CFD APPROACH FOR PREDICTION OF EFFICIENCY OF FRANCIS TURBINE

Sanjay Jain
Assistant Professor
Mechanical Engg Dept,
Institute of Technology,
Nirma University,
Ahmedabad-382481(Gujarat)

R. P. Saini
Associate Professor
Alternate Hydro Energy Centre,
Indian Institute of Technology,
Roorkee-247667(Uttarakhand)

Arun Kumar
Head
Alternate Hydro Energy Centre,
Indian Institute of Technology,
Roorkee-247667(Uttarakhand)

ABSTRACT

Turbine is the most critical component in hydropower plants because it affects the cost as well as overall performance of the plant. Hence, for the cost effective design of any hydropower project, it is very important to predict the hydraulic behavior and efficiency of hydro turbines before they are put in actual use. Experimental approach of predicting the performance of hydro turbine is costly and time consuming compared to CFD approach. The aim of the paper is to predict the performance and efficiency of Francis turbine using CFD approach and to validate the same with model testing results. The overall efficiency of the turbine is determined based on the fundamental equation i.e. ratio of output to input power. The various parameters used in the equation depend on the type of boundary conditions used for the numerical simulation. Two sets of boundary conditions viz. (i) pressure inlet and pressure outlet and (ii) mass flow inlet and pressure outlet were used. The overall efficiency predicted with CFD approach was compared with the model testing results obtained from the manufacturer and very good agreement was found. CFD approach may be helpful in improvement of the existing efficiency measuring techniques and evaluation of the performance of hydro turbines.

1. INTRODUCTION

There are many components in hydropower plant but turbine is the heart of any hydropower plant because it affects the cost as well as overall performance of the whole plant. Typical cost distribution for low, medium and high head hydropower projects is shown in Fig. 1[1]. In case of high head plants the turbine cost is less compared to the cost of civil components as it is very difficult to carry out construction work in hilly areas. But for medium and low head hydropower plants, the typical turbine cost may vary from 15 to 35 percentage of the whole power project cost. Thus, for the cost-effective design of hydropower project it is very crucial to understand the flow characteristics in different parts of the turbine i.e. how energy transfer and transformation take place in the different parts, which help in predicting their performance in advance before manufacturing them. The normal practice to predict the efficiency of a hydro turbine is based on theoretical approach or experimental model testing. Theoretical approach for prediction of efficiency just gives a value; but it is unable to identify the main cause for the poor performance. Conversely, model testing is considered to be costly and time consuming process.

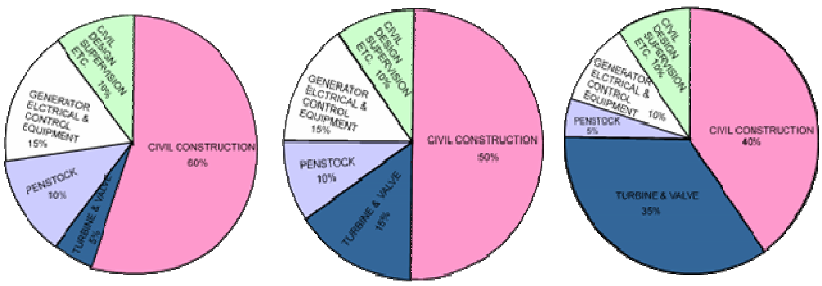

Fig. 1. Typical plant cost for high, medium and low head hydropower projects

Computational Fluid Dynamics (CFD) is the present day state-of-art technique in fluid flow analysis. It has wide range of applications-like aerodynamics of aircraft and vehicles, flow analysis of turbo-machinery, hydrodynamics of ships, power plants, automobiles, process industries, marine engineering, biomedical engineering etc. Also, CFD analysis is considered as a powerful alternative design tool to provide insight into flow characteristic in hydropower components. Many investigators have applied CFD as a numerical simulation tool for the analysis of Francis turbine such as for prediction of part load performance, cavitation behavior, rotor-stator interaction etc. A team from Sulzer Hydro and Sulzer Innotec. [2] modeled a complete Francis turbine-*from the inlet of the spiral casing to the draft tube outlet*- using a 3D Navier Stokes code which can be used to design new runners that match existing components more accurately, at a lower cost than by using model tests.

Ciocan et al. [3] presented a CFD methodology to study the unsteady rotating vortex in the draft tube of a Francis turbine at part load conditions and associated experimental study of the flow phenomena. They performed unsteady Reynolds-Averaged Navier-Stokes (URANS) simulation for the flow and validated the same with experimental results.

Bajic [4] introduced a novel technique for diagnostics of turbine cavitation in a Francis turbine which enables identification of different cavitation mechanisms functioning in a turbine and delivers detailed turbine cavitation characteristics, for each of the mechanisms or for the total cavitation. He conducted the experiments and reported that how a turbine cavitation behavior can be improved and how a turbine operation can be optimized with respect to cavitation erosion.

This paper presents the CFD approach for prediction of efficiency of a 3 MW capacity Francis turbine. The numerical simulations were carried out using commercial CFD package Fluent for the prediction of overall efficiency. The overall efficiency of hydro turbine was determined based on the fundamental equations. The various parameters used in the equations depend on the type of boundary conditions used for the numerical simulation. Two sets of boundary conditions were used. The comparison of CFD results with the model testing results obtained from the manufacturer is also presented.

2. DIFFERENT EFFICIENCIES OF HYDRO TURBINES[5]

For hydropower plants, a gross head is defined as the difference between the head race level and the tail race level when no water is flowing through different components. When water flow through the system the hydraulic losses are occurring which can be categorized as major losses and minor losses. The head loss occurring due to friction in different components is known as major losses whereas the head losses occurring due to either change in direction or cross sectional area of flow i.e. due to bends & pipe fittings, at entrance & exit of penstock etc. are known as minor losses. The minor losses are very small compared to major losses hence can be neglected. Accordingly, the net or effective head acting on the turbine can be found by subtracting hydraulic losses from the gross head.

The total quantity of the water issuing from the jet may not strike the turbine blades. A part of the total discharge may leak through the gap between the runner blades and guide vanes and some flow may escape through the stuffing box around the shaft. The leakage loss is taken into account by considering mechanical efficiency (η_v) which is defined as:

$$\eta_v = \frac{\text{volume of water actually striking the runner blades available at the turbinee shaft}}{\text{total water supplieed by the jet}}$$

$$= \frac{Qa}{Q}$$

When water flow through the turbine hydraulic losses may occur due to blade friction, eddy formation in different components, change in flow direction as well as due to loss in kinetic energy at the exit of the turbine. In power generation, the hydraulic losses are taken into account by considering the hydraulic efficiency (η_h) which is defined as:

$$\eta_h = \frac{\text{power developed by the turbine runner}}{\text{power supplied by the water jet at entrance of the turbine}}$$

$$= \frac{\rho Q a\,(Vw1 \pm Vw2)u}{\rho g Q a H} = \frac{(Vw1 \pm Vw2)u}{gH} = \frac{Hr}{H}$$

where, Hr = $\frac{(Vw1 \pm Vw2)u}{g}$ represents the energy transfer per unit weight of water and is known as 'Euler head' or 'Runner head'.

The power developed by a turbine runner is decreased by mechanical losses caused by friction between the rotating parts (shaft and the runner), friction between the stationary part (bearing and sealing) and by friction in the elements that transmit power. Due to these losses, the power available at the turbine shaft is less than the power developed by the turbine runner. The mechanical losses are taken into account by considering mechanical efficiency (η_m) which is defined as:

$$\eta_m = \frac{\text{power available at the turbinee shaft}}{\text{power developed by the turbine runner}}$$

$$= \frac{2\pi NT/_{60}}{\rho g Q a H r}$$

Overall efficiency (η_o) of the turbine is the product of hydraulic, mechanical and volumetric efficiency. It may be defined as:

$$\eta_o = \frac{\text{power available at the turbinee shaft}}{\text{power supplied by the water jet at entrance of the turbine}}$$

$$= \frac{2\pi NT/_{60}}{\rho g Q H} = \eta_h \times \eta_m \times \eta_v$$

There are various approaches of finding efficiency of the turbine like based on input and output power from the system, based on percentage head drop in different components, based on the losses occurring in different components etc. Drtina and Sallaberger [6] discussed the basic principles of hydraulic turbines, with special emphasis on the use of CFD as a tool which is being increasingly applied to gain insight into the complex three-dimensional phenomena occurring in these machines. They calculated efficiency of Francis turbine based on the pressure losses occurring in the different components.

Patel and Satanee [7] carried out CFD analysis of Francis turbine for the improvement of efficiency, cavitation performance and dynamic behavior. They calculated efficiency based on the percentage head drop in different components. They put thin vane in the bend portion of the draft tube and found that secondary flow in the draft tube can be minimized by providing vane in the draft tube.

Jain et al. [8] carried out the flow analysis of Francis turbine at design and off-design conditions. They calculated the overall efficiency of turbine based on output and input power. They used three different turbulence models namely; standard k-ε, Renormalization group (RNG) k-ε and k-ω shear stress transport (SST) models. They found that k-ω SST model is better suited for simulation in hydro turbines compared to other two models. They observed low pressure zone near the inlet potion of the draft tube at part load operating conditions.

3. COMPUTATIONAL FLUID DYNAMICS

CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The physical aspect of fluid flow in hydro turbines is governed by two fundamental conservation laws:

(i) Conservation of Mass: In all real life conditions mass is always conserved on macro as well as micro levels. The generalized mass conservation equation in differential form is given below:

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

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

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

CFD is the art of replacing these partial differential equations in these fundamental governing equations with discretized algebraic forms, which in turn are solved to obtain numbers using computer 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.

4. CFD PREDICTION OF EFFICIENCY OF HYDRO TURBINE – A CASE STUDY

The CFD approach for prediction of efficiency of a 3 MW capacity horizontal axis Francis turbine is presented. The rated head and discharge for the turbine were 48 m and 7.2 m^3/s respectively. The computational model consists of spiral casing, 18 numbers of stay vanes and guide vanes, runner having 13 numbers of blades and a draft tube. The geometry of the turbine was created in GAMBIT, which is a preprocessor of FLUENT. The computational domain was meshed using unstructured grid consists of triangular and tetrahedral element. The total number of mesh elements was around 2.5 million for the entire assembly. The assembly drawing and grid for the Francis turbine are shown in Fig.s 2 & 3 respectively. The flow in the runner was computed in the moving reference frame, while the flow in the stationary components was calculated in the stationary reference frame. The steady state simulations were carried out at design and off-design conditions using Reynolds averaged Navier-Stokes (RANS) equations with different turbulence models. To consider turbulence effect in the flow shear stress transport k-ω model was used. The simulations were carried out between 50% and 85% guide vane openings at 4 different operating points to cover wide range of discharge.

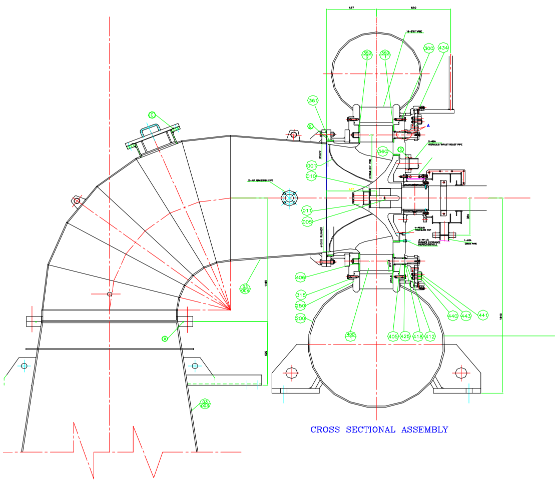

Fig. 2. Assembly drawing of Francis turbine

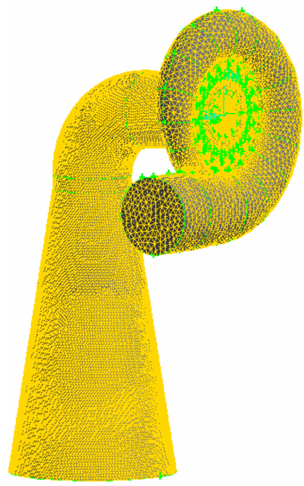

Fig. 3. Grid for Francis turbine

The overall efficiency of hydro turbine was determined based on the fundamental equations. The various parameters used in the equations depend on the type of boundary conditions used for the numerical simulation. Literature suggest different sets of boundary conditions for the CFD analysis of hydro turbines e.g. total pressure inlet & static pressure outlet, mass flow inlet & static pressure outlet [9]. Patel and Satanee [7] used mass flow inlet and pressure outlet boundary conditions for the numerical simulation of Francis turbine. Ruprecht et al. [10] used mass flow inlet and outflow boundary conditions. For the present analysis two sets of boundary conditions viz. (i) pressure inlet and pressure outlet and (ii) mass flow inlet and pressure outlet were attempted and presented in this section.

4.1 Simulations with first set of boundary conditions

For the first set of boundary conditions, *i.e. pressure inlet and pressure outlet*, following input and output parameters were used:

- Input parameters: total pressure inlet (Pt1) was defined at the turbine casing inlet, total pressure considering draft tube submergence was defined at draft tube outlet (Pt2) and turbine runner was defined in moving reference frame with rotational speed (N). Gird interface was defined between stationary & rotating part i.e. between casing & runner as well as between runner & draft tube.
- Output parameters (generated by Fluent): the volume flow rate (Q) was calculated based on the mass fluxes entering and leaving the turbine and torque (T) acting on the turbine was calculated based on the total moment acting on the rotating runner which was a resultant of pressure and viscous moments.

However, after few sets of numerical simulations the results were found diverging and hence simulations were stopped. And for the rest of the analysis second set of boundary conditions were used and presented in the next section.

4.2 Simulations with second set of boundary conditions

For the second set of boundary conditions mass flow inlet was specified at casing inlet and pressure outlet was specified at draft outlet. Grid interface was defined between casing & runner as well as between runner & draft tube. The various boundary conditions are shown in Fig. 4.

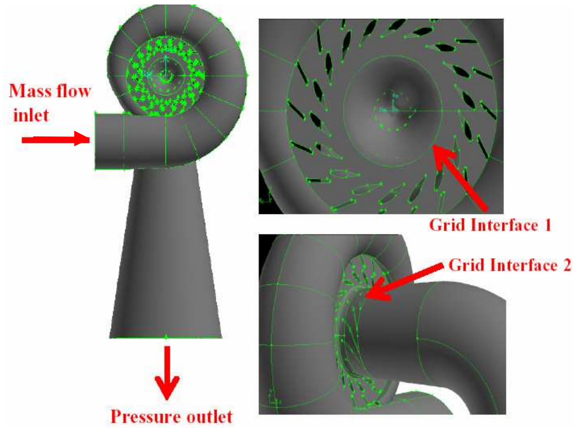

Fig. 4. Boundary conditions

The overall efficiency of the Francis turbine is calculated based on the fundamental equation, i.e. ratio of output power from the turbine to input power supplied to the turbine.

$$\eta_o = \frac{T\,\omega}{Q\,(pt1 - pt2)}$$

In above equation, T is the net torque acting on the runner (N-m), ω is the angular speed (radian), Q is discharge through turbine (m^3/s), p_{t1} is total pressure at the inlet to the casing (Pa) and p_{t2} is the total pressure at the exit of draft tube (Pa). The net torque acting on the runner is a resultant of pressure and viscous moments and is calculated by taking surface integral of cross product of stress tensor and radius vector.

$$T = \int(\vec{r} \times (\tau . n) dS) . a$$

Input parameters:
Mass flow rate (Q) was defined at the turbine casing inlet and total pressure (Pt2) considering draft tube submergence was defined at draft tube outlet. Turbine runner was defined in moving reference frame with rotational

speed (N = 600 rpm) and casing & draft tube were considered in stationary reference frame. The range of input parameters is given in Table I.

Table I: Input parameters

Discharge , Q (m^3/s)	Angular speed, ω (rad/sec)	Pressure outlet, Pt2 (Pa)
8.00	62.8	32127
7.20	62.8	32127
5.93	62.8	32127
4.71	62.8	32127

Output parameters:
Based on the boundary conditions applied in the input parameters, the mass and momentum conservation equations were solved iteratively and various output parameters were generated. The head acting on the turbine (H) is calculated based on the total pressure acting on the turbine and torque (T) acting on the turbine is calculated based on the total moment acting on the rotating runner which is a resultant of pressure and viscous moments. The range of output parameters obtained from FLUENT is given in Table II.

Table II: Output parameters generated by FLUENT

Pressure at casing inlet, Pt1 (Pa)	Torque components			Torque, T (N-m)
	Tx (N-m)	Ty (N-m)	Tz (N-m)	
532456.66	- 889.86	49.11	- 51888.93	51896.59
553658.83	- 650.80	25.25	- 47738.43	47742.88
575276.75	- 445.10	227.10	- 40069.43	40072.55
600116.75	- 1903.68	1465.25	- 31624.23	31715.35

Based on the above parameters obtained from FLUENT software, the turbine power output and overall efficiency of turbine were worked out and compared with the model testing results obtained from the manufacturer as shown in Fig.s 5 & 6 respectively. If volumetric and mechanical efficiency of the turbine are known, hydraulic efficiency can also be worked out using above data. It can be seen that as discharge passing through turbine increases the turbine power output also increases. The overall efficiency of turbine increases with increasing discharge, reaches maximum at design discharge and then starts decreasing. The power output predicted by CFD shows very good agreement with the model testing results obtained from the manufacturer. But some deviation in the overall efficiency was observed; however the trend of both the curves was exactly the same. The deviation in the efficiency may be due to various assumptions, discretization errors, modeling errors and round off errors.

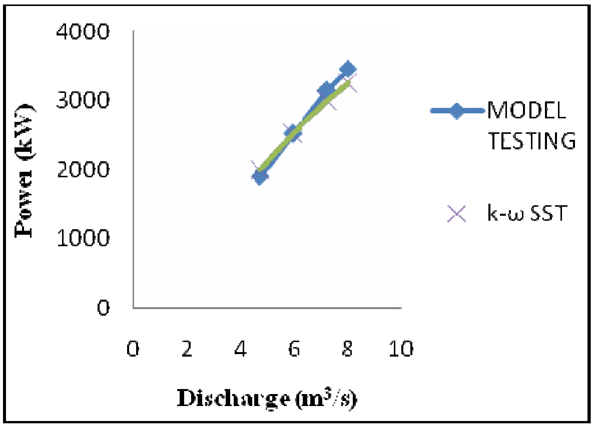

Fig. 5 Power versus discharge curve

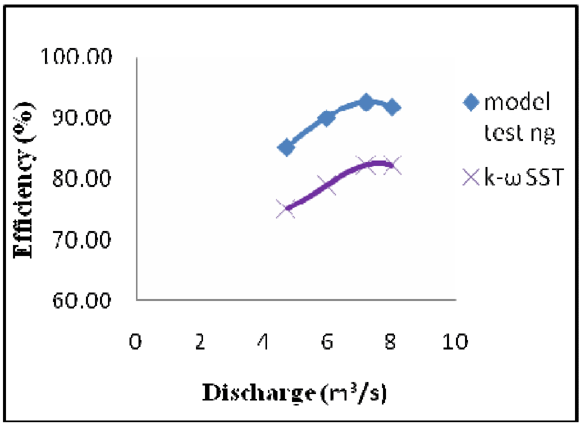

Fig. 6 Efficiency versus discharge curve

5. CONCLUSIONS

The experimental approach of evaluating the performance of hydro turbine is costly as well as time consuming. Conversely, CFD approach is faster and very large amount of results can be produced at virtually no added cost. The CFD approach for prediction of efficiency of Francis turbine was presented in this paper. The numerical simulations were carried out using two sets of boundary conditions viz. (i) pressure inlet and pressure outlet and (ii) mass flow inlet and pressure outlet. However, it was felt that second set of boundary conditions, *i.e. mass flow at casing inlet and total pressure at draft tube outlet,* were better suited for the CFD analysis of Francis turbine. The overall efficiency of turbine was predicted using CFD approach and compared with the model testing results obtained from the manufacturer and very good agreement was found. It can be concluded that CFD approach complements the other approaches, as CFD approach helps in reduction in cost of model testing and saving in time which leads to cost-effective design of the system. CFD approach may be helpful in improvement of the existing efficiency measuring techniques and evaluation of the performance of hydro turbines to enhance the viability of hydropower development.

REFERENCES

1. Saini, R. P., ppt on "Selection of Hydro Turbines", 2006.
2. A Team from Suzler Hydro and Suzler Innotec, "Design by numbers [hydraulic turbines]," International Water Power & Dam Construction, v 50, n 3, 1998.
3. Ciocan, G. D., Iliescu, M. S., Vu, T. C., Nennemann, B., and Avellan, F., "Experimental study and numerical simulation of the FLINDT draft tube rotating vortex", ASME J. Fluids Eng., 129, pp. 146-158, 2007
4. Bajic, B., 2002, "Multidimensional Diagnostics of Turbine Cavitation,", ASME J. Fluids Eng., 124, pp. 943-950.
5. Kumar, D. S., Book on "Fluid Mechanics and Fluid Power Engineering".
6. Drtina, P., and Sallaberger, M., "Hydraulic Turbines – Basic Principles and State-of-the-art Computational Fluid Dynamics Applications", Proceedings of Instn Mech engrs, Vol 213, Part C, 1999.
7. Patel K., and Satanee, M., "New Development of High Head Francis Turbine at Jyoti Ltd. for Small Hydro Power Plant", Himalayan Small Hydropower Summit, Dehradun, 2006.
8. Jain, S. V., Saini, R. P. and Kumar, A., "CFD Based Flow Analysis of Francis Turbine", International Conference on Energy Engineering, Pondicherry, 2009.
9. Best practice guidelines for turbomachinery CFD, 2007.
10. Ruprecht, A., Heitele, M., Helmrich, T., Moser, W., and Aschenbrenner, T., "Numerical Simulation of a Complete Francis Turbine including Unsteady Rotor/Stator Interactions.

BIODATA OF THE AUTHORS

"Prof. Sanjay Jain graduated in Mechanical Engineering from the Gujarat University in 2000. He obtained M. Tech. in Alternate Hydro Energy Systems from Alternate Hydro Energy Centre at Indian Institute of Technology, Roorkee. From 2000 to 2001 he worked at Saurashtra Chemicals Ltd, Porbandar, Gujarat as a maintenance engineer. In 2001 he joined Mechanical Engineering Department at Institute of Technology, Nirma University where he dealt with various subjects and projects related to Fluid Mechanics and Hydraulic Machines".