HYDRAULIC PERFORMANCE OF ELBOW DRAFT TUBE FOR DIFFERENT GEOMETRIC CONFIGURATIONS USING CFD

Vishnu Prasad
Associate Professor
Department of Civil Engineering
M.A. National Institute of Technology, Bhopal (India) – 051

Ruchi Khare
Assistant Professor
Department of Civil Engineering
M.A. National Institute of Technology, Bhopal (India) – 051

Abhas Chincholikar
PG Student
Department of Civil Engineering
M.A. National Institute of Technology, Bhopal (India) – 051

SYNOPSIS

The hydraulic reaction turbines are provided with draft tube to recover the part of kinetic energy coming out of runner. The energy recovery depends on the design of draft tube. The numerical flow simulation for 3D viscous turbulent flow has been carried out in elbow draft tube by varying its parameters like length and height at different mass flow rate using Ansys CFX code. The draft tube efficiencies and losses are computed from pressure and velocity distributions and presented graphically to study the effect of geometrical parameters on draft tube performance. The predicted geometrical parameters from numerical simulation for the best performance are closely matching to the geometry of draft tube used in most of the hydro power stations.

Keywords: Draft tube, hydraulic turbine, numerical simulation, hydro power, efficiency, energy, elbow.

1. INTRODUCTION

The draft tube used in hydraulic reaction turbine has gradual increase in cross sectional area from its inlet to outlet. It is one of the important component of reaction turbine and connects runner exit to tail race. The main functions of draft tube is to allow the installation of turbine above the tail race level without loss of head and to convert major part of kinetic energy coming out of runner into pressure energy. In mixed flow reaction turbines, kinetic energy from runner is up to 15% whereas in low head and high speed axial flow turbines, kinetic energy leaving the runner may go up to 50% of total input energy. The recovery of kinetic energy is achieved by increasing the cross-sectional area of the draft tube in the flow direction. Initially, the straight conical tubes with inlet and outlet areas of different cross-section were used.

The hydrodynamic investigations on straight draft tubes were carried out between 1903 and 1907 and investigators adopted walls of draft tube parallel to streamline based on theoretical solutions. A large number of investigations were carried out on straight diffusers during the period 1909 to 1929 leading the design of various draft tubes. The use of straight tubes was restricted to turbines of medium and small diameters because due to increase of the diameter of the runner D_1, the length of the tube became so large that it is irrational to construct such tubes. The recovery of the kinetic energy of axial and rotational flow can be best achieved in bell mouth tubes. The use of such tubes for large runner diameters has again restriction due to support problem of such large dimensions and weight. All these problems are overcome by elbow draft tube for large diameter hydraulic turbines [3, 4].

The determination of optimum shape and dimension of the draft tube and its element is a very difficult problem and has not been solved until now. The height of curved draft tube has a great influence on the efficiency and power output of turbine. It is seen that reduction of height of draft tube from h= 1.915 D_1 to 1.54D_1 in Kaplan turbine, efficiency is reduced by 5% and when height is increased from 1.915 D_1 to 2.3 D_1 efficiency is increased by 3.5%. At Volozsky hydro electric station, it was found that the use of standard draft tube of height h=2.24 D_1 instead of 1.915D_1 has made it possible to obtain an additional power output of 100 to 150 kWh [2].

Later model testing has been used to study the influence of the draft tube geometry more accurately. The dimensions for various draft tubes are given based on comprehensive hydro dynamical calculations and experimental studies[2]. These dimensions were then used in the preliminary design phase of draft tubes and model tests are adopted to avoid serious mistakes. Scale-up formulas are then employed to translate the model design to the prototype. A model test approach is, however, both time consuming and expensive. The computational fluid dynamics (CFD) became an alternative and very attractive tool for flow simulations. The CFD codes have now also matured to provide substantial insight to hydro turbine design and development applications.

The numerical simulation in elbow draft tube of using Finite Element based Numerical Flow Simulation with modeling with standard k-ε model and the extended k-ε model of Chen-Kim and Very Large Eddy Simulation (VLES) known as promising tool for prediction of unsteady phenomena. All steady state calculations converged well. Slight difference of results between k-ε and Chen and Kim model is noticed, especially in vortex behind the runner [8]. The parallel performance of CFD software CFX5.7.1 on homogeneous computer networks has been tried to capture in efficiency improvement by modifying the shape of draft tube. The same inlet velocity profile is used in geometry modifications. It is found that there was no noticeable improvement in pressure recovery factor or flow field between original and redesigned draft tube geometry in CFD simulations for steady and unsteady flow analysis as compared to experiments [9]. The performance of separate draft tube has been done using CFD simulations in order to clarify the best inflow conditions. The detailed unsteady CFD simulations were carried out in order to recognize draft tube separation phenomenon more precisely [10]. In present paper, 3D viscous flow simulations has been carried out in elbow draft tube for different geometry to study the effect of height and length on performance of draft tube.

