater than that of the model in the present case. Thus, we had better carry out the numerical simulation for the prototype to understand the unsteady flow inside.

Nilsson et al [46] studied that a parallel multiblock finite volume CFD (Computational Fluid Dynamics) code CALC-PMB (Parallel MultiBlock) for computations of turbulent flow in complex domains has been developed and used for the computations of the flow through a Kaplan water turbine. The computations are including both the guide vanes and the runner, where the runner computations get the inlet boundary conditions from the circumferentially averaged properties of the guide vane computations Four different operating conditions have been computed and the results from the computations are compared. The computational results are in accordance with observations done by the turbine manufacturer. This work is focused on tip clearance flow, which reduces the efficiency of the turbine by about 0.5 %.

Gagnon et al [47] analyzes unsteady rotor-stator interactions in a propeller turbine. The flow behaviour is studied with numerical simulations using a 3-D Navier-Stokes solver and a $k - e$ turbulence model (code ANSYS CFX). They used the ANSYS CFX code for three operating conditions. Analysis pointed out two types of interactions, namely: potential and wake interactions. Blade torque and guide vanes forces spectrums indicate phenomena of low intensities since the turbine operates at low head and since the gap between stationary and rotating parts is relatively large. Main phenomena are of potential natures and also induced by wakes behind guide vanes. Interaction intensities are increased as we move away from optimal operation point. An experimental investigation is currently being prepared for validation and will provide insight for improving CFD simulations of axial hydraulic machinery in future studies.

Grafenberger et al [48] studied that the same optimization procedure is adapted to the design of optimal blades of Francis runners, in a multi-objective, multi-operating point way. The optimization targets, such as cavitation safety and desired load distributions along the blade from the leading to the trailing edge, are defined and brought in the form of objective functions. Emphasis is laid on the problem formulation of Francis and Kaplan runner blades, followed by comments on possible extension to Pelton turbines. A two-level optimization scheme is then set up and solved by employing two-level hierarchical metamodel-assisted evolutionary algorithms (HMAEA). On the low level, a low CPU cost exploration of the search space is carried out on a less accurate CFD tool, i.e. an Euler equations' solver running on coarse grids. The high level utilizes a more accurate and CPU demanding simulation process, by focusing on the best performing areas of the design space identified and communicated by the low level.

1.12 OBJECTIVE OF PRESENT STUDY

Investigation of flow condition of hydro turbines is very important to know the efficiency of the turbine. Model testing is generally adopted to analyze the flow conditions. However this is not economically viable especially for micro hydro turbines. Further the part load efficiency of low head turbines (Propeller) is considered to be poor. It is therefore, there is need to develop a Kaplan runner for micro hydro turbine, in order to improve the part load efficiency.

Keeping this in view, the design of a Kaplan runner has been carried out for micro hydro turbine, based on CFD analysis with following objectives:-

- To identify the system and operating parameter for a Kaplan runner of micro turbine.
- To design the Kaplan runner for micro turbine based on the selected parameters.
- To develop a CFD model.
- To carry out the CFD analysis of Kaplan runner.
- To optimize the operating parameters of Kaplan turbine for micro hydro range.

1.13 DISSERTATION OUTLINE

The organization of the report is as follows:

Chapter 1 the current chapter briefly describes the importance of CFD analysis of hydro turbine for micro hydro power and also presented the literature review related to the CFD analysis of hydro turbines. The chapter also describes objective of work based on the literature review and finally the organisation of the dissertation is outlined.

Chapter 2 describes about the Kaplan turbine and its components. The purpose of the components describes in this chapter and also describes working proportions of Kaplan turbine.

Chapter 3 describes the sizing of the all components of Kaplan turbine for rated head and rated discharge.

Chapter 4 presents CFD analysis of Kaplan turbines, which consist 3D modelling of Kaplan turbine and numerical simulation on Kaplan turbine.

Chapter 5 presents the computation of efficiency of Kaplan turbine at design condition and part load conditions. Plot the part load efficiency curve for Kaplan turbine.

Chapter 6 concludes the work contained in the main body of the dissertation

CHAPTER 2
KAPLAN TURBINE

2.1 KAPLAN TURBINE

A Kaplan turbine is type of propeller turbine which was developed by the Austrian Engineer V. Kaplan. It is the axial flow turbine, which is sustain for relatively low heads, and hence required a large quantity of water to develop large amount of power. It is also a reaction type turbine and hence it operates in an entirely closed conduit from the head race to tail race. The main components of a Kaplan turbine are scroll casing, stay ring, arrangement of wicket gates and draft tube. Between the wicket gate and the runner the water in a Kaplan turbine turn through a right-angle into the axial direction and then passes through the runner. The runner of a Kaplan has three, four or six (or eight in some exceptional cases) blades and it closely resembles a ship's propeller. The blades are attached to hub or boss are so shaped that water flows axially through the runner. Ordinarily the runner of a propeller turbine is fixed, But the Kaplan turbine runner blades can be turned about their own axis, so that their angle of inclination may be adjust while the turbine is in motion. This adjustment of runner blade is usually carried out automatically by means of a servo motor operating inside the hollow coupling of turbine and governor shaft. When both wicket gate angle and runner blade angle may thus be varied, a high efficiency can be maintained over a wide range of operating condition. In other words even at part load, when a lower discharge is flowing through the runner, high efficiency can be attained in the case of a Kaplan turbine. It will be observed that although the corresponding change in the flow through runner does affects the shape of the shape of the velocity triangles, yet as the blade angles are simultaneously adjusted, the water under all the working conditions flows through the runner blades without shock. As such the eddy losses which are inevitable in Francis and propeller turbines are almost completely eliminated [11].

2.2 COMPONENTS OF KAPLAN TURBINE

The components of Kaplan turbine described below;

2.2.1 Wicket Gate

Wicket gate in Kaplan is same as the wicket gates in Francis turbine. The only difference is that the wicket gate in Francis usually used for guiding the water but in the case of Kaplan wicket gate used for controlling the work. The wicket gates or guide vane are fixed between two rings in the form of a wheel known as guide wheel. The wicket gate have a hydrofoil section which allow water to pass over them without forming eddies and with minimum friction losses. Each wicket gate can be rotated about its pivot centre which is connected to the regulating ring by means of a link and a lever and hence the required quantity of water can be supplied to the runner by opening or closing the wicket gates. The wicket gates are generally made of cast steel.

2.2.2 Stay Vanes

In case of big units, the stay vanes are provided at the inside circumference of the casing. The stay vanes provide strength to the casing and also direct the water towards wicket gates.

2.2.3 Runner

The flow in the runner of a Kaplan turbine is purely axial. There is axial entry and exit of water. The width of the runner depends on the specific speed. The high specific speed runner is wider than the one which has a low specific speed because the high specific speed runner has to work with a large amount of water. The runners are made of cast iron for small output, cast steel for large output and stainless steel or a non-ferrous metal like bronze, when the water is chemically impure and there is a danger of corrosion.

2.2.4 Shaft and Bearing

The runner is keyed to the shaft which may be vertical or horizontal. The shaft is generally made of steel and is forged. It is provided with a collar for transmitting the axial thrust to the bearing. The turbine is generally provided with one bearing. In vertical shaft turbine, bearing carries full runner load and acts as thrust cum supporting bearing. Right selection of bearing is therefore extremely important.

2.2.5 Draft Tube

The water after passing through the runner flows to the tail race through draft tube. A draft tube is a pipe or passage of gradually increasing cross section area which connects the runner exit to tail race. It may be made of cast or plate steel or concrete. It must be alright and under all conditions of operation its lower and must be submerged below the level of water in the tail race. The following are the purposes of the draft tube:

(i) It permits a negative or suction head to be established at the runner exit, thus making it possible to install the turbine above the tail race level without loss of head.

(ii) It converts major part of kinetic energy coming out of runner into pressure energy i.e. it acts as a recuperator. In mixed flow reaction turbines, kinetic energy from runner is up to 15% whereas in low head and high speed axial flow turbines, kinetic energy leaving the runner may go up to 50% of total input energy. The recovery of kinetic energy is achieved by increasing the cross-sectional area of the draft tube in the flow direction [48].

The following types of draft tubes are used in the Small Hydropower Stations.

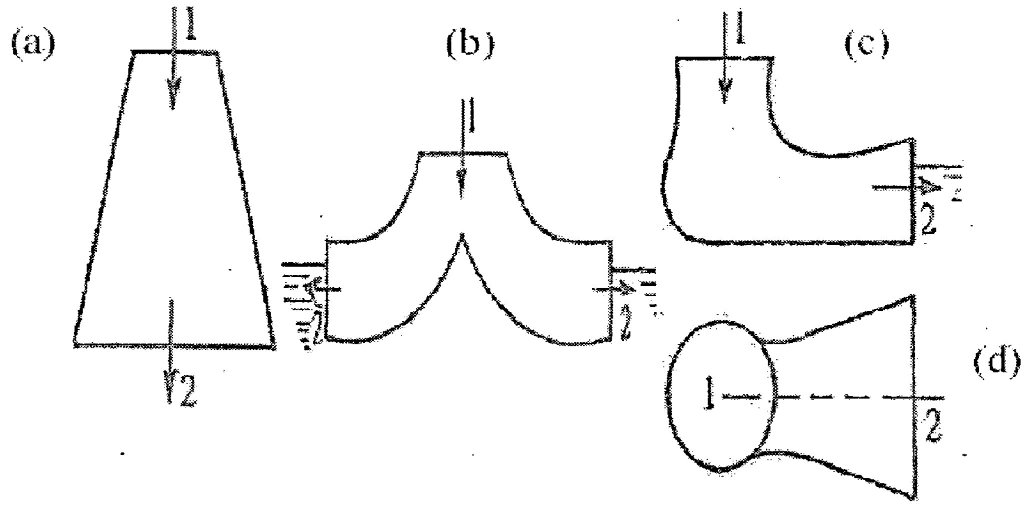
(a) Straight Divergent Tube

(b) Moody Spreading Tube

(c) Simple Elbow Tube

(d) Elbow Type with a Circular Inlet and a Rectangular outlet section

Figure 2.1 shows different types of draft tubes which are employed in the field to suite particular conditions of installation. The types (a) and (b) are most efficient but the types (c) and (d) have an advantage that they require less excavation for their installation. It has been observed that for the straight divergent type draft tube the central cone angle should not more than 8°. This is because if this angle is more than 8°the water flowing through draft tube will not remain in contact with its inner surface, with the result that eddies that eddies are formed and efficiency of draft tube is reduce.



Fiure.2.1: Different types of draft tube (a) Straight divergent tube (b) Moody spreading tube (c) Simple elbow type (d) Elbow type with circular inlet and rectangular outlet section [50]

2.2.6 Casing

There is different type of casing used in axial flow turbines which are listed below;

(i) Spiral casing: To avoid loss of efficiency, the flow of water from the penstock to the runner should be such that it will not form eddies. In order to distribute the water around the guide ring evenly, the scroll casing is designed with cross sectional area reducing uniformly around the circumference, maximum at the entrance and nearly zero at the tip. This gives a spiral shape and hence the casing is named as spiral casing.

(ii) Tubular casing: The casing of turbine is in the form of tube called tubular casing and turbine called tubular turbine. A tubular turbine is an axial turbine with either adjustable or non adjustable runner vans and hence it is similar to Kaplan or propeller turbines. However, in a tubular turbine the scroll casing is not provided but the runner is placed in tube extended from the head water to tail water as shown in fig. And hence it is called tubular turbine. It is a low head turbine which can be used under head. The tubular turbines may have either vertical or inclined or horizontal disposition of shaft.

The tubular turbine is of two types based on the shape of casing which is listed below:

i. S-type tubular turbine

The shape of the casing is like S so it is called S type tubular turbine, which is shown below in Figure 2.2;

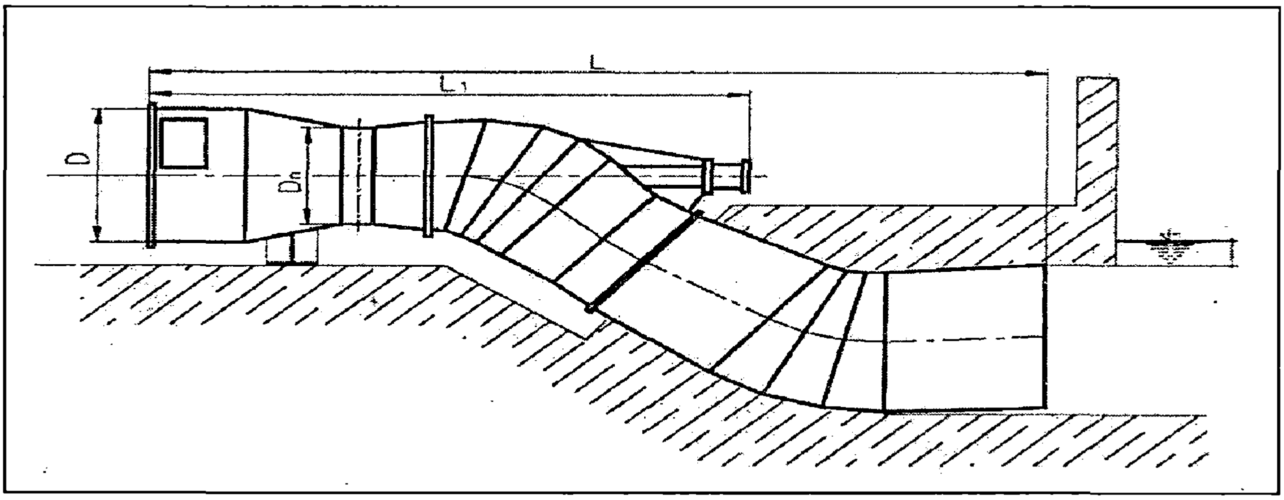


Figure 2.2: S type tubular turbine [51]

ii. L-type tubular turbine

The shape of the casing is like L shape so it is called L type turbine, which is shown below in Figure 2.3;

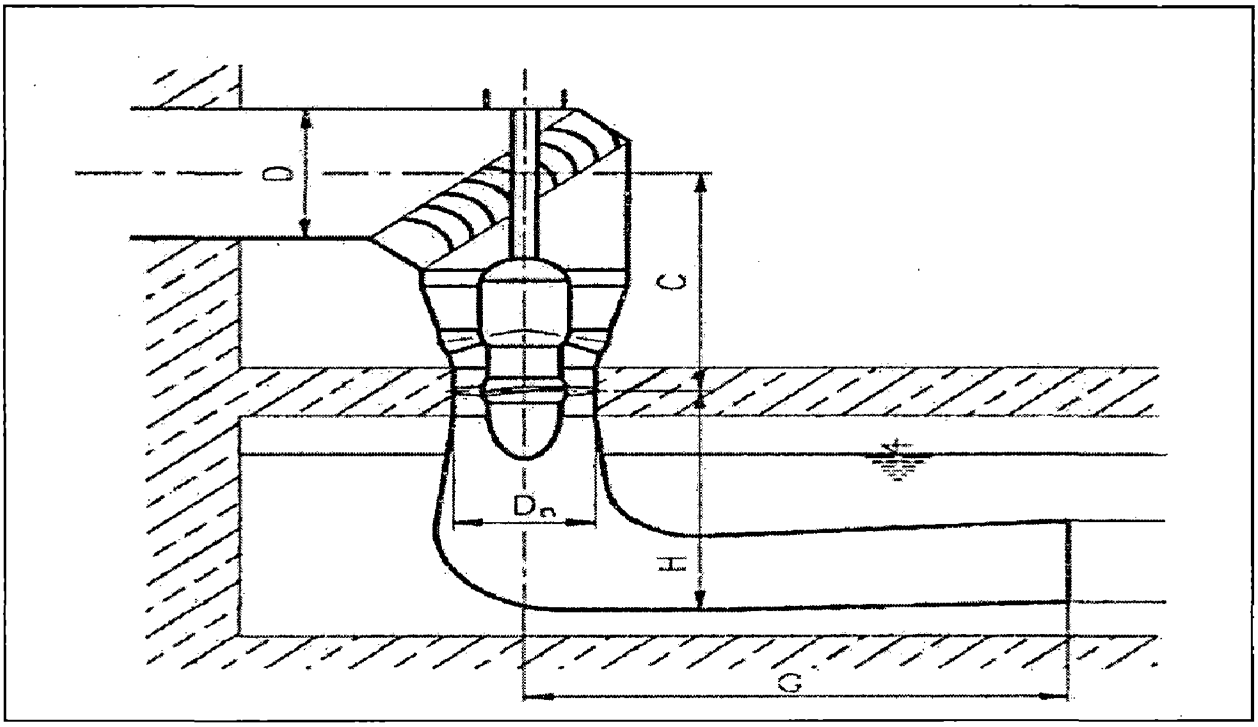


Figure 2.3: L type tubular turbine [51]

For the present study, S Type tubular casing with Kaplan runner has been used.

