DESIGN OPTIMISATION OF CONICAL DRAFT TUBE OF HYDRAULIC TURBINE

Dr. Ruchi Khare, Dr. Vishnu Prasad, Mitrasen Verma
Department of Civil Engineering
M.A. National Institute of Technology
Bhopal ,India
ruchif4@rediffmail.com

ABSTRACT

The optimum performance of draft tube is an important aspect in the design of hydraulic turbine, which can be achieved by varying the shape and size of draft tube. The recovery of kinetic energy leaving the runner determines the performance of draft tubes. The cross-sectional area at exit of draft tube is dependent of length and angle of diffuser and must be chosen to ensure maximum recovery with minimum loss. Conical draft tubes gives better efficiency in comparison to elbow draft tubes because of more recovery of vortex flow coming out of runner. In this paper, the numerical simulation of conical draft tube is done using CFD code ANSYS CFX 13.0 for different length and diffuser angle of the draft tube. The performance of draft tube is analyzed by calculating head loss, head recovery coefficients and efficiency of draft tube from simulation results.

Key words: Conical draft tube, draft tube performance, numerical simulation, whirl

INTRODUCTION

The efficiency of hydraulic turbine depends on the performance of its each component i.e. casing, stay ring, distributor, runner and draft tube. The draft tube, which converts the kinetic energy coming out of the runner into pressure energy, is an important component of the turbine. The hydraulic characteristics of any draft tube depend on its shape and dimensions and the flow pattern at its entrance [7]. Straight conical draft tube is the simplest type of draft tube and it has excellent hydraulic characteristics [3]. This draft tube has been eventually used for small and medium size vertical turbines. The low specific speed mixed flow turbines and consequently the smaller value of the kinetic energy of the flow leaving the runner permits the use of short draft tube [2]. Hence straight conical draft tubes can be used for runner diameter as large as D_1=1.6 m when the conditions of construction permit the use of increased depth for foundation. The length of conical tube for minimal losses varies with the diameter of the turbine [1]. In certain cases it is possible to make use of straight tubes even for large runner diameters provided ,the foundation can be laid well underground [5].

The objective of present work is to optimize the geometry of conical draft tube by varying the length and diffuser angle because these parameters influence the efficiency of the draft tube. For particular conditions at the draft tube inlet, there is an optimum value of these two parameters. Due to increase in the length of draft tube, the friction losses increase and thus reduce the draft tube efficiency and finally turbine efficiency [8]. If diffuser angle increases, then flow separation may occur at the walls of draft tube and eddies are generated in the flow passage of draft tube which further increases the losses and reduces the efficiency [6]. So it is required to find optimal length and diffuser angle which provides minimum losses and the highest possible efficiency. Different geometries of draft tube have been created by changing the lengths and diffuser angles. The numerical analysis have been done by using ANSYS 13 CFX code and if optimum values of length and diffuser angle are found for the maximum efficiency and head recovery for given boundary conditions.

GEOMETRIC MODELLING AND BOUNDARY CONDITIONS

Straight conical draft tube has diverging passage. The geometric modeling of this draft tube is carried for five values of L/D_3 ratio (ie,10, 12, 15, 19, and 23) and three values of diffuser angle. The geometric dimensions of conical draft tube are shown in fig. 1

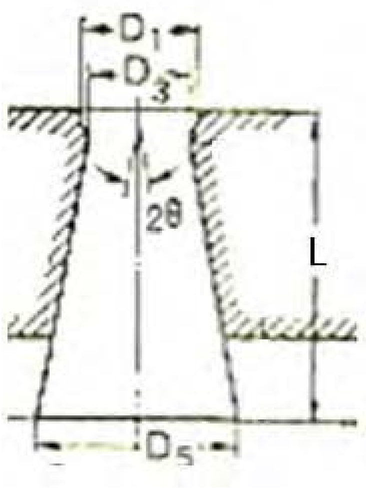

Figure 1. Straight conical draft tube [1]

