EFFECT OF DIFFUSER LENGTH ON PERFORMANCE CHARACTERISTICS OF ELBOW DRAFT TUBE WITH DIVIDING PIER

Rahul Bajaj\textsuperscript{1}, Dr. Ruchi Khare\textsuperscript{2}, Dr. Vishnu prasad\textsuperscript{3}

M.Tech. Student\textsuperscript{1}, Asst. Professor\textsuperscript{2}, Professor\textsuperscript{3}
Department of Civil Engineering
M.A. National Institute of Technology, Bhopal (M.P.)

ABSTRACT

The hydraulic turbines extract the energy of flowing water and converts into mechanical energy. The reaction turbine has components namely casing, stay ring, guide vane, runner and draft tube. Each component plays some role in performance of turbine. Out of above component casing, stay ring and distributor guide the flow while in runner and draft tube energy transfer and conversion takes place. In reaction turbine, significant part of input energy goes out of runner unutilized in form of kinetic energy. Draft tube are provided at exit of runner to connect turbine and tail race providing closed conduit flow of varying cross sectional area. The development in the design of turbines leads to different shape of draft tube to recover as much energy as possible. The elbow draft tube is mostly used with large reaction turbine. It is found that geometry of draft tube, flow space affects its performance to large extent but due to limitation of experimental investigation, the mesh were limited to few shapes. The growth on computational power has made it possible to investigate the many alternative design of draft tube. In the present paper, numerical simulation has been done for a large elbow draft tube with pier in diffuser. The length of diffuser has been changed to see its effect on the performance i.e. head and efficiency of draft tube.

Keywords: Draft Tube, CFD, Diffuser, Pier, Kinetic Energy, Efficiency.

INTRODUCTION

Draft tube is an important component of hydraulic reaction turbine and improves the performance of turbine by converting the kinetic energy entering in draft tube from runner, into pressure energy. [1] In axial flow reaction turbine, the kinetic energy of water leaving the runner is
up to 50% of total input energy whereas in case of mixed flow reaction turbine it is up to 15% of total energy.

In beginning of 19th century, the hydrodynamic investigation was carried by F. Frankl and A.V. Milovich on straight draft tube and they determine the solution to the problem of determining the shape of draft tube on the basis of theoretical investigation. Therefore large numbers of investigations were carried out on straight diffuser between 1909 and 1929 and emphasized the need to design various shapes of draft tubes. The use of straight tubes was restricted to turbine of medium and small diameters owing to the diameter of runner \( D_1 \) increases, the length of the tube becomes so large it is irrational to construct. To overcome these problems, bell mouth tubes were invented and recover kinetic energy of axial and rotational component of flow. With the rapid development of large capacity mixed flow hydraulic turbine, bell mouth draft tube was also absolute because large runner diameter leads increase in dimensions and weight of draft tube. Later on all these problems were overcome by elbow draft tube which is suitable for large diameter hydraulic turbine. In case when diffuser length exceeds 10 to 12 m, the dividing pier is provided which gives the better flow distribution and consequently improves the performance of draft tube and power characteristics of the turbine.[2]

In hydraulic turbine, determination of optimum shape and dimension as well as prediction of flow behavior is very difficult task. To overcome these problems the numerical simulation has become more informative tool.[3] At the best operating condition the flow enters mostly axially minimum swirl and radial velocity in a draft tube but at off design condition, the flow enters with significant swirl velocity.

Since from more than a decade’s Computational Fluid Dynamics (CFD) is most effective tool and widely used by researcher for predicting performance and flow behavior in a domain and optimization of design of hydraulic turbine components. Conventional model testing is costly and time consuming and to overcome this, CFD is an alternative tool which is a cost effective and minimized laboratory testing of models.[4]

**GEOMETRY AND MESH GENERATION**

The elbow draft tube has three parts namely cone, elbow and diffuser. In large draft tube piers of thickness \( b = 0.25 \) \( D_1 \) are used to improve the performance of turbine. The geometric dimensions of elbow draft tube are shown in fig. 1.