2.3 WORKING PROPORTIONS OF KAPLAN TURBINE

In general the main dimensions of Kaplan turbine runner are same established by a procedure similar to that for Francis turbine runner. However, the following are main deviations;

i. Choose an appropriate value of the ratio n= (d/D), where d is the diameter of hub and D is diameter of runner. The value of n usually varies from 0.35 to 0.70.

ii. The discharge Q flowing through the runner is given by

$$Q = \frac{\pi}{4}(D^2 - d^2)V_f = \frac{\pi}{4}(D^2 - d^2)\varphi\sqrt{2gh} \qquad \text{...... (2.1)}$$

The value of flow ratio φ for a Kaplan turbine is around 0.70.

iii. The runner blade of Kaplan turbine are wrapped or twisted, the blade angle being greater at outer tip than at the hub. This is because the peripheral velocity of the blade being directly proportional to radius, it will vary from section to section along the blade, and hence in order to have shock free entry and exit of water the blades with angles varying from section to section[11].

CHAPTER 3

DESIGN OF KAPLAN TURBINE

3.1 GENERAL

In the present study, the Kaplan turbine with tubular casing for micro hydro range has been designed. Kaplan turbine having rated head and rated discharge of 1.5 m and 7.03 m^3/s respectively is considered.

3.2 DESIGN OF KAPLAN TURBINE

The main characteristics are the data on which the design of the runner is based. To calculate the sizing of turbine or the dimensions of turbine some adaptation mechanism are needed. In Figure 3.1, a sketch of a Kaplan turbine is given. On this sketch, those heads and points which play a significant role in this dissertation are marked.

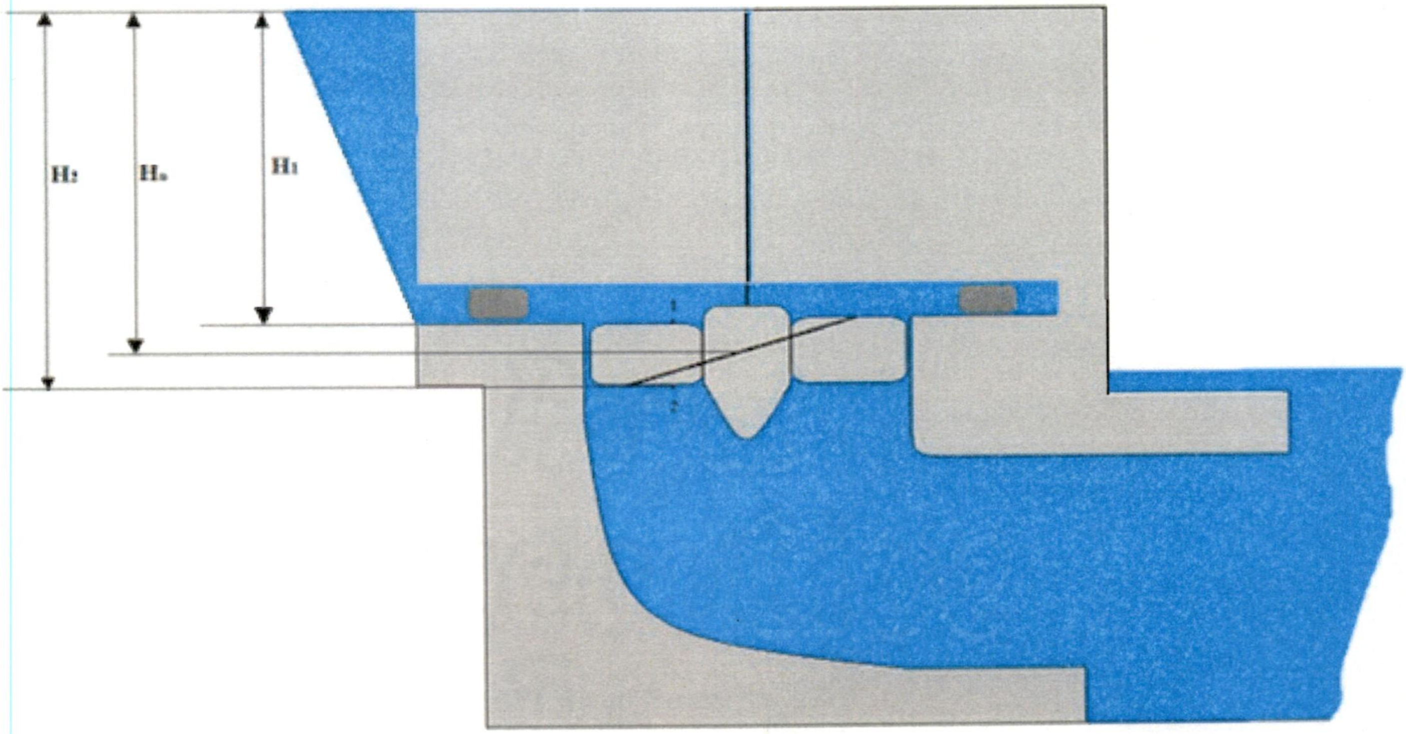


Figure 3.1: Sketch of Kaplan turbine

3.2.1 Estimation of Power

The power or potential of the site has been computed by using following equation:

$$P = Q*H_n*g*\rho$$

Where:

Q = Rated Discharge (m^3/s)

H_n = Rated head (m)

ρ = Water density (kg/m^3)

g = Acceleration of gravity

The values of the different parameters are selected as given below;

$Q = 7.03\ m^3/s$

$H_n = 1.5\ m$

$\rho = 998\ kg/m^3$

$g = 9.81\ m/s^2$

$P = 7.03*1.5*9.81*998 = 103446\ W$

For the present study, the rated potential of the plan has been computed as 100 kW as the convenience for the design calculation of Kaplan turbine.

3.2.2 Specific Speed and Rotational Speed

The different types of water turbines can be classified by their specific speed. Different definitions of the specific speed exist which can be found in the technical literature. As stated in the "Guide on how to develop a small hydropower plant", the specific speed is a dimensionless parameter and characterizes the hydraulic properties of a turbine in terms of speed and discharge capacity; it is based on similitude rules [50].

A significant point for specific speed is that it is independent of dimension or size, both of actual turbine and of the specific turbine. It therefore means that all turbine of the same geometry shape, working under the same value of k_u and φ, and thus having the same efficiency, will have the same specific speed, no matter what their sizes be and what power they develop under what head. As such it may be started that N_s represented the specific speed of the actual turbine as well as of the specific turbine [11]. The equation of the specific speed has been given below;

$$Ns = \frac{N\sqrt{(P*1.358)}}{Hn^{\frac{5}{4}}} \tag{3.1}$$

Where:

N_s is the Specific speed of the turbine.

N is Rotational speed of the turbine.

Take the value of P as 100 kW and rated head H_n as 1.5m, the specific speed is calculated in the terms of rotational speed are given below;

$$Ns = \frac{N\sqrt{(100*1.358)}}{1.5^{\frac{5}{4}}} \tag{3.2}$$

$$Ns = 7.02N \tag{3.3}$$

The Equation (3.3) is the relationship between specific speed and rotational speed. Rotational has been fixed by varying the rotation speed in the Equation (3.3) and according to the specific speed fix the value of rotational speed. Table 3.1 shows the variation of specific speed with rotational speed.

Table 3.1: Variation of specific speed with rotational speed

N(rpm)	Ns
100	702
125	877.5
140	982.8

The specific speed of the Kaplan turbine varies between 340 and 1000. According to the variation in specific speed, the values of rotational speed vary from 100 rpm to 140 rpm. For present study, rotational speed has been taken as 125 rpm for which the specific speed carried out to be 877.5.

3.2.3 Runner Diameter and Number of Blades of Runner

Diameter of runner has been calculated by the equation which is given below [51];

$$D = \frac{84.6 \times 0.0233 \times N_S^{2/3} \times H_n^{1/2}}{N} \tag{3.4}$$

D = 1.77 m

Where, D is the diameter of runner.

According to variation of head and specific speed, the *no. of blades* and the *ratio between diameters of hub to tip* is decided. The Handbook of Turbo machinery by Earl Logan and Ramendra Roy describe the variation of head and specific speed [52].

a. According to the parameters for designing of Kaplan turbine, the number of blades has been selected as 4 [52].
b. Hub to Tip ratio i.e. d/D has been taken as 0.35, where d is the diameter of hub and D is the diameter of runner. By this relation diameter of hub has been calculated as 0.62 m.

3.2.4 Blade Design

For the blade design number of factor play significant role, the leading edge is thicker then the trailing edge for streamlined flow. Furthermore, the blade should to be as thin as possible to improve the cavitation characteristics that mean it is thicker near the hub and thinner towards the tip. In addition, the blade has to be distorted on the basis of the tangential velocity.

3.2.4.1 Velocity and angle of distortion (180°-β_∞)

The velocity triangles, which occur on the blade, play a significant role in determining its distortion. When a cylindrical cut is set at the runner and the cut is developed into a drawing pane, a grating like that shown in Figure 3.2 occurs. Velocity triangle 1 occurs directly before the grating and the velocity triangle 2 occurs directly after the grating. The meridian components w1m and w2m are equal. The medial relative velocity can be determined via the average of w1 and w2 and its direction is specified due to the angle β_∞. Value t represents the grating partition and value l denotes the chord.

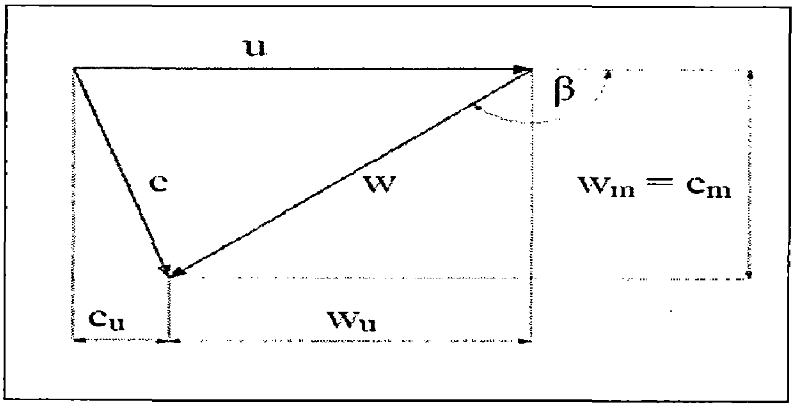


Figure 3.2(a): velocity triangle and its components

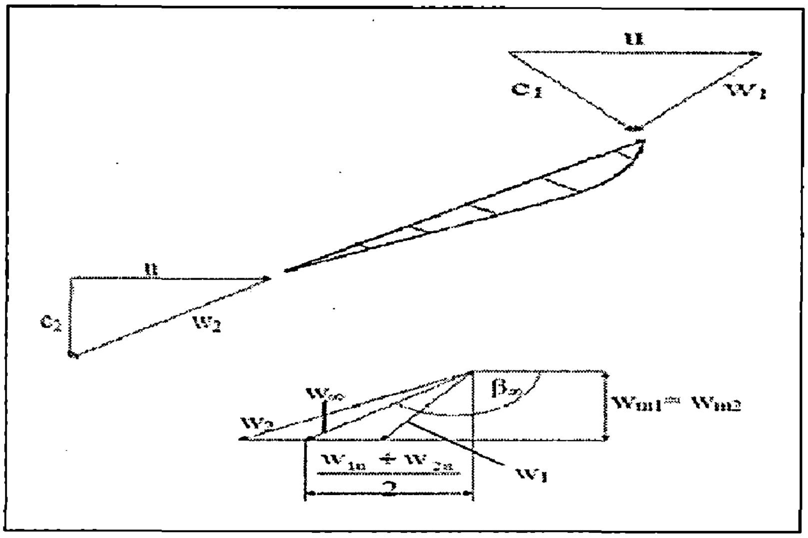


Figure 3.2(b): Inlet and outlet velocity triangle at blade

Where:

c = Absolute velocity

c_u = Tangential component of absolute velocity

c_m = Meridian component of absolute velocity

u = Tangential velocity

w = Relative velocity

w_m = Meridian component of relative velocity

w_u = Tangential component of relative velocity

and c_1 ,u_1, w_1 are the velocity at point 1 show in figure 3.1

Similarly c_2, u_2, w_2 are the velocity at point 2 show in figure 3.1

The value of parameter given below

$H_n = 1.5$ m

$H_1 = 1.40$ m

$H_2 = 1.60$ m

Tangential velocity $u = \frac{\pi \times D \times N}{60} = \frac{\pi \times 1.77 \times 125}{60} = 11.56$ m/s

$$c_{u1} = \frac{H_1 \times g}{u} = 1.23 \text{ m/s}$$

$$c_{u2} = \frac{H_2 \times g}{u} = 1.32 \text{ m/s}$$

$$w_{u1} = c_{u1} - u = -10.33 \text{ m/s}$$

$$w_{u2} = c_{u2} - u = -10.24 \text{ m/s}$$

$$w_{u\infty} = \frac{w_{u1} + w_{u2}}{2} = -10.29 \text{ m/s}$$

$$A_\infty = \frac{\pi(D^2 - d^2)}{2} = 2.16 \text{ m}^2$$

$$w_m = \frac{Q}{A\infty} = 3.25 \text{ m/s}$$

$$w_1 = \sqrt{(w_{u1}^2 + w_m^2)} = 10.82 \text{ m/s}$$

$$w_2 = \sqrt{(w_{u2}^2 + w_m^2)} = 10.74 \text{ m/s}$$

$$w_\infty = \sqrt{(w_{u\infty}^2 + w_m^2)} = 10.79 \text{ m/s}$$

$$\beta_\infty = \cos^{-1}(w_{u\infty}/ w_\infty) = 161.8° \approx 162°$$

$$(180°-\beta_\infty) = 180° - 162° = 18°$$

To define the distortion of the blade divide the blade in six different radiuses and the velocity triangle of the six different radiuses of the blade is determined in Table 3.2. The angles β_∞ for each radius give conclusions on the distortion of the blade [52]. The cylindrical cuts of the blade is shown in Figure 3.3