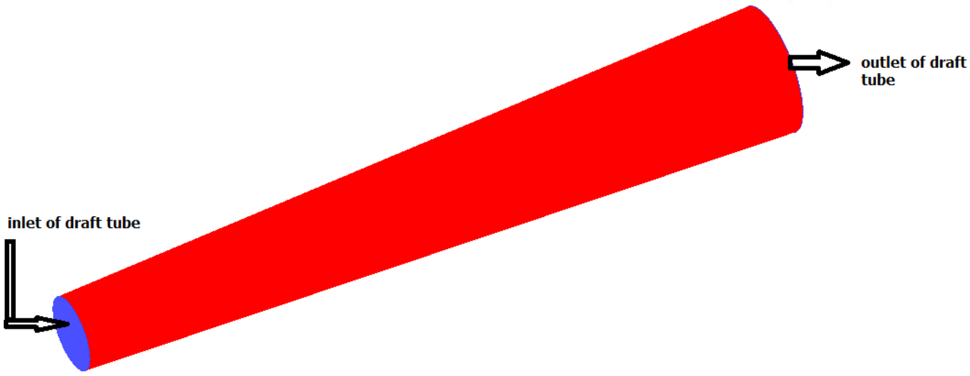

Figure 2. Isometric view of modeled draft tube

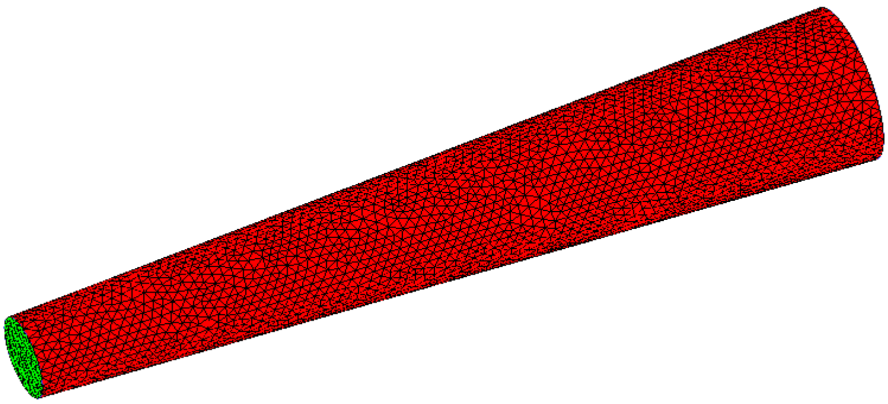

Figure 3. Meshing of the draft tube

The meshing of the draft tube flow domain is generated in ANSYS ICEM CFD as shown in fig.3. The unstructured triangular elements on surfaces and tetrahedral in flow domain are used in draft tube meshing. The fine mesh is generated near surfaces in order to capture the boundary layer and turbulence as compared to interior domain. The cylindrical velocity components are given as the inlet boundary condition at the inlet of the draft tube. Shear stress transport (SST) turbulence model in Ansys CFX code has been used for the analysis. The Y plus value in this analysis is less than 200 which is acceptable for automatic wall function. The wall of the draft tube is assumed as smooth with no slip conditions.

FORMULAE USED

The related formulae used in calculation of different parameters are as follows.

Head loss

$$H_{LD} = \frac{TP_{in} - TP_{out}}{\rho g} \qquad (1)$$

Head loss coefficient

$$\xi = \frac{\rho g H_{LD}}{TP_{in}} \qquad (2)$$

Head recovery

$$\Delta h_d = \left(\frac{v_3^2}{2g} - \frac{v_5^2}{2g} \right) - H_{LD} \qquad (3)$$

Head recovery coefficient

$$\zeta = \frac{\left\{ \left(\frac{\rho v_3^2}{2} - \frac{\rho v_5^2}{2} \right) - \rho g H_{LD} \right\}}{TP_{in}} \qquad (4)$$

Efficiency

$$\eta_D = \frac{\Delta h_d}{\frac{v_3^2}{2g}} X 100 \qquad (5)$$

where,

H_{LD} = head loss in draft tube (m)
TP_{in} = total pressure at draft tube inlet (Pa)
TP_{out} = total pressure at draft tube inlet (Pa)
ρ = specific mass of fluid (Kg/m^3)
v_3 = velocity at draft tube inlet (m/sec)
v_5 = velocity at draft tube outlet (m/sec)
g = acceleration due to gravity (m/sec^2)

RESULTS AND DISCUSSIONS

The 3D turbulent flow simulation has been carried out in straight conical draft tube with five h/D ratios of 10, 12, 15, 19, 23 and three diffuser angles 4°, 5°, 6° for cylindrical velocity components as, Axial component= 9.17 m/s , Radial component=0.142151 m/s, Tangential component=4.35075 m/s (Ruchi, 2011).

The head loss, recovery and efficiency are computed and presented in graphical form in fig 4 to fig 6. It is seen from fig. 4 that the variation pattern of head loss varies with the angle of diffuser of the draft tube cone. There is sharp drop in head loss at large cone angle as compared to small angles and this is due to the flow separation at higher diffuser angle. The head loss is more dependent on length of draft tube for a limited range of L/D_3 ratio. The starting point of drop in head loss shifts towards higher L/D_3 ratio as the cone angle decreases. It is also observed that there is negligible change in head loss at diffuser angle of 4 degree for range of L/D_3 considered for simulation and for all cone angles after L/D_3 value of 19.

