PERFORMANCE EVALUATION OF HYDRO TURBINE USING CFD ANALYSIS

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

ANIL KUMAR

ALTERNATE HYDRO ENERGY CENTRE
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE -247 667 (INDIA)
DECEMBER, 2007

CANDIDATE'S DECLARATION

I hereby declare that the work presented in the dissertation entitled **"PERFORMANCE EVALUATION OF HYDRO TURBINE USING CFD ANALYSIS"** submitted in partial fulfillment of the requirements for the award of degree of **Master of Technology in Alternate Hydro Energy System** in the Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee, is an authentic record of my own work carried out from July 2006 to Dec 2007 under the guidance and supervision of Dr. R.P.Saini, Associate Professor, Alternate Hydro Energy Centre and Shri. Arun Kumar, Head, Alternate Hydro Energy Centre.

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

Date: December31, 2007
Place: Roorkee

Anil Kumar
(ANIL KUMAR)

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

(Dr. R.P.Saini)
Associate Professor
Alternate Hydro Energy Centre,
Indian Institute of Technology, Roorkee

(Arun Kumar)
Head
Alternate Hydro Energy Centre,
Indian Institute of Technology, Roorkee

ACKNOWLEDGEMENT

I heartily like to acknowledge my sincere gratitude and indebtedness to **Dr. R. P. Saini** Associate Professor and **Sri Arun Kumar** Head, Alternate Hydro Energy Centre, I.I.T. Roorkee for the precious guidance and kind information, continuous help and the affectionate treatment.

Moreover, I wish to express my profound gratitude again to **Sri Arun Kumar**, Head, Alternate Hydro Energy Centre, Indian Institute of Technology, Roorkee for providing all the facilities, which have made it possible for me to complete this report. The cooperation they gave is greatly appreciated and also same gratitude to **Dr. R. P. Saini** for very informative guidance. At last, I wish to express my sincere gratitude to Staff of **Corporate R & D, BHEL, Hyderabad** for software support and guidance provided during course of work.

Finally, my sincere regards to friends and staff at the department who have directly and indirectly helped me in completing this Report.

Anil Kumar

Date: December, 2007 **(ANIL KUMAR)**

ABSTRACT

The present work is aimed to carry out the Performance evaluation of typical medium head Francis Turbine model using Computational fluid Dynamics (CFD) Analysis. The analysis is conducted to check the hydrodynamic performance of the runner. For this purpose Complete Francis turbine comprising spiral casing, stay vane, guide vane, runner and draft tube is modeled with the aid of CAD drawings. The model thus constituted resembles a numerical test rig for turbine performance evaluation. The next step involved is to generate a computational grid. The software used for this purpose is CFX-TURBOGRID. The solutions are obtained by proper specifications of the boundary conditions. Computational fluid Dynamics (CFD) simulation of turbine is carried out using TASC flow package and the results of analysis for 54 operating points with changing Guide vane opening, varying mass flow/turbine rpm, are compiled and optimum operating regime is identified. I have been performed the work of modeling, meshing and analysis in Corporate R & D, BHEL, Hyderabad .The results provided by the analysis are compared with experimental value. Peak efficiency for each guide vane opening is found out and highest efficiency around 94.00% is obtained.

CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1
INTRODUCTION AND LITERATURE REVIEW

1.1 INTRODUCTION

Energy is critical not only for economic growth but also for social development. The energy demand is growing day by day. In order to meet the increased in demand all the sources of the energy are required to exploit. Indian per capita consumption of electricity continues to be around 520 kWh per annum. Due to the rapid depletion of the fossils fuels we need to look, for other source of energy specially the renewable energy sources. Hydro represents non-consumptive, non-radioactive, and non-polluting use of water resources for free energy requirements.

India is blessed with many rivers, natural streams, cannel networks and mountains offering tremendous hydro potential of major, small, mini and micro hydropower. Among all the renewable energy sources available, small hydropower is considered as the most promising proven and cost effective source of the energy.

India has a history of 100 years of Small Hydro since its first installation of 130 kW at Darjeeling in the year 1897.India has one of the world's largest Irrigation Canal networks with thousands of Dams and Barrages. It has monsoon fed, double monsoon fed as well as snow fed rivers and streams with perennial flows. An identified potential of more than 10,000 MW [1] of small hydro exists in India, though overall potential of 15,000 MW [1] is anticipated. The installed capacity as on 31.3.2005 is 1705.23 MW [1] with an additional 479.29 MW [1] under construction. Worldwide installed capacity of small hydro today is around 50,000 MW [1] against an estimated Potential of 180,000 MW [1].

1.2 SMALL HYDRO POWER

Small hydro power development has been taking place steadily since a very long time, where encouraging factors existed. This development has mostly taken place in remote mountainous areas to meet the local energy demand and in the plains, through development of canal drops wherever found attractive. The cost of low head/small hydro-electric schemes depend largely on the civil engineering works, like dam, water conductor system and power house, etc. The sitting of low head hydro-electric schemes on large irrigation canal offer a good scope to utilize the falls on the canal for generation of electricity more economically. In world all over the world there is a general tendency to define small hydropower with power output. Different countries follow different definitions for small hydro. As a matter of fact, small hydro power (SHP) cannot be defined in terms of power output or physical size of the power station, due to the influence of head on the scheme. A definition is needed which combines both the power output and physical size factors to define the small hydro projects. In the Indian context, small power projects costing less than or equal to Rs 25000 million does not require Central Electricity Authority (CEA) clearance. Environmental Clearance is not required for SHP project below and up to 25 MW

Central Electricity Authority (CEA), Government of India and Bureau of Indian Standard (12800: Part III) classified Small hydropower schemes as follows [1]

Table 1.1 Various Capacity of Small Hydro Power [1]

Type	Station capacity	Unit rating
Micro	Up to 100 kW	Up to 100 kW
Mini	101 kW to 2000 kW	101 kW to 1000 kW
Small	2001 kW to 25000 kW	1001 kW to 5000 kW

1.2.1 Different Types of SHP Schemes

Small hydropower can also be broadly categorized in three types as follows

1.2.2 Run-of-River Schemes

Run-of-River hydroelectric schemes are those, in which water is diverted from stream without creating any storage in the river. In such schemes, power is generated from flowing water and available head. The output of a run-of-river plant is subjected to the instantaneous flow of the stream.

1.2.3 Canal Based Schemes

Canal based small hydropower scheme is planned to generate power by utilizing the fall in the canal. These schemes may be planned in the canal itself or in the bye pass channel.

1.2.4 Dam Toe Based Schemes

Dam Toe Based schemes are those in which water is stored in a large reservoir. Level of the water in the dam decides the head available for power generation. This scheme is very reliable for power generation across the year. Basically head available at irrigation dams are used in power generation.

1.3 HYDRAULIC TURBINES

The purpose of a hydraulic turbine is to transform the water potential energy to mechanical rotational energy. The various types of turbines used in small hydro power schemes are discussed as follow.

1.3.1 Pelton Turbine

The water pressure is converted into kinetic energy before entering the runner. The kinetic energy is in the form of a high-speed jet that strikes the buckets, mounted on the periphery of the runner. Turbines that operated in this way are called impulse turbines. The most usual impulse turbine is the Pelton. The modern Pelton runner is a tangential turbine with discharge buckets as shown in Fig. 1.1 in Pelton runner one or more jets of water impinge on a wheel containing many hemispherical curved buckets. It has a high efficiency of about 88% at the rated output and can maintain same efficiency at part load conditions. It is simple in design and cheap in construction making it more suitable for micro hydro of high head. Movement of water in the runner passage takes place with a free surface contacting the ambient air, which implies that the energy available is extracted from the flow at atmospheric pressure. There is no pressure change across the runner blades. The work is done on the runner by the fluid due to the change in angular momentum and the motion of the runner blades. Pelton wheels typically have more than the one nozzle and some have 'throttleable' nozzles for power control. Pelton turbines are not used at lower heads because their rotational speed becomes very low. If the runner size and low speed do not pose a problem for particular installation, then pelton turbine can be used efficiently with fairly low heads.

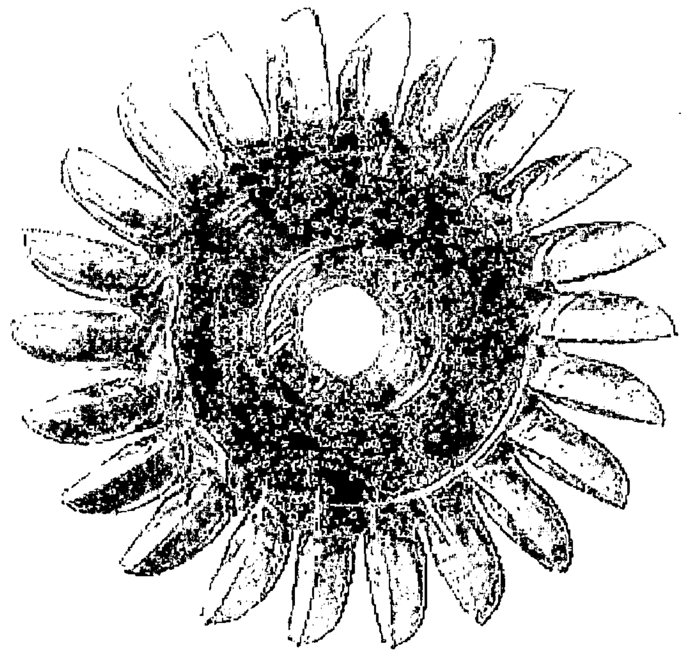


Fig. 1.1 A Pelton Runner [2]

1.3.2 Turgo Impulse Turbine

The efficiency of turgo impulse turbine is lower than for the pelton and francis turbines. Turgo impulse turbine has a higher rotational speed for the same flow and head as Compared to the pelton. Construction of turgo turbine is an impulse machine similar to the Pelton but is designed to have a higher specific speed. In this turbine the jet aims to strike the plane of the runner on one side and exist on the other. Therefore, the flow rate is not limited by the discharged fluid interfering with the incoming jet. As a consequence, a turgo turbine can have a smaller diameter runner than the pelton for an equivalent power. With smaller faster spinning runners, it is more likely to be possible to connect turgo turbines directly to the generators rather than having to go via a costly speed increasing transmission. Water impingement on a typical turgo runner is shown in Fig 1.2. turgo impulse is efficient over wide range of speeds and shares the general characteristics of impulse turbines listed for pelton.

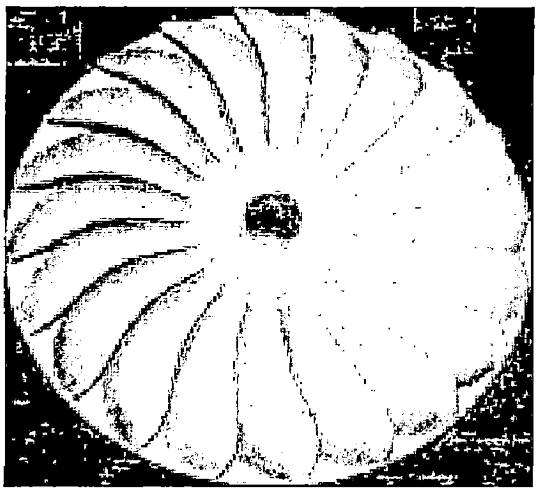


Fig. 1.2 A Turgo Impulse Turbine Runner [3]

1.3.3 Cross Flow Turbines

Cross Flow Turbines are also called 'Michell-Banki' Turbines. It has a drum shaped runner consisting of two parallel discs connected together near the rims by a series of curved blades. A Cross Flow Turbine always has its runner shaft horizontal. It can operate with heads of 5 to 200m.A typical Cross Flow Turbine is shown in fig. 1.3 bellow. One of the simplest and most efficient types of water turbine for small-scale use is the Australian Michell or 'Banki' turbine or Cross Flow Turbines. In the operation of a Cross Flow Turbine a rectangular nozzle directs the jet onto the full length of the runner. The water strikes the blades and imparts most of its kinetic energy (2/3). It then passes through the runner and strikes the blade again on exit, impacting the remaining energy (1/3) before leaving the turbine. At low flows, the water can be channeled through either two- thirds or one third of the runner, thereby sustaining relatively high turbine efficiency.

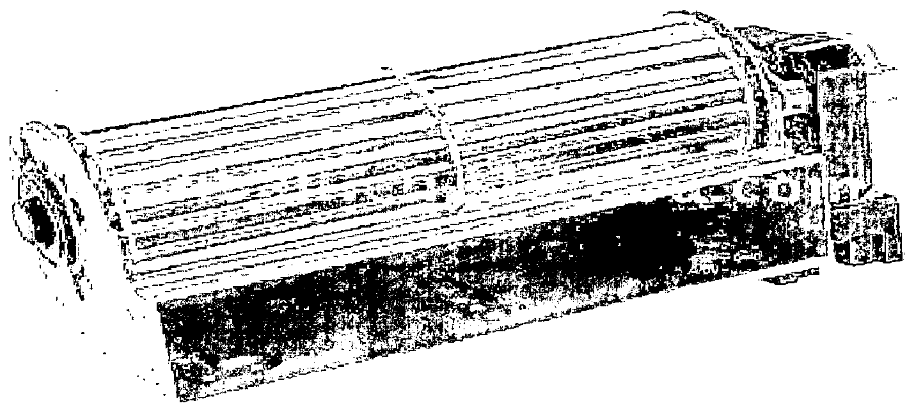


Fig. 1.3 A Cross Flow Turbine Runner [3]

1.3.4 Francis Turbines

Francis turbines are reaction turbines, with fixed runner blades and adjustable guide vanes, used for medium heads. In this turbine the admission is always radial but the outlet is axial. Fig 1.4 shows a horizontal axis Francis Turbine. Their usual field of application is from 25 to 350 m head. Francis turbine can be either volute cased or open flume machines. The spiral casing is tapered to distribute water uniformly around the entire parameter of the runner. The guide vanes feed the water into the runner at the correct angle. The Francis Turbine is generally fitted with adjustable guide vanes. These regulate the water flow as it enters the runner and are usually linked to the governing system which matches the flow to turbine loading in the same way as a spear valve or a deflector plate in pelton turbine. Generally it has an efficiency of about 90% at the rated output.