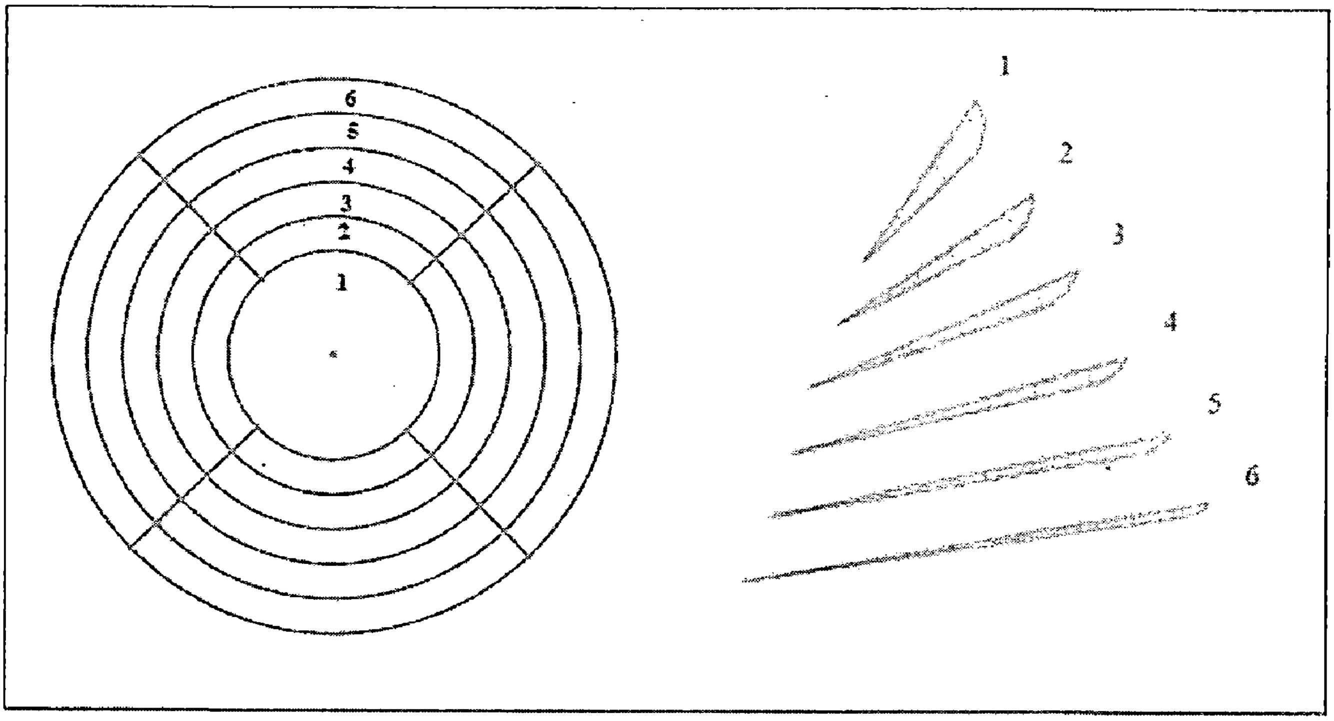


Figure 3.3: Cylindrical cuts of the blade

Table 3.2: Velocity and angles of the occurring velocity triangles

S. No.	Parameters	Values					
1	D (m)	1.77	1.54	1.31	1.08	0.85	0.62
2	H_1(m)	1.45	1.45	1.45	1.45	1.45	1.45
3	H_2 (m)	1.55	1.55	1.55	1.55	1.55	1.55
4	C_{u1}(m/s)	1.23	1.41	1.66	2.02	2.56	3.51
5	C_{u2}(m/s)	1.32	1.51	1.78	2.16	2.74	3.76
6	U(m/s)	11.56	10.06	8.56	7.05	5.55	4.05
7	W_{u1}(m/s)	-10.33	-8.64	-6.89	-5.04	-2.99	-0.54
8	W_{u2}(m/s)	-10.24	-8.55	-6.78	-4.90	-2.81	-0.29
9	$W_{u\infty}$(m/s)	-10.29	-8.60	-6.84	-4.97	-2.90	-0.42
10	W_m(m/s)	3.25	3.25	3.25	3.25	3.25	3.25
11	W_∞(m/s)	10.79	9.19	7.57	5.94	4.36	3.28
12	Angle(degree)	18°	20°	26°	33°	48°	82°

3.2.5 Dimensions of Draft Tube

For the present study, sizing of draft tube has taken from the Indian Standard 12800 (part-1) [53]. All the dimensions of the draft tube are given as empirical relation between the diameter of runner and the dimensions of draft tube. These dimensions have been shown in the Figure 3.4.

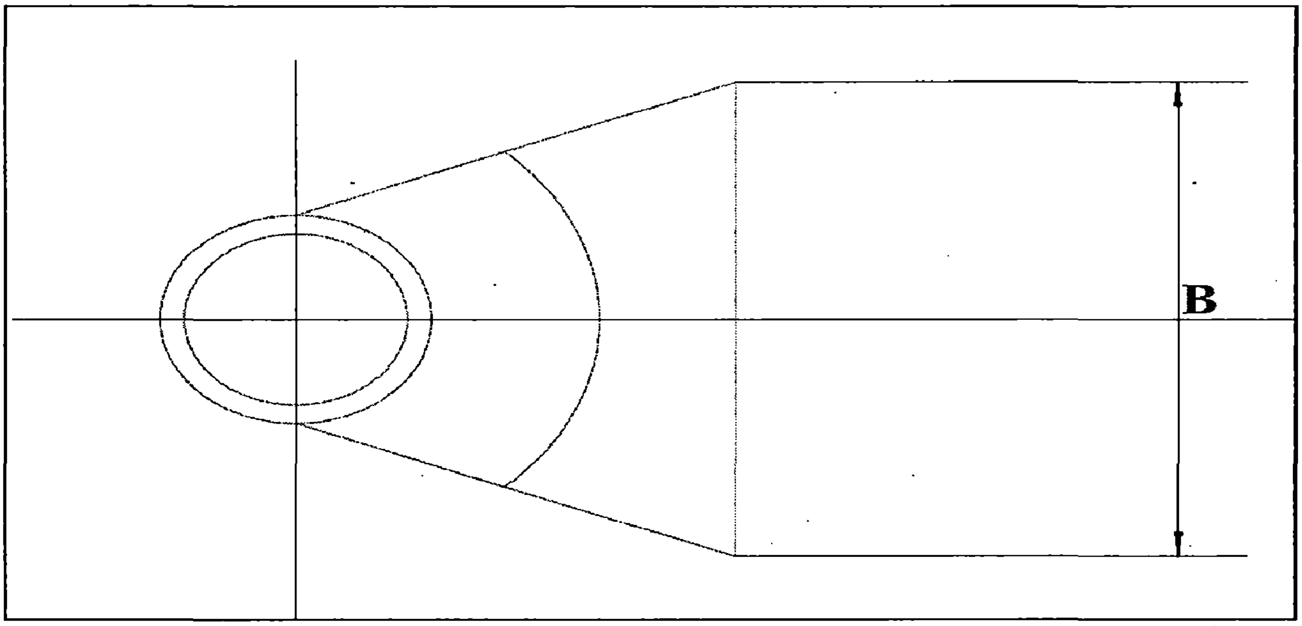


Figure 3.4(a): Top view of draft tube

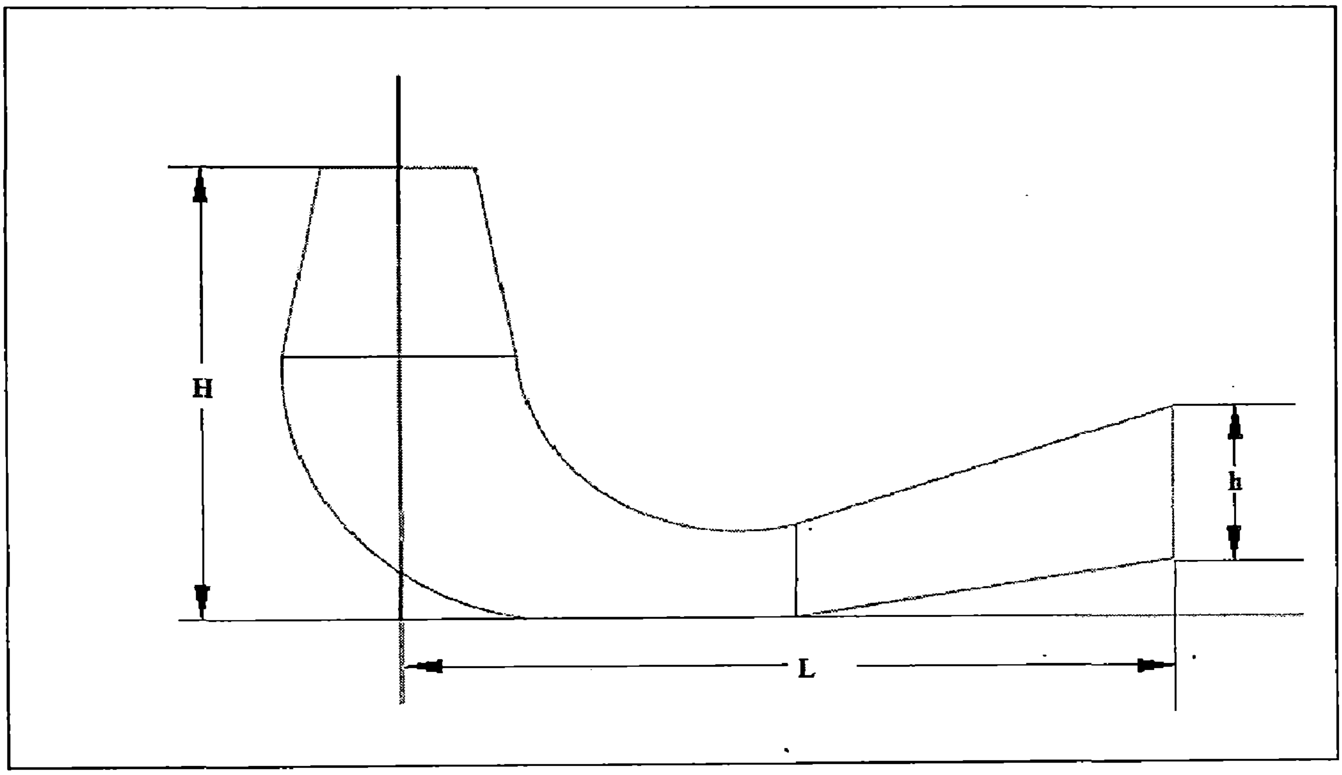


Figure 3.4(b): Front view of Draft tube

The empirical relation between the diameter of runner and the dimension of the draft tube are given below:

i. Height of the tube at exit end (h)

$h = 1.25 * D$

$h = 2.4$ m

ii. Depth (H)

$H = 1.2 * D$

$H = 2$ m

iii. Length (L)

$L = 5 * D$

$L = 8.85$ m

iv. Clear width of the draft tube at the exit end (B)

$B = 2.6 * D$

$B = 4.6$ m

3.2.6 Design of Wicket Gates

The dimensions of the wicket gates are given below;

i. Diameter (D_g)

Diameter of wicket gates has been calculated by formula which is given below

$$D_g = \frac{60 * K_{ug} * \sqrt{2gH}}{\pi N} \tag{3.5}$$

$D_g = 1.86$ m

Where, K_{ug} is speed ratio, varies from 1.3 to 2.25, increase with the specific speed [54]. For the present study, the value of the speed ratio (K_{ug}) has been taken 2.25.

ii. Height (B_g)

In the case of Kaplan turbine the value of the ratio of Height of the wicket gates and diameter of runner vary 0.10 to 0.3 according to the specific speed. For the present study, the value of the ratio has been taken 0.23.

i.e. $B_g/D = 0.23$

so $B_g = 0.40$ m

iii. Length (L_g)

The empirical relation between the length and diameter of the wicket gate is given below;

$L_g = 0.3 * D_g = 0.56$ m

iv. No. of wicket gates

Number of wicket gates varies 8 to 24, as increasing diameter of runner.

Let the no. of wicket gate 12 which is suitable for the runner diameter of Kaplan turbine [52].

CHAPTER 4

CFD ANALYSIS OF KAPLAN TURBINE

4.1 GENERAL

As per literature review propeller turbine has low part load efficiency and Kaplan turbine have better part load efficiency. For micro hydro range variation of parameter is high. So propeller turbine is not give its full efficiency as it worked at part load. So there is need of Kaplan turbine to extract full potential.

This chapter described modeling of the Kaplan turbine based on the sizing of Kaplan turbine which describe in chapter 3 and simulation of flow in the turbine. There are number of CAD software available in market which is used for modeling of turbine i.e. CATIA, PRO-E, GAMBIT, SOLIDWORKS etc. In this dissertation PRO-E software has been used for modeling for turbine parts. After developing 3D modeling of turbine, 3D model has been saved into .igs format of file.

Simulation of flow in the turbine has been done with the help of ANSYS-14 software. When 3D model created in Pro-E saved in .igs format which can read by ANSYS software. ANSYS software has many tools which is used in CFD analysis. The .igs file imported in the geometry on the workbench tool. Then define the geometry material in geometry section. Then send file into the mesh in which meshing of model has been generated. Simulation occurred in FLUENT software which is one of the tools of ANSYS software. The flow in the runner was calculated in moving reference frame, while flow in the stationary component was calculated in stationary reference frame.

The overall efficiency of Kaplan turbine was determined based on fundamental equations. The various parameters used in equation depend on the type of boundary condition used for the numerical simulation. As per literature review different sets of boundary condition for CFD analysis of hydro turbine e.g. total pressure inlet & static pressure outlet, mass flow inlet & static pressure outlet. This dissertation described, simulation has been done by mass flow inlet and pressure outlet set of boundary condition.

4.2 KAPLAN TURBINE CONSIDERED FOR THE PRESENT STUDY

A 100 kW Kaplan turbine with tubular casing under rated head 1.5 m and 7.03 m^3/s discharge has been consider in the present study. The other parameters of the Kaplan turbine have been computed by using design in the Chapter 3. Salient feature of the Kaplan turbine considered are given in Table 4.1.

Table 4.1 Parameters of turbine and its values

S. No.	PARAMETERS	VALUES
1	Rated Head	1.5 m
2	Rated Discharge	7.03 m^3/s
3	Rated Turbine output	100 kW
4	Rated Speed	125 rpm
5	No. Runner blade	4 number
6	Diameter of runner	1.77 m
7	Diameter of hub	0.67 m
8	Diameter of wicket gate	1.86 m
9	No. of wicket gate	12 number
10	Height of wicket gate	0.23 m
11	Length of the wicket gate	0.56 m

4.3 METHODOLOGY

The present work has been followed the certain methodology which provided the optimum approach to finish the work. For CFD analysis of hydro turbine *first step* to generate CAD model which has been done be PRO-E software, this stage is preprocessor stage for the simulation. *Second step* to assemble the parts and generate the mesh different turbine components with the help of ANSYS software, which is also a pre processer stage for simulation. *Third step* is solving the equation with help of boundary condition and initial condition with the help of FLUENT software, this stage is called solver stage. *Final step* analyze the result, this stage is called post processing stage. The methodology of CFD analysis has been shown by flow chart given below in Figure 4.1.

