International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 07 | July - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

PERFORMANCE EVALUTATION *OF HYDRO TURBINES BY CFD*

Mr. Pranjul[1],Mr. Aditya veer Gautam[2], Mr. Anurag kumar[3],

Mrs. Shivangi Dixit[4], Mr. Rajneesh Kumar[5]

1Research Scholar, Mechanical Engineering, FET, Rama University, Kanpur
2,3,4,5, Assistant professor, FET, Rama University, Kanpur

---***---

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 inCorporate 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.

Keywords- Small hydro, Pelton turbine, CFD, ANSYS

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].

1.1 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] .

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

Table 1 Various Capacity of Small Hydro Power [1]

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

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 07 | July - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

Table 3 Various Hydro Turbines for Small Hydro Power

1.2 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 x m x g
Power P = T x ω

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

Unit Dis charge Q₁₁ = Q x 1000 /D$^2\sqrt{H}$

Unit Speed N₁₁: N x D / $\sqrt{H}$

where: 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.

1.3 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

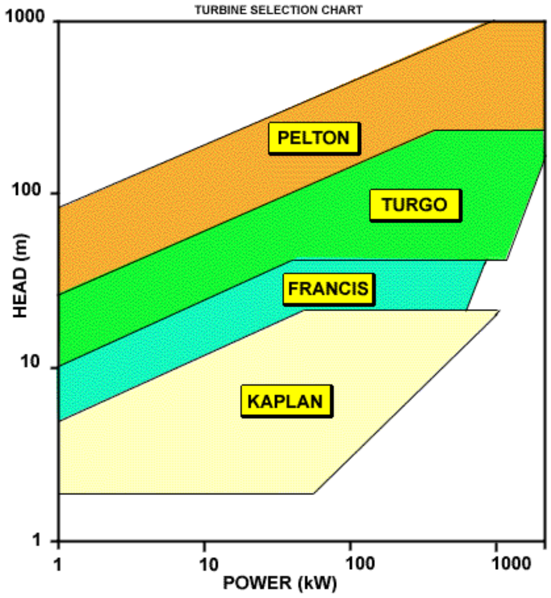

Fig. 1 Turbine Selection Chart [3]

1.4 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

1.5 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

1.6 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

1.7 PERFORMANCE EVALUATION OF 90 MW FRANCIS TURBINE USING CFD

In the present work of studythe 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 scale down up to rate output of 90 kW for analysis purpose. The Scale ratio for analysis is 1:100.

1.8 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

1.9 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 inlet	:	spiral casing
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.

2. 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. ANALYSIS

STEP- 1 In this step analysis is carried out for spiral casing ,stay vanes and guide rings, as shown in Fig 4 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.

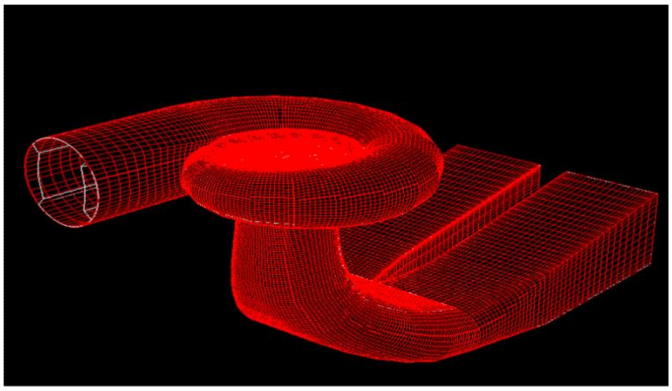

Fig 2 Generation of the Grid under Step-1

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.

Table No. 4 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

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 outletto the guide ring. The boundary conditions are given in Table 5.

Table.5 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 0f 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. 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.

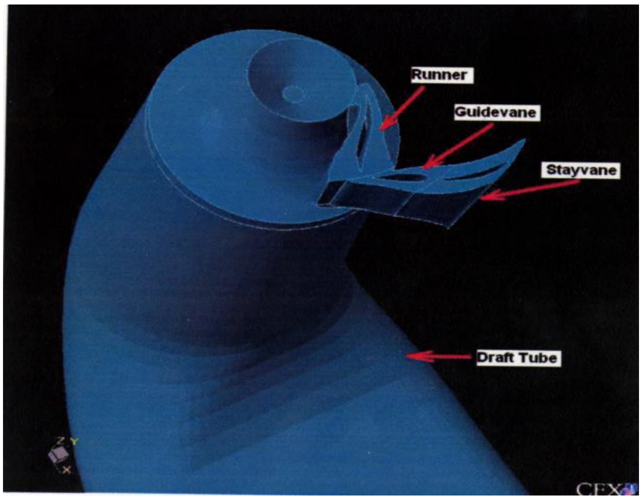

Fig.3 Model for Step -2

3.4 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.

3.5 Graphical Presentation of Results

The various plots obtained from post processing of the results are shown as follow:
Figure 4 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 cavitation's phenomenon over the runner blades. The cavitation's creates many problems like erosion of metal from blade surface, create cracks etc.

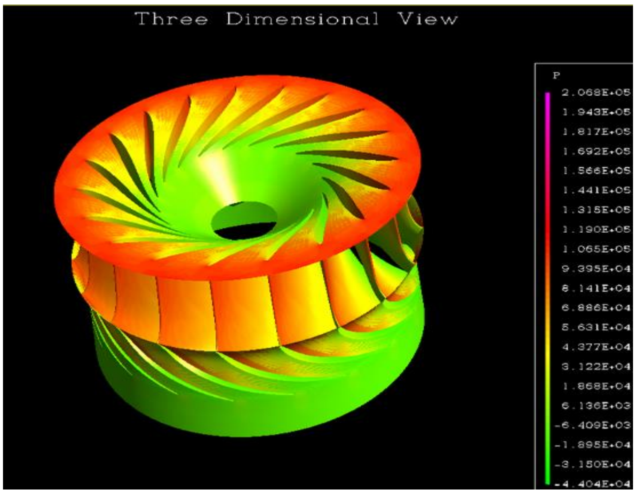

Fig.4 Static Pressure Variation

Figure 5 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 cavitation's effects. The performance of turbine is greatly affected by cavitation's phenomenon.

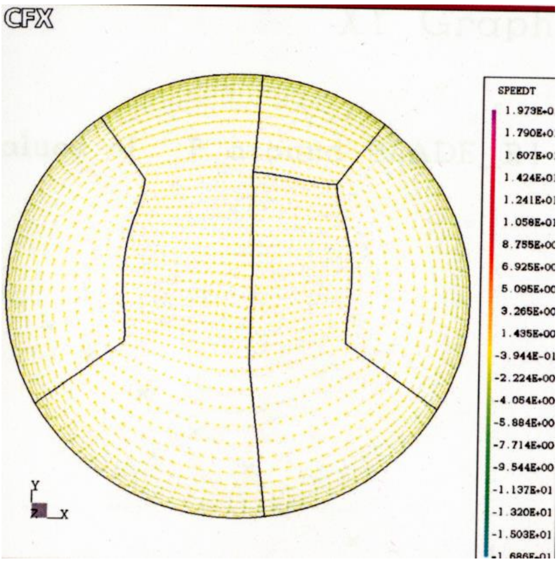

Fig 5 Velocity Plot of Tangential Velocity

Figure 6 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.

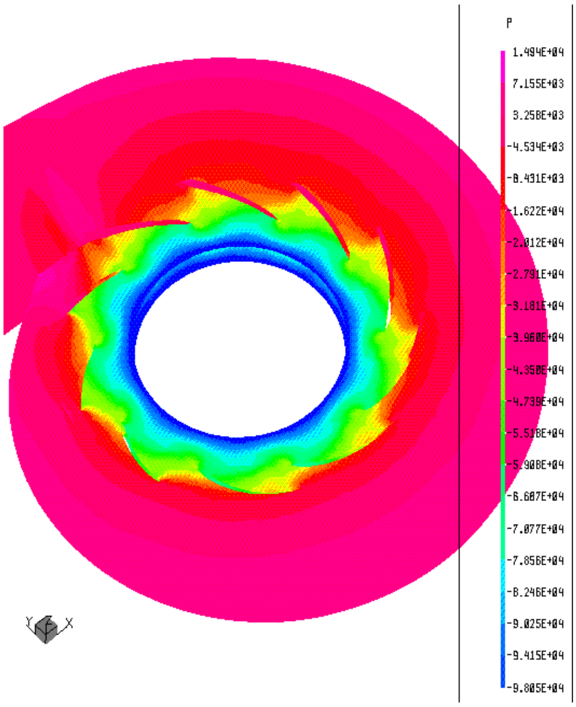

Fig. 6 Pressure Distribution in Spiral Casing

Figure 7 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.6 pressure distribution around spiral casing.

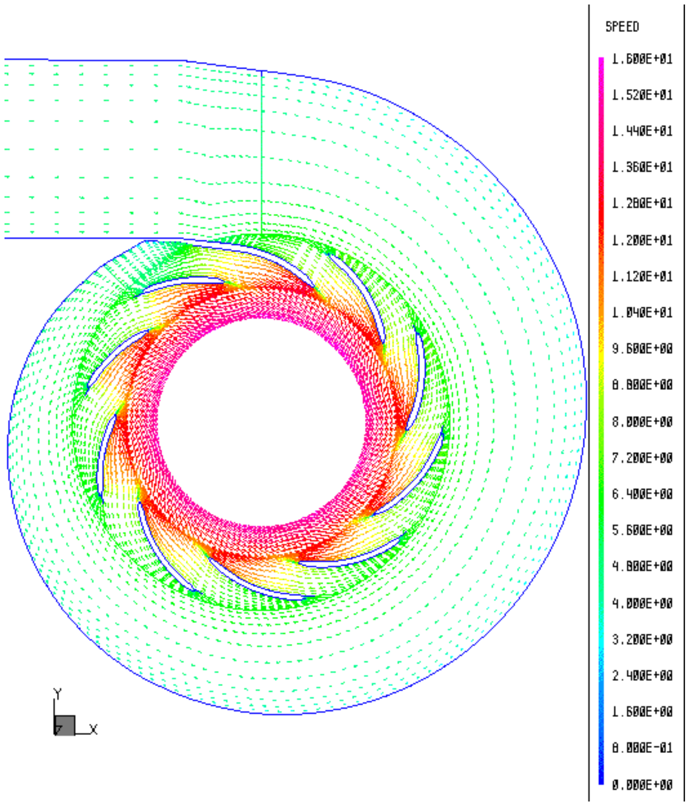

Fig7 . Velocity Distribution in Spiral Casing of Francis Turbine

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.8.

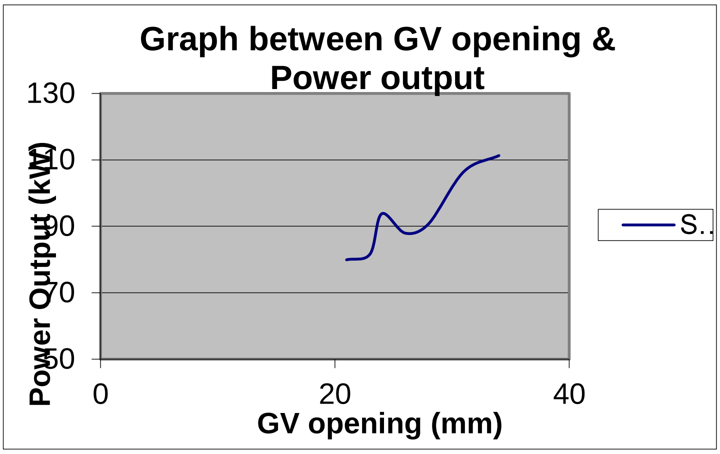

Fig8 .Plots between Guide Vanes Opening Positions and Power Output

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.

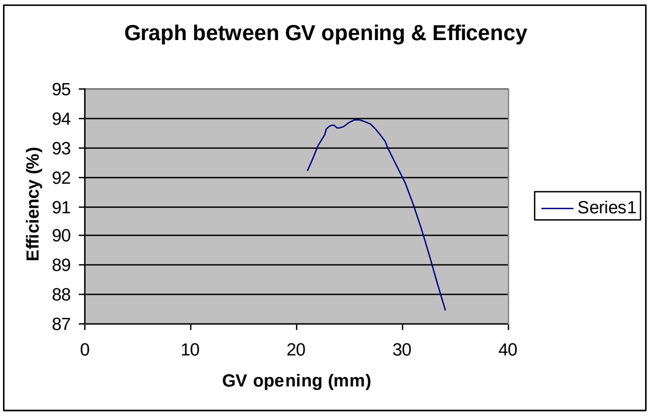

Fig9 .Plots between Guide Vanes Opening Positions (mm) & Efficiency (%)

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**

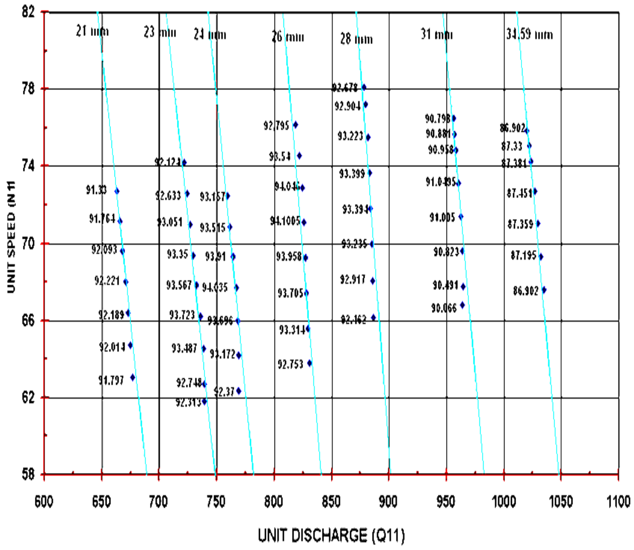

Fig.10 Plots between Unit Speed (N11) and Unit Dischargem (Q11) at Fixed Guide Vanes Opening Positions

4. 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 cavitation, 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.

REFERENCE

1) Central Electricity Authority (CEA) website
2) Bureau of Indian Standard (12800: Part III).
3) www.jyoti.com
4) AjitThakker, Fergal Hourigan “Computational fluid dynamics analysis of 0.6 m, 0.6 hub-to-tip ratio impulse turbine with fixed guide vanes”, engineering lage Journal, 2002.
5) P Drtina& M SallabergerSulzer 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. GuoyiPeng ,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).
8) T. Behr J. Schlienger A. I. Kalfas R. S. Abhari “Fluid Dynamics and Performance of Partially and Fully Shrouded Axial Turbines” University of Rhode IslandKingston, Kingston, R.I.,1983.
9) J D Denton and W N Dawes “Computational fluid dynamics for turbo machinery design” The Pennsylvania State University, December 2001.
10) John D.Anderson “Computational fluid dynamics analysis of a typical medium-head francis turbine” International StudentEdition” Toronto, Nelson, 2007.
11) Water Power 1983 “American International Conference on small scale Hydropower” Tennessee Valley Authority (TVA) and U.S. Corps of Engineers, WashingtonDC
12) Suhas V. Patankar “Numerical Heat Transfer and Fluid flow”, Hemisphere Publishing Corporation, 1992.
13) Personal communication with Corporate R &D ,BHEL,Hyderabad.
14) www.fluentusers.com
15) www.cfd-online.com.
16) www.bhel.com
17) www.microhydropower.net/turbines.html.

18) www.eee.ntu.ac.uk/reasearch/microhydro/picosite
19) Naidu, Dr. B.S.K (2005), "Small Hydro Highest Density, Non- Conventional, Renewable Energy Source."
20) CEA (1982), "Guidelines for Development of Small Hydro Electric Schemes", Government of India, New Delhi.
21) BHEL journal, Volume 26, No.4, December 2005.
22) BHEL journal, Volume 26, No.3, December 2005.
23) FLUENT Documentation, 2005.