The modeling of elbow draft tube has been done in ICEM CFD for 3 length \( L/D_1 \) ratio [2]. Model of elbow draft tube for \( h/D_1 = 2.3 \) and \( L/D_1 = 15 \) is shown in fig.2. The draft tube is having 250 mm diameter at inlet and the analysis is done for \( L = 5 \) \( D_1 \), \( L = 10 \) \( D_1 \) and \( L = 15 \) \( D_1 \) diffuser length.

![Fig. 1: Geometric parameter of elbow draft tube with dividing pier [2]](image-url)
In three dimensional CFD simulations, the fluid domain can be discretized by tetrahedral or hexahedral mesh. Unstructured mesh which mainly consists of tetrahedral elements while structured mesh consists of hexahedral elements.[5] Ansys CFX uses finite volume method (FVM) for the discretized domain and N-s equations is solved at every node of the cell.

In the present work, meshing of draft tube is done in Ansys ICEM CFD as shown in fig. 3. For this flow domain, unstructured three dimensional tetrahedral meshing has been adopted.

**BOUNDARY CONDITIONS**

The numerical flow simulation is carried out for 13 whirl component varying from 3.0 m/s to -3.0 m/s at an interval of 0.5 m/s at 3 different lengths of diffuser i.e. L = 5 * D₁, L = 10 * D₁ and L = 15 * D₁. Cylindrical velocity component at inlet of draft tube is specified as axial component = -15.41 m/s, radial component = 0 m/s and 13 whirl component at an interval of 0.5 m/s as inlet boundary condition. Average static pressure is specified at outlet of draft tube as outlet boundary condition. All the walls in the geometry is assumed as smooth and no slip. Shear Stress Transport (SST) K-w turbulence model is used with wall function as automatic in Ansys CFX code. Convergence criteria are set to be $10^{-6}$ as RMS value and 500 as maximum iteration.
COMPUTATION OF PARAMETERS

The following formulae are used to assess performance of draft tube using the velocity and pressure distribution from numerical simulation.

Head loss in draft tube

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

Head recovery in draft tube

$$H_{RD} = \frac{(V_{03}^2 - V_{05}^2)}{2g} - H_{LD}$$

Draft tube efficiency

$$\eta_D = \frac{2gH_{RD}}{V_{03}^2} \times 100$$

Relative head loss

$$H_{RL,LD} = \frac{2gH_{LD}}{V_{03}^2} \times 100$$

RESULTS AND DISCUSSIONS

The 3D flow analysis has been carried out in elbow draft tube with three L / D₁ ratios of 5, 10 and 15 for 13 whirl component varying from 3.0 m/s to -3.0 m/s at an interval of 0.5 m/s for constant n₁’=135.6 rpm. Fig. 4 shows the various sections at which pressure and velocity contours are taken.

Fig. 4: Pressure contour for different section (L=5D₁)
Fig. 5.(a): Pressure contour at elbow section for $L = 5 \, D_1$ and theta component $= -1 \, \text{m/s}$

Fig. 5.(b): Pressure contour at elbow section for $L = 10 \, D_1$ and theta component $= -1 \, \text{m/s}$

Fig. 5.(c): Pressure contour at elbow section for $L = 15 \, D_1$ and theta component $= -1 \, \text{m/s}$
Fig. 6.(a): Velocity contour at elbow section for $L= 5 \, D_1$ and theta component= -1 m/s

Fig. 6.(b): Velocity contour at elbow section for $L= 10 \, D_1$ and theta component= -1 m/s

Fig. 6.(c): Velocity contour at elbow section for $L= 15 \, D_1$ and theta component= -1 m/s
It is seen in figure 5.(a),(b) and (c) that maximum pressure variation is obtained at outer side of elbow and in fig. 6. (a),(b) and (c) that at the periphery, velocity is zero and velocity is increases as moves from outer side to inner side of elbow.