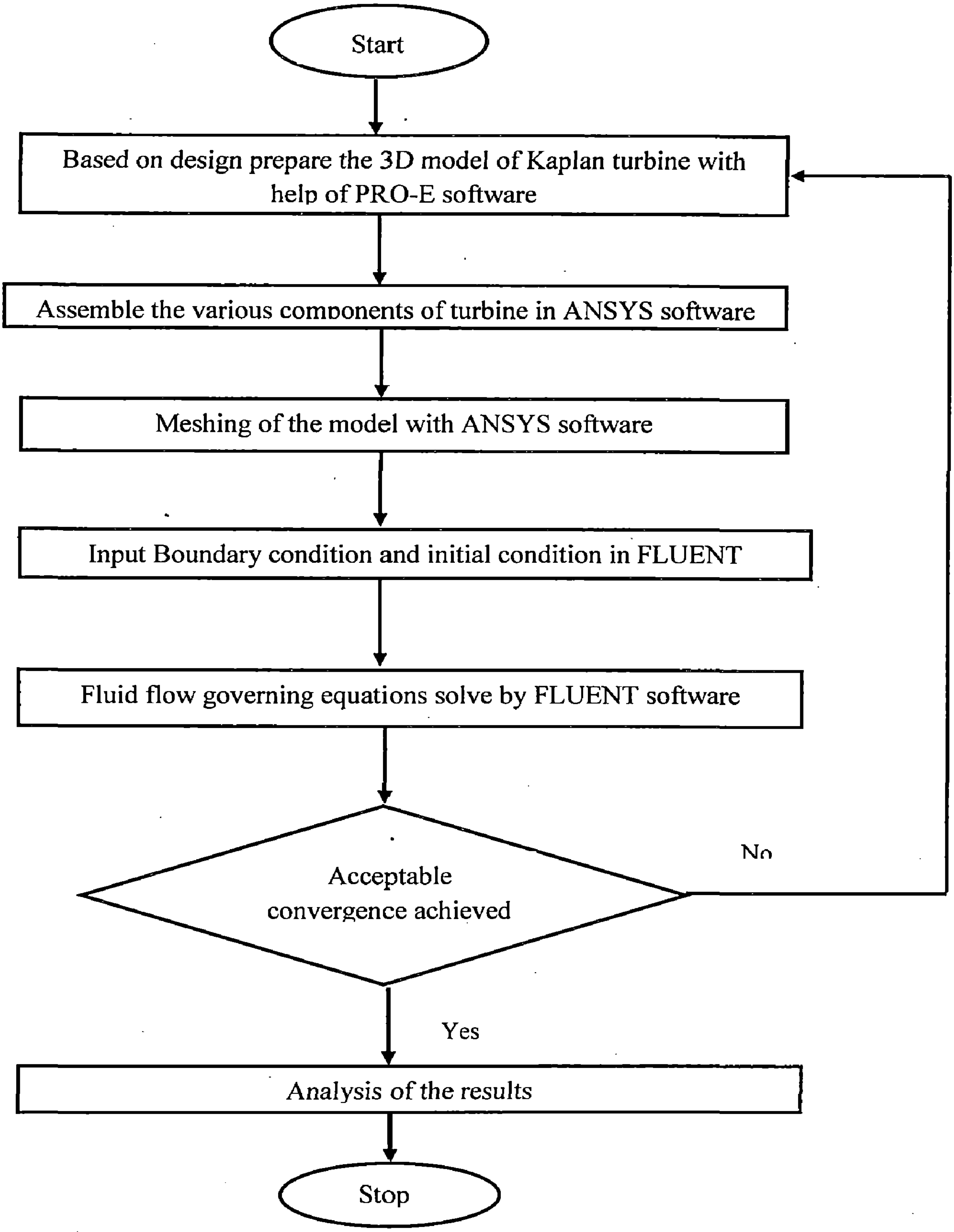


Figure 4.1: Flow chart for methodology

4.4 3D MODELING OF TURBINE COMPONENTS

3D modeling of hydro turbine is the first step of CFD analysis. For the present study PRO-E software has been used for creating 3D model of turbine. PRO-E is most user friendly software though which complicated geometry easily can be created.

4.4.1 Tools for Creating Part on PRO-E

There are many tools used in creating the 3D model of turbine. Some of basic useful tools which are described as below;

i. Open the PRO-E part modeling

Open the PRO-E software which has been provide the pro-e work screen and first step open new file which gives window in which select the part modeling. Set the parameters on which the model has been prepare. In the present study, mmns_part_solid has been used. It means unit of length is on millimeter and unit of mass is on Newton.

ii. Work screen of PRO-E

The work screen generated after selecting the part modeling. It is a platform on which model has been developed. In work screen various tools such 3 principal orthogonal planes, file management, view management, datum creation tool and feature creation toolbar are described which is used for creating 3D model of turbine parts.

iii. Sketching toolbar

For prepare any 3D model first work to prepare the 2D sketch which is created by sketching tools. First of all select plan in which the 2D sketch need to prepare then click on the sketch tool icon in the work screen sketch window is come, this will lead to sketch orientation on which the 2D sketch has been prepared. In the present study line, rectangle, circle, arc and circle etc tools have been used.

iv. 3D part generation tool bar

The 2D sketch has been converted in to 3D model with the help of 3D generation tools. Extrude, extrude cut, sweep, sweep blend, revolve and chamfer tools has been used for generation of 3D models of turbine.

v. Edit Toolbar

The final group of buttons is used for editing and modifying existing features. Used for removing extra part in the model of the turbine and also used for the developing symmetrical bodies such as the blades of turbine and blades of wicket gate. Extrude cut and Pattern command has been used in the present study [55].

4.4.2 Creating 3d Model for Wicket Gate

Step 1: Start PRO-E and Select the part modeling.

Step 2: Select the sketcher and front plane on which sketch is generated.

Step 3: Make sketch and click ok button.

Step 4: Revolve the sketch along X-axis and make 3D model of body on which wicket gate is placed. Figure 4.2 shows the body on which wicket gate is placed.

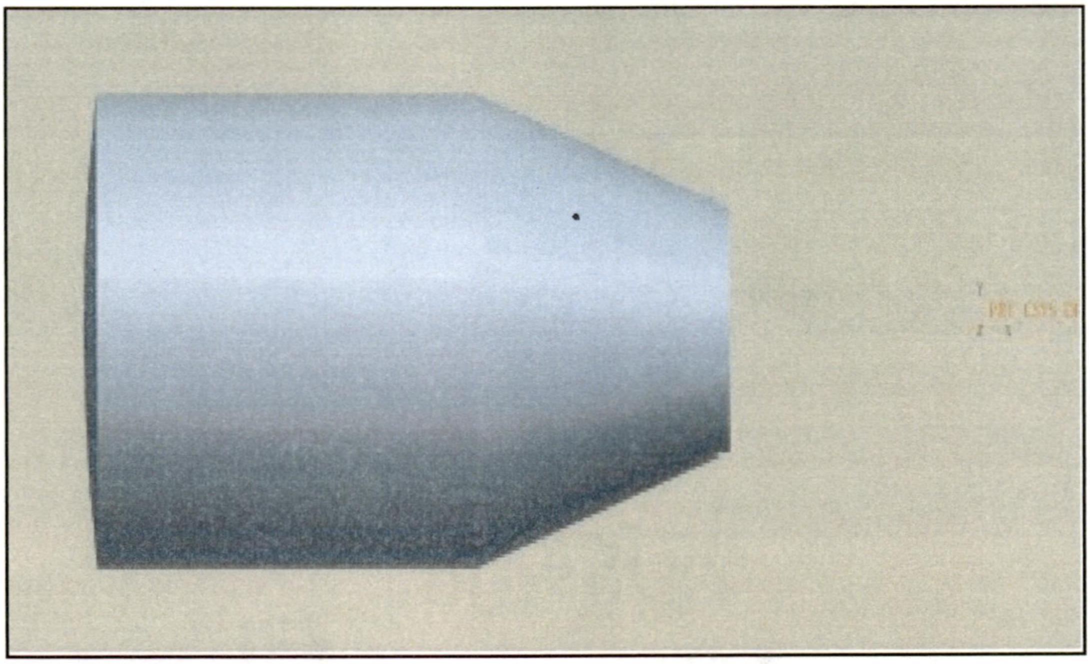


Figure 4.2: Body on which wicket gate is placed

Step 5: Make straight line on the plane normal to the inclined plane.

Step 6: With help of swept blend thin protrusion command final model of wicket gate has been made. Figure 4.3 shows the 3D model of wicket gate.

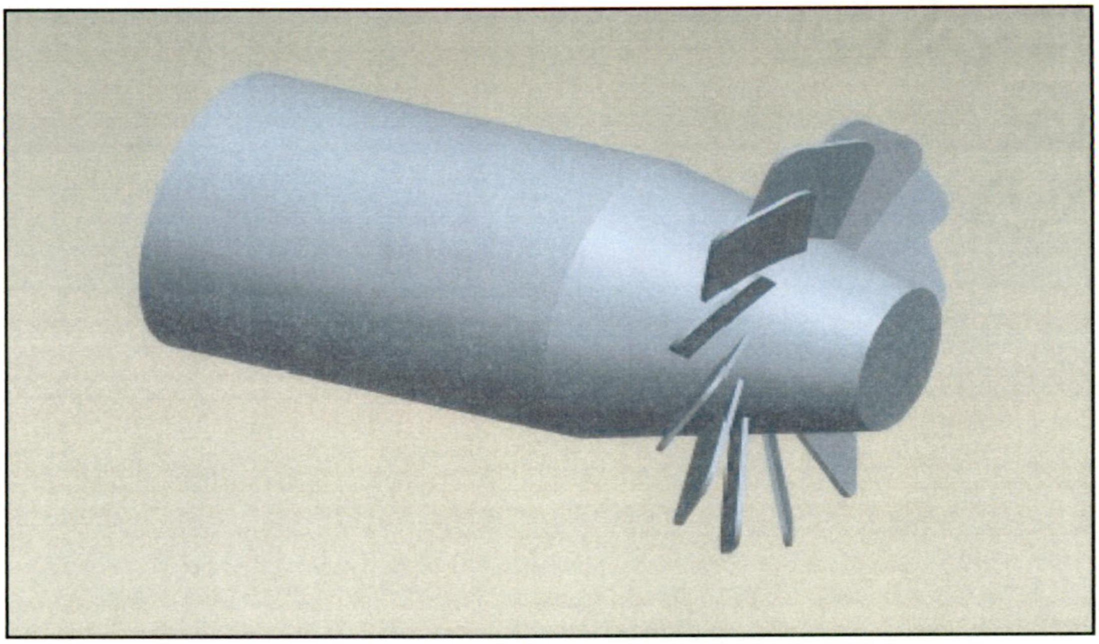


Figure 4.3: 3D model of wicket gates

4.4.3 Creating the 3d Model of Tubular Casing with Draft Tube

Step 1: Start PRO-E and Select the part modeling.

Step 2: Select the sketcher and front plane on which sketch is generated.

Step 3: Make sketch and click ok button.

Step 4: With the help of swept blend protrusion command to make the tubular casing. Figure 4.4 shows the 3D model of tubular casing of Kaplan turbine.

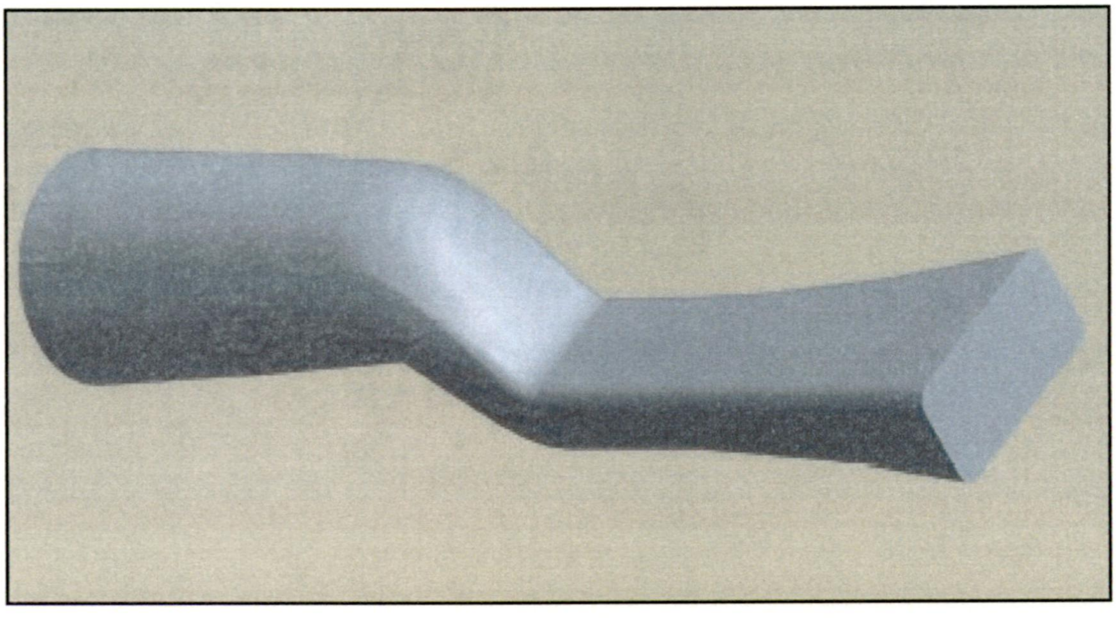


Figure 4.4: 3D model of tubular casing with draft tube

4.4.4 Creating the 3 Model of Blade

Step 1: Start work bench on ANSYS-14 software.

Step 2: Open the bladegen module and click on create all blades.

Step 3: Edit bladegen and select new bladegen file and window of initial Meridional congufuration Dialog open.

Step 4: Provide the blade dimensions, blade angle and no. of blades has been show in Table 4.2.

Table 4.2: Blade parameters

Parameters	Values
Hub radius	0.31 m
Runner blade radius	0.885 m
Thickness	0.03 m
Blade angle	18°
Number of blades	4 number

Step 5: Select model properties: open the model property dialog box the parameters have been shown in shown in Table 4.3.

Table 4.3: Turbine parameter type

Parameters	Type
Component type	Turbine
Configuration Type	Axial
Rotation type	Negative
Model units	meter

Step 6: click ok button then the blades of turbine has been generated.

Step 7: Send the data to geometry module on work bench and generate the blades on geometry module which has been show in Figure 4.5.

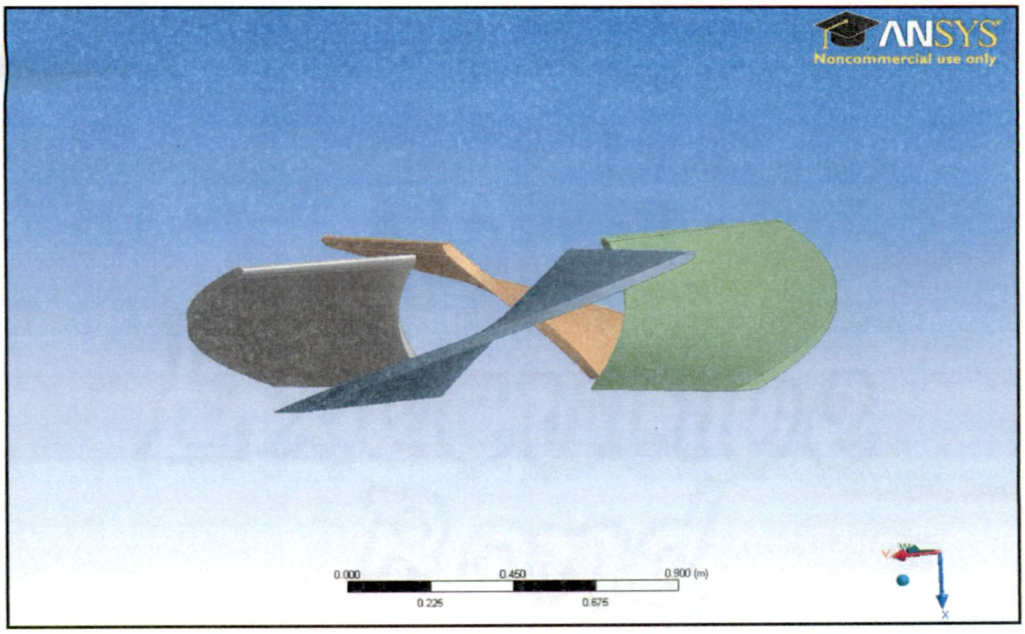


Figure 4.5: Blades generated by ANSYS software

Step 8: Export the model in .IGS file and save the file.