2. GEOMETRIC MODELLING AND BOUBDRY CONDITIONS

The elbow draft tube has three parts namely cone, elbow and diffuser. The geometric dimensions of elbow draft tube are shown in fig.1.

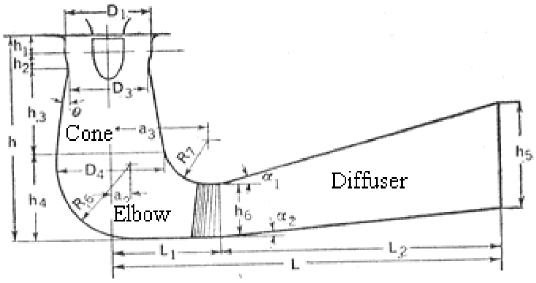

Fig.1- Geometric parameters of draft tube [2]

The geometry modeling of draft tube is carried for four height h/D_1 and five lengths L/D_1 ratios [2]. The isometric view of the modeled tube for h/D_1=2.24 and L/D_1=5.5 is shown in fig.2.

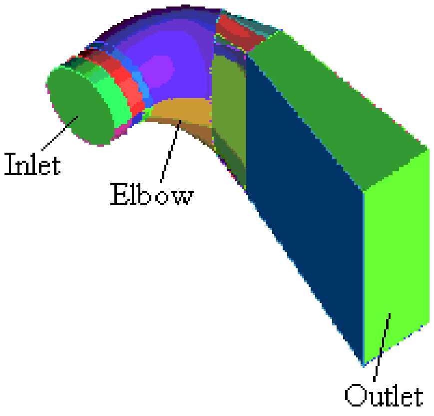

Fig.2 – Isometric view of modeled draft tube

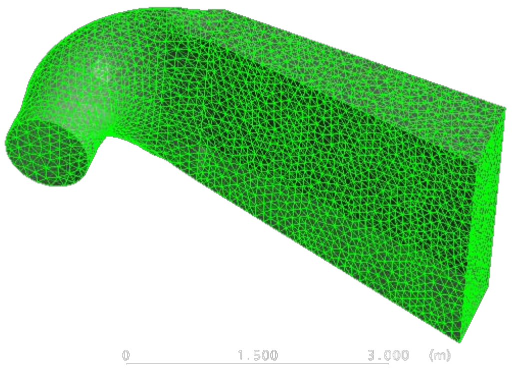

Fig.3- Meshing of draft tube

The meshing of the draft tube flow domain is generated in ANSYS ICEM CFD as shown in fig.3. The mesh generation is done to convert the large domain into number of small elements. Each element consists of nodes at which flow variables are calculated. The unstructured triangular elements on surfaces and tetrahedral in flow domain are adopted in draft tube meshing. The fine mesh is done near the surfaces in order to capture the boundary layer and turbulence as compared to interior domain. This has reduced the number of grids points and hence computational time.

The mass flow rate normal to surface at inlet of draft tube cone is specified as inlet boundary condition. The static pressure at outlet of draft tube is specified as outlet boundary condition. The walls of draft tube are assumed to be smooth with no slip. Shear Stress Transport (SST) κ-ω turbulence model in Ansys CFX code has been used for analysis due to boundary curvature in elbow and diffusing flow.

3. FORMULAE USED

The following formulae are used to compute flow parameter for draft tube performance using pressure and velocity obtained from the numerical simulation:

Head loss in draft tube

$$H_{LD} = \frac{P_{03} - P_{05}}{\gamma} \qquad (1)$$

Head recovery in draft tube:

$$H_{RD} = \frac{(V_3^2 - V_5^2)}{2g} - H_{LD} \qquad (2)$$

Efficiency of draft tube

$$\eta_D = \frac{2gH_{RD}}{V_3^2} x100 \qquad (3)$$

Relative Loss

$$H_{RLD} = \frac{2gH_{LD}}{V_3^2} x100 \qquad (4)$$

where P_0 and V are the average values of total pressure and velocity. γ and g are specific weight of water and acceleration due to gravity respectively. The subscripts 3 and 5 stands for corresponding values at inlet and outlet of draft tube.

4. RESULTS AND DISCUSSIONS

The viscous 3D turbulent flow analysis simulation has been carried out in elbow draft with four h/D_1 ratios of 1.54, 1.915, 2.24, 3.0 and five L/D_1 ratios of 4.0, 4.5, 5.0, 5.5, 6.0 for three mass flow rates of 15000, 20000 and 25000 Kg/s.