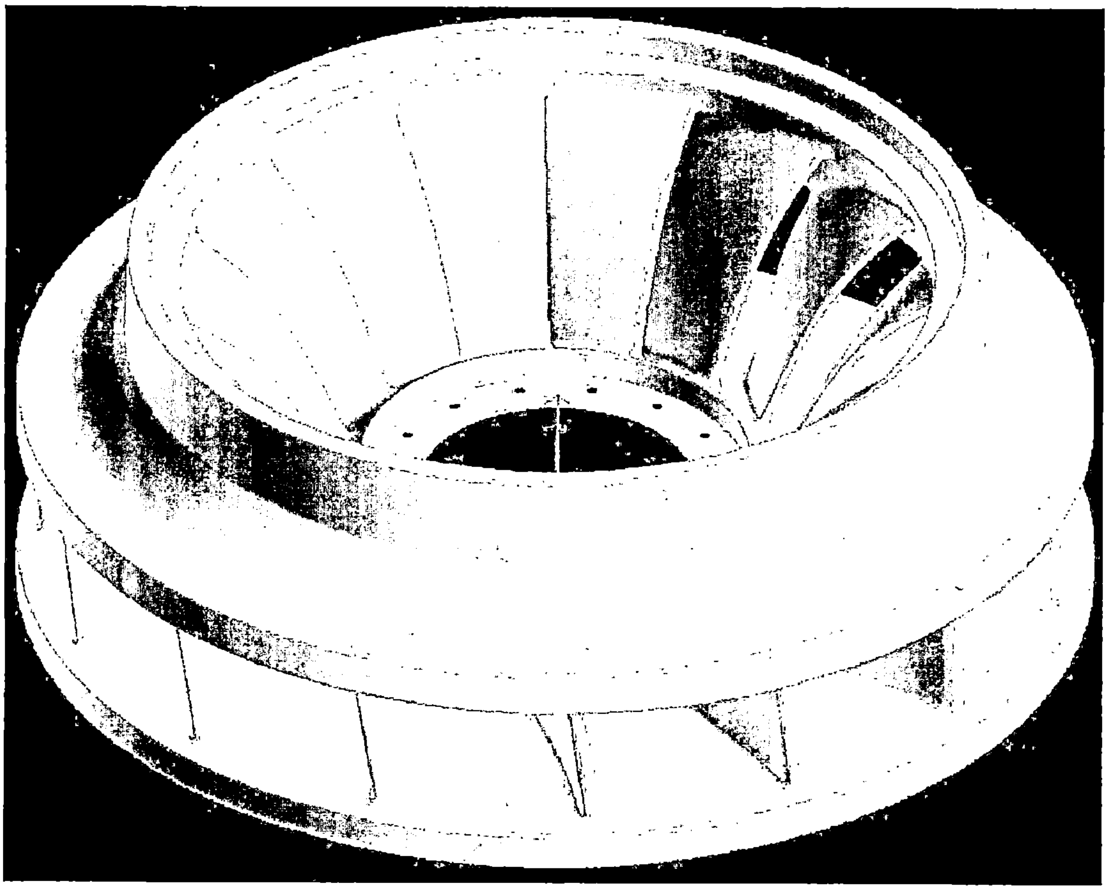


Fig. 1.4 A Francis Turbine Runner [3]

1.3.3 Kaplan And Propeller Turbines

Kaplan and propeller turbines are axial-flow reaction turbines; general used for low heads from 2 to 40 m. the Kaplan turbine has adjustable runner blades and may or may not have adjustable guide-vanes. If both blades and guide-vanes are adjustable it is described as "double-regulated'. If guide-vanes are fixed it is "single-regulated". Generally it has an efficiency of about 93% at the rated output. Fixed runner blade Kaplan turbines are called propeller turbines. The propeller turbine, which can be used in most of the axial flow turbine configurations. Unfortunately, this simple fixed runner blade, fixed guide vane turbine is only of a limited use, as its efficiency falls off sharply from its nominal operating conditions. However, most of the disadvantages of the simple propeller turbine can be overcome by the use of adjustable blades. The basic propeller turbine consists of a propeller fitted inside the continuation of the penstock tube. The turbine shaft passes out of the tube at the point where the tube changes the direction. The propeller has usually three to six blades and the water flow is regulated by the static blades or swivel gates.

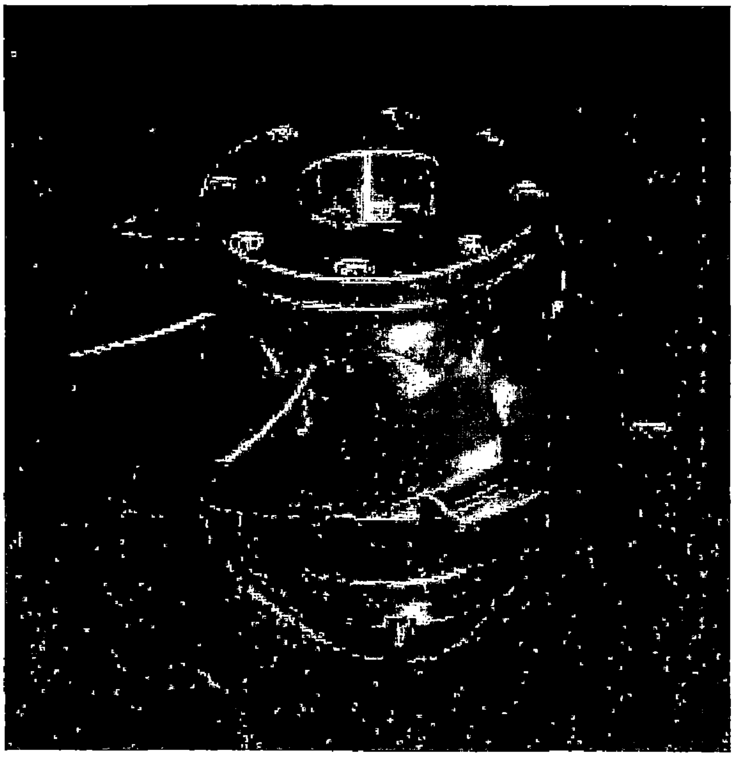


Fig. 1.5 A Kaplan Turbine Runner [3]

Turbine application ranges in general are given below in Table 1.2.

Table 1.2 Various Hydro Turbines for Small Hydro Power

Turbines used	Head	Specific speed
Pelton	High (180-350)	10-40
Francis	Medium (20-180)	40-250
Kaplan, semi Kaplan	Low (5-20)	300-1000
Bulb turbines, Tubular turbines	Ultra low (1-5)	200-300

The types of machines in low-head schemes are generally 'S' type tubular turbines (Kaplan or Semi-Kaplan) with generator outside the water passage. Bulb turbines are horizantal units which have propeller runners generally directly connected to the generator. The generator is enclosed in water tight enclosure (bulb) located in the turbine water passageway. This is available with fixed or variable pitch blades and with or without a wicket gates. Francis and impulse turbines are preferred in medium and high head schemes. Micro Pelton and Turgo impulse is seen in high head as well as micro hydro sets. Orientation of the turbine is chosen mainly based on the space availability, size, maintainability, and civil works considerations.

1.4 PERFORMANCE EVALUATIONS

Water at standard temperature & pressure is the working medium. All inputs are given in SI units (i.e., length in meters, mass in kg, time in seconds).The fluid is assumed to be incompressible.

Formulae used in calculating important performance parameters:

Efficiency $\eta = P / H \times m \times g$

Power P = T x ω

Net Head H = Total head at inlet- Total head at outlet.

Unit Dis charge $Q_{11} = Q \times 1000 / D^2 \sqrt{H}$

Unit Speed N_{11}: $N \times D / \sqrt{H}$

where,

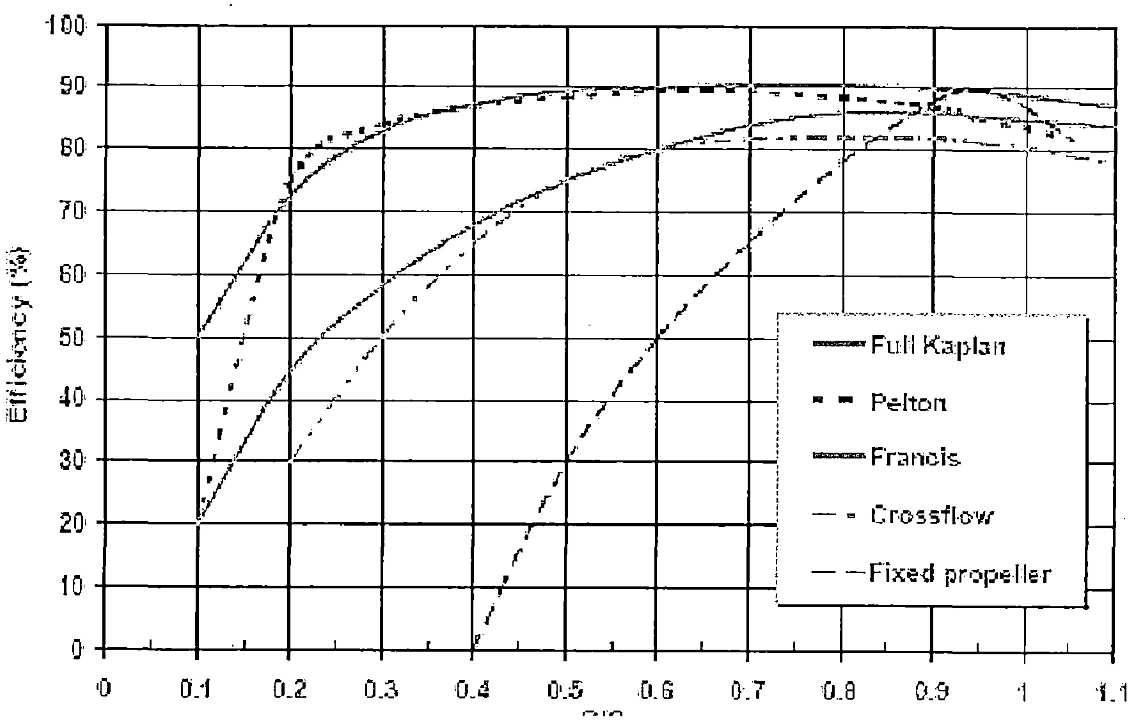
H: Head, Q: Discharge, D: Runner Dia.

The performance characteristics represent in a graphical form the relationships between the variables relevant to hydraulic machines. Each and every hydraulic machine has its own set of characteristics which represent its performance. In case of turbines, the output is power developed at a given speed and the fundamental turbine characteristic consists of plot of power against speed at constant head.

1.5 EFFICIENCY OF TURBINES

The relative efficiencies of different turbines can be compared at

- Design point
- At reduced flow rate (Part Load)



Q/Q0 (Unit Discharge)

Fig.1.6.Efficiencies Comparison of Different Turbines at Reduced Flow Rates [2]

From the efficiencies comparison chart, it is evident that Pelton and Kaplan turbines retain their high efficiencies when running below design flow, but the efficiency of cross flow and Francis turbines falls sharply as they are operated below half of their design flow.

1.6 TURBINE SELECTION

The purpose of all turbines is to convert the energy from the falling water into the rotating shaft power. This is one of the most efficient ways of getting energy from water. To select a turbine for a specific situation is a great challenge as it will directly affect the total power generate from that scheme.

1.6.1 Selection Criteria of Turbines

The selection of turbines depends upon following factors.

- Site characteristics
- Head of the hydro scheme
- Flow rate available in the scheme
- Desired runner speed of the generator
- The probability of operating the turbine at reduced flow rates

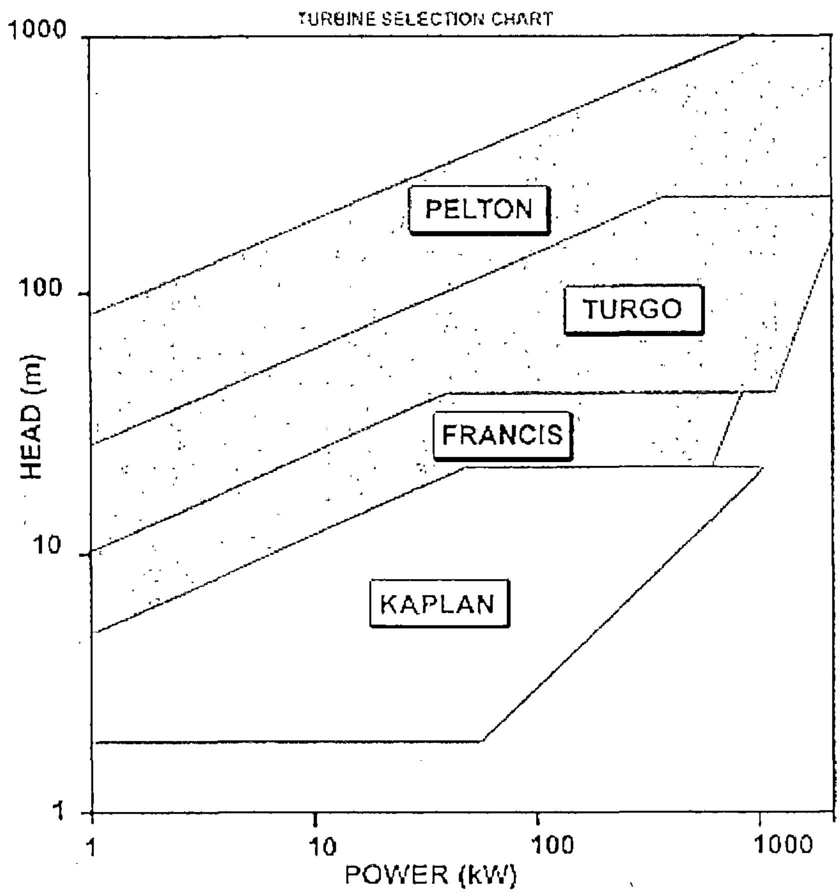


Fig. 1.7 Turbine Selection Chart [3]

1.7 COMPUTATIONAL FLUID DYNAMICS

1.7.1 Scope of CFD

Computational Fluid Dynamics (CFD) is a branch of Fluid Mechanics that resolves fluid flow problems numerically. The physical laws governing a fluid flow problem are represented by a system of partial differential equations including the continuity equation, the Navier-Stokes equations. The numerical analysis resolves the equations by accurate and complex numerical schemes.

It nicely and synergistically complements the other two approaches of pure theory and pure experiment, but it will never replace either of these approaches. There will always be a need for theory and experiment. The future advancement of fluid dynamics will rest upon a proper balance of all three approaches, with computational fluid dynamics helping to interpret and understand the results of theory and experiment, and vice versa . The Fig.1.4 shows the three dimensions in fluid dynamics.