Step 9: Open the .IGS file in PRO-E software and with the edit feature tool generate datum planes.

Step 10: Select the sketch tool and select Datum plane. Make the 2D sketch of hub and make 3D model of hub by revolving the sketch the runner model has been completed which is shown below in Figure 4.6.

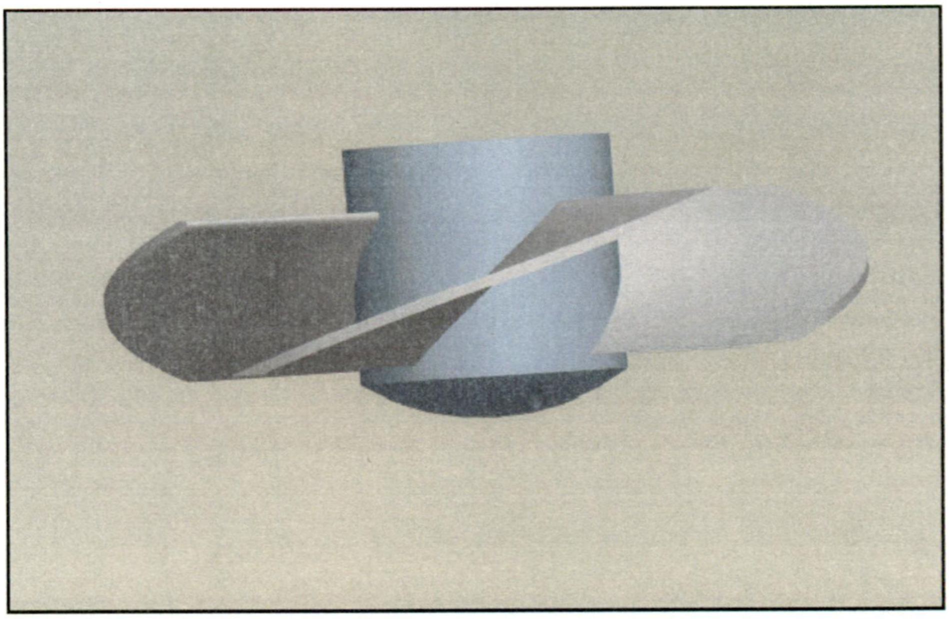


Figure 4.6: Kaplan runner

4.5. MESHING OF THE 3D MODELS

Meshing or grid generation of 3D model is most important and time consuming task in CFD analysis process. As the complexity of the geometry unstructured grid consists of the triangular and tetrahedral element is used.

For the present study ANSYS mesh module has been used for the CFD FLUENT meshing. ANSYS software provides many modules for the present study geometry, mesh and FLUENT module has been used for simulation.

4.5.1 Steps for Mesh Generation in 3D Model

Step 1: Open the ANSYS Workbench which has been provided the window in which provided the information about whole project.

Step 2: Open the geometry module and Import .igs file 3D model generated by modeling software PRO-E.

Step 3: Import all 3D model of Kaplan Turbine and assemble the parts in geometry module. Assembly show in Figure 4.7;

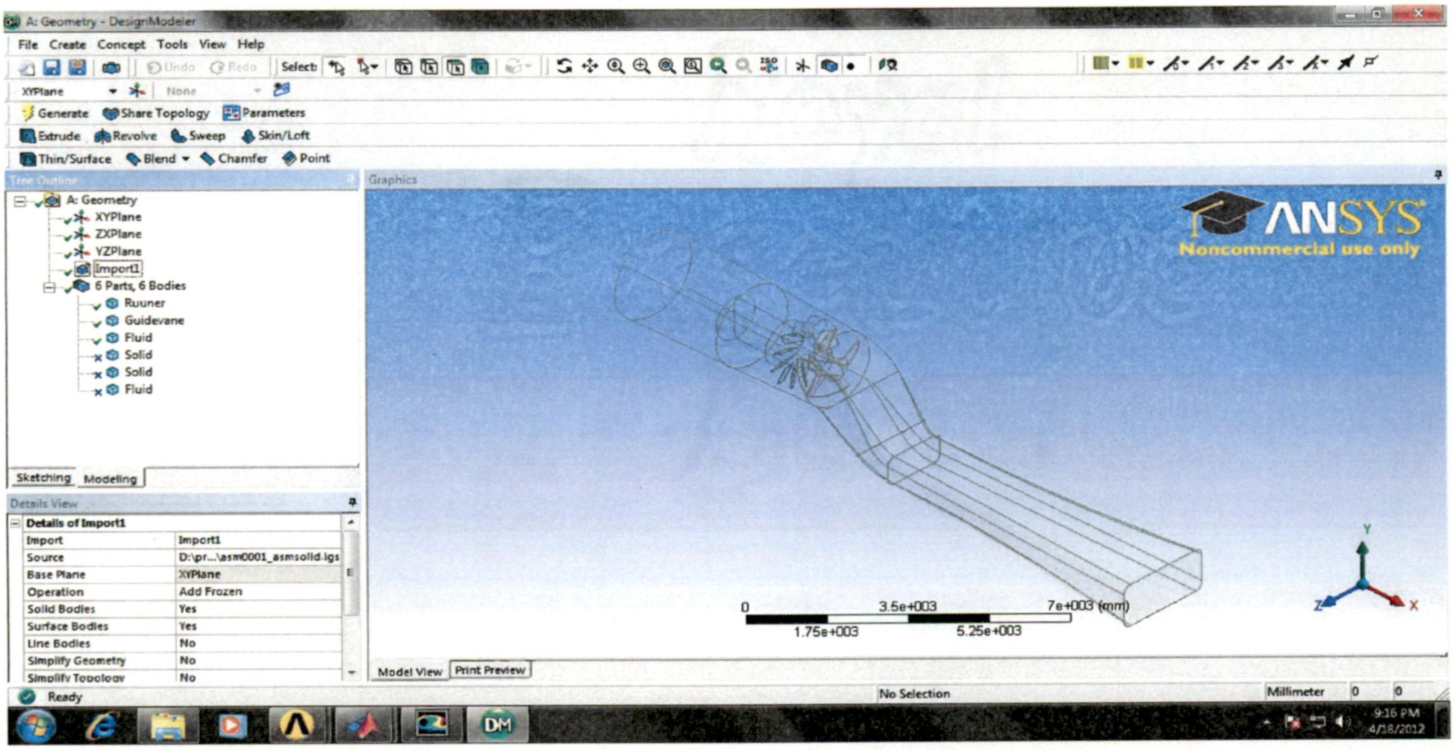


Figure 4.7: Assembly of turbine in Geometry module

Step 4: Send the geometry in the mesh module and double click on the mesh module this has been provided the window where mesh has been generated and edited.

4.5.2 Meshed 3D Model of Turbine Parts

i. Tubular casing

The meshed model of tubular casing is shown below in Figure 4.8. Total no. of nodes and element are 1065899 and 754929 respectively, generated by the meshing of the Tubular casing.

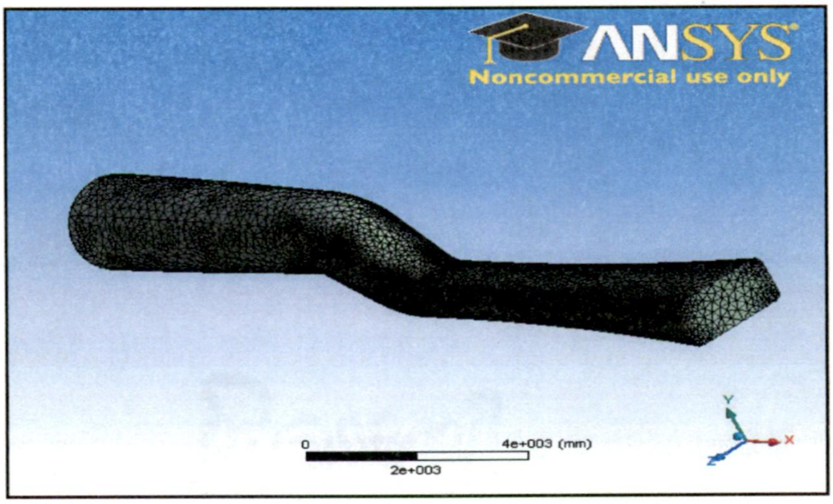


Figure 4.8: Meshed model of Tubular casing

ii. Wicket gates

The meshed model of wicket gates is shown below in Figure 4.9. Total no. of nodes and element are 2283589 and 1523396 respectively, generated by the meshing of the wicket gates.

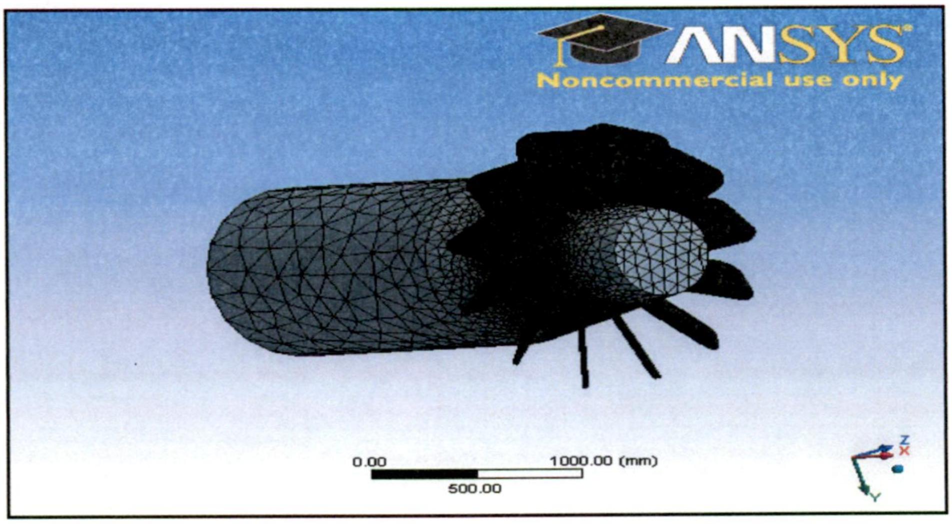


Figure 4.9: Meshed model of wicket gate

iii. Runner

The meshed model of runner is shown below in Figure 4.10. Total no. of nodes and element are 732441 and 486116 respectively, generated by the meshing of the runner.

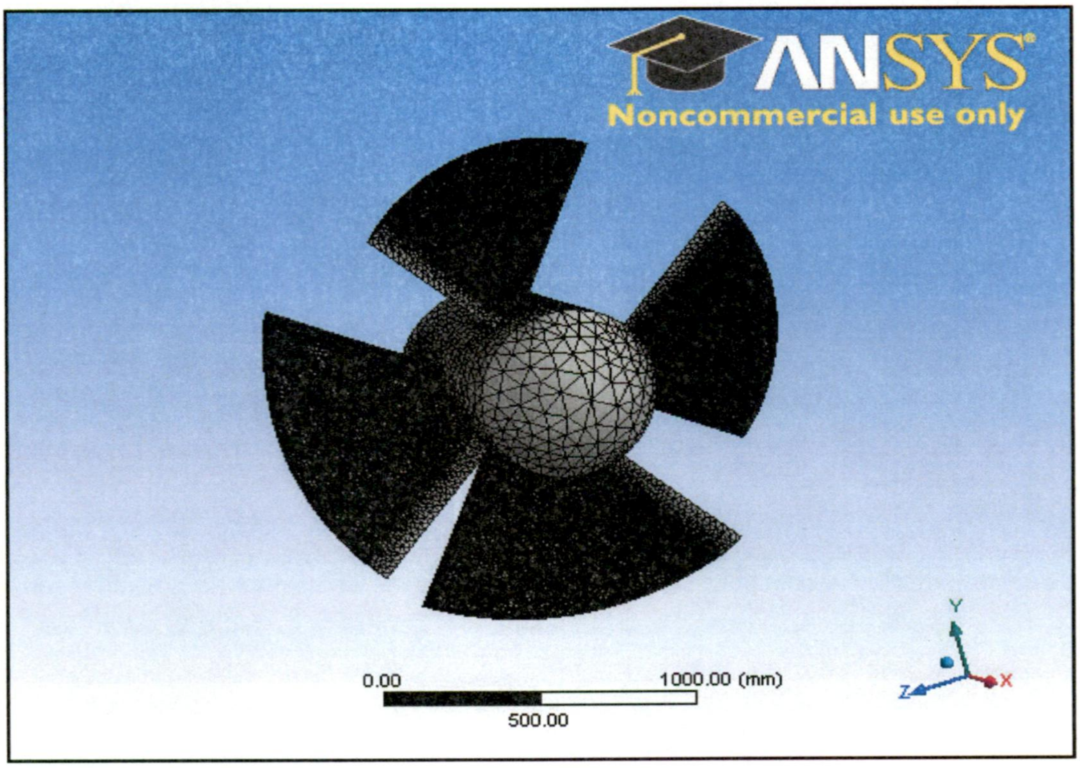


Figure 4.10: Meshed model of Runner

3D modeling and the meshing of the turbine part is the pre-processing stage for the CFD simulation. Next stage is the solver stage in which the fluid flow governing equation has been solve with the help of FLUENT module in ANSYS software.

4.6. SOLVER EXECUTION

4.6.1 Steps Involve in Solver Execution in FLUENT [56]

a. In the solver execution menu is laid down such that order of operation is generally left to right. Following are the steps in the solver execution.

   (i) Importing and scaling the mesh file.

   (ii) Selecting the physical models.

   (iii) Defining the material properties.

(iv) Defining initial operating conditions and cell zone condition.

(v) Defining boundary conditions.

(vi) Defining Solution method and control.

(vii) Initialization solution.

(viii) Run calculation.

4.6.2 Steps Execution

Following steps are followed for execution.

Step I.

i. Send the mesh file on the FLUENT module in the work bench.
ii. Update the mesh and open the FLUENT module.
iii. Open FLUENT launcher.
iv. Checking grid.
v. Scaling mesh, since model is created in mm. hence before analysis unit must be changed to m in FLUENT. Since in FLUENT defaults unit is S.I.

Step II.

FLUENT in ANSYS 14 provide many turbulence model such as standard K- ε, RNG K- ε, Realizable K- ε, K-ω etc. In the present study, Standard k- ε has been used for simulation. Lipej [21] applied standard k-ε model to simulate the axial hydraulic turbines and Liu et al [24] also applied standard k-ε model to predict performance of Kaplan turbine. Model constant is given below in Table 4.4.

Table 4.4: Model constant for Standard k-epsilon model

Model constant	Value of constant
C2-Epsilon	1.9
TKE Prandtl Number	1
TDR Prandtl Number	1.2
Energy Prandtl Number	0.85
Wall Prandtl Number	0.85

Step III. Selection of material

Define the material types and property. Physical model may require inclusion of additional materials and dictates which properties need to be defined. Material properties defined in Materials panel: Single-phase, single species flows: Define fluid/solid properties. Defining materials: here material has been taken water liquid with density 998.2 kg/m^3, and viscosity 0.001003 kg/m-s.

Step IV. Operating conditions and Cell zone conditions

i. Defining operating conditions: operating condition has been taken as operating pressure 101325 Pascal, gravitational acceleration 9.81 m/s^2 along negative y-direction and Operating temperature 300 K.
ii. Defining cell zone condition: Cell zone condition defines the moving frame and stationary frame i.e. define the motion for the moving component of the turbine and define the stationary component of the turbine. There are three components for which define the cell zone conditions for present study.
    a. *Runner*: It is moving frame, it has been provided rotation speed 13.09 rad/s on sub pad of the cell zone condition.
    b. *Wicket gate:* wicket gate are stationary so used stationary frame, cell zone condition provided no translational velocity and rotational velocity for wicket gate.
    c. *Tubular casing with draft tube:* It is moving frame, for the present study there is translational velocity 1.63 m/s provided in the positive X-direction.