**Fig. 7.(a):** Pressure contour at inlet of diffuser for \( L = 5 \, D_1 \) and theta component= -1 m/s

**Fig. 7.(b):** Pressure contour at inlet of diffuser for \( L = 10 \, D_1 \) and theta component= -1 m/s

**Fig. 7.(c):** Pressure contour at inlet of diffuser for \( L = 15 \, D_1 \) and theta component= -1 m/s

**Fig. 8.(a):** Velocity contour at inlet of diffuser for \( L = 5 \, D_1 \) and theta component= -1 m/s

**Fig. 8.(b):** Velocity contour at inlet of diffuser for \( L = 10 \, D_1 \) and theta component= -1 m/s
From fig 7.(a),(b) and (c), it is seen that the maximum pressure zone occurs at divider and in fig 8.(a),(b) and (c) minimum velocity zone is occurs at divider and periphery of diffuser.

Fig. 8.(c): Velocity contour at inlet of diffuser for L= 15 D₁ and theta component= -1 m/s

Fig. 9.(a): Pressure contour at diffuser for L= 5 D₁ and theta component= -1 m/s

Fig. 9.(b): Pressure contour at diffuser outlet for L= 10 D₁ and theta component= -1 m/s

Fig. 9.(c): Pressure contour at diffuser outlet for L= 15 D₁ and theta component= -1 m/s

Fig. 10.(a): Velocity contour at diffuser outlet for L= 5 D₁ and theta component= -1 m/s
It is observed from fig 9.(a),(b) and (c) that pressure variation is nearly constant and in fig 10.(a),(b) and (c), velocity is minimum at upper side of diffuser due to sharp curvature at divider.
The pressure contour for a draft tube is shown in Fig. 11. (a), (b) and (c) at L/D₁ = 5, 10 and 15 respectively. From these contours it can be seen that after achieving a optimum pressure at the inlet of dividing pier, pressure is nearly constant throughout the length of draft tube, there is sudden drop in pressure due to curvature at elbow section, where as.

Fig. 11.(c): Pressure Contour at L= 15 D₁ and theta component= -1 m/s

Fig. 12.(a): Velocity stream line at L= 5 D₁ and theta component= -1 m/s

Fig. 12.(b): Velocity stream line at L= 10 D₁ and theta component= -1 m/s
The streamline patterns for a draft tube is shown in Fig.12. (a), (b) and (c) at L/ D₁ = 5, 10 and 15 respectively. From the above figure it can be seen that the amount of whirl increases with increase in diffuser length. The whirl is higher in the left of the dividing pier as compared to whirl in right of dividing pier, it may be because of eccentricity of the dividing pier with the centre line of draft tube inlet. The velocity reduces from inlet to outlet which confirms the characteristic of draft tube.

The draft tube efficiency and relative loss in each numerical simulation are computed and graphical representation is shown in fig. 13 and fig. 14. As shown in fig.13 the maximum draft tube efficiency obtained at 0 m/s whirl component in case of L/D₁ = 10. All these numerical simulation shows that when diffuser length is L > 10 D₁, the highest efficiency is achieved. From fig. 14, it is seen that, minimum relative loss is for L/D₁ = 5 and 10.

**Fig. 12.(c):** Velocity stream line at L=15 D₁ and theta component= -1 m/s

**Fig. 13:** Efficiency of Draft Tube at different L / D₁ ratios

**Fig. 14:** Relative Loss in Draft Tube at different L / D₁ ratios
CONCLUSION

From numerical simulation of elbow draft tube with dividing pier it may be observed that that maximum efficiency is achieved at $L = 10 \cdot D_1$ length of the draft tube. The comparative study of the pressure variations and velocity contours at the inlet section of draft tube and just after the elbow section shows that location of dividing pier effects the velocity distribution significantly. Efficiency of draft tube is affected due to whirl component.

REFERENCES