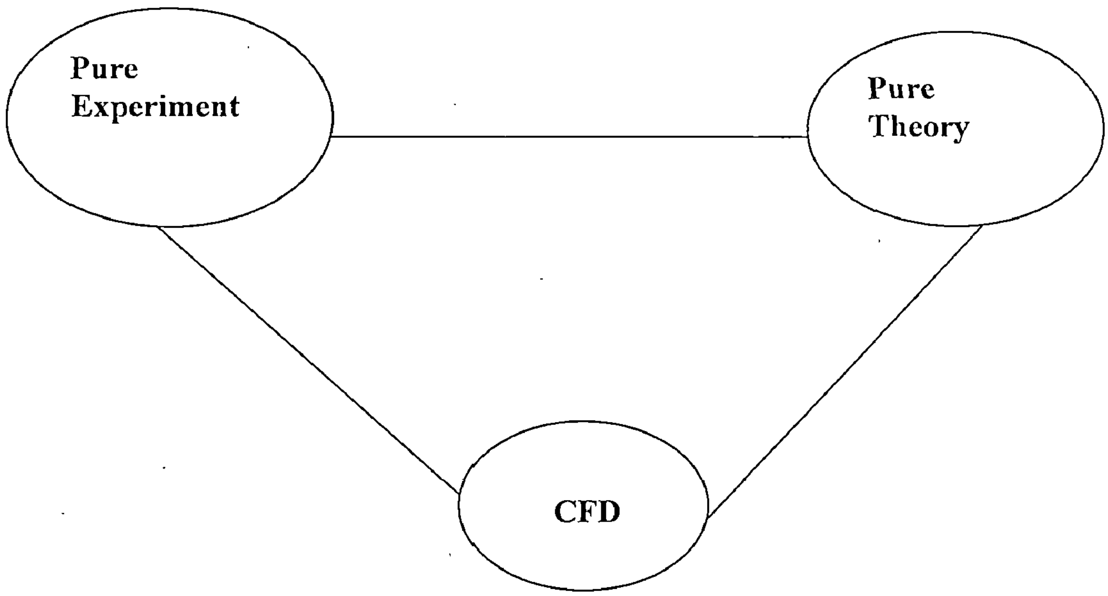


Fig.1.8 Three Dimensions of Fluid Mechanics

Computational fluid dynamics as an engineering tool for simulating viscous 3D flows in various machines is gaining wide acceptance among the design of power plant equipment like turbines. In hydro turbines, the blades are the key component to convert potential energy of water into kinetic energy which is used to drive the generator to produce electric power. Therefore, focused research work is going on at great pace for development of improved blade profile through extensive use of CFD tools with limited experiments. In CFD flow field is calculated numerically by means of color visualizations, quantitative analysis a view of the flow field and better understanding of the fluid mechanics can be obtained. This technique also enables a detailed comparison to experiments to attain an even better understanding of the flow field. The method has become an essential part of design, testing and optimizing process.

1.7.2 Uses of Computational Fluid Dynamics (CFD)

a) Design overview: parts of the system can be visualized by CFD analysis which is sometimes not possible by any other mean to know what phenomenon is happening inside the system.
b) Initial predictions: CFD may be applied to predict how a design will perform and test many variations to reach an optimal result.
c) Efficiency: time and money are saved with shorter design cycle due to better and faster analysis.

1.8 LITERATURE REVIEW

- P Drtina & M Sallaberger Sulzer Hydro Zürich,Switzerland "Hydraulic Turbines—Basic Principles and State-of-the-art Computational Fluid Dynamics Applications". This paper discusses the basic principles of hydraulic turbines, with special emphasis on the use of computational fluid dynamics (CFD) as a tool which is being increasingly applied to gain insight into the complex three-dimensional (3d) phenomena occurring in these types of fluid machinery. The basic fluid mechanics is briefly treated for the three main types of hydraulic turbine: Pelton, Francis and axial turbines. From the vast number of applications where CFD has proven to be an important help to the design engineer, two examples have been chosen for a detailed discussion. The first example gives a comparison of experimental data and 3d euler and 3d navier-stokes results for the flow in a Francis runner. The second example highlights the state-of-the-art of predicting the performance of an entire Francis turbine by means of numerical simulation.

- A Lipej Turboinštitut Rovšnikova ,Ljubljana, Slovenia "Optimization Method for the Design of Axial Hydraulic Turbines" Computational fluid dynamics (CFD) is becoming an increasingly reliable tool for the design of water turbines. Using different CFD codes, it is possible to find out and compare criteria for classifying runner blade geometry regarding the strengths of their characteristics. The final decision of runner geometry, with demanding energetic and cavitation characteristics, always remains for the design engineer. To reach the final result, the engineer has to compare the flow analysis results of a great number of different geometries. To replace a part of this work, an optimization algorithm has been developed. This optimization procedure helps to check many more geometries with less human work. In this paper, a multi objective genetic algorithm for the design of axial runners is presented. For the flow analysis, the CFX-TASC flow code, this makes it possible to start the optimization procedure with a relatively high level of efficiency and transforms prescribed genetic parameters to the runner geometry.

- Ajit Thakker, Fergal Hourigan "Computational Fluid Dynamics Analysis of a 0.6 m, 0.6 hub-to-tip Ratio Impulse Turbine With Fixed Guide Vanes" This paper presents the comparison of a three-dimensional Computational Fluid Dynamics (CFD) analysis with empirical performance data of a 0.6 m Impulse Turbine with Fixed Guide Vanes . Pro-Engineer, Gambit and Fluent 6 were used to create a 3-D model of the turbine. A hybrid meshing scheme was used with hexahedral cells in the near blade region and tetrahedral and pyramid cells in the rest of the domain. The turbine has a hub-to-tip ratio of 0.6 and results were obtained over a wide range of flow coefficients. Satisfactory agreement was obtained with experimental results. The model yielded a maximum efficiency of approximately 54% as compared to a maximum efficiency of around 49% from experiment. A degree of insight into flow behavior, not possible with experiment, was obtained. Sizeable areas of separation on the pressure side of the rotor blade were identified toward the tip. The aim of the work is to benchmark the CFD results with experimental data and to investigate the performance of the turbine using CFD and to with a view to integrating CFD into the design process.

- Cherny S.G,Sharov S.V, SkorospelovV.A, Turuk P.A."Methods for Three-Dimensional Flows Computation in Hydraulic Turbines". This consider the complex problem of numerically simulating a flow in the fluid passage of a hydraulic turbine that consists of a spiral casing, a cascade of stay vanes and guide apparatus vanes, a runner, and a draft tube. Methods of its solution are based on solution algorithms of the Euler equations and the Reynolds equations, as well as on tools of the geometric support of simulation used for the development of geometric models of turbine components, grids calculation in them, an interchange of flow parameters between fluid passage segments and flow visualization in them. We give the results of the flow calculations in the fluid passage of a real hydraulic turbine.

- Guoyi Peng , Shuliang Cao , Masaru Ishizuka , Shinji Hayama . "Design optimization of axial flow hydraulic turbine runner: Part II - multi-objective constrained optimization method" This paper is concerned with the design optimization of axial flow hydraulic turbine runner blade geometry. In order to obtain a better design plan with good performance, a new comprehensive performance optimization procedure has been presented by combining a multi-variable multi-objective constrained optimization model with a Q3D inverse computation and a performance prediction procedure. With careful analysis of the inverse design of axial hydraulic turbine runner, the total hydraulic loss and the cavitations coefficient are taken as optimization objectives and a comprehensive objective function is defined using the weight factors. Parameters of a newly proposed blade bound circulation distribution function and parameters describing positions of blade leading and training.

- T. Behr J. Schlienger A. I. Kalfas R. S. Abhari "Fluid Dynamics and Performance of Partially and Fully Shrouded Axial Turbines". This paper illustrated a unique comparative experimental and numerical investigation carried out on two test cases with shroud configurations, differing only in the labyrinth seal path. The blade geometry and tip clearance are identical in the two test cases. The geometries under investigation are representative of an axial turbine with a full and partial shroud, respectively. Global performance and flow field data were acquired and analyzed. Computational simulations were carried out to complement the investigation and to facilitate the analysis of the steady and unsteady flow measurements. A detailed comparison between the two test cases is presented in terms of flow field analysis and performance evaluation. The present analysis has shown that an integrated and matched blade-shroud aerodynamic design has to be adopted to reach optimal performances.

- J D Denton and W N Dawes "Computational fluid dynamics for turbo machinery design". This paper presents a review of the main CFD methods in use, discusses their advantages and limitations and points out where further developments are required. The paper is concerned with the application of CFD and does not describe the numerical methods or turbulence modeling in any detail .Computational fluid dynamics (CFD) probably plays a greater part in the aerodynamic design of turbo machinery than it does in any other engineering application. For many years the design of a modern turbine or compressor has been unthinkable without the help of CFD and this dependence has increased as more of the flow becomes amenable to numerical prediction. The benefits of CFD range from shorter design cycles to better performance and reduced costs and weight.

1.9 OBJECTIVE OF PRESENT STUDY

In order to achieve maximum power output from the hydro turbines, a three dimensional Computational Fluid Dynamics (CFD) models to simulate the flow through the turbines for performance analysis, without performing actual model test, and improving the design of hydro turbines for medium head projects has been attempted. The Hydropower industry is faced today challenge in a cost effective manner of development, evaluation and implementation of innovative concepts for hydro turbines a more general and economic approach is to employ three dimensional computational fluid dynamics (CFD) models to simulate the flow through the turbines for performance analysis without performing actual model test.

CHAPTER 2
COMPUTATIONAL FLUID DYNAMICS

2.1 GENERAL

CFD is the art of replacing the integral or the partial derivative of the following fundamental laws equations:

a) Mass is conserved.
b) Newtons second law.
c) Energy is conserved.

Above three laws gives the fundamental governing equation for CFD. The partial derivative or the integral of these equations discretize into algebraic forms, which in turn are solved to obtain number for the flow field values at discrete points in time and/or space. The end product of CFD is indeed a collection of numbers, in contrast to a closed form analytical solution.

2.2 RELEVANCE OF CFD TO HYDRO TURBINES

The application of CFD in the field of Hydro turbines is probably one of the earliest, and the main reason for this may be attributed to the following:

- The flow phenomenon in terms of physics in Hydro turbines is compressible, unsteady and turbulent and the geometry is complex with the flow being highly 3-dimentioanl.
- Hydro turbines are not standardized and need specific tailor made design to suit specific site and customer practices.
- Hydro turbines already operate at high efficiencies and any improvements in these need very thorough understanding of the flow and loss mechanisms.
- There is enormous experimental data available, in that, model testing is part of the design cycle and thesis can be used to validate CFD results.
- CFD techniques are now being regularly applied in the development of new blade profiles to cut down the experimental cycle.

2.3 BASIC ASPECTS OF CFD

a) Governing equations

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is

$$\frac{\partial \rho}{\partial t} + \nabla .(\rho V) = 0 \qquad (2.1)$$

Conservation of momentum equation is

$$\rho \frac{\partial V}{\partial t} + \rho(V.\nabla)V = -\nabla p + \rho.g + \nabla.\ \tau_{ij} \qquad (2.2)$$

Where,

Velocity of fluid, m/sec	=	V
Time factor, seconds	=	ςij
Pressure,N/m sq.	=	p
Density of fluid, kg/m^3	=	ρ

These equations along with the conservation of energy equation form a set of coupled, nonlinear partial deferential equations. It is not possible to solve these equations analytically for most engineering problems.

It is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD). The strategy of CFD is to replace the continuous problem domain with a discrete domain using a grid. In the continuous domain, each flow variable is defined at every point in the domain.

b) Dicretization. The solution of flow which is governed by the Naviers- stokes equation or as the case may be, the eulers equation and these are partial differential equations. The partial derivatives in these equations are replaced by approximate algebraic difference quotients, where these algebraic difference quotients are expressed strictly in terms of the flow field variable at two or more of the discrete grid points. These algebraic equations can be solved for the values of flow field variables at discrete grid points only. This method is called the method of difference.

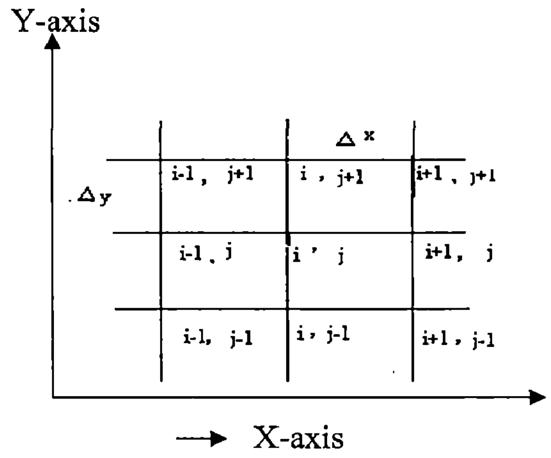


Fig 2.1 Discrete Grid Points

d) Explicit & Impicit Approaches:

i) In explicit method the each difference equation contains only one unknown and therefore can be solved explicity for this unknown in a straight forward manner.

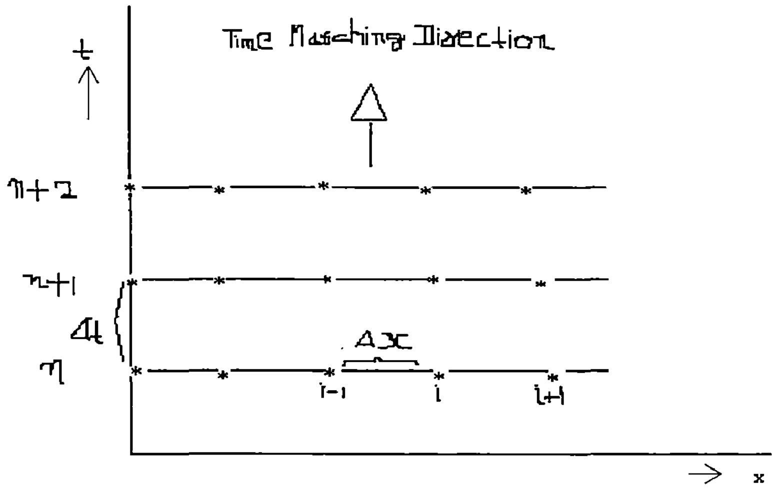


Fig 2.2 Time Marching

Assume T is known at all grid points at time level n. according to time marching T at all grid point at time level n+1 are calculated from the known values at time level n. similarly for n+2 it is calculated from n+1 time level and so on.

i) In implicit approach the unknown must be obtained by means of simultaneous solution of the difference equations applied at all the grid points arrayed at a given time level. Thus it needs complex set of calculation.

e) Grid generation: The arrangement of discrete points throughout the flow field is simply called a grid. The basic grid is of rectangular type. But for all physical applications are not analyzed using rectangular grid and we need their transformation and for that we need a transformation for the derivatives in the standard partial differential equations.

Different types of grids are used like stretched grids, elliptical grids and adaptive grids as per need of application.

After grid generation the mesh is to generate on these grid points. These can be visualized as a mesh of finite volume cells. These meshes can be structured or unstructured type meshes.

f) Software used:

CFX-TASCFLOW, like other commercial CFD codes, offers a variety of boundary condition options such as velocity inlet, pressure inlet and pressure outlet. It is very important to specify the proper boundary condition.

2.4 TURBULENCE MODELING

There are two different states of flows that are easily identified and distinguished laminar flow and turbulent flow. Laminar flows are characterized by smoothly varying velocity fields in space and time in which individual laminae move past one another without generating cross currents. These flows arise when the fluid viscosity is sufficiently large to damp out any perturbations to the flow that may occur due to boundary imperfections or other irregularities. These flows occur at low-to-moderate values of the Reynolds number. The turbulent flows are characterized by large, nearly random fluctuations in velocity and pressure in both space and time. These fluctuations arise from instabilities that grow until nonlinear interactions cause them to break down into finer and finer whirls that eventually are dissipated (into heat) by the action of viscosity.

Turbulent flows occur in the opposite limit of high Reynolds numbers. The typical time history of the flow variable u at a fixed point in space the dashed line through the curve indicates the average velocity. We can define three types of averages.

1. Time average
2. Volume average

- DISCRETIZATION SCHEME.
  1. Upwind Difference.
  2. Modified Linear Profile with Physical Advection correction.