Step V. Boundary Condition

There are two boundary conditions are given for the present model, i.e. mass flow inlet and pressure outlet For the present study, mass flow rate 7030 kg/s has been given at the inlet of tubular casing and second boundary condition pressure condition at the outlet of the draft tube has been 15092 Pascal.

Step VI. Solution method and control

i. Solution Method: SIMPLEC solution method with skewness factor 1 has been used for present study. For the solution of the moment second order upwind method has been used and for the turbulence first order upwind method has been used.

ii. Solution control: Under relaxation factor for pressure 0.3, Density 1, body force 1, momentum 0.7 and Turbulent kinetic energy 0.8 has been used for present study.

Step VII. Solution Initialization.

Initialization is the step before running the calculation. There are two type of solution Initialization has been provided by FLUENT software these are Hybrid Initialization and Standard Initialization. Hybrid initialization has been used for the present study.

Step VIII. Run calculation

In the present study, 1000 numbers of iteration have been used for each simulation [57]. After the iterations, the results have been obtained in the form of static pressure, total pressure, velocity of different components and the torque at the runner. The results have been discussed in the Chapter 5.

CHAPTER 5

RESULTS AND DISCUSSION

5.1 GENERAL

The detailed methodology adopted to carry out the flow analysis of Kaplan turbine using commercial CFD package Fluent which is one of the module of ANSYS software, was presented in Chapter 4. The simulations were carried out at four different wicket gate openings and each par load condition has four simulations for four different runner blade angles. These simulations have been carried out with the help of the standard k-epsilon turbulence model. In this chapter, the complex flow structure is predicted inside the different components at design and different operating points. The efficiency of Kaplan turbine has been evaluated with the help of input and output parameters of FLUENT. The operating characteristic curves, predicted by the numerical simulations were drawn and discussed in this chapter.

5.2 COMPUTATION OF EFFICIENCY

The efficiency of a turbine with an incompressible working fluid is defined as the ratio of the work delivered to the rotor to the energy available from the fluid stream. This ratio can be expressed as:

$$\eta = \frac{T\omega}{Q(P_{t1}-P_{t2})} \tag{5.1}$$

Where, T is the net torque acting on the runner (N-m), ω is the angular speed (radian), Q is discharge through turbine (m^3/s), P_{t1} is total pressure at the inlet of the turbine and P_{t2} is the total pressure at the exit of the runner.

Values of different parameters which are required to calculate efficiency of Kaplan turbine can be obtained with the help of report sub pad in post processor of FLUENT software. The input parameters for calculating efficiency are shown below;

(a) Average total pressure at turbine outlet (P_{t2}), (Pa)

(b) Discharge through turbine (Q),(m^3/s)

(c) The angular speed of the runner (ω), (rad/s)

The output parameters for calculating the efficiency are shown below;

(d) Total pressure at turbine inlet (P_{t1}), (pa)

(e) The net torque acting on the runner (T), (N-m)

5.3 ANALYSIS OF SIMULATION FOR DESIGN CONDITION AT 85% OPENING

Simulations were carried out for the Kaplan turbine under 7.03 m^3/s discharge and 1.5 m head as design parameters. For design condition wicket gate opening is taken 85% and the runner blade angle is equal to 18° at the outer distortion. The design condition has been provided the maximum efficiency for the turbine out of all the different operating conditions.

5.3.1 Contour for Design Condition

The total pressure variation on the turbine for wicket gate opening at 85% is shown below in Figures 5.1, 5.2 and 5.3. Figure 5.1 shows the pressure variation on runner, it has been found that the maximum pressure is observed at the tip of the runner. Increase in pressure was observed from hub to tip of the runner.

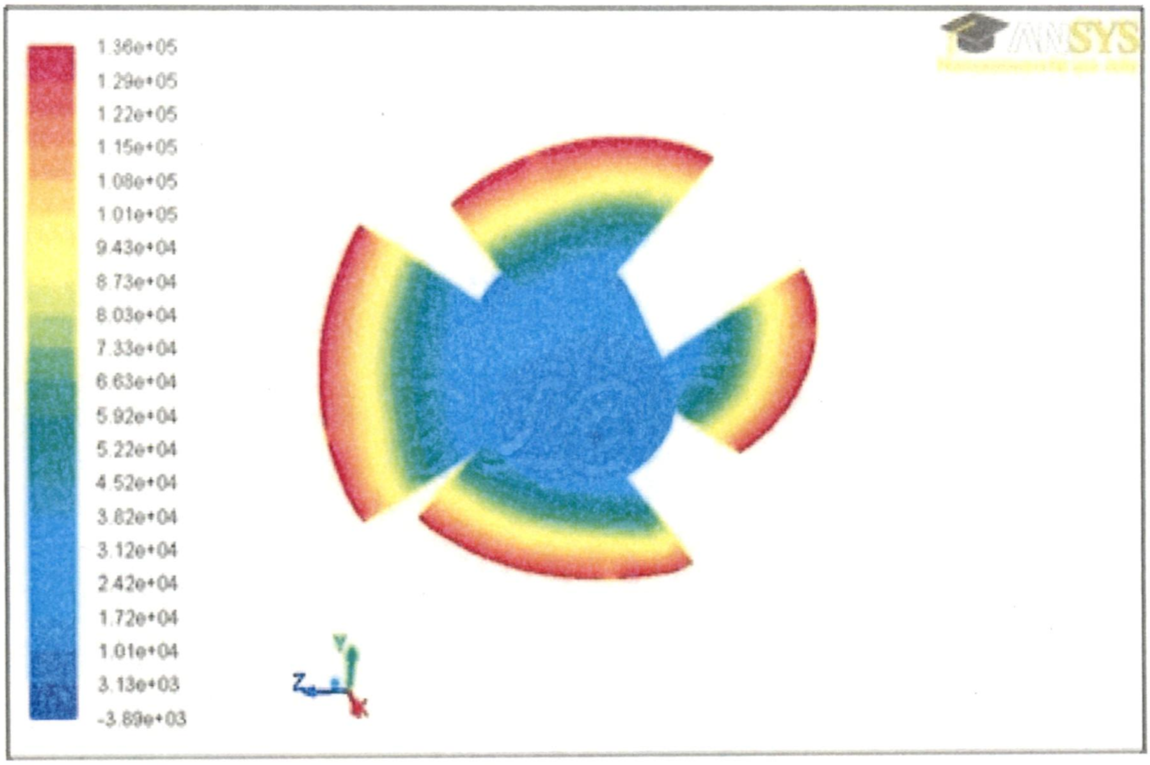


Figure 5.1: Total Pressure variation on Runner

The variation of pressure in the wicket gate is shown in Figure 5.2. It has been found that the maximum pressure is obtained at the upper portion of the wicket gate. The pressure variation in the wicket gate is found to be lesser than pressure variation in the runner and tubular casing.

Figure 5.2: Total pressure variation on wicket gate

Figure 5.3 shows the variation of total pressure in tubular casing with draft tube. It has been found that the maximum pressure obtained at the inlet of the casing. The total pressure has the minimum value at the inlet of draft tube.

Figure 5.3: Total pressure variation on casing and draft tube

5.3.2 Efficiency at Design Condition

The values of input parameters used for simulation in FLUENT software i.e. pressure at outlet of the turbine, angular speed and discharge are given in Table 5.1

Table 5.1: The values of input parameters for design condition

Parameters	Discharge(Q) (m^3/s)	Angular speed(ω) (rad/s)	Pressure at outlet of turbine(P_{t2}) (Pa)
Value	7.03	13.09	15092

The output parameter torque acting on the runner and the pressure at inlet of the turbine has been calculated by FLUENT software are given in Table 5.2

Table 5.2: The output parameters generated by FLUENT for design condition

Pressure at inlet of turbine(P_{t1}) (Pa)	**Torque components**			**Torque, T (N-m)**
	Tx (N-m)	**Ty (N-m)**	**Tz (N-m)**	
136710.5	-33.38	-114.67	-59487.15	59487.27

The efficiency of the Kaplan turbine for design condition has been found to be 91% from Equation (5.1).

5.4 ANALYSIS OF SIMULATION FOR DIFFERENT PART LOAD CONDITIONS

In the present study, the simulations were carried out between 75% to 55% of the wicket gates opening at 3 different points to cover the wide range of discharge for the part load conditions. Simulations have been performed for four different runner blade angles at each wicket gate opening to determine the part load efficiency of Kaplan turbine as discussed below;

5.4.1 Wicket Gate at 75% Opening

Four different runner blade angles of 17°, 18°, 19° and 20° have been used for four simulation of the Kaplan turbine at 75% of wicket gate opening. Table 5.3 gives input parameter for the simulation of different runner blade angles, Table 5.4 gives the output parameters generated by the FLUENT software and the value of efficiency generated through Equation (5.1) based on the input and output parameters are given in Table 5.5.

Table 5.3: Input parameter for wicket gate opening at 75%

Runner Blade angle	Discharge(Q) (m^3/s)	Angular speed(ω) (rad/s)	Pressure at outlet of turbine(Pt2), (Pa)
17°	6.02	13.09	15092
18°	6.02	13.09	15092
19°	6.02	13.09	15092
20°	6.02	13.09	15092

Table 5.4: Output parameter generated by FLUENT software for wicket gate opening at 75%

Runner Blade angle	Pressure at inlet of turbine, (P_{t1}) (Pa)	Torque components			Torque, T (N-m)
		Tx (N-m)	Ty (N-m)	Tz (N-m)	
17°	132678.7	-13.12	-172.05	-45964.37	45964.69
18°	127882.99	-210.55	-1235.71	-44722.36	44740.16
19°	126652.80	-97.69	14.07	-46388.59	46388.69
20°	125375.44	1.44	-43.83	-44548.76	44548.76

Table 5.5: The variation of efficiency with runner blade angle for wicket gate opening at 75%

Runner Blade angle	Efficiency (%)
17°	84.99
18°	86.22
19°	90.41
20°	87.83

The efficiencies of the Kaplan turbine for 75% of the wicket gate position has been found 84.99%, 86.22%, 90.41% and 87.83% for the runner blade angle of 17°, 18°, 19° and 20°

respectively. It has been seen that for the blade angle 19° almost same value of efficiency has been observed as the design conditions for opening of wicket gate at 85%.

5.4.1.1 The efficiency curve for wicket gate opening at 75%

Figure 5.4 shows the variation of the efficiency with respect to runner blade angle for wicket gate opening at 75%. It has been seen that efficiency for wicket gate opening at 75% increases as the runner blade angle and the maximum efficiency 90.41% has been obtained at 19° runner blade angle.

Figure 5.4: Efficiency curve v/s Runner blade angle curve

5.4.1.2 Contours for 19° runner blade angle

The total pressure variation on the turbine for wicket gate at 75% opening is shown below in Figures 5.5, 5.6 and 5.7. Figure 5.5 shows the pressure variation on runner which is observed on similar line as at the design condition. It has been found that the maximum pressure is observed at the tip of the runner. Increase in pressure was observed from hub to tip of the runner.

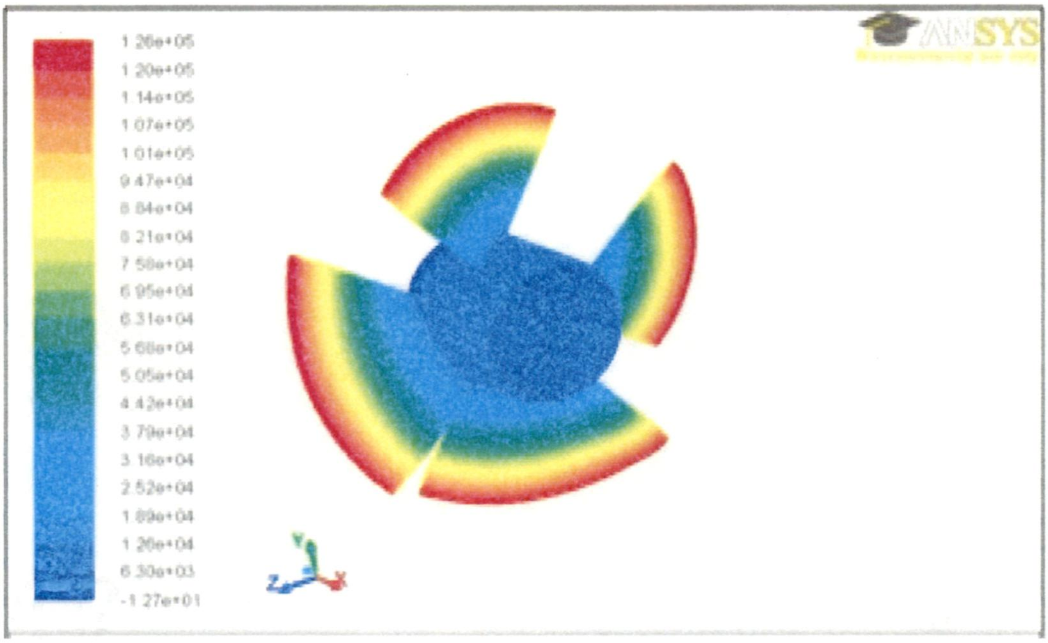


Figure 5.5: Total pressure variation on Runner

Figure 5.6 shows the variation of total pressure in tubular casing with draft tube. It has been found that the maximum pressure obtained at the inlet of the casing. The total pressure has the minimum value at the inlet of draft tube.

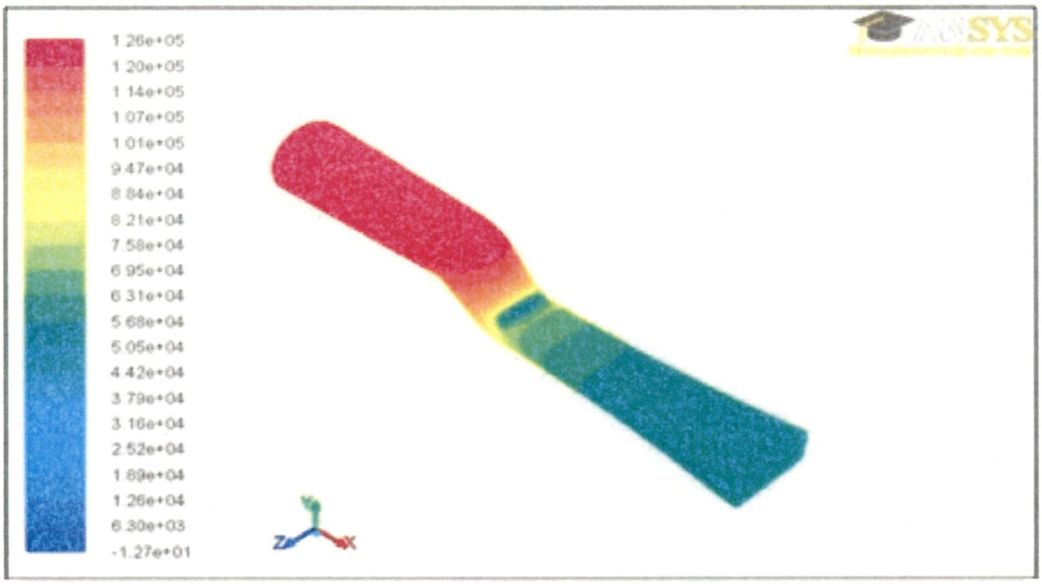


Figure 5.6: Total pressure variation on Tubular Casing

The variation of pressure in the wicket gate is shows in Figure 5.7. It has been found that the maximum pressure obtained at the upper portion of wicket gate. The pressure variation in the wicket gate is found to be lesser than pressure variation in the runner and tubular casing.