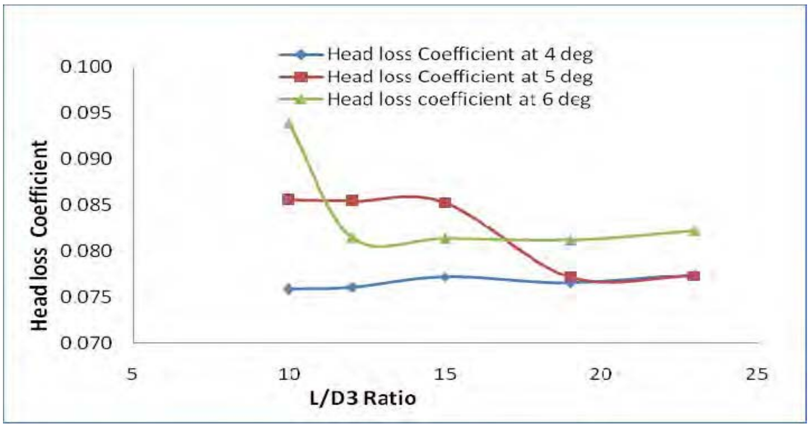

Figure 4. Head loss variation with length at different diffuser angle

The variation of head recovery with length of draft tube for different diffuser angle is plotted in fig.5. It is seen that head recovery depends on both length and diffuser angle of draft tube. The head recovery is more influenced at smaller length but has very negligible change after the length equal to 19D_3. It is also observed that as the value of diffuser angle increases, the rate of head recovery also increases at any L/D_3 ratio. The rate of change in recovery due to cone angle is more at small length of draft tube.

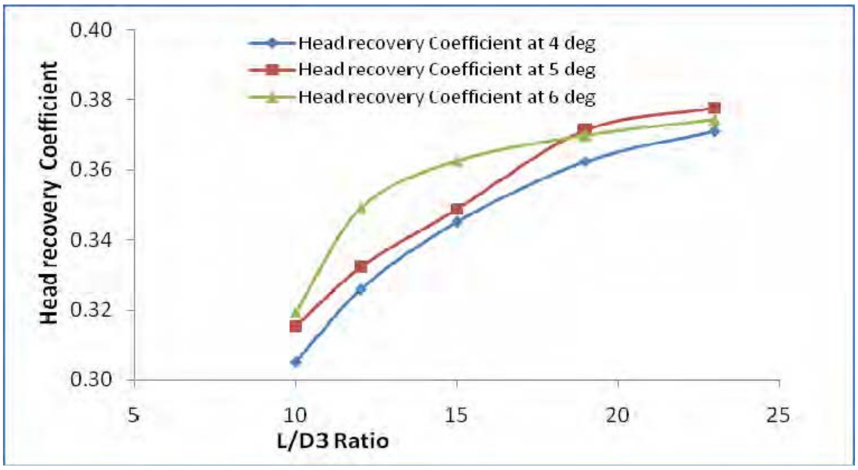

Figure 5. Head recovery with L/D_3 ratio at different diffuser angle

The efficiency variations in fig. 6 indicate that efficiency has gradual increase as the L/D_3 ratio is increased up to 19 but after this, increase in efficiency is very less due to small increase in recovery.