2.5 DISCRETIZATION USING THE FINITE-DIFFERENCE METHOD

2.5.1 Discretization

The fundamental ideas underlying CFD by applying them to the following simple 1D equation is as shown in the below,

$$\frac{du}{dx} + u^{m} = 0\,; 0 \le x \le 1\,; u(0) = 1$$

By considering case where $m = 1$ the equation is linear. Then considering the $m = 2$ case where the equation is nonlinear. Driving the discrete representation of the above equation with $m = 1$ on the following grid,

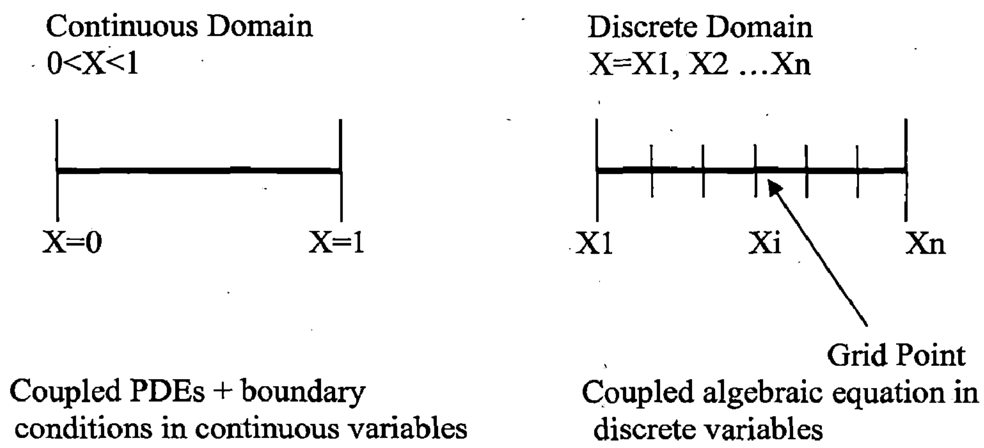


Fig 2.3 Continuous and Discrete Domain

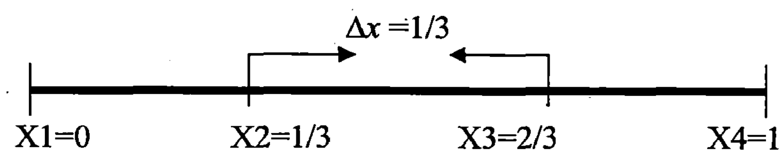


Fig 2.4 Discrete Representation

The grid has four equally spaced grid points with Δx being the spacing between successive points as show in the Fig.2.4. The governing equation is valid at any grid point, was given by

$$\left(\frac{du}{dx}\right)_i + u_i = 0 \tag{2.1}$$

Subscript i represent the value at grid point x_i. To get an expression for $(du/dx)_i$ in terms of u at the grid points, by expanding u_{i-1} in a Taylor's series

$$u_{i-1} = u_i - \Delta x\, (du/dx)_i + O(\Delta x)^2 \tag{2.2}$$

Rearranging the Eq. 2.2 gives the following Eq. 2.3

$$\left(\frac{du}{dx}\right)_i = \frac{u_i - u_{i-1}}{\Delta x} + O(\Delta x) \tag{2.3}$$

The error in $(du/dx)_i$ due to the neglected terms in the Taylor's series is called the truncation error. The truncation error is of the order $O(\Delta x)$, this discrete representation is termed first order accurate.

Using (2.1) in (2.3) and excluding higher-order terms in the Taylor's series, we can get the following discrete equation:

$$\frac{u_i - u_{i-1}}{\Delta x} + u_i = 0 \tag{2.4}$$

The error in $(du/dx)_i$ due to the neglected terms in the Taylor's series is called the truncation error. The truncation error is of the order $O(\Delta x)$, this discrete representation is termed first order accurate. Using (2.1) in (2.3) and excluding higher-order terms in the Taylor's series, we can get the following discrete equation:

$$\frac{u_i - u_{i-1}}{\Delta x} + u_i = 0 \tag{2.5}$$

This method of deriving the discrete equation using Taylor's series expansion is called the finite-difference method. The most of commercial CFD codes use the finite-volume or finite-element methods, which are better suited for modeling flow past complex geometries. FLUENT code uses the finite-volume method and ANSYS uses the finite-element method.

2.5.2 Discrete System and Boundary Conditions

The discrete equation obtained by using the finite-difference method was,

$$\frac{u_i - u_{i-1}}{\Delta x} + u_i = 0 \tag{2.6}$$

Rearranging Eq. (2.6),

$$-u_{i-1} + (1 + \Delta x)u_i = 0 \tag{2.7}$$

Applying this equation to the 1D grid shown earlier at grid points $i = 2, 3, 4$ gives

$$\begin{aligned} -u_1 + (1 + \Delta x)u_2 &= 0 \; (i = 2) \\ -u_2 + (1 + \Delta x)u_3 &= 0 \; (i = 3) \\ -u_3 + (1 + \Delta x)u_4 &= 0 \; (i = 4) \end{aligned} \qquad \text{(2.7i, 2.7ii, 2.7iii)}$$

The discrete equation cannot be applied at the left boundary (i=1) since u_{i-1} is not defined here. Instead, we use the boundary condition to get u_1=1.

Equations (2.7i) form a system of four simultaneous algebraic equations in the four unknowns u_1, u_2, u_3 and u_4. It's convenient to write this system in matrix form,

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ -1 & 1+\Delta x & 0 & 0 \\ 0 & -1 & 1+\Delta x & 0 \\ 0 & 0 & -1 & 1+\Delta x \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{2.8}$$

In a general situation, apply the discrete equations to the grid points (or cells in the finite-volume method) in the interior of the domain. For grid points (or cells) at or near the boundary, apply a combination of the discrete equations and boundary conditions. In the end, obtain a system of simultaneous algebraic equations with the number of equations being equal to the number of independent discrete variables. The process is essentially the same as for the model equation above with the details being much more complex.

FLUENT, like other commercial CFD codes, offers a variety of boundary condition options such as velocity inlet, pressure inlet and pressure outlet. It is very important to specify the proper boundary condition

2.5.3 Solution of Discrete System

The discrete system represented by Eq. (3.16) form 1D example can be inverted to obtain the unknowns at the grid points. Solving for u_1, u_2, u_3 and u_4 in turn and using

$\Delta x = 1/4$, we get

$u_1 = 1$ $u_2 = 3/4$ $u_3 = 9/16$ $u_4 = 27/64$ (2.9)

The exact solution for the 1D example is calculated to be $uexact = \exp(-x)$. Fig.2.5 shows the comparison of the discrete solution obtained on the four-point grid with the exact solution. In a practical CFD application, have thousands to millions of unknowns in the discrete system, by using the Gaussian elimination procedure naively to invert the matrix, it will long time. Lot of work goes into optimizing the matrix inversion in order to minimize the CPU time and memory required. The matrix to be inverted is sparse most of the entries in it are zeros since the discrete equation at a grid point or cell will contain only quantities at the neighboring points or cells. A CFD code would store only the non-zero values to minimize memory usage. It would also generally use an iterative procedure to invert the matrix, the longer one iterates, the closer one gets to the true solution for the matrix inversion.

2.6COMPUTATIONAL FLUID DYNAMICS OF FRANCIS TURBINE

The flow through a Francis turbine is known to be highly three dimensional, unsteady and turbulent making the modeling of the flow very demanding in terms of computational resources. Some of the important factors to be considered in the analysis of the flow are the adverse pressure gradient, the presence of high turbulence stress. Experimental results can give accurate results at a limited number of points in space, using equipment that is often expensive to purchase and requires skill to calibrate and use successfully. If a number of flow situations are to be considered, say for a number of physical geometries to be created and then tested. This increases the time and expense of testing. The development of high-speed digital computer during the last decade has brought about a great impact on the way principles from the science pf fluid mechanics; heat transfer and combustion are applied to engineering design practice. The availability of large computer memory together with high storage capacity of data has further made it possible to solve practical problems in very short time at a very little cost.

Essentially there are three methods for determine the solution to flow problems viz. Experimental, Analytical and Numerical. The Analytical methods aim at getting a closed form solution in the entire domain assuming the process to follow continuum hypothesis. These are generally restricted to simple geometry, simple physics and generally linear problems. Once the problem becomes complex, the various assumptions that are needed to be made to obtain a closed form solution, entails loss of accuracy of the critical parameter of interest. This leads them to be used as a check on the accuracy of a numerical procedure but makes them mainly unsuitable for the analysis of real engineering problems.

However, they give the direction and general nature of the solution. Hence over the years, scientists and engineers have resorted to experimental techniques concentrating in the regions of interest. These experimental techniques have their inherent problems viz. that they are equipment oriented, and they need large resources of hardware, time and operating costs. Their applications are also limited due to scaling considerations.

Further theses involve certain measurement difficulties and handling of large quantity of data. Numerical methods have emerged as a third method and have overcome the restrictions in both experimental and analytical methods. They involve the discretization of the governing mathematical equations in a way such that the numerical solutions can be obtained. This approach forms the core of Computational Fluid Dynamics, commonly known as CFD. It is a new methodology that has emerged during the last decade as a basic engineering tool to understand and solve complex problems in fluid mechanics and heat transfer in the industry .In this computational approach, the equations that govern the process of interest are solved at discrete locations in the domain numerically in an iterative manner. Further, the popularity of CFD has been possible due to great developments in computing algorithms that have enabled fast Graphic User Interface that makes the interpretation and Visualization of the results easy.

CFD methods have their own disadvantage in terms of specifications of proper boundary conditions, truncation errors, convergence problems, right choice of turbulence models and parameters, right choice of discretization method etc. This applications of CFD to practice problems need understanding of basic theory to overcome the above mentioned problems.

2.7 SCOPE OF PRESENT WORK STUDY

The hydro turbine is one of the most critical components of hydro power plant. The hydro turbines for hydro power plant are selected according to the available head and discharge, which may not come under any standard size available with the manufacture. It is therefore the performance analysis of turbine is carried out and prediction of characteristics of the prototype is possible. The present work is aimed to carry out the Performance evaluation of typical medium head Francis Turbine model using Computational fluid Dynamics (CFD) Analysis. The analysis is conducted to find out hydrodynamic performance of the runner. For this purpose a Francis turbine comprising spiral casing, stay vane, guide vane, runner and draft tube is modeled with the aid of CAD drawings. The model thus resembles a numerical test rig for turbine performance evaluation. The results obtained by the analysis are compared with experimental value.

2.8 JUSTIFICATION OF CFD ANALYSIS FOR HYDRO TURBINE

1) CFD techniques are now used as practical tools for estimation of performance parameter.

2) Head turbines are used in canal based system there is low head is available so performance and efficiency is largely depend upon flow pattern of water inside runner. So runner profile is very crucial for better efficiency.

3) CFD is also facilitates to find performance of turbine without actual model test for design improvement.

4) The most important advantage of computational prediction is its low cost. In most applications, the cost of a computer run is many orders of magnitude lower than the cost of a corresponding experimentation investigation. This can reduce or even eliminate the need for expensive or large-scale physical test facilities. This factor assumes increasing importance as the physical situation to be studied becomes larger and more complicated. Further whereas the prices of most items are increasing, computing cost is likely to be even lower in the future.

5) A computational investigation can be performed with remarkable speed. A designer can study the implication of hundreds of different configurations in less than a day and choose the optimum design. With the ability to reuse information generated in other stages of the design. With the ability to reuse information generated in other stages of the design process, rapid evaluation of design alternatives can be made. On the other hand, a corresponding experimental investigation would take a long time.

6) A computer solution of problem gives detailed and complete information .It can provide the values of all relevant variables (such as velocity, pressure, temperature, concentration, turbulence intensity) throughout the domain of interest. This provides a

better understanding of the flow phenomenon and the product performance because knowledge of such values is not restricted to those areas that can be instruments during testing. For this reason, even when an experiment is performed, there is great value in obtaining a companion computer solution to supplement the experimental information.

7) In a theoretical calculation, realistic conditions can be easily stimulated. The re is no need to resort to small scale or cold models. Through a computer program, there is little difficulty in having very large or very small dimensions, in treating very low or very high temperature, in handling toxic or flammable substances, or in following very fast or very slow processes.

8) A prediction method is sometimes uses to study a basic phenomenon, rather than a complex engineering application. In the study of phenomenon, one wants to focus attention on a few essential parameters and eliminates all irrelevant features. Thus many idealizations are desirable for example two dimensionality, constant density, an adiabatic surface, or infinite reaction rate. In a computation, such conditions can be easily and exact setup, whereas even careful experimental can barely approximate the idealization.

CFD can also be used to investigate configurations that may be too large to test or which pose a significant safety risk; including pollutant spread a nuclear accident scenarios. This can often provide confidence in operation, reduce or eliminate the cost of problem solving during installations, reduce product liability risks.

CHAPTER 3
MODELING, MESHING AND THE SOLVER

3.1 GENERAL

In the previous chapter Computational Fluid Dynamics (CFD) has been discussed. In order to carry out the CFD for the designed runner, modeling, meshing and analysis required for the analysis has been presented under this chapter.

Francis turbine has been taken for present study. The modeling of Francis turbine has been carried out under this chapter.

3.2 CFD PROBLEM APPROACH

The basic steps involved in solving any CFD problem is as follows

- Identification of flow domain
- Grid generation
- Specification of boundary conditions and initial conditions
- Selection of solver parameters and convergence
- Results and post processing
- Macro development for engineering analysis

3.3 ADOPTED PROBLEM APPROACH

The aim of project is to understand the hydro turbine performance. For this basically 3 steps are required.

1. Modeling of components
2. 3d-grid generation
3. Analysis

3.4 SOLID MODELLING

The first step in the present analysis was to develop a model of the blade, which needs to be analyzed. This is done by the solid modeling of the blade with the help of blade geometry as the basic requirement.

Solid modeling is the unambiguous representation of the solid parts of an object, that is, models of solid objects suitable for computer processing. It is also known as volume modeling.

3.5 MODELING AND 3D-GRID GENERATION –COMPONENT WISE

Modeling of all major components of hydro turbine is nothing but extraction of geometry from drawings. Computer modeling of flow requires the fluid domain to be discretized by creating nodes. Various grid techniques are used for different components as per suitability and ease. These are described below.

3.5.1. Spiral Casing

First the sections along radial direction are made in AUTOCAD*. Create IGES files then imported into IDEAS*.These sections are placed angular wise and loft between the sections to create volumes. The surfaces are extracted from volumes and input to the ICEM-CFD*. This CFD software was used to generate 3D-grid. O-grid is made along the sections of spiral to improve the skew angle.