Figure 5.7: Total Pressure variation on the wicket gates

5.4.2 Opening of Wicket Gate at 65%

On similar lines as discussed in the section 5.4.1, four different runner blade angles i.e. 18°, 19°, 20° and 21° have been considered for four simulations of the Kaplan at 65% of the wicket gate opening. Table 5.6 gives input parameters for the simulation of different runner blade angles, Table 5.7 gives the output parameters generated by the FLUENT software and value of efficiency generated through Equation (5.1) based on the input and output parameters are given in Table 5.8.

Table 5.6: Input parameter for wicket gate opening at 65%

Runner Blade angle	Discharge(Q) (m^3/s)	Angular speed(ω) (rad/s)	Pressure at outlet of turbine, (Pt2) (Pa)
18°	4.92	13.09	15092
19°	4.92	13.09	15092
20°	4.92	13.09	15092
21°	4.92	13.09	15092

Table 5.7: Output parameter generated by FLUENT software for wicket gate opening at 65%

Runner Blade angle	Pressure at inlet of turbine, (p_{t1}) (Pa)	Torque components			Torque, T (N-m)
		Tx (N-m)	Ty (N-m)	Tz (N-m)	
18°	136067.22	-33.28	-174.52	-37360.59	37361.09
19°	134852.76	-104.93	-224.13	-38502.65	38505.42
20°	131909.80	-61.72	-120.94	-39682.43	39682.66
21°	136344.28	-59.52	-31.99	-38776.38	38176.43

Table 5.8: The variation of efficiency with runner blade angle for wicket gate opening at 65%

Runner Blade angle	Efficiency (%)
18°	80.16
19°	82.55
20°	85.38
21°	82.08

The efficiencies of the Kaplan turbine for wicket gate position at 65% has been found as 80.16%, 82.55%, 85.38% and 82.08% at runner blade angle 18°, 19°, 20° and 21° respectively. It has been observed that for the blade angles 20°, the efficiency attains its maximum value at 65% opening of the wicket gate.

5.4.2.1 The efficiency curve for opening of wicket gate at 65%

Figure 5.8 shows the variation of the efficiency with respect to the runner blade angle for wicket gate opening at 65%. It has been seen that efficiency for wicket gate opening at 65% increases as the runner blade angle and the maximum efficiency 85.38% has been obtained at 20° runner blade angle.

Figure 5.8: Efficiency v/s Runner blade angle curve

5.4.2.2 Contours for 20° runner blade angle for wicket gate opening at 65%

The total pressure variation on the turbine for the wicket gate opening at 65% is shown below in Figures 5.9, 5.10 and 5.11. Figure 5.9 shows the pressure variation on runner which is observed on similar line as at the design condition. It has been found that the maximum pressure is observed at the tip of the runner. Increase in pressure was observed from hub to tip of the runner.

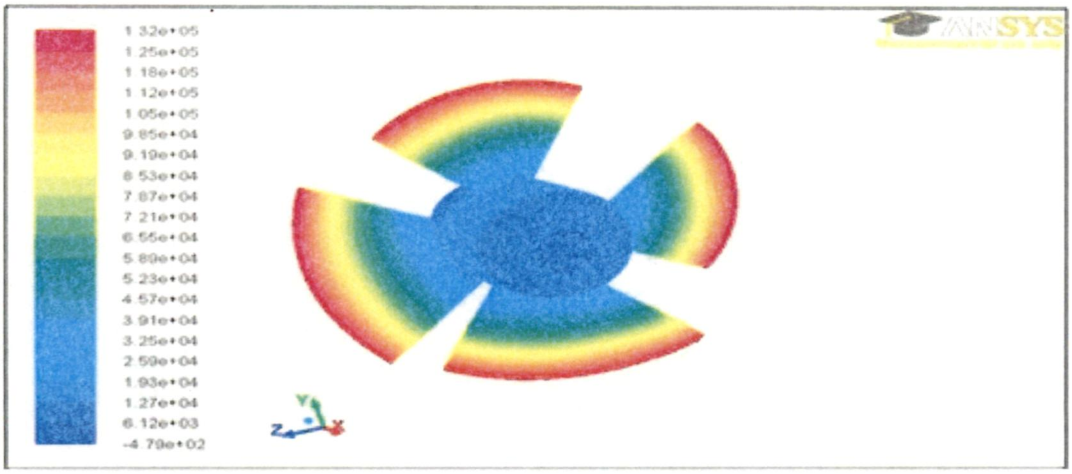


Figure 5.9: Total pressure variation on runner

Figure 5.10 shows the variation of total pressure in tubular casing with draft tube. It has been found that the maximum pressure obtained at the inlet of the casing. The total pressure has the minimum value at the inlet of draft tube.

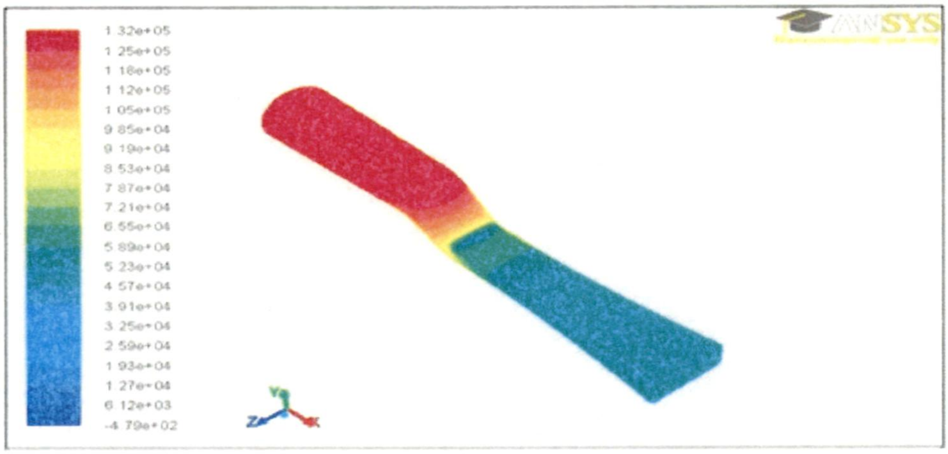


Figure 5.10: Total pressure variation on tubular casing

The variation of pressure in the wicket gate is shown in Figure 5.11. It has been found that the maximum pressure obtained at the upper portion of wicket gate. The pressure variation in the wicket gate is found to be lesser than pressure variation in the runner and tubular casing.

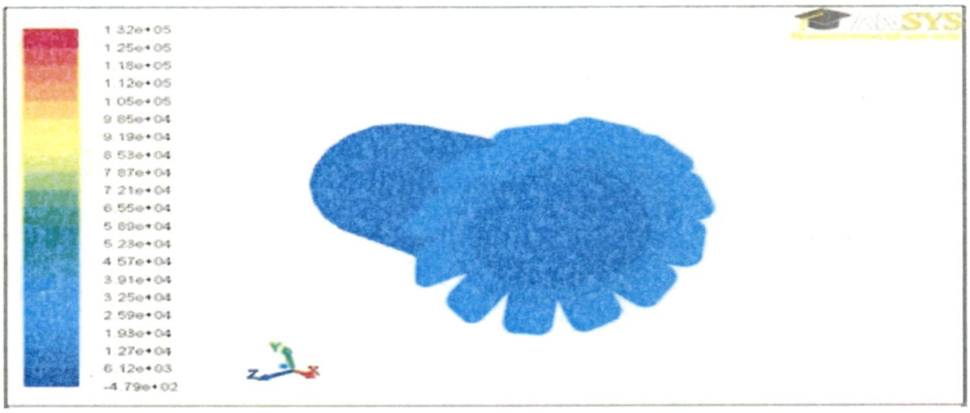


Figure 5.11: Total pressure variation on wicket gate

5.4.3 Opening of Wicket Gate at 55%

On similar lines as discussed in the section 5.4.1, four different runner blade angles i.e. 19°, 20°, 21° and 22° have been considered for four simulations of the Kaplan turbine at 55% of the wicket gate opening. Table 5.9 gives input parameters for the simulation of different blade angles, Table 5.10 gives the output parameters generated by the FLUENT software and value of efficiency generated through the Equation (5.1) based on the input and output parameters are given in Table 5.11.

Table 5.9: Input parameter for wicket gate opening at 55%

Runner Blade angle	Discharge, (Q) (m^3/s)	Angular speed, (ω) (rad/s)	Pressure at outlet of turbine, (P_{t2}), (Pa)
19°	3.87	13.09	15092
20°	3.87	13.09	15092
21°	3.87	13.09	15092
22°	3.87	13.09	15092

Table 5.10: Output parameter for wicket gate opening at 55%

Runner Blade angle	Pressure at inlet of turbine, (P_{t1}) (Pa)	Torque components			Torque, T (N-m)
		Tx (N-m)	Ty (N-m)	Tz (N-m)	
19°	143339.50	-61.38	-85.30	-30795.29	26795.46
20°	140447.86	-1.56	-258.01	-31736.92	27237.93
21°	138486.98	-28.03	-19.17	-32953.63	27853.68
22°	141367.23	-37.24	-437.89	-32135.93	27377.83

Table 5.11: The variation of efficiency with runner blade angle for wicket gate opening at 55%

Runner Blade angle	Efficiency (%)
19°	70.67
20°	73.49
21°	76.35
22°	73.33

The efficiencies of the Kaplan turbine for wicket gate position at 55% has been found as 70.67%, 73.49%, 76.35% and 73.33% at runner blade angle 19°, 20°, 21° and 22° respectively. It has been observed that for a blade angle 21°, the efficiency attains its maximum value at 55% opening of wicket gate.

5.4.3.1 The efficiency curve for 55% opening of wicket gate

Figure 5.12 shows the variation of the efficiency with respect to the runner blade angle for the wicket gate opening at 55%. It has been observed that efficiency for wicket gate opening at 55% increases as the runner blade angle and the maximum efficiency 76.35% has been obtained at 21° runner blade angle.

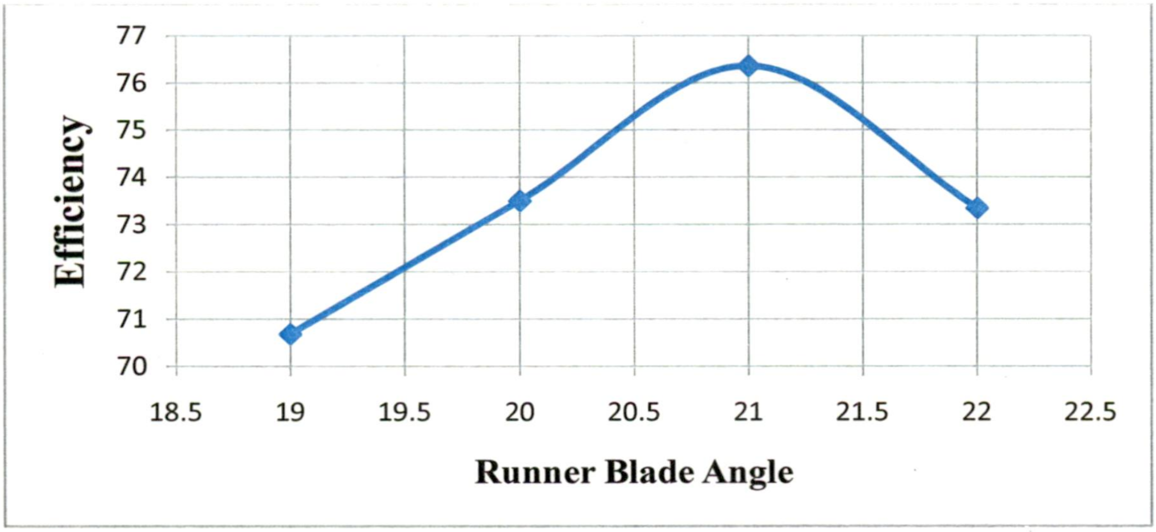


Figure 5.12: Efficiency v/s Runner blade angle curve

5.4.3.2 Contours for 20° runner blade angle for wicket gate opening at 55%

The total pressure variation in the turbine is shown below in the Figures 5.13, 5.14 & 5.15. Figure 5.13 shows the pressure variation on runner which is observed on similar line as at the design condition. It has been found maximum pressure is observed at the tip of the runner. Increase in pressure was observed from hub to tip of the runner.

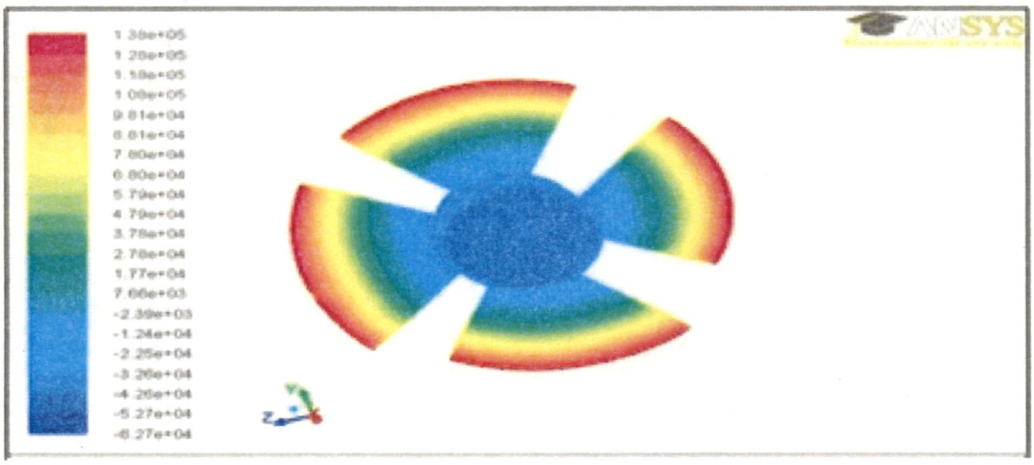


Figure 5.13: Total pressure variation on runner

Figure 5.14 shows the variation of total pressure in tubular casing with draft tube. It has been found that the maximum pressure obtained at the inlet of the casing. The total pressure has the minimum value at the inlet of draft tube.

Figure 5.14: Total pressure variation on casing

The variation of pressure in the wicket gate is shown in Figure 5.15. It has been found that the maximum pressure obtained at the upper portion of wicket gate. The pressure variation in the wicket gate is found to be lesser than pressure variation in the runner and tubular casing.

Figure 5.15: Total pressure variation on wicket gate

5.5. EFFICIENCY CURVE FOR DIFFERENT OPERATING CONDITIONS

The variation in the turbine efficiencies with respect to runner blade angles are shown in Figure 5.16 for different wicket gate openings. Maximum efficiency has been obtained at 19° runner blade angle corresponding to 75% wicket gate opening and for 65% opening of wicket gate the maximum efficiency has been obtained at 20° runner blade angle. Further, it has been seen that for 55% opening of wicket gate the maximum efficiency has been obtained at 21° runner blade angle. The part load efficiency curve has been made by joining the optimum efficiency of different wicket gate openings. The first point of the efficiency curve is the efficiency of turbine design condition which has been found as 91% at 85% wicket gate opening in the present study.