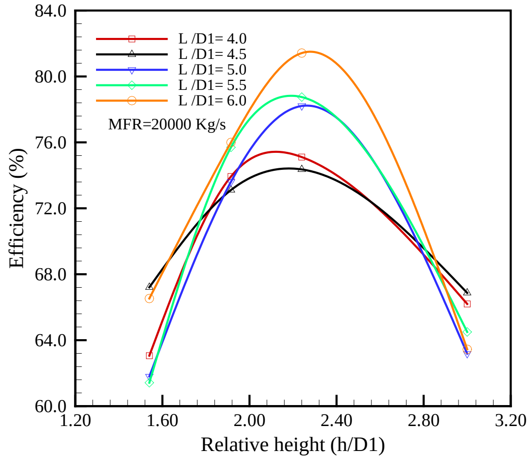

Fig.4- Efficiency at different L/D_1 ratios

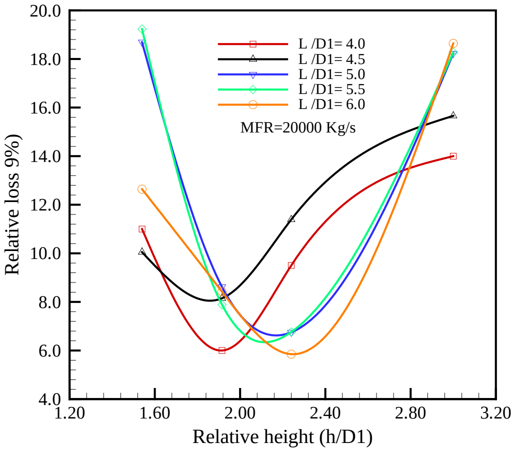

Fig.5- Loss at different L/D_1 ratios

The efficiency and loss in each simulation are computed and presented graphical form in fig.4 to fig.9. It is seen from fig.4 that efficiency of draft tube for constant mass flow rate (MFR) of 20000Kg/s has parabolic variation giving maximum efficiency at the height ratio of 2.24. This may be due to insufficient length for conversion of kinetic energy at low height ratios and large eddy formation and flow separation at more height ratios. The variation of loss is reverse to that of efficiency but minimum loss points lie between h/D_1 ratio of 1.915 and 2.24 as shown in fig.5 due to more frictional loss at low height ratios and eddy losses at more height ratios.

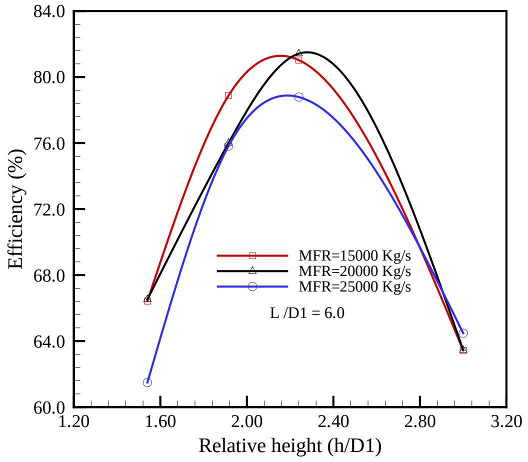

Fig.6- Efficiency at different mass flow rate

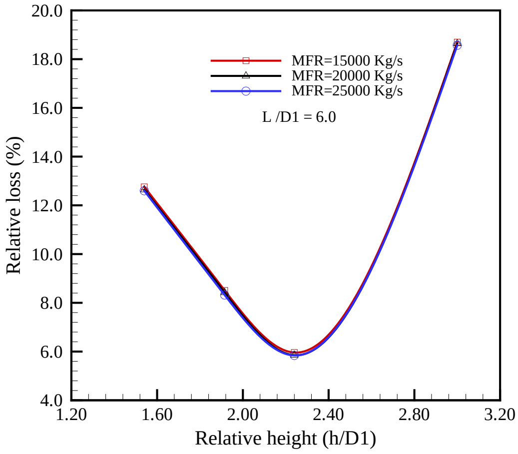

Fig.7-Loss at different mass flow rate

The value of maximum efficiency increases with increase in L/D_1 ratios and the best efficiency is obtained at length ratio L/D_1 of 6.0. It is also observed from analysis that mass flow rate has little effect at constant length ratio. The efficiency slightly decreases with increase in mass flow rate as seen in fig.6. There is no effect of mass flow rate on loss variation in fig.7.

Similarly mass flow rate has no effect on efficiency and loss characteristics of draft tube as seen in fig.8 and fig.9 respectively for constant h/D_1 ratio because of increase in kinetic energy at inlet with increase in mass flow rate and accordingly increase in head recovery and losses.