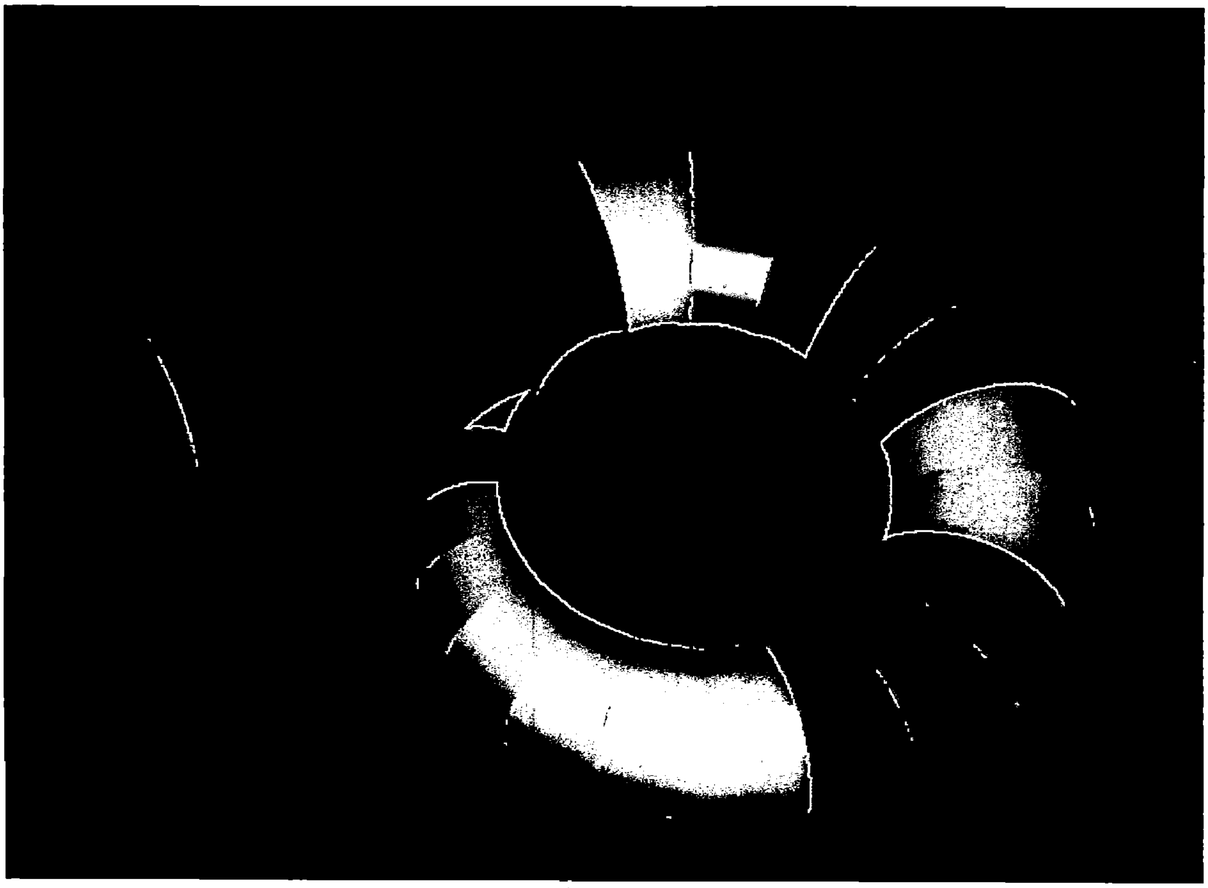


Fig.3.1. Model of Spiral Casing

3.5.2 Stay Vanes and Guide Vanes

There are 12 stay vanes and 16 guide vanes. The stay vanes have variable height on the inlet side to fill the gap between the upper and lower spiral casing surfaces. The guide vanes are identical. CAD software is used to create profiles. The hub, shroud and profile points data is collected and named as hub.curve, shroud.curve and profile.curve and these files are imported to TURBO-GRID. This makes the grid generation task simpler and faster and it is very suitable for blade type configurations. Generic single block grid template was selected to create grid.

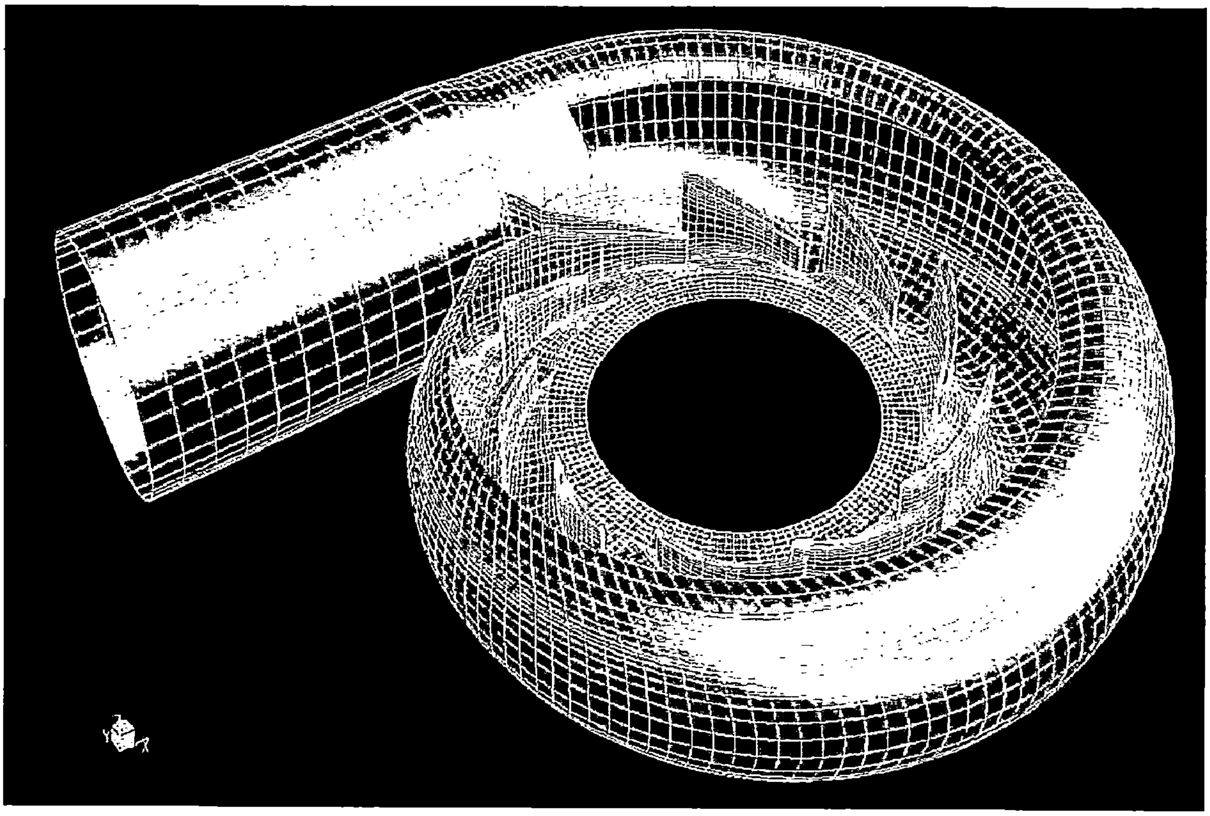


Fig.3.2. Spiral Casing of Francis Turbine with Stay Vanes

3.5.3 Runner Vanes

There are totally 11 runner vanes in Francis turbine. Runner blade is having complex surfaces. The broad section data points of blade are taken and created the sections along axial direction. All the sections are not in the same plane. Created the geometry using IDEAS. The sections are lofted to create volumes. Finally volume surrounded by surfaces is obtained. Points on hub, shroud and profiles along stream wise direction are collected and then input to the TURBO-GRID.

Hub to shroud the profile shape is not constant so to capture the geometry more number of profiles (maximum 10 or 11) is considered along span. The periodic symmetry of the runner blade passage is of great use in the analysis, which works with less number of nodes, as only one runner passage is sufficient with periodic symmetry. This is possible CFX-TURBOGRID software. In this periodic boundary surfaces are automatically created. Generic-single block grid template was used.

3.5.4 Draft Tube

Draft tube CAD model is generated in IDEAS. It is easy to visualize the solid model for which geometry input is given directly from engineering drawing. It is not a single component. It is an assemble of upper cone, lower cone, draft tube bent and diffuser. To construct the geometry AUTOCAD tool is also helpful. The boundary surfaces are taken from IDEAS (IGES) and imported to ICEM-CFD. O-grid is made to improve skew angle.

Generating the grid in draft tube cone: shaft is entered to some extent of cone, where there is no flow. So selected the region carefully while creating the inflow boundary condition of draft tube. One more important thing is considered that is TASCFLOW should not accept if axis of rotation is passing through the grid (i, j, k=0). So created two volumes whose inner volume diameter is equal to the shaft diameter where there is no flow. For outer volume created grid for quarter portion of cone and then rotated the grid by suitable angle to get complete volume mesh.

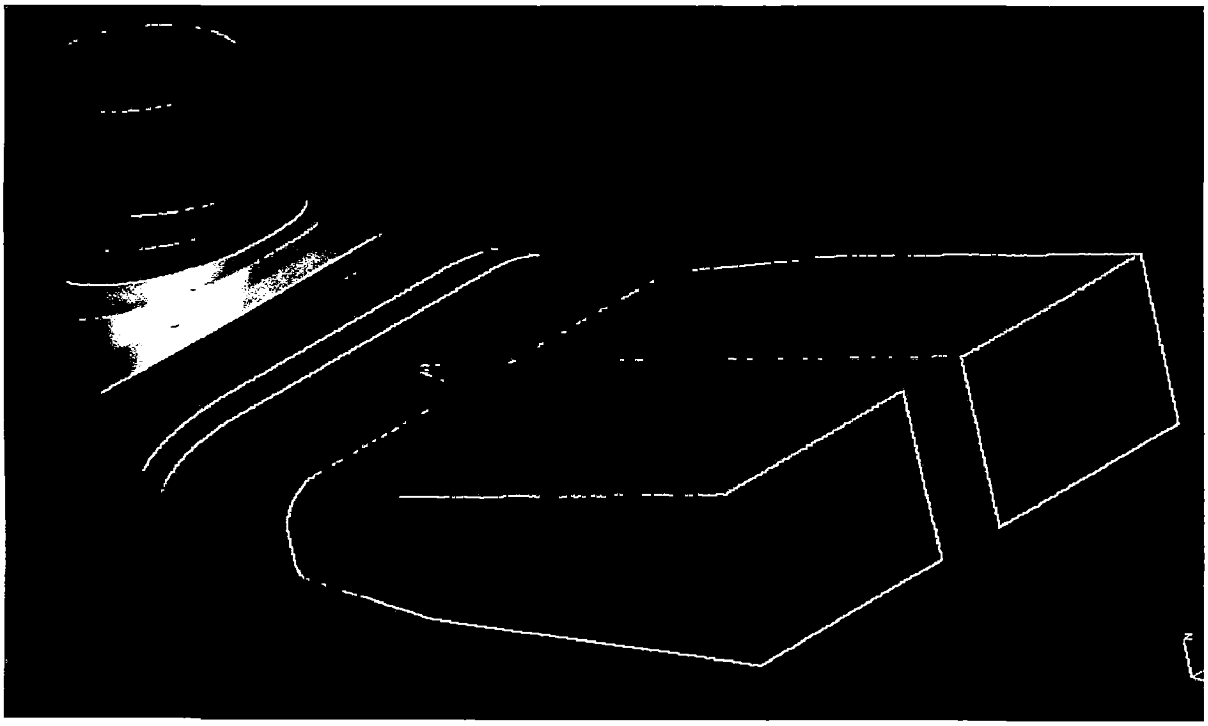


Fig.3.2. Model of Draft Tube

3.5.5. 3D-Grid Generation- Build Case

Using TASC-merge option merges the grids of spiral, stay vanes and guide vanes and saved in to one GRD file. The 3D-grid is as shown in figure. This grid is assembled by using BUILDCASE option in TASCFLOW as three components 1.stationary component consisting of spiral casing, stay vanes, guide vanes 2.rotating component i.e. runner blade grid of one sector and 3.stationary component: draft tube. Thus it is a multi frame of reference problem. There is a change in speed of rotation from stationary component to rotating component vice-versa.

The inlet of runner grid and exit of guide vane grid is interfaced as stage interface and one more is between the runner grid exit and draft tube inlet. The whole grid is formed in MFR with two stage interfaces. Runner grid is in rotating frame of reference with speed of rotation corresponding to model blade rotation speed. Created the GGI interfaces between the spiral outlet and stay vane inlet, spiral main and nose, stay vane outlet and guide vane inlet, draft tube cone and bent.

3.6 CFD ANALYSIS OF FRANCIS TURBINE A GENERAL APPROACH

CFD analysis was made for understanding flow through the turbine. CFX-TASCFLOW software tool was used for analysis purpose. The present work is aimed to carry out the Performance evaluation of typical medium head Francis Turbine model using Computational fluid Dynamics (CFD) Analysis. The analysis is conducted to find out hydrodynamic performance of the runner. For this purpose a Francis turbine comprising spiral casing, stay vane, guide vane, runner and draft tube is modeled with the aid of CAD drawings. The model thus resembles a numerical test rig for turbine performance evaluation. The results obtained by the analysis are compared with experimental value.

3.7 PERFORMANCE EVALUATION OF 90 MW FRANCIS TURBINE USING CFD

In the present work of study the Francis Turbine with following operating condition and Dimensions has been taken from actual BHEL,Bhopal made Francis Turbine for analysis purpose, Which is supplied to Srinagar Hydro power plant. The Francis turbine is modeled for analysis purpose to rate output of 90 kW for CFD analysis purpose. The Scale ratio for modeling the component from the prototype is 1:100.

3.7.1 OPERATING CONDITIONS

- Rated Head = 18 meter
- Rated Speed = 777 RPM
- Rated discharge = 0.586 m^3/s (586 kg/sec)
- Runner Diameter = 400 mm
- Working fluid Water at STP
- Rated output = 90 kW
- Specific speed of turbine = 174

3.7.2 PRE-PROCESSING

Preprocessing involves the following steps.

a) Boundary conditions

For inflow and outflow selected the corresponding regions. Mass flow at inlet and pressure at outlet was specified as boundary conditions.

Inflow	:	spiral casing inlet
Outflow	:	draft tube outlet
Wall	:	smooth

Mass Flow Rate	:	586 kg/s
Outlet Static Pressure	:	0 Pa

b) Zones and attributes

Working fluid was selected i.e. water at STP in this case.

Working Fluid	:	water at STP (SI)
Density	:	998.2 kg/m^3
Viscosity	:	0.000923 $N\text{-}Sec/m^2$
Conductivity	:	0.597 W-m/k
Specific Heat (C_P)	:	4182KJ/kg-k
Specific Heat (C_V)	:	4182KJ/kg-k

c) RF setup

Specified the speed of rotating frame of reference i.e. runner speed.

Speed of Runner	:	-777.6Rpm.

d) Turbulence model

The flow is considered as viscous flow so selected suitable model.

Turbulence Model	:	Standard k-ε model
Eddy Viscosity Ratio	:	10
Intensity	:	0.05

e) Grid transformation

Scaled the grid to convert model to proto type.

Model runner diameter is 2.150m and prototype diameter is 0.400m so scale factor is model/runner.

Scale Factor	:	5.382857

f) Check quality

Skew angle, aspect ratio and volumes.

g) Initial guess generator

Realistic initial values were given for better convergence.

U	:	3
V	:	4
W	:	-2

h) Write preprocessing

After specifying all conditions write GCI file by using write preprocessing command.

The convergence tolerances used range from 1.0e-03 to 5.0e-04.