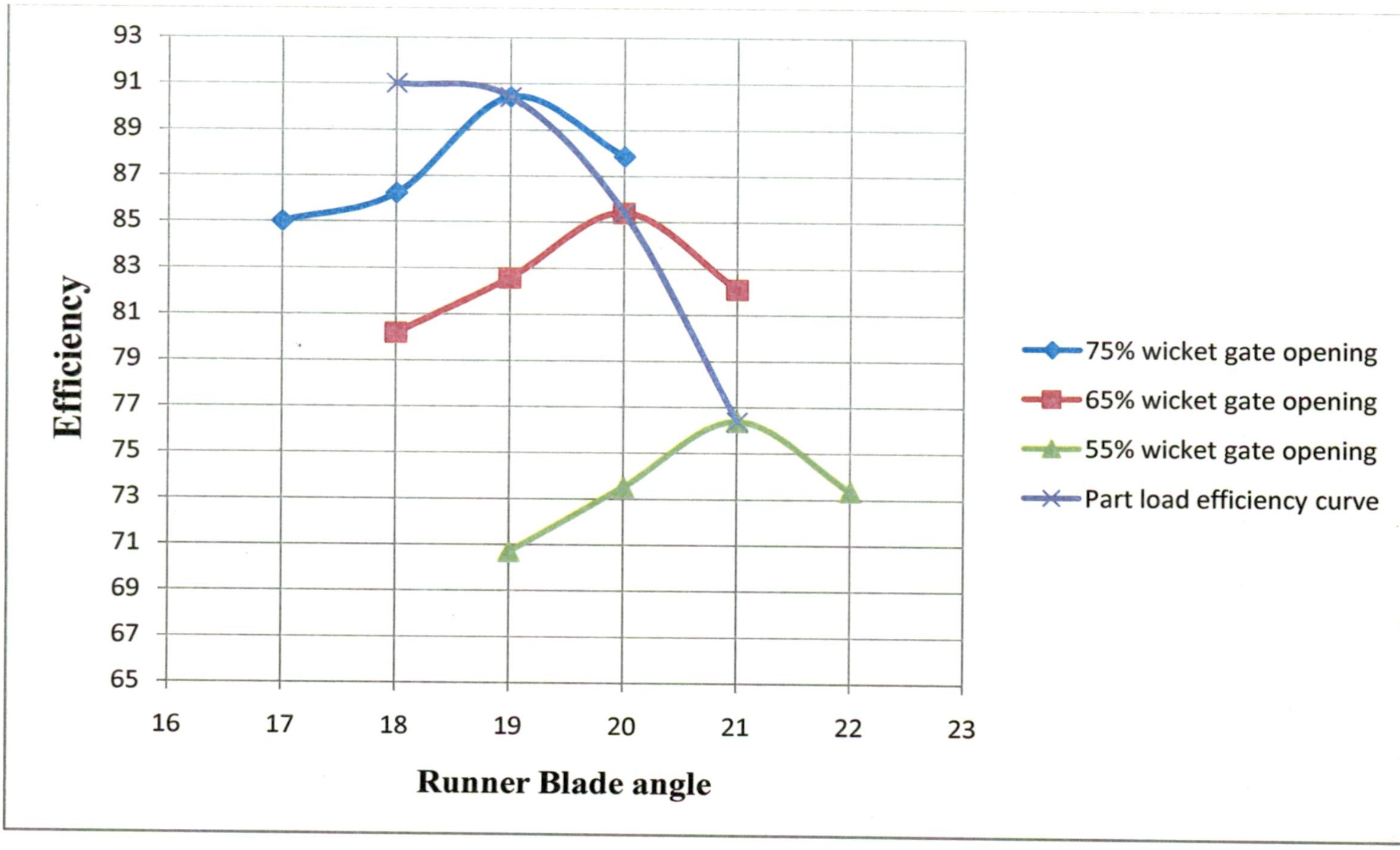


Figure 5.16: Efficiency curve for different operation conditions

CHAPTER-6

CONCLUSIONS AND RECOMMENDATIONS

6.1 CONCLUSIONS

In the present study, the operating parameters of the Kaplan turbine have been optimized using commercial CFD package ANSYS-14. The turbine having rated capacity of 100 kW at rated head and discharge of 1.5 m and 7.03 m^3/s respectively. The simulations have been carried out for three-dimensional with standard k- ε turbulence model. The following conclusions were drawn from present study:

(i) Based on CFD analysis it has been found that the total pressure variation is high on the runner and near the wall of the casing. In the casing, maximum value of total pressure has been observed while minimum value of total pressure obtained at wicket gate.

(ii) For the different part load conditions, the different values of efficiency have been obtained as 90.41%, 85.38% and 76.38% at different position of wicket gate i.e. 75%, 65% and 55% respectively.

(iii) 90.41% efficiency has been obtained for 75% wicket gate position at 19° runner blade angle, 85.38% efficiency has been obtained for 65% wicket gate position at 20° runner blade angle and 76.35% efficiency has been obtained for 55% wicket gate position at 21° runner blade angle.

(iv) The efficiency of the Kaplan turbine has been found to be maximum as 91% for design condition i.e. at rated discharge 7.03m^3/s and at rated head 1.5 m at 85% wicket gate opening.

6.2 RECOMMENDATIONS

In present analysis, flow is assumed to be steady state. Instead, in future unsteady flow analysis can be carried out. Also, 3D model can be created by other CAD software such as CATIA, SOLIDWORKS etc. Meshing can be generated by different meshing module such ICEM-CFD and TURBO-mesh. For post processing analysis CFX module of ANSYS-14 software can be used.

1. Yukse, I., "As a renewable energy hydropower for sustainable development in Turkey" Renewable and Sustainable Energy Reviews 14 (2010) 3213–3219.
2. http://www.cea.nic.in/reports/monthly/executive_rep/mar12/1-2.pdf accessed on 28th April 2012.
3. Van de Vate, J.F., (2002) Full-energy-chain greenhouse-gas emission: a comparison at power, hydropower, solar power and wind power, International Journal of Risk Assessment and Management, Vol. 3, No. 1 pp. 59-74.
4. www.powermin.nic.in (accessed on 25th April 2012)
5. IEA, International Energy Agency. World energy outlook 2009 (accessed on 28th April 2012).
6. http://www.mnre.gov.in/prog-smallhydro.htm (accessed on 30th April 2012).
7. Guidelines for Development of Small Hydro Electric Scheme, CEA (1982).
8. www.etsap.org.E07 –Hydropower-GS-gct.pdf
9. Khennas, S. and Barnett, A., "Micro-hydro power: an option for socio-economic development" World Renewable Energy Congress VI
10. Wagner, H. J. and Mathur, J., "Introduction To Hydro Energy System" Springer Heidelberg Dordrecht London New York.
11. Modi,P. N. & Seth, S. M., "Hydraulic and Fluid Mechanics" Seventh Edition(1985), 1025
12. Jain, S., Saini, R. P. & Kumar, A., "CFD approach for prediction of efficiency of Francis turbine" IGHEM-2010, oct. 21-23, AHEC, IIT Roorkee, Indian
13. Date, A. W., "Introduction to Computational Fluid Dynamics" Cambridge University Press 2005
14. Ferziger, J.H. and Peric, M., "Computation Methods for Fluid Dynamics" 3rd edition-revised (2004), Springer (India) Private Limited, New Delhi.
15. Manual "FLUENT 6.3.26 Software and GAMBIT 6.3 Software."
16. FLUENT Documentation, 2005
17. Prasad, V., Gahlot, V. K. & Krinnamachar, P., "CFD approach for design optimization and validation for axial flow hydraulic turbine" Indian Journal of Engineering & Material secince Vol. 16, August 2009,pp 229-236

18. Drtina, P. and Sallaberger, M., "Hydraulic turbines—basic principles and state-of-theart computational fluid dynamics applications" Proc Instn Mech Engrs Vol 213 Part C 1999
19. Wu, J., Shimmei, K., Tani, K., Niikura, K. & Sato, J., "CFD-Based design optimization for hydro turbines" Journal of Fluid Enginnering, Feb 2007 Vol. 129,pp159-168
20. Thakker, A., Frawley, P., & Abugihalia,Y., "Experimental and CFD analysis of 0.6m Impulse turbine with fixed guide vanes" The International Society of Offshore and polar Engineers ISBN 1-1880653-52-4(Vol.1)
21. Lipej, A., "Optimization method for the design of axial hydraulic turbines" Journal of Power and Energy 2004 218: 43
22. Thum, S., & Schilling, R., "Optimization of Hydraulic Machinery Bladings by Multilevel CFD Techniques" International Journal of Rotating Machinery 2005:2, 161–167
23. Viscanti, N., Pesatori, E. & Turozzi, G., "Improvement of a Francis runner design" 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, October 14-16, 2009, Brno, Czech Republic
24. Liu, S., Wu, S., Nishi, M. & Wu, Y., "Flow Simulation and Performance Prediction of a Kaplan Turbine" The 4th International Symposium on Fluid Machinery and Fluid Engineering November 24-27, 2008, Beijing, China
25. Kim, Y. T., Nam, S. H., Cho, Y. J., Hwang, Y., C., Choi,Y. D., Nam, C. D. & Lee, Y. H., "Tubular-Type hydro turbine performance for variable guide vane opening by CFD" The fifth international conference on fluid mechanics, Aug 15-19,2007, Shanghai, Chaina
26. Choi, Y. D., Kim, C. D., Kim, Y. T., Song, J. I. & Lee, Y., "A performance study on a direct drive hydro turbine for wave energy converter" Journal of Mechanical Science and Technology 24 (11) (2010), pp 2197-2206
27. Khare, R., Prasad, V. & Kumar, S., "Derivation of Global Parametric Performance of Mixed Flow Hydraulic Turbine Using CFD" Hydro Nepal issue no. 7 July, 2010
28. Carija, Z. & Mrsa, Z., "Complete Francis turbine flow simulation for the whole range of discharges" 4th International Congress of Croatian Society of Mechanics September, 18-20, 2003 Bizovac, Croatia

29. Nicolle, J, Labbe, P, Gauthier, G & Lussier, M "Impact of blade geometry differences for the CFD performance analysis of existing turbines" Earth and Environmental Science 12 (2010) 012028
30. Kyriacou, S. A., Weissenberger, A., Grafenberger, P. & Giannakoglou, K. C., "Optimization of hydraulic machinery by exploiting previous successful designs" Earth and Environmental Science 12 (2010) 012031
31. Choi, Y. D., Yoon, H. Y., Inagaki, M. S., Ooike, S., Kim, Y. J. & Lee, Y. H., "Performance improvement of a cross-flow hydro turbine by air layer effect" Earth and Environmental Science 12 (2010) 012030
32. Cherny, S., Chirkov, D., Lapin, V., Lobareva, I., Sharov, S. and Skorospelov, V., "3D Euler flow simulation in hydro turbines unsteady analysis and automatic design" Russian Foundation for Basic Research(project no. 04-01-00246) and Integration Basic Research Programme No 27 of the SB RAS
33. Khare, R., Prasad, V. & Kumar, S., "CFD approach for flow characteristics of hydraulic Francis turbine" International Journal of Engineering Science and Technology Vol. 2(8), 2010, 3824-3831
34. Ying, H., Heming, C., Jil, H. & Xirong, L., "Numerical simulation of unsteady turbulent flow through a Francis turbine" Wuhan University Journal of Natural Sciences 2011, Vol.16 No.2, pp 179-184
35. Lain, S., Garcia, M., Quinterol, B. & Orrego, S., "CFD Numerical simulations of Francis turbines" Rev. Fac. Ing. Univ. Antioquia N. ° 51 pp. 24-33. Febrero, 2010
36. Choi, Y. D., Lim, J. I., Kim Y. T. & Lee, Y. H. "Performance and internal characteristic of a cross flow hydro turbine by the shape of nozzle and runner blade" Journal of Fluid Science and Technology Vol. 3,No. 3,2008
37. Xiao, H. and Yu, B. "3D-Viscous Flow Simulation and Performance Prediction of a Complete Model Francis Turbine" In: Proc. Of International Conference on Mechanic Automation and Control Engineering (MACE), 2010, pp. 3942 – 3944.
38. Haurissa, J. and Soenoko, J. "Performance and flow characteristics Latitude of nozzle in turbine blades second level" Journal of Economics and Engineering, ISSN: 2078-0346, No.4, December, 2010.
39. Choi, Y. D., Lim, J. I., Kim, C. G., Kim, Y. T. & Lee, Y. H. "CFD analysis for the performance of cross flow hydraulic turbine with variation of blade angle" In Proc. of Fifth International Conference on Fluid Mechanics, Aug 15-19 2007, Shanghai, Chaina.

40. Vn, T. C., Koller, M., Gauthier, M. & Deschenes, C., "Flow simulation and efficiency hill chart prediction for a Propeller turbine" In Proc. Of Earth and Environmental science 12(2010)012040.
41. Keck, H. & Sick, M., "Thirty years of numerical flow simulation in hydraulic turbomachines" Acta Mechanica, 2008, Vol. 201, No. 1-4, pp 211-229.
42. Daneshkah, K. and Zangeneh, M., "Parametric design of a Francis turbine runner by means of a three-dimensional inverse design method" In Proc. Of Earth and Environmental science 12(2010)012058.
43. Wu, Y., Liu, S., Wu, X., Dou, H., Zhang, L. & Tao, X., "Turbulent flow computation through a model Francis turbine and its performance prediction" In Proc. Of Earth and Environmental science 12(2010)012004.
44. Lipej, A. & Poloni, C., "Design of Kaplan runner using multiobjective genetic algorithm optimization" Journal of Hydraulic Research. VOL. 38, 2000, NO. 1,pp 73-79.
45. ShuHong, L., Jie, S., ShangFeng, W. & YuLin, W., "Numerical simulation of pressure fluctuation in Kaplan turbine Science in China Series E: Technological Sciences, Aug. 2008, vol. 51,no. 8,pp 1137-1148.
46. Nilsson, H. & Davidson, L., "A Numerical Comparison of Four Operating Conditions in a Kaplan Water Turbine, Focusing on Tip Clearance Flow" In Proc. of the 20th IAHR Symposium, August 6-9 2000, Charlotte, North Carolina.
47. Gagnon, J.M. & Deschenes, C., "Numerical Simulation of a Rotor-Stator Unsteady Interaction in a Propeller Turbine"
48. Grafenberger, P., Parkinson, E., Georgopoulou, H.A., Kyriacou, S.A. & Giannakoglou, K.C., "Constrained Multi-Objective Design Optimization of Hydraulic Components Using a Hierarchical Metamodel Assisted Evolutionary Algorithm. Part 2: Applications" In Proc. of 24th Symposium on Hydraulic Machinery and Systems
49. Prasad, V., Khare, R. and Chincholikar, A., "Hydraulic Performance Of Elbow Draft Tube For Different Geometric Configurations Using CFD" IGHEM-2010, oct. 21-23, AHEC, IIT Roorkee, India.
50. Layman's Guide Book, on "How to develop a small hydro site", European Small Hydropower Association (ESHA) (1998), pp. 25-40
51. http://www.zregdansk.pl/index_en.php?n=81

52. Flaspohler, T., "Design of Kaplan runner turbine for small hydroelectric power Plants" Phd. Thesis, Tampere University of applied science 2007.
53. IS 12800 (Part-I), "Guidelines for turbine, preliminary dimensions and layout of surface hydro-electric power house" 1993
54. Jain, S., "CFD based flow analysis of hilly small hydropower station" M.tech. Dissertation AHEC IIT Roorkee, 2007
55. www.cadlab.tuc.gr/courses/cad/wf2tutorials/wf2%20tutorial.pdf
56. www.ossberger.de
57. http://hpce.iitm.ac.in/website//Manuals/Fluent_6.3/fluent6.3/help/pdf/tg/pdf.htm

LIST OF PUBLICATION

[1] Mishra, A., Saini, R.P. & Singhal, M.K. "CFD based performance analysis of Kaplan turbine for micro hydro power", International Conference on Mechanical and Industrial Engineering (ICMIE – 2012), Goa, 16 June, 2012.