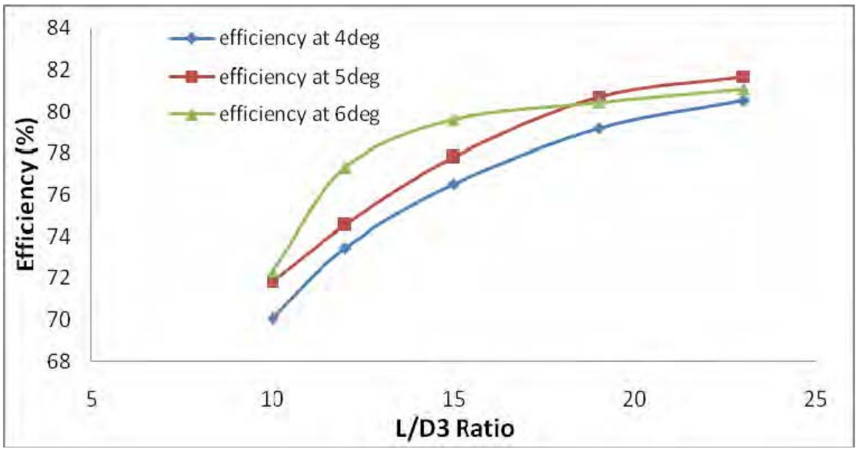

Figure 6. Variation of efficiency at different L/D_3 ratio and diffuser angle

The comparison of stream lines shown in fig.7 for three diffuser angles and constant length 19 D_3 of draft tube depicts that amount of whirl is increasing with increase in diffuser angle, but the highest head recovery is obtained from the draft tube of geometrical configuration with 5 degree diffuser angle and 19D_3 length (fig.5 and fig.6). It is seen from pressure contours in fig.8 that pressure variation is minimum for 5 degree diffuser angle.

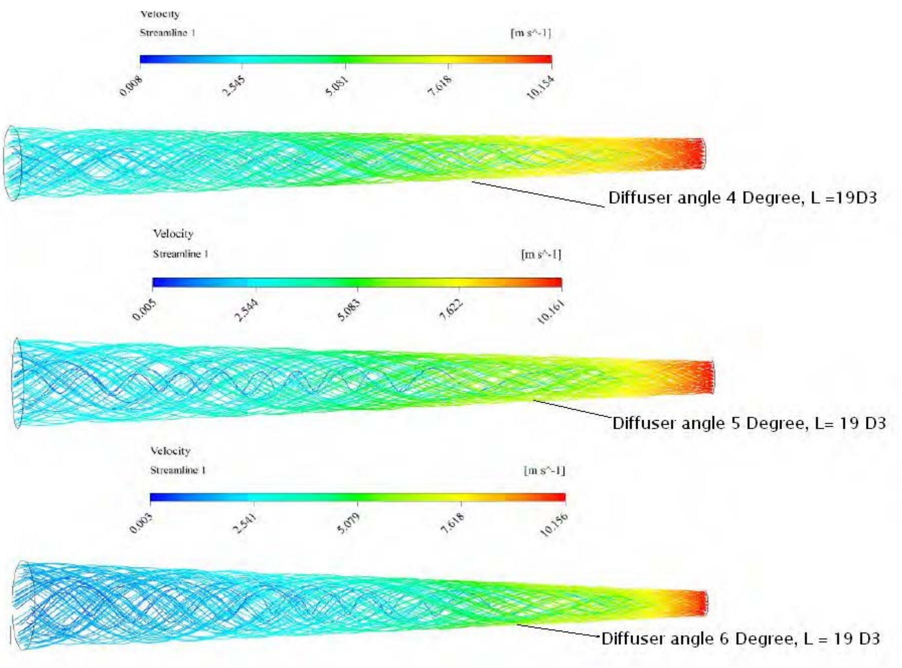

Figure 7. Velocity stream lines for different diffuser angle at constant length 19D3

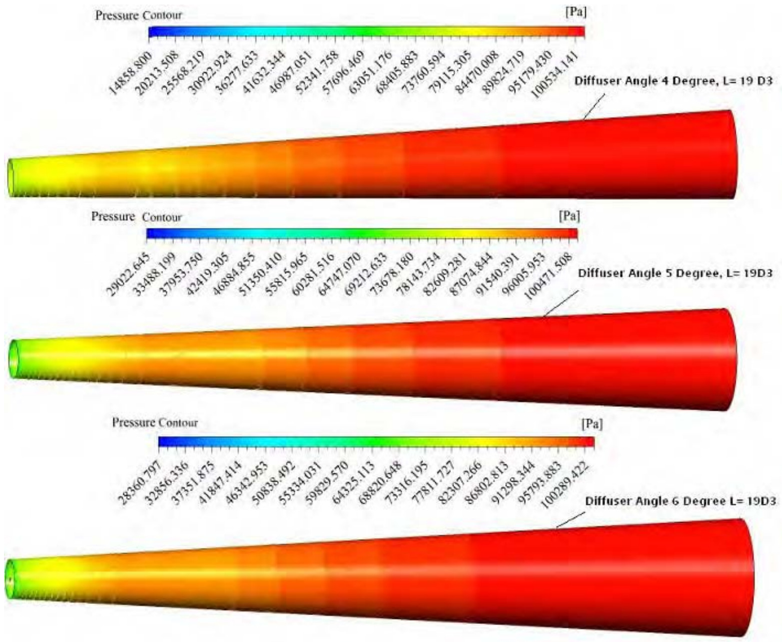

Figure 8. Pressure contours for different diffuser angle at constant length 19D3

CONCLUSION

It is seen from the numerical simulation of conical draft tube that both length and diffuser angle has significant effect on performance of straight conical draft tube. There is no significant variation in head loss, recovery and efficiency of draft tube with length beyond $19D_3$. Therefore, increase in length of draft tube beyond $19D_3$ length will not be economical and also lead problem of cavitation in turbine. Secondly maximum efficiency is achieved at diffuser angle of 5 degree. Hence the length of draft tube corresponding to 19 D_3 is the optimum length of draft tube. The most of the hydro power plants have used straight conical draft tube around the length $19D_3$, diffuser angle 3.6° to 6° and hence the results from numerical simulation are validated. It may be concluded that CFD is very effective tool for numerical flow simulation in complex flow domains with reasonable accuracy.

REFERENCES

[1] Gubin M. F., "Draft tubes of Hydro-Electric Stations" Amerind Publishing Co. Pvt. Ltd., New Delhi.1973

[2] Modi P.N., Seth S.M. , "Hydraulics and Fluid Mechanics", Standard Book House, New Delhi.1991

[3] Carija Z., Mrsa Z. and Dragovic, "Turbulent Flow Simulation In Kaplan Draft Tube" 5th international Congress of Croatian Society of Mechanics Trogir/Split, Croatia 21-23 Sept. 2006

[4] Ruchi Khare," Hydrodynamic Parametric Study of Mixed Flow Hydraulic Turbine", Ph.D. Thesis, MANIT, India,2011

[5] Rudolf Pavel,"Simulation of Vortex Breakdown in an Enclosed Cylinder as a Preliminary Study of the Draft tube Vortex rope creation" 6th IASME/WSEAS International Conference on Fluid Mechanics and Aerodynamics (FMA'08) Rhodes, Greece.2008

[6] Sylvain Tridon, Stéphane Barre, Gabriel DanCiocan,Laurent Tomas,2010,"Experimental analysis of the swirling flow in a Francis turbine draft tube: Focus on radial velocity component determination", Journal of Mechanics B Fluids, ISSN: 977546, Volume: 29, Issue: 4, Publisher: Elsevier Masson SAS, Pages: 321-335 ,2010

[7] Szlivka ferenc and éder Gergely, "Optimization of Diffuser with CFX Technology" V European Conference on Computational Fluid Dynamics,ECCOMAS CFD , J. C.F. Pereira and A. Sequeira (Eds) Lisbon, Portugal, 14–17 June 2010

[8] Vishnu Prasad, Ruchi Khare and Abhas Chincholikar, "Numerical Simulation for Performance of Elbow Draft Tube at Different Geometric Configurations" Proceedings of 8th International Conference by IGHEM, IIT, Roorkee,2010

Optimization methodology assessment for the inlet velocity profile of a hydraulic turbine draft tube: part IIâ€"performance evaluation of draft tube model. Author & abstract. Download.Â Previously, three steps of optimization methodology to minimize the energy losses were studied: the inlet velocity profile parameterization, the numerical optimization set-up and the objective function validation. In the latter step, a global optimization method called Multi Island Genetic Algorithm (MIGA) was considered, which requires a large number of iterations before producing a reliable result.Â Specifically, the draft tube calculations were performed on a sequence of five different grids each having approximately twice the number of elements compared to the previous. Hydraulic turbine Conical-duct diffuser Automatic numerical optimization CFD Multi-Island Genetic Algorithm. Technical Editor: Jader Barbosa Jr. This is a preview of subscription content, log in to check access.Â Fares R, Chen X, Agarwal R (2011) Shape optimization of an axisymmetric diffuser and a 3D hydro-turbine draft tube using a genetic algorithm. AIAA Paper, p 1243Google Scholar. 7. Ferrando LÃ³pez L (2006) Surface parameterization and optimum design methodology for hydraulic turbinesGoogle Scholar. 8. Fluent Inc. (2007) GambitGoogle Scholar. The draft tube design of a hydraulic turbine, particularly in low to medium head applications, plays an important role in determining the efficiency and power characteristics of the overall machine, since an important proportion of the available energy, being in kinetic form leaving the runner, needs to be recovered by the draft tube into static head. For large units, these efficiency and power characteristics can equate to large sums of money when considering the anticipated selling price of the energy produced over the machineâ€™s life-cycle.Â IRJET- Introduction and Design of Conical Type Draft Tube. Read more. Optimum Design of Conical Draft Tube by Analysis of - IJLTEMAS. Read more.