3.7.3 SOLVING

Solving the problem involves the following steps.

a. Solver parameters

Write mass flow scalar fields to RSO (PMASS)

Write surface area fields to RSO (PAREA)

Write BC information to RSO (BCINFO)

Number of Steps	:	1000
Residual Convergence Criteria	:	0.001
Fluid Time Step (DTIME)	:	1e-3

Frequency of Intermediate Start Files: 20

b. Run the solver monitor.

3.7.4 POST PROCESSING

The performance of hydro turbine is studied by using macro programs.

The various plots are drawn and discussed under Results & Discussion in next chapter.

CHAPTER 4
ANALYSIS

4.1 GENERAL

In the previous chapter the modeling, meshing and grid generation for the Francis turbine has been presented. In order to carry out the analysis the following steps and tabulation of results of different CFD runs have been presented under this chapter.

4.2 MODELING CONSIDERATIONS

The usual practice is that all the functional components of the turbine are included in the model, except for the runner where only one blade sector is considered by exploiting the cyclic symmetry of the turbine runner. Thus the model consists of spiral casing, all stay vanes, & all guide vanes, a single runner passage and draft tube. The following approach has been considered.

1) The primary objective is to evaluate the flow direction at inlet to the stay vanes, over a range of operating conditions for analysis of Spiral Casing, all Stay vanes and all Guide vanes.

2) The flow angle obtained from Step-1 is now used as an inlet direction for the current step. Here a sector, which comprises one stay vane, one guide vane and one runner passage and draft tube is modeled. The analyses are then carried out for range of operating conditions. This result yields the hydrodynamic performance characteristics of the turbine, which help in saving huge part of computational time.

4.3 ANALYSIS

STEP- 1

In this step analysis is carried out for spiral casing ,stay vanes and guide rings, as shown in Fig 4.1 for the following three Guide vane openings positions.

a. The maximum guide vane opening i.e. the ao of 20 deg is the first operating condition for which the analysis is carried out.
b. The second guide vane opening condition is ao equals to 23 deg.
c. The third guide vane opening condition is ao equals to 26 deg.

Fig 4.1 Generation of the Grid under Step-1

4.3.1. Quality Aspect in Grid Generation

There are various aspects of grid generation that is examined before taking a CFD run. While discretizing it may so happen that in the computational domain, for some of the finite volume cells the cell depth may become zero resulting in zero volume cells. The existence of such a zero volume cell renders the grid useless foe CFD run. At times, there can be negative volume cells. The negative volumes cells are a result of conflict in coordinate conventions. TASC flow uses structured grids. Hence there are two conventions to be taken care 'the XYZ coordinate convention and the IJK (grid index) convention'. If one of the convention become left handed while other is right handed negative volumes are generated. The presence negative volume results in error and CFD run cannot proceed.

Skew angle is another cause of concern for CFD analysis. Ideal situation is when all angles, of any finite volume cell, are 90 degrees. This is seldom the case. Most of the time the angles get distorted based on the geometrical shape of the domain under consideration. Such distortions are permitted in the range of 20 deg to 160 deg. Any angle below 20 deg or above 160 deg is a warning situation in TASC flow. This becomes a severe warning if the minimum falls below 10 deg. In the present study, it is ensured that all skew are above 10 deg .Any negative volumes found in the component grids are cleared by suitable change in grid. The other important factor in grid quality is the aspect ratio defined as the ratio of longest side to the smallest side of volume cell. The ideal value for aspect ratio is 1.However, TASCflow is a robust package and tolerates aspect ratios as high as 100.Aspect ratios of all the component grids are maintained well below 100.The component grids are integrated using the build case feature of TASCflow. The grid quality levels achieved in all the component grids are summarized in the Table 4.1.

Table No. 4.1. Software Package used for Modeling & Meshing.

SI.NO.	Component Name	Modeling Package	Meshing Package	Minimum Skew Angle	Maximum Aspect Ratio	Grid Size (nodes)
1	Stay vane (Single blade)	Auto cad, I-DEAS (Version :2006)	Turbo grid	13.76	55.84	72292
2	Guide vane (Single blade)	Auto cad, I-DEAS (Version :2006)	Turbo grid	19.84	47.77	114421
3	Runner (Single blade)	Auto cad, I-DEAS (Version :2006)	Turbo grid	15.72	87.75	133176
4	Draft Tube	Auto cad, I-DEAS (Version :2006)	Turbo grid	28.94	54.98	345594

4.3.2. Boundary Conditions

The boundary conditions for the analyses are the rated mass flow at the inlet to the spiral casing and a static pressure of zero Pa (Pascal) at outlet to the guide ring. The boundary conditions are given in Table 4.2.

Table.4.2. Boundary Conditions & Results of Step -1

SI.No.	GV opening (deg)	Inlet Boundary Condition(Kg/s)	Outlet Boundary Condition(Pa)	Flow angle at spiral casing outlet(deg)	Average flow angle at spiral casing outlet(deg)
1	20	457	0	-21.91	-21.88
2	23	512	0	-21.87	
3	26	560	0	-21.85	

STEP-2

Based on the results obtained under step -1 i.e. flow angle to the inlet of stay vane values are used as direction. The spiral casing analysis can be eliminated after find out outlet angle of flow at exit of spiral casing. The current analysis can be carried out using a suitable sector of stay vane and the runner.

For this case , 12 stay vanes with varying height & equally distributed in the 360 deg circumference, 16 guide vanes and 12 runner blades were considered. For modeling average height of stay vane and guide vane are assumed to be same. This gives us a pitch ratio Of 1:1.Only one runner blade passage is modeled and frozen rotor interface is used between guide vane and runner vane and between runner vane and draft tube.

The model of step -2 are shown in fig 4.2. It consists of one runner vanes, guide vanes, stay vanes along with the draft tube assembly. For analysis purpose we are utilizing symmetry principle for modeling of assembly.

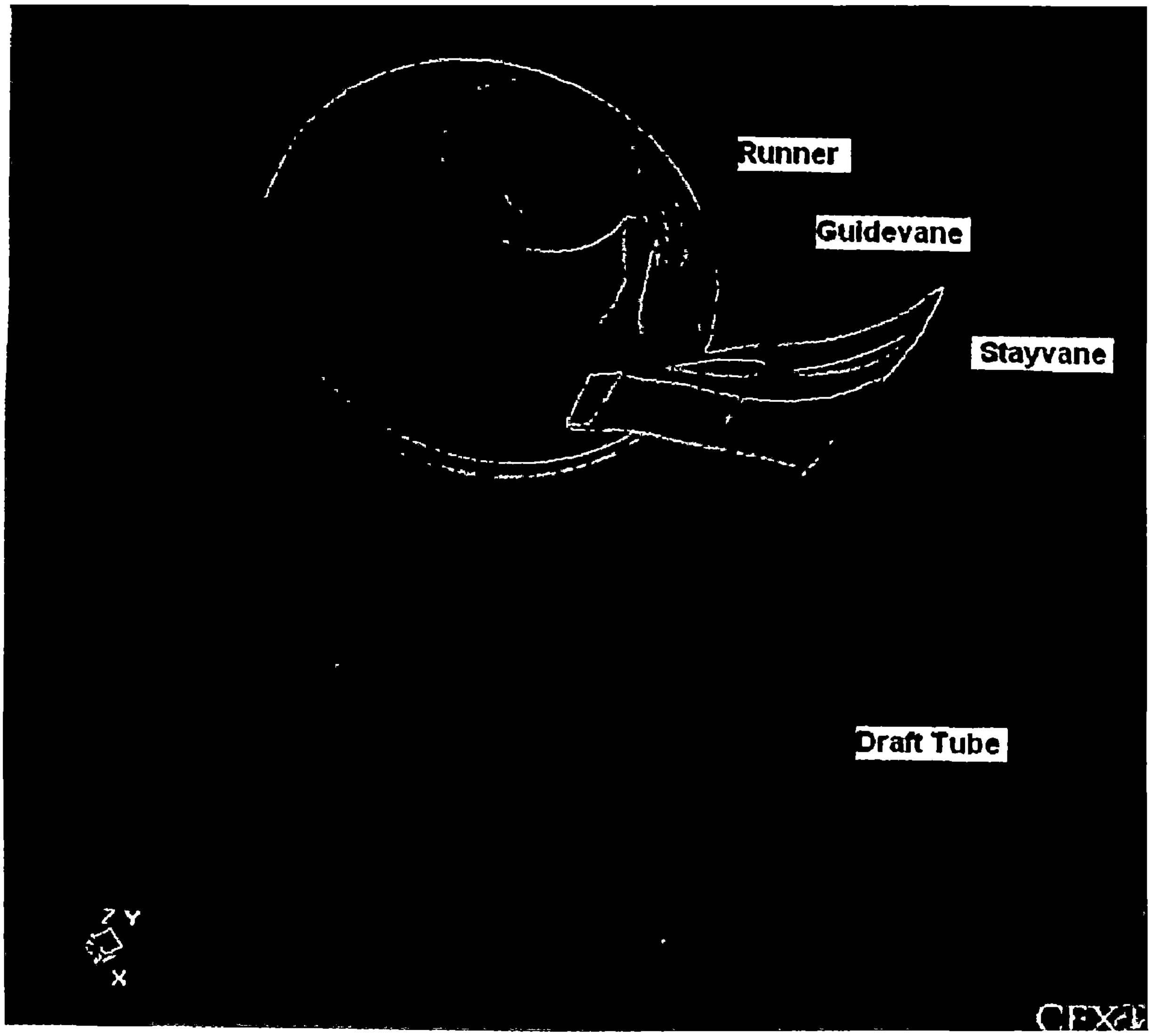


Fig.4.2. Model for Step -2

4.3.3. Boundary Conditions

The boundary conditions here are Inlet mass flow rate at the inlet to the stay vane with the prescribed direction components obtained from results of step-1.The direction in cylindrical components for a typical case are specified as follows.

Axial Components of flow: 0

Radial components of flow: Sin (-21.88) = -0.92794

Tangential components of flow: Cos (-21.88) = 0.37273

At the outlet of Draft Tube a static pressure: 0 Pa.

Ordinary general grid Interface is specified stay vane & guide vane. Frozen rotor interface is used between guide vane and runner vane and between runner vane and Draft tube.

4.4 RESULTS AND DISCUSSIONS

A total of 54 runs are taken for 21 mm ,23 mm,24 mm,26 mm,28 mm,30mm,32mm,34mm of different guide vane opening covering the wide range of operation of the turbine. Peak efficiency for each of the above Guide vane opening is obtained. The results are compiled using macros are given in Table.4.3.

Table.4.3. CFD Runs Complied Results

Sl.No.	OPENING	RPM	N11	Q11	HEAD	TORQUE	POWER	EFFICIENCY	CONVERGENCE
	mm				m	N-m	kW	%	LEVEL
GUIDE VANE OPENING = 21 MM									
1	21	680	63.0837	676.808	18	1095.76	78.029	91.7966	1.00E-04
2	21	700	64.7222	674.546	18	1074.139	78.739	92.0142	1.00E-04
3	21	720	66.3673	672.478	18	1052.729	79.374	92.1888	1.00E-04
4	**21**	**740**	**67.9905**	**670.306**	**18**	**1031.284**	**79.92**	**92.2206**	**1.00E-04**
5	21	760	69.5889	668.009	18	1009.654	80.355	92.0926	1.00E-04
6	21	780	71.1559	665.539	18	987.965	80.664	91.7635	1.00E-04
7	21	800	72.7206	663.169	18	965.169	80.858	91.33	1.00E-04
GUIDE VANE OPENING = 23 MM									
8	23	650	61.7943	739.308	18	1170.113	79.647	92.3129	1.00E-04
9	23	660	62.7148	738.953	18	1158.926	80.099	92.7476	1.00E-04
10	23	680	64.5384	738.074	18	1136.509	80.93	93.487	1.00E-04
11	**23**	**700**	**66.2265**	**735.74**	**18**	**1113.855**	**81.65**	**93.7228**	**1.00E-04**
12	23	720	67.8093	732.399	18	1090.995	82.259	93.5665	1.00E-04
13	23	740	69.4038	729.36	18	1067.895	82.754	93.35	1.00E-04
14	23	760	71.0084	726.586	18	1044.395	83.12	93.0513	1.00E-04
15	23	780	72.6065	723.889	18	1020.606	83.365	92.6331	1.00E-04
16	23	800	74.2134	721.411	18	996.434	83.477	92.1243	1.00E-04

GUIDE VANE OPENING = 24 MM										
17	24	680	62.3723	768.818	18	1294.972	92.214	92.3703	1.20E-04
18	24	700	64.2136	768.898	18	1269.482	93.058	93.1716	1.70E-04
19	24	720	65.9756	768.053	18	1243.896	93.788	93.6957	2.00E-04
20	**24**	**740**	**67.7094**	**766.933**	**18**	**1218.203**	**94.402**	**94.0345**	**2.30E-04**
21	24	760	69.3168	764.513	18	1192.093	94.857	93.91	2.70E-04
22	24	780	70.8675	761.541	18	1165.68	95.214	93.5149	2.90E-04
23	24	800	72.4646	759.235	18	1139.074	95.427	93.1571	2.90E-04
GUIDE VANE OPENING = 26 MM										
24	26	660	63.7511	830.615	18	1240.258	85.721	92.7534	1.00E-04
25	26	680	65.5923	829.467	18	1214.404	86.477	93.3138	1.00E-04
26	26	700	67.4184	828.201	18	1188.282	87.106	93.7054	1.00E-04
27	26	720	69.2427	826.984	18	1161.802	87.598	93.9582	1.00E-04
28	**26**	**740**	**71.0778**	**825.958**	**18**	**1134.93**	**87.949**	**94.1005**	**1.00E-04**
29	26	760	72.8872	824.695	18	1107.804	88.167	94.0456	1.00E-04
30	26	780	74.5586	821.976	18	1080.71	88.274	93.5399	1.00E-04
31	26	800	76.1578	818.615	18	1053.902	88.292	92.7951	1.00E-04
GUIDE VANE OPENING = 28 MM										
32	28	680	66.1599	886.021	18	1252.557	89.194	92.4618	1.00E-04
33	28	700	68.0573	885.389	18	1224.502	89.761	92.9169	1.00E-04
34	28	720	69.9504	884.74	18	1196.314	90.2	93.2347	1.00E-04
35	28	740	71.8262	883.912	18	1168.156	90.524	93.3939	1.00E-04

36	**28**	**760**	**73.6781**	**882.841**	**18**	**1140.248**	**90.749**	**93.3998**	**1.00E-04**
37	28	780	75.486	881.313	18	1112.751	90.891	93.2225	1.00E-04
38	28	800	77.2632	879.509	18	1085.662	90.952	92.9038	1.00E-04
39	28	810	78.1324	878.423	18	1072.3	90.956	92.6784	2.00E-04
GUIDE VANE OPENING = 31 MM									
40	31	710	66.806	963.909	18	1405.491	104.5	90.0664	1.00E-04
41	31	720	67.7937	964.574	18	1390.583	104.848	90.491	1.00E-04
42	31	740	69.5998	963.508	18	1360.971	105.465	90.823	1.00E-04
43	31	760	71.3734	962.059	18	1331.811	105.995	91.0048	1.00E-04
44	**31**	**780**	**73.1155**	**960.272**	**18**	**1303.138**	**106.442**	**91.0495**	**1.00E-04**
45	31	800	74.8249	958.153	18	1274.9	106.806	90.958	1.00E-04
46	31	810	75.6728	957.049	18	1260.995	106.961	90.8806	3.00E-04
47	31	820	76.5202	955.964	18	1247.305	107.106	90.7975	1.50E-04
GUIDE VANE OPENING = 34 MM									
48	34	720	67.6098	1034.82	18	1444.403	108.905	86.9019	1.00E-04
49	34	740	69.3299	1032.47	18	1416.528	109.771	87.1946	1.00E-04
50	34	760	71.0129	1029.7	18	1389.302	110.57	87.3596	1.00E-04
51	**34**	**780**	**72.6819**	**1026.88**	**18**	**1362.546**	**111.295**	**87.4505**	**1.00E-04**
52	34	800	74.2864	1023.31	18	1336.698	111.983	87.3807	1.00E-04
53	34	810	75.0923	1021.64	18	1323.777	112.287	87.3322	1.00E-04
54	34	820	75.8893	1019.89	18	1310.976	112.574	87.2562	1.00E-04

4.4.1. Graphical Presentation of Results

The various plots obtained from post processing of the results are shown as follow:

Figure 4.3 shows the static pressure variation over the runner blades. The zones of localized low pressure can be observed from this plot. This low pressure zones are responsible for cavitations phenomenon over the runner blades. The cavitations creates many problem like erosion of metal from blade surface, create cracks e.t.c.