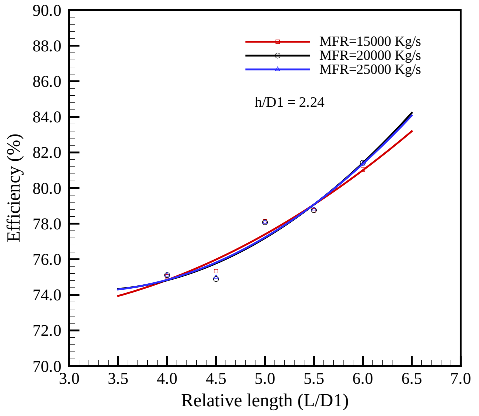

Fig.8- Efficiency at different mass flow rate

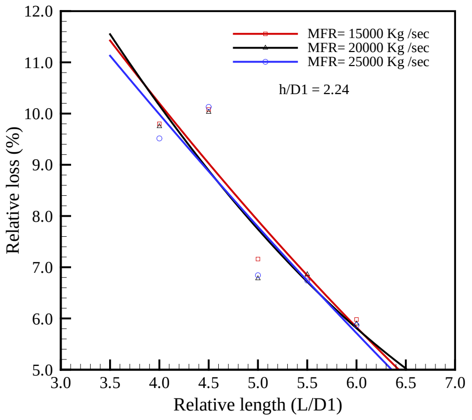

Fig.9- Loss at different mass flow rate

In fig.9, the efficiency gradually increases with increase in L/D_1 ratio. This may be due to increased length of diffuser for conversion of energy with increased flow area. There is opposite trend for the losses to that of efficiency with L/D_1 ratio for all three mass flow rates.

5. CONCLUSIONS

It is seen from the simulation in draft tube that both height and length of draft tube has significant effect on performance of elbow draft tube. The mass flow rate has nearly no effect on efficiency and loss in draft tube. The best performance is achieved close to height ratio of 2.24 and length ratio L/D_1 of 6.0. It is found that most of the hydro power plants has used elbow draft around the to height ratio of 2.24 and length ratio L/D_1 of 6.0 and hence the results from numerical simulation have also been validated. It may be concluded that CFD is very effective tool for numerical flow simulation in complex flow domains with reasonable accuracy.

6. REFERENCES

[1] G.T. Csanady, *Theory of Turbo machines*, McGraw Hill, New York, 1964.

[2] M.F. Gubin, *Draft Tubes of Hydraulic Station,* Amerind Publishing Company Pvt. Ltd, 1973.

[3] M. Nechleba, *Hydraulic Turbine-Design and Equipment*, ARTIA Prague, 1980.

[4] Raabe Ing. Joachim, *Hydro Power- The design, Use and Function of Hydro Mechanical Hydraulic and Electrical Equipment*, VDI-Verlag, GmbH, Dusseldorf, 1985,

[5] C. A. J. Fletcher, *Computational Techniques for Fluid Dynamics*” Vol. 1, Springer Pub., 1991.

[6] Lars Davidson, Hakan Nilsson, Walter Gyllenram, Numerical Investi-gations of Unsteady Flow in Draft Tubes, Water Turbine, *Collaborative R&D Program- Program Status Report*, Elforsk, GE Energy Sweden, , 2004.

[7] *Ansys CFX10.0 User Manual*, Ansys Inc., 2005.

[8] Buntić Ogor I., Dietze S., Ruprecht A, Numerical Simulation of The Flow in Elbow Draft Tube of Turbine-99, *Proceeding of the Third IAHR/ERCOFTAC Workshop on Draft Tube Flow,* Porjus, Sweden, 2005, pp-1-8.

[9] Hellstrom J.G.I., Marjavaara B.D., Lundstorm T.S., Parallel CFD Simulations of an Original and Redesigned Hydraulic Turbine Draft Tube, *Journal of Advances in Engineering Software*, Vol. 38, 2007, pp338-344.

[10] Ales Skotak Jere and Obrovsky, Low Swirl Flow Separation in a Kaplan Turbine Draft Tube, *2nd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems Timisoara, Romania,* 2007.

AUTHORS BIODATA

Dr. Vishnu Prasad, Associate Professor in Maulana Azad National Institute of Technology Bhopal, India (MANIT). Presently persuing research in hydraulic turbines and water resources. He has 20 years of experience in teaching, consultancy and research at S.V. National Institute of Technology, Surat and MANIT, Bhopal. He had been to UK for training on model testing and CFD for six months. He has published 47 papers in International/National journals/conferences and guided 50 M.Tech dissertations.
Email: vpp7@yahoo.com

Ms Ruchi Khare, Assistant. Professor in Maulana Azad National Institute of Technology, Bhopal, India (MANIT) is presently pursuing research in hydraulic turbines. She has 12 years of experience in teaching, consultancy and research at MANIT, Bhopal. She has published 20 papers in International/National journals/conferences.
email: ruchif4@rediffmail.com

Abhas Chincholkar, was regular student of M.Tech in Hydro Power Engineering at MANIT, Bhopal.