Enhancement in the Design of Elbow Draft Tube as Compare to Draft Tube by Using CFD Simulation

Jitendra Kurre[1], Vinay Kumar, Shashank S. Mishra[3]

[1] CSVTU Bhilai (C.G.)
Shri Shankaracharya Group of Institutions
Junwani Bhilai (C.G.)
jitendrakurre18@gmail.com

[2]CSVTU Bhilai (C.G.)
Shri Shankaracharya Group of Institutions
Junwani Bhilai (C.G.)
vinay2606@yahoo.com

[3]CSVTU Bhilai (C.G.)
Shri Shankaracharya Group of Institutions
Junwani Bhilai (C.G.)
Shashank_sscet@yahoo.com

Abstract: ***Draft tube is a divergent tube one end is connected to the outlet of the turbine and other end is immersed well below the water level. The major function of the draft tube is to increase the pressure from the inlet to the outlet of the draft tube as it flows through it and hence increase it more than atmospheric pressure. The other function is to safely discharge the water that has worked on the turbine to tail race. With the use of very low head and high speed turbines, the kinetic energy leaving the runner became higher and the height of the runner above the tail race became smaller.***
This is achieved by increasing the cross-sectional area of the draft tube in the flow directions. My aim is to be maximizing the pressure in the outlet of the draft tube. We have to take one factor to optimise the model of elbow draft tube in hydro power plant. The experimental values are taken fromreference paper [1]. Analysing of pressure and velocity contour in the previous results and our result is validated. The pressure and velocity contour are to be taken for different diffuser angle to optimise and investigate better model of elbow draft tube for maximum efficiency.

Keywords: about four key words separated by commas**.**

1. Introduction

All the design of a hydropower system, the draft tube is an important component that significantly affects both the efficiency and cost, especially in low-head systems. Because of the effects on overall efficiency, even a slight increase in performance could result in a substantial energy savings. Draft tubes can be large and expensive, therefore more compact designs offer the potential of lower cost. The optimum trade-off between efficiency and cost requires a thorough knowledge of diffuser performance. For conventional systems, designers have a large amount of experience, but the possibility for improvement is still there.

The Curved draft tube is the basic type used in vertical hydraulic turbines of medium and large capacities. At present there is no theoretical method of determining the optimal dimensions of the curved draft tubes. A large number of different modifications in the basic shapes are existent. All variants have the following main parts as shown in
1) The initial part
2) The initial cone
3) The elbow
4) The Outflow diffuser.
The initial part of the diffuser is similar to that of the straight conical draft tubes. The initial cone has straight sides and connects the runner wheel chamber to the inlet section of the elbow. The flow deflects from the vertical to the horizontal plane in the elbow. Sometimes the flow is turned to some extent as observed in horizontal plane in elbow. The outflow diffuser is used to connect the elbow with the tailrace of hydroelectric station. On the basis of hydraulic characteristics all these parts are different from one another. The major part of kinetic energy is recovered in the cone and thus its length is designated to be maximum possible. This makes it possible to have small velocities at the inlet to the elbow and thus reduces hydraulic losses.
The primary function of the elbow is to turn the flow from vertical to horizontal direction with a minimum loss of energy. Draft tubes are designed in such a way that the recovery of kinetic energy takes place in not only in the initial cone but also in the elbow, which is conical along its length. The third part of the tube, the outflow diffuser also recovers a part of kinetic energy but to a smaller extent than initial cone as the velocity at the inlet section of the diffuser is considerably reduced. In addition the flow in the diffuser is influenced by the flow characteristics at the exit of the elbow. When the outflow velocity from diffuser is high flow dividers are introduced in the diffuser. On the whole curved draft tubes differ completely from straight ones in their hydraulic indices, especially at non optimal conditions of operating turbines. The dimensions of curved draft tube may be divided into two groups:
1. External

2. Internal

1.1 ELBOW OF CURVED DRAFT TUBE

It is one of the most complex elements of the draft tubes where there is maximum occurrence of hydraulic losses. The losses in the elbow of curved draft tube accounts for nearly 20 % or more of the total hydraulic losses in the draft tube. These losses are caused by the turning of the flow as well as due to complex shape of the elbow. The cross-section usually changes from circular to rectangular along the length. In hydroelectric stations we often come across situations where the outflow diffuser is inclined towards the tail race of the station. In these cases the upper surface of the diffuser is horizontal or may even have a positive slope towards the tail race; the bottom of the diffuser also slopes towards the tail race.

2. Governing equation for viscous flow model

The physical aspects of any fluid flow are governed by the following three fundamental principles:
(1) Mass is conserved
(2) F = ma (Newton's second law)
(3) Energy is conserved.

These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations.

2.1 Continuity Equation: The governing flow equation, which results from the application of law of mass conservation to any one of the four models the flow, is called quantity equations. The continuity equation suggests that the mass of fluid following is constant with the time.
The general form of continuity equation in 3D Cartesian coordinates is,

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho V_x)}{\partial x}+\frac{\partial(\rho V_y)}{\partial y}+\frac{\partial(\rho V_z)}{\partial z}=0$$

Where, V_x, V_y and V_z are components of velocity in x, y and z direction respectively.

2.2 Momentum Equation: The resulting Equation from the principle F=m*a (Law of momentum/Newton's Second Law) is called Momentum Equation. This equation tells us about various forces acting on the flow field in various directions.
According to Newton's Second law of motion, inertia force acting on a body in any direction is equal to resultants of all body and surface forces in that direction. The gravity force, pressure and shear forces as surface forces are commonly considered on fluid element.

$$\frac{\partial V_x}{\partial t}+V_x\frac{\partial V_x}{\partial x}+V_y\frac{\partial V_x}{\partial y} = F_x-\frac{1}{\rho}\frac{\partial P}{\partial x}+\frac{\mu}{\rho}\left(\frac{\partial^2 V_x}{\partial x^2}+\frac{\partial^2 V_x}{\partial y^2}+\frac{\partial^2 V_x}{\partial z^2}\right)$$

$$\frac{\partial V_y}{\partial t}+V_x\frac{\partial V_y}{\partial x}+V_y\frac{\partial V_y}{\partial y}+V_z\frac{\partial V_y}{\partial z}=F_y-\frac{1}{\rho}\frac{\partial P}{\partial x}+\frac{\mu}{\rho}\left(\frac{\partial^2 V_y}{\partial x^2}+\right.$$
∂2Vy∂y2+∂2Vy∂z2

Where F_x, F_y, and F_z are gravity forces per unit