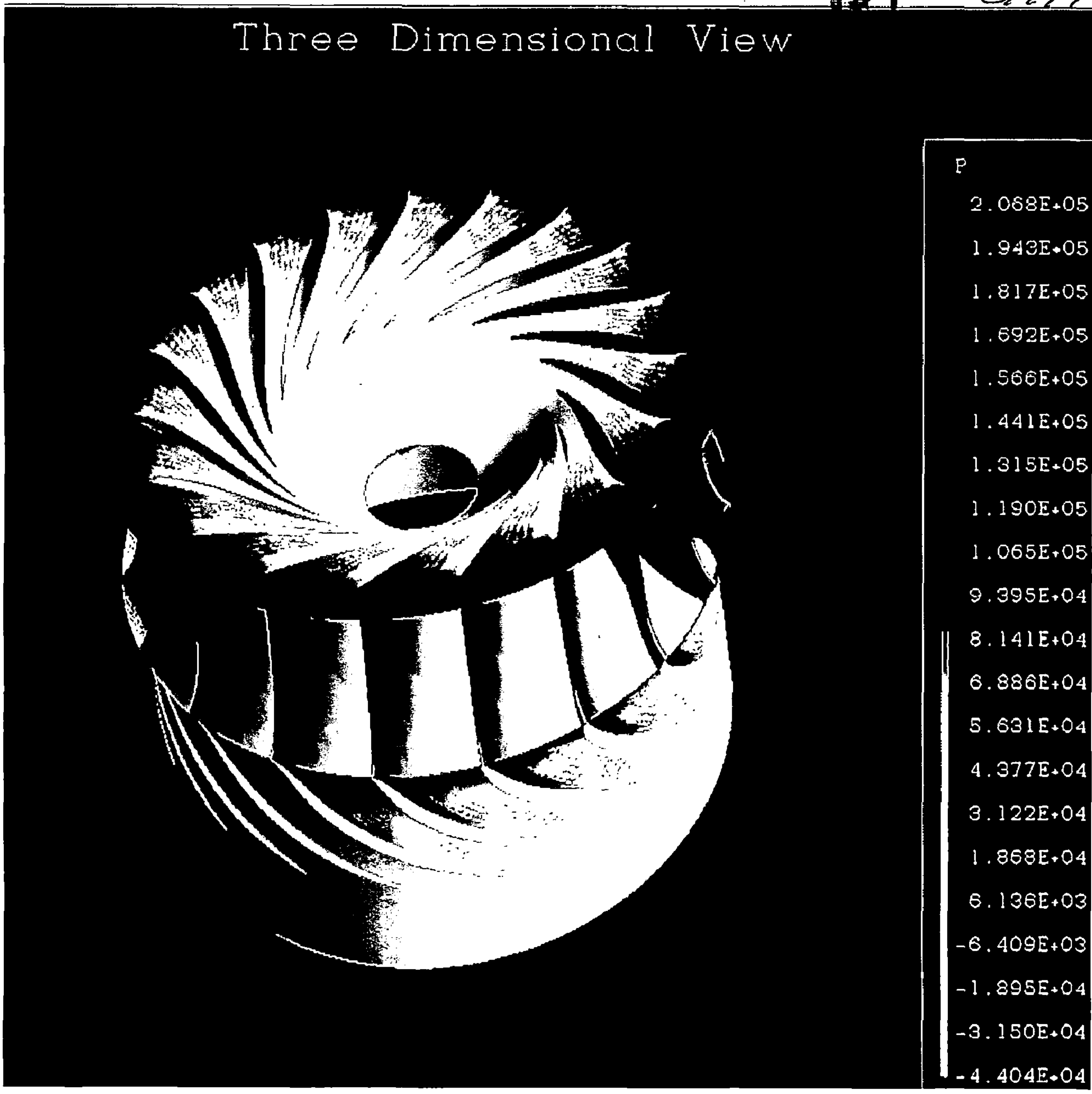


Fig 4.3.Static Pressure Variation

Figure 4.4 shows the tangential velocity plots at Draft Tube outlet. It can be observed that the flow attachment towards the wall is high. This implies that the velocity vector concentration around the draft tube outlet is high results the low pressure zones formation, which will cause the cavitations effects. The performance of turbine is greatly affected by cavitations phenomenon.

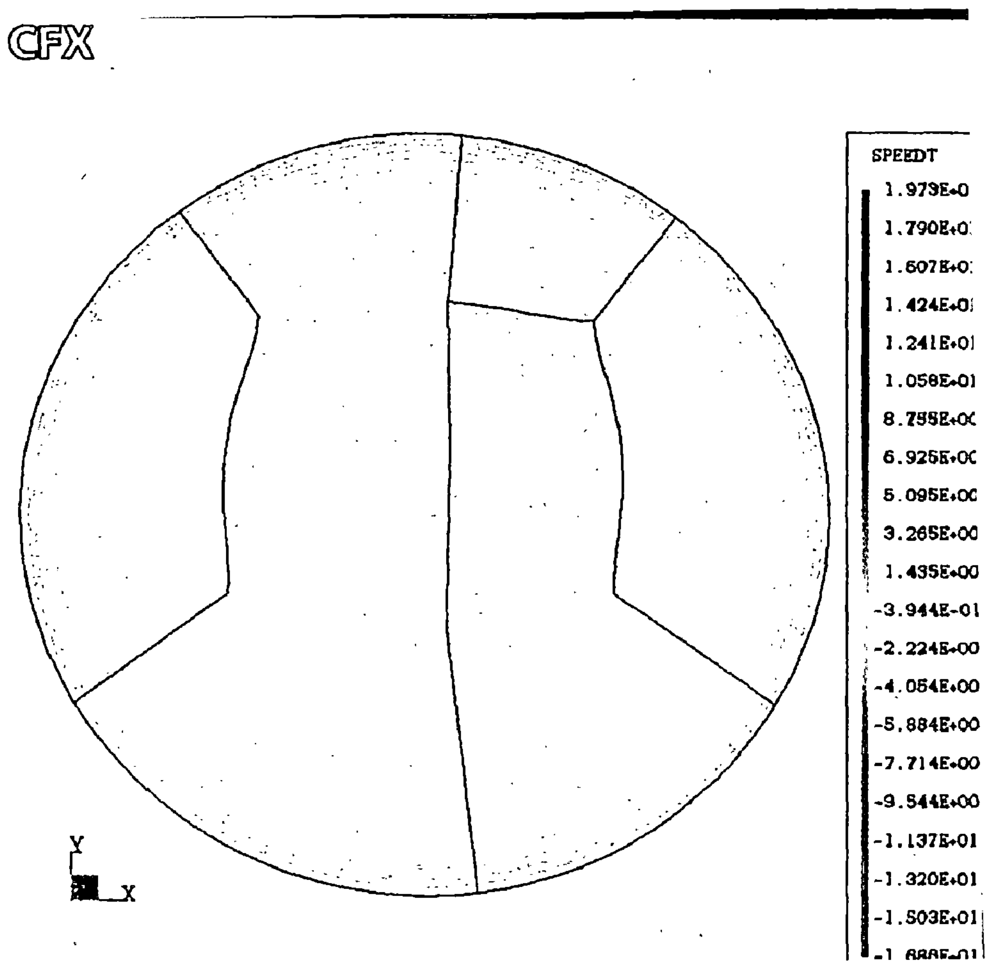


Fig .4.4. Velocity Plot of Tangential Velocity

Figure 4.5 shows the pressure distribution in spiral casing. The zones of localized low pressure can be observed from this plot. The low pressure zones more increasing towards center. The distribution of water across the runner is totally depends upon the pressure distribution in spiral casing which greatly effects the performance of turbines as for better performance it has required the equal distribution of water across the runner periphery.

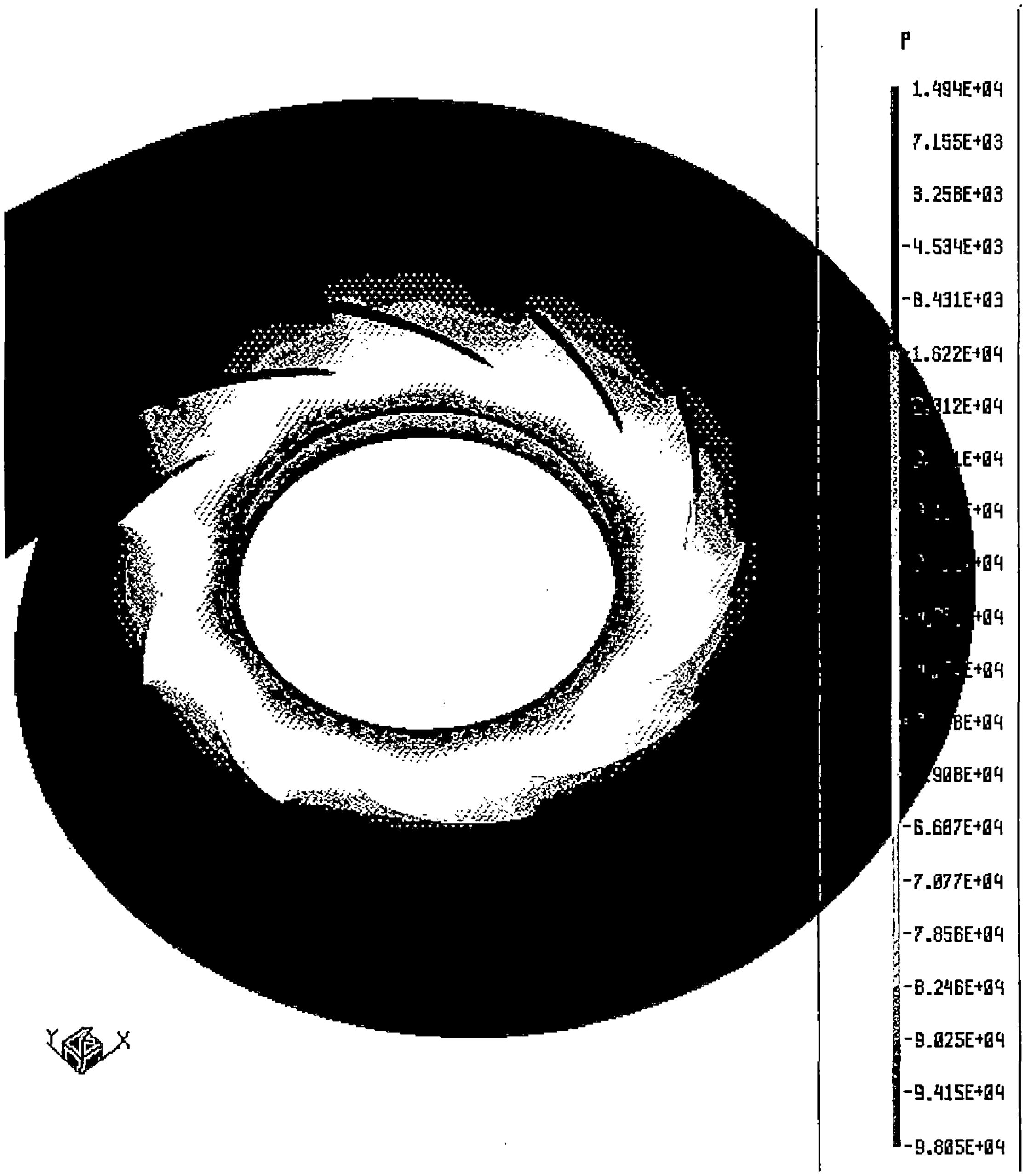


Fig.4.5. Pressure Distribution in Spiral Casing

Figure 4.6 shows the velocity distribution in spiral casing of Francis turbine. It can be observed that the velocity vectors are more concentrated at wall of spiral casing as well as at bend position of spiral casing. This implies that the velocity vector concentration around the walls and bends of spiral casing is high results the low pressure zones formation. This can be observed from above Fig.4.5 pressure distribution around spiral casing.

