International Conference on Energy Engineering held at Pondicherry Engineering College, Puducherry, January 2009

CFD BASED FLOW ANALYSIS OF A FRANCIS TURBINE

Sanjay Jain[1] R.P.Saini[2] Arun Kumar[3]

[1]Nirma University of Science & Technology, Ahmedabad-38248, Gujarat, email: svjain5@yahoo.com
[2]Indian Institute of Technology, Roorkee-247667, Uttarakhand, email: saini.rajeshwer@gmail.com
[3]Indian Institute of Technology, Roorkee-247667, Uttarakhand, email: aheciitr@gmail.com

ABSTRACT

Flow analysis in hydropower plants has long been a rigorous subject of research. The cost-effective design of hydro turbine is a challenging task and requires critical analysis of the highly complex flow being turbulent and three dimensional in nature. The flow analysis inside the hydro turbine through experiments is very difficult, time consuming and expensive. Use of Computational Fluid Dynamics (CFD) for the analysis of flow in hydro turbines has increased in recent years due to the development of the high speed computers, evolution of CFD and its easy use.

Flow analysis of Francis turbine carried out using commercial CFD package "FLUENT" is presented in this paper. The computational domain consists of spiral casing, 18 numbers of stay vanes and guide vanes, runner having 13 numbers of blades and a draft tube. The steady state simulation has been carried out at design and off-design conditions using Reynolds averaged Navier-Stokes (RANS) equations with three different turbulence models. To consider the rotational effect, moving reference model with grid interface between stationary and rotating components has been used. Based on the analysis, it is found that non-uniformities are created in different components at off-design conditions which result in decrease in efficiency. In the draft tube, regions of secondary flow, low pressure and vortices are observed at different operating conditions. The CFD results are compared with the results of model testing provided by the manufacturer and are found in very good agreement.

1. INTRODUCTION

Energy plays a significant role in the economic and technological advancement of modern society. The economic development plans implemented since independence have necessarily required increasing amount of energy. As a result consumption of energy in all forms has been steadily rising all over the country. This growing consumption of energy has also resulted in the country becoming increasingly dependent on fossil fuels such as coal, oil and gas. Rising prices of oil and gas and potential shortage in future lead to concern about the security of energy supply needed to sustain our economic growth. Increased use of fossil fuels also causes environmental problems both locally and globally. Against this background, the country urgently needs to develop a sustainable path of energy development. Promotion of energy conservation and increased use of renewable energy sources are the effective methods for sustainable energy supply.

Hydro power is probably the most oldest and yet the most reliable source of all renewable energy, with bulk of its potential yet to be harnessed in many countries. In hydropower sector, among different components turbine is the most critical component because it directly affect the cost of civil works, overall performance as well as life of whole project. Flow analysis of hydro turbines is often a challenging task as it requires critical analysis of the highly complex flow which is turbulent and three dimensional in nature and having rapidly changing flow passage. The flow analysis inside the hydro turbine is very difficult, time consuming and expensive through experiments. Use of Computational Fluid Dynamics (CFD) as a tool for the analysis of flow in hydro turbines has increased in recent years due to the development of the high speed computers and evolution of CFD.

For more than a decade CFD is used in the field of hydraulic machinery in research and development as well as in design. Starting with Q3D-Euler and 3D-Euler today usually the Reynolds averaged Navier-Stokes equations together with a robust model of turbulence (usually the k-ε model) is used [1]. Many researchers have used CFD as a tool to analyze the flow inside the hydro turbine. Michel et al. [2] described Alstom's experience in various hydropower stations of different countries, in the hydro turbine rehabilitation field. They found that, in most of the cases runner and guide vanes can be easily replaced, but modifications affecting the mechanical structure of the machine such as stay vanes reshaping or the concreted parts such as spiral case or draft tubes modifications were rare and only happened in extreme cases. They felt that, to achieve the better performance, deep study in the form of CFD calculations and then a checking by model test is generally necessary.

A team from Sulzer Hydro and Sulzer Innotec. [3] modelled 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.

Guo et al. [4] performed Large-Eddy Simulation of Non-Cavitating Cavitating Flows in the draft tube of a Francis turbine. They found that, under the part load conditions, the whirl of a vortex is produced in the draft tube and under certain operation conditions it may cause flow instability that leads to vibration or noise. They also mentioned that the operation range of the turbine is strongly related to the cavitation phenomena, which may occur either in the vanes of the runner or in the stationary parts.

In this paper, flow analysis of 3.14 MW Francis turbine carried out using commercial CFD package "FLUENT" is presented. The salient features of the turbine are given in Table 1.

Table 1 Salient features of Francis turbine

Turbine Capacity	3.14 MW
Rated Head	48 m
Rated Discharge	7.2 m^3/s
Rated Speed	600 rpm
Specific Speed	266.17 (SI units)
Guide Vane Opening	80.93 mm
Runner inlet diameter	0.71 m
Runner outlet diameter	1.01 m

2. COMPUTATIONAL FLUID DYNAMICS

Today, CFD has become an equal partner with pure theory and pure experiment in the analysis and solution of fluid dynamics problem. Compared to theoretical approach to solve fluid flow problems, the CFD approach has the advantage that it can provide a solution for a much more complex problem. And compared to experimental approach, CFD approach can produce extremely large volumes of results at virtually no added expense. However, future advancement of fluid dynamics will rest upon a proper balance of all the three approaches, with computational fluid dynamics helping to interpret and understand the results of theory and experiment, and vice versa.

CFD is a numerical technique to obtain an approximate solution numerically. We have to use a discretization method, which approximate the differential equations by a system of algebraic equations, which can then be solved on a computer. The approximations are applied to small domain in space and/or time so that the numerical solution provides results at discrete locations in space and/or time.

The physical aspect of any fluid flow in hydro turbines is governed by the following two fundamental principles:

(i) Conservation of Mass
(ii) Conservation of Momentum

These fundamental principles can be expressed in terms of partial differential equations. CFD is a numerical technique to replace these partial differential equations of fluid flow into the algebraic equations by numbers and discretizing them in space and/or time domain. With the advent of high speed digital computers, CFD has become a powerful tool to predict flow characteristics in various problems in an economical way.

3. STEADY STATE NUMERICAL SIMULATION

The computational domain consists of spiral casing, 18 numbers of stay vanes and guide vanes, runner with 13 blades and a draft tube. Total 12 sets of numerical steady state simulations were carried out to get the performance data of Francis turbine. The flow in the rotor was computed in the moving reference frame, while the flow in the stationary components was calculated in the stationary reference frame.

3.1 Modeling and Mesh Generation

The geometry of the computational domain of the Francis turbine is created using bottom-up approach in GAMBIT which is a preprocessor of FLUENT. As the geometry is complex unstructured grid consists of triangular and tetrahedral element is used. The total number of mesh elements was around 2.5 million for the entire assembly. A grid around the runner in Fig. 1 and grid for whole assembly is shown in Fig. 2. It can be seen that, finer mesh is used near the runner blade profile and size of mesh elements increases away from these profiles.

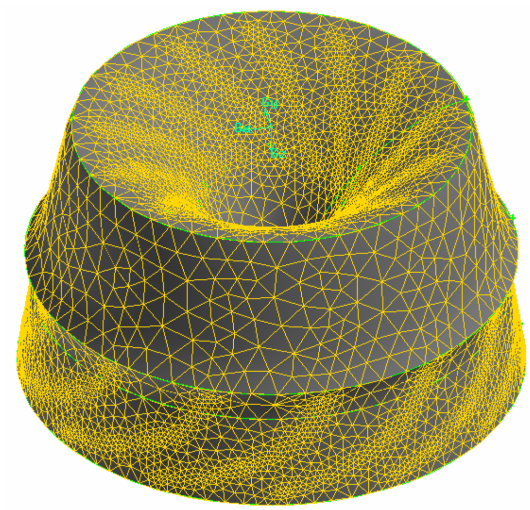

Fig. 1 Grid for the runner

3.2 Boundary Conditions and Operating points

The simulations were carried out over a four different guide vane openings with three different turbulence models namely standard k-ε model, RNG k-ε model and SST k-ω model. The flow is assumed to be steady state with incompressible single phase fluid. Mass flow rate was specified at the spiral casing inlet while static pressure was defined at the outlet boundary. Between inter acting components grid inter face is used as shown in Fig. 3. Runner is defined in moving reference frame.

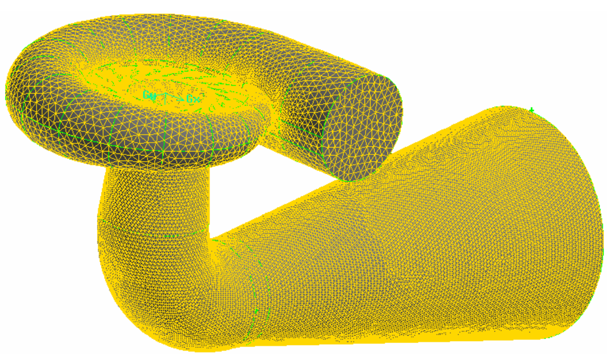

Fig. 2 Grid for whole assembly

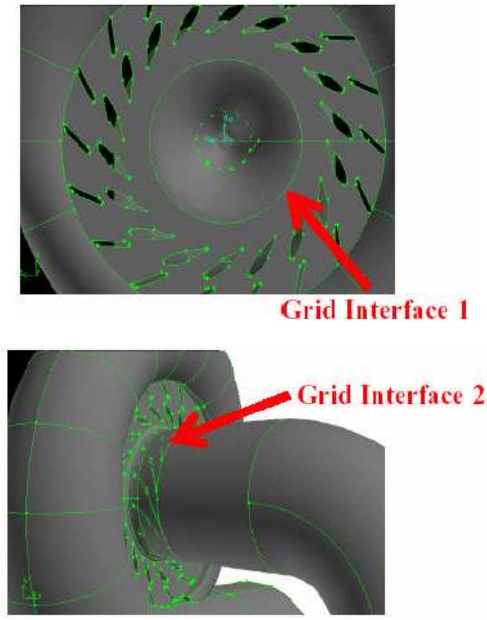

Fig. 3 Boundary conditions between interacting components.

4. RESULTS AND DISCUSSION

To predict the turbine performance, it is very essential to know, how energy transfers in the different components of the turbine. This phenomenon can be understood through the study of flow field inside the turbine. The results of CFD analysis correspond to rated conditions have been presented for different components in the form of pressure and velocity contours as well as velocity vectors. The detail description of result is as follows.

4.1 Casing, stay vanes and guide vanes

The spiral casing is designed such that, as the water flow through the casing, some amount of water enters through the stay vane passage and simultaneously the area of casing decreases such that at the runner inlet velocity becomes same throughout the circumference. The variation of static pressure in the plane passing through centre of spiral casing and across the axis of rotation is shown in Fig. 4. It can be seen that, as the water flow towards the runner the velocity increases gradually in radial direction which shows uniform conversion of pressure energy into kinetic energy. In the vicinity of the casing high velocity gradient has been observed which is due to the viscous effect of the fluid. Near the nose vane, abrupt changes in velocity and pressure are observed, which may be due to rebound of flow from the nose vane.

The velocity vectors in the volute passage of casing is shown in Fig. 5. It can be seen that velocity vectors are quite uniform and do not show pattern of secondary flow. Similar results were reported by Patel and Satanee [5].

4.2 Runner

The variation of static pressure and velocity in water around runner passage are shown in Figs. 6 and 7 respectively. It can be seen that, as the water flow though the runner passage the pressure and velocity gradually decreases which is due to the conversion of pressure and kinetic energies into mechanical energy in the form of runner torque. It is seen that, on pressure side of the blade higher pressure is generated whereas on suction side of the blade lower pressure is generated. It is also observed that, due to forced vortex flow through the runner low pressure zone is created below the hub of the runner.

4.3 Draft tube

The variation in static pressure and velocity vectors in the flow through draft tube are shown in Figs. 8 and 9 respectively. It is seen that, the pressure distribution is uniform up to the bend portion. Further, near the bend portion flow has been observed to be accelerated due to change in flow direction which results in lower pressure in this region; hence this region may be prone to cavitation. The secondary flow and vortices are also observed near the inlet portion of the draft tube at part load operating conditions.