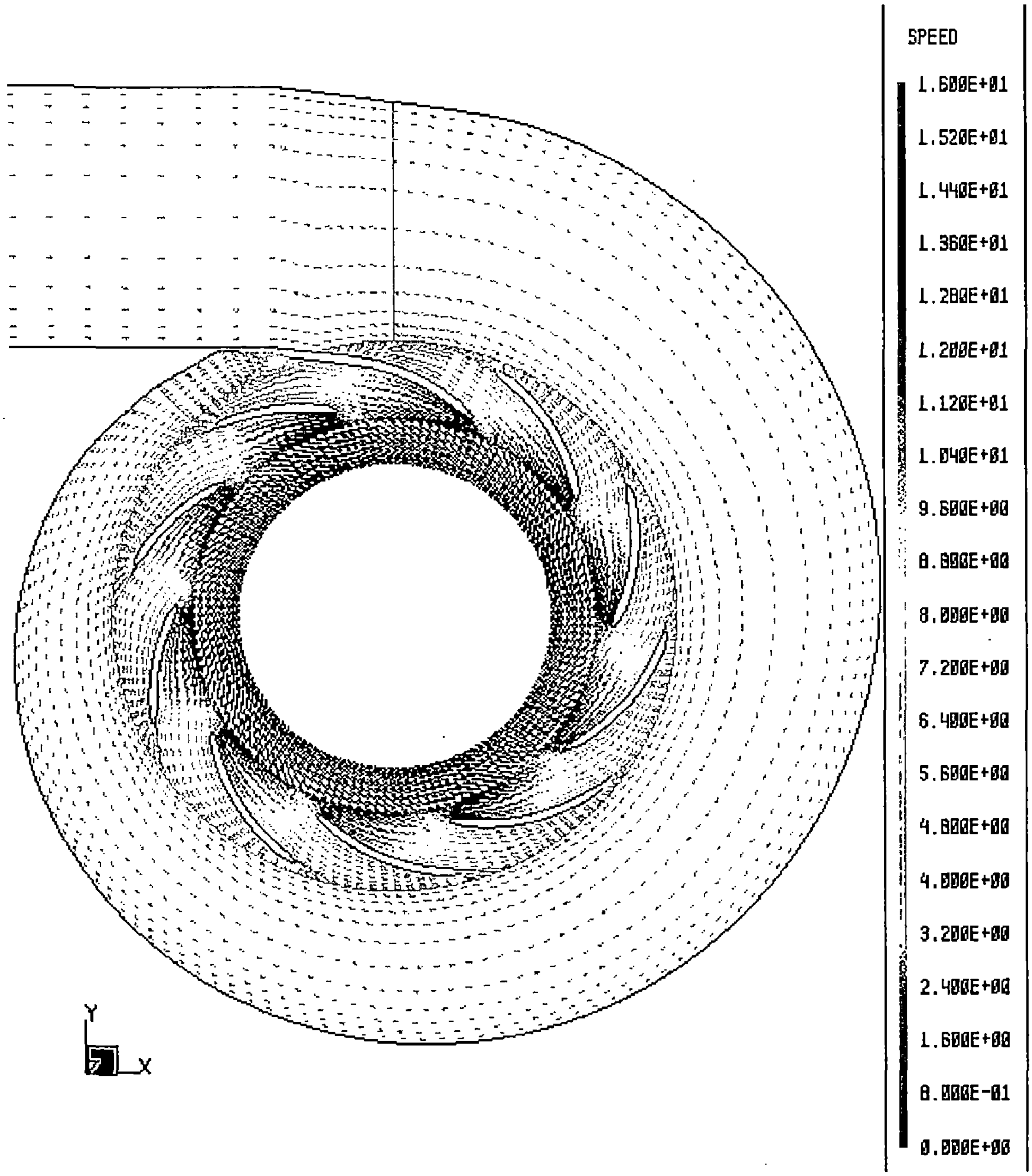


Fig.4.6. Velocity Distribution in Spiral Casing of Francis Turbine

Plots between different guide vanes opening positions and power output from Francis turbine at different RPM of runner of turbine is shown in fig no.4.7.

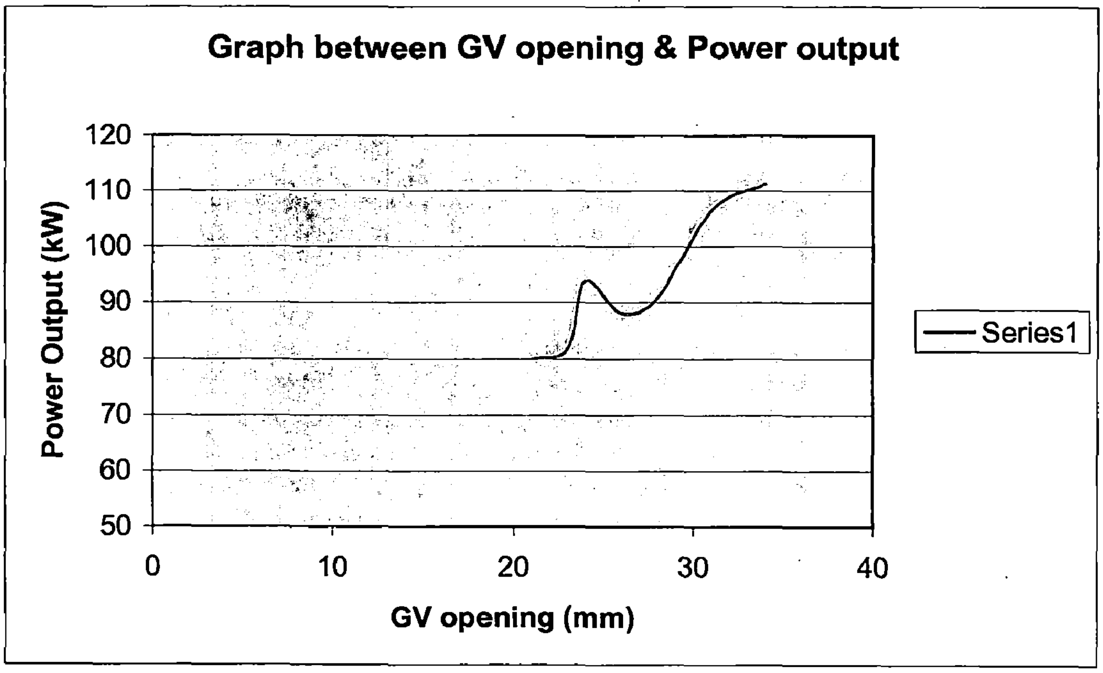


Fig.4.7. Plots between Guide Vanes Opening Positions and Power Output

Plots between different guide vanes opening positions and efficiency of Francis turbine at different RPM of runner of turbine is shown in Fig .4.8.

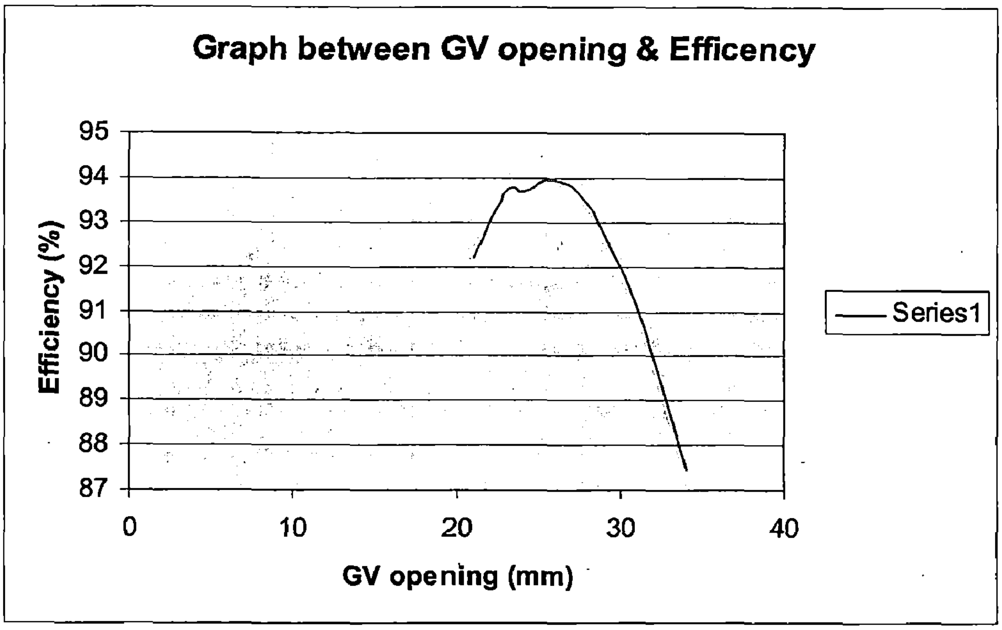


Fig.4.8. Plots between Guide Vanes Opening Positions (mm) & Efficiency (%)

Plots between unit speed (N11) and unit discharge (Q11) at fixed guide vanes opening position is known as Hill chart. With this plot it can be observed that at fixed guide vane opening the maximum efficiency achieved at a particular operating points. By fixing operating condition we can achieve maximum efficiency and best performance from turbine.

HILL CHART

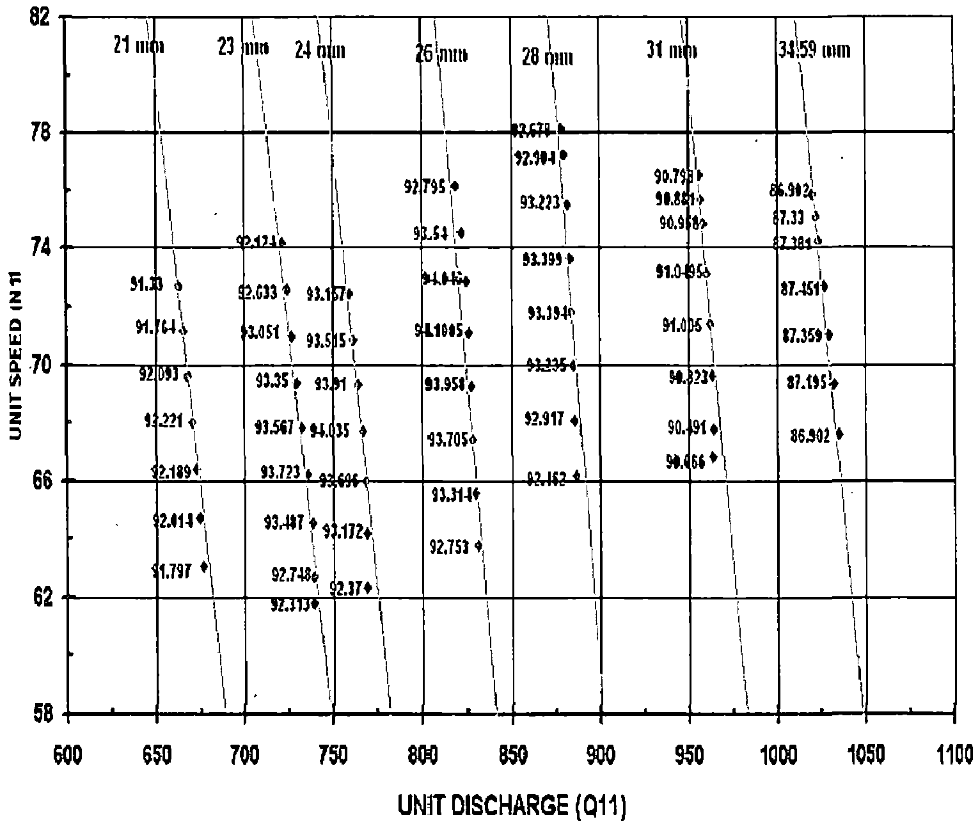


Fig.4.9. Plots between Unit Speed (N11) and Unit Discharge (Q11) at Fixed Guide Vanes Opening Positions

5.1 CONCLUSIONS

In the present dissertation work performance evaluation of Francis turbine by using computational fluid dynamics analysis has been carried out. Such evaluation is in term of design of turbines blades profile and improving its efficiency, following conclusions are drawn from the present study.

1) The maximum efficiency of designed turbine has been obtained at guide vane opening of 60 % (24 mm) as 94.00%.
2) This being model efficiency the prototype efficiency can be in variance than this efficiency.
3) The CFD analysis of Francis turbine provides us the pressure distribution across the whole turbines. It can be easily observed that the localized low-pressure zones inside the turbine may lead to many problems like cavitations, which shall result in loss of efficiency.
4) It facilitates the flow simulation in turbine. Plots of flow simulation shows the velocity vector inside the turbine which is an indication of flow separations such separation leads to loss of efficiency.
5) The abrupt changes on the borders (i.e. the walls and the outlet) are because of the fact that the atmospheric conditions interfere with the computations in the immediate vicinity of the borders.
6) The modeling considerations and meshing can affect the results. A CFD analysis result depends on correct meshing of the assembly of components of turbine and its mesh distortion. Hence results obtained above are subject to limited correction of meshing.

5.2 RECOMMENDATIONS FOR FUTURE WORK

1) Using different CFD codes, it may be possible to find out and compare criteria for classifying runner blade geometry regarding the strengths of their characteristics.
2) There is a scope of flow analysis by changing the design of the blades.

REFERENCES

1) Central Electricity Authority (CEA) website

2) Bureau of Indian Standard (12800: Part III).

3) www.jyoti.com

4) Ajit Thakker, Fergal Hourigan "Computational fluid dynamics analysis of 0.6 m, 0.6 hub-to-tip ratio impulse turbine with fixed guide vanes", engineering village Journal, 2002.

5) P Drtina & M Sallaberger Sulzer Hydro Zürich,Switzerland " Hydraulic turbines-basic principles and state-of-the-art CFD applications"Journal of Energy Engineering, Volume 121, No. 1, April, 1995.

6) A Lipej Turboinštitut Rovšnikova ,Ljubljana, Slovenia "Optimization method for the design of axial hydraulic turbines" Engineering Experiment Station, Oregon, State Coll., Corwallis, Oregon.

7) Cherny S.G.; Sharov S.V.; Skorospelov V.A.; Turuk P.A "Methods for three-dimensional flows computation in hydraulic turbines" ASME, NY, 1982.

8) Guoyi Peng , Shuliang Cao ,Masaru Ishizuka , Shinji Hayama "Design optimization of axial flow hydraulic turbine runner: Part II - multi-objective constrained optimization method" JSME International Journal, Ser. 11, 34(1).

9) T. Behr J. Schlienger A. I. Kalfas R. S. Abhari "Fluid Dynamics and Performance of Partially and Fully Shrouded Axial Turbines" University of Rhode Island Kingston, Kingston, R.I.,1983.

10) J D Denton and W N Dawes "Computational fluid dynamics for turbo machinery design" The Pennsylvania State University, December 2001.

11) John D.Anderson "Computational fluid dynamics analysis of a typical medium-head francis turbine" International Student Edition" Toronto, Nelson, 2007.

12) Water Power 1983 "American International Conference on small scale Hydropower" Tennessee Valley Authority (TVA) and U.S. Corps of Engineers, Washington DC

13) Suhas V. Patankar "Numerical Heat Transfer and Fluid flow", Hemisphere Publishing Corporation, 1992.

14) Personal communication with Corporate R & D ,BHEL,Hyderabad.

15) www.fluentusers.com

16) www.cfd-online.com.

17) www.bhel.com

18) WWW.MICROHYDROPOWER.NET/TURBINES.HTML.

19) www.eee.ntu.ac.uk/reasearch/microhydro/picosite

20) Naidu, Dr. B.S.K (2005), "Small Hydro Highest Density, Non- Conventional, Renewable Energy Source."

21) CEA (1982), "Guidelines for Development of Small Hydro Electric Schemes", Government of India, New Delhi.

22) BHEL journal, Volume 26, No.4, December 2005.

23) BHEL journal, Volume 26, No.3,December 2005.

24) FLUENT Documentation, 2005.

NOMENCLATURE

kW	=	Kilowatt
MW	=	Megawatt
R_1	=	Outer Radius of the Runner
R_2	=	Inner Radius of the Runner
β_1	=	Outer Blade Angle
β_2	=	Inner Blade Angle
r_b	=	Curvature Radius of the Blade
r_p	=	Pitch Circle Radius
δ	=	Segment Angle of the Blade
ϕ	=	Angle in radius between two points on the Spiral and the origin of the Spiral
e	=	Natural Logarithm
K	=	Cotangent of the angle between the tangent to the logarithmically Spiral
Q	=	Discharge
b	=	Inlet Width
ϕ^o	=	Admission Arc Angle
H	=	Net Head
α	=	Angle of Absolute Velocity
L	=	Admission Arc Length
P	=	Pressure
V	=	Velocity
u_i	=	Velocity at the i^{th} Grid Point
Δx	=	Distance between two consecutive Grid Points.
$\hat{n}$	=	Outward Normal to the Surface
C	=	Courant Number
$\vec{V}$	=	Net Velocity Vector