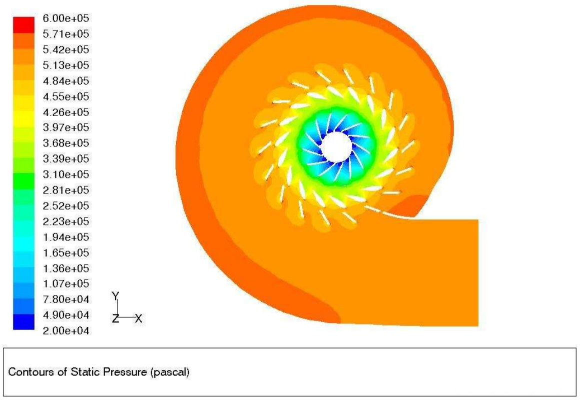

Fig. 4 Contours of static pressure in the central plane of casing

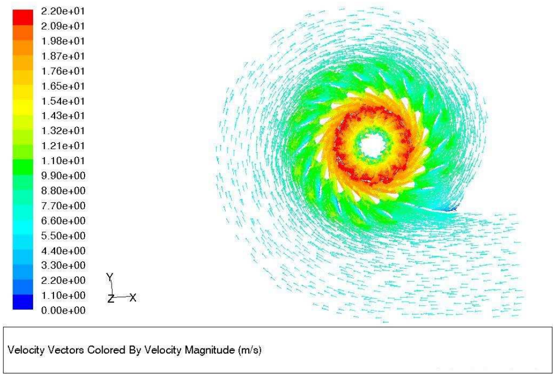

Fig. 5 Velocity vectors in volute passage of casing

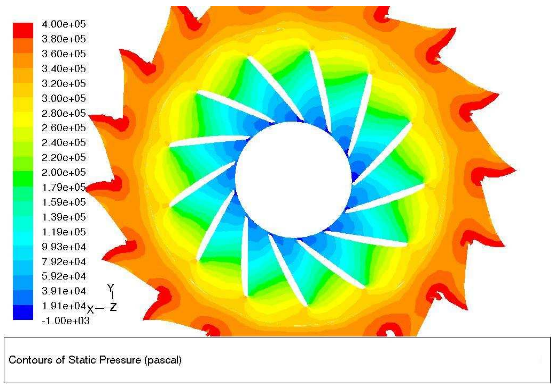

Fig. 6 Contours of static pressure around runner (central plane of casing)

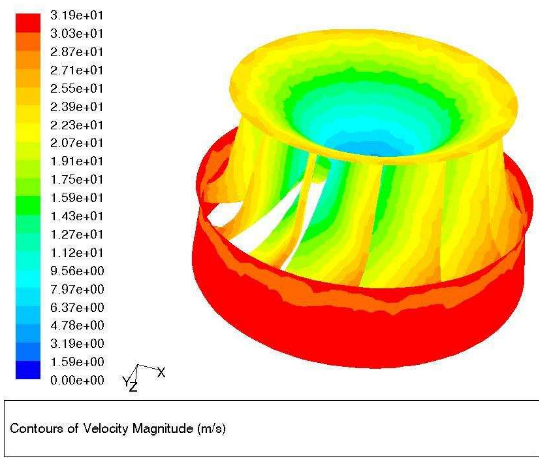

Fig. 7 Contours of velocity around runner

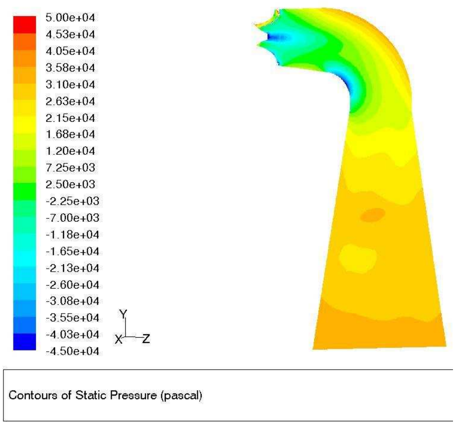

Fig. 8 Contours of static pressure in the flow through draft tube

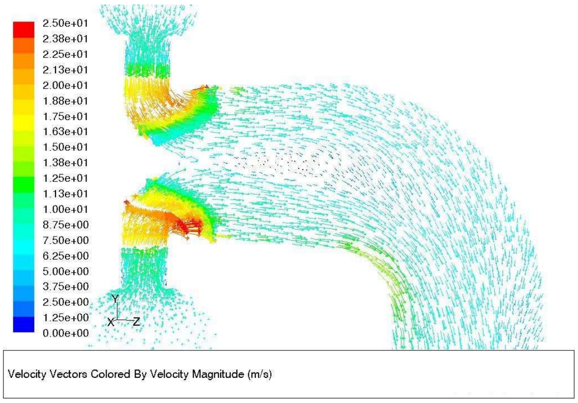

Fig. 9 Velocity vectors in the flow through draft tube

5. OPERATING CHARACTERISTIC CURVES

In hydro turbines, basically three independent parameters are there namely speed (N), head (H) and discharge (Q). For operating characteristic curves, the speed and head are kept constant and the variation of power and efficiency with respect to discharge is plotted. The results obtained with three different turbulence models were compared with the results of model testing and it seems that SST k-ω model is better suited for the flow analysis in hydraulic turbo-machinery. The comparison between CFD results and the results of model tests is also presented.

5.1 Shaft Power Vs Discharge Curve

Fig. 10 shows variation of turbine power output with increase in discharge. To plot this curve, the speed and head on the turbine are kept constant and variation of shaft power output is plotted with increase in discharge by opening the guide vanes. It can be seen that as discharge increases, the power output from turbine increases continuously even beyond the rated conditions because more amount of input is provided to the turbine by the water. Comparison of CFD results with that of model testing shows good agreement.

5.2 Efficiency Vs Discharge Curve

Fig. 11 shows variation of turbine efficiency with increase in discharge. As mentioned, the speed and head on the turbine are kept constant and variation of efficiency is plotted with increase in discharge by opening the guide vanes. It can be observed that as discharge increases, the efficiency increases, reaches maximum at rated conditions and then decreases when discharge increases beyond rated conditions, i.e. parabolic profile. As such, there is some deviation in CFD results compared to results of model tests but the trend is exactly the same, which shows quite good agreement between CFD results and results of model testing.

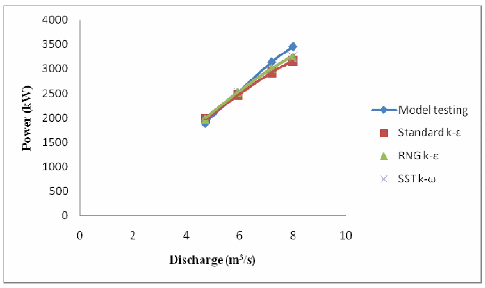

Fig. 10 Power versus discharge curve for different operating conditions

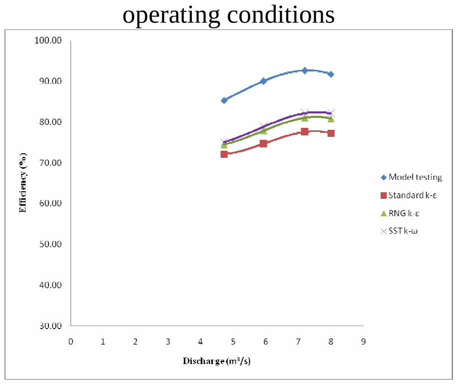

Fig. 11 Efficiency versus discharge curve for different operating conditions

6. CONCLUSIONS

Flow analysis of Francis turbine carried out using commercial CFD package "FLUENT" is presented in this paper. The simulations were carried out at four different operating points with three different turbulence models. The following conclusions are drawn from the analysis:

i. SST k-ω turbulence model has been found to provide better results in comparison with standard k-ε and RNG k-ε models.

ii. High velocity gradient has been observed in the flow in the vicinity of the casing, which is due to viscous effect of the fluid.

   Further, near the nose vane portion, abrupt change in velocity and pressure has been observed which may be due to rebound of flow from the nose vane.

iii. It is seen that the pressure side of the blade is subjected to higher pressure compared to suction side and low pressure zone is created below the hub of the runner.

   This may be due to forced vortex flow through the runner.

iv. In the draft tube, the regions of secondary flow, back flow and vortices are observed at lower discharge than rated discharge.

   It is further found that, due to acceleration of the flow in the draft tube, pressure in the bend portion drops below the vapor pressure of water and hence this region may be prone to cavitation.

v. The CFD results are compared with the model test results and some deviation is observed. However, the trends of power and efficiency variation are similar as that of model testing. It can be concluded that, the CFD results shows very good agreement with results of model testing.

REFERENCES

1. Ruprecht, A., "Unsteady Flow Analysis in Hydraulic Turbomachinery," 20th IAHR Symposium on Hydraulic Machinery and Cavitation, Charlotte, 2000.
2. Michel, B.,Couston, M., Francois, M., and Sabourin, M., "Hydro turbines rehabilitation," IMechE Event Publications, v 2004 6, Hydropower Developments - New Projects, Rehabilitation, and Power Recovery - IMechE Conference Transactions, pp. 3-12, 2004.
3. A Team from Suzler Hydro and Suzler Innotec, The article on "Design by numbers [hydraulic turbines]," International Water Power & Dam Construction, v 50, n 3, March 1998, 20-2.
4. Guo, Y., Kato, C., and Miyagawa, K., "Large-Eddy Simulation of Non-Cavitating and Cavitating Flows in the Draft Tube of a Francis Turbine"
5. Patel, K., and Satanee, M., " New Development of High Head Francis Turbine at Jyoti Ltd. For Small Hydro power Plant," Proceedings of Himalayan Small Hydropower Summit (HSHS), Dehradun, 2006.
6. Jain, S.K., Saini, R.P., and Kumar, A., "CFD Based Flow Analysis of Hilly Small hydropower Station," M.Tech. dissertation, AHEC, IIT, Roorkee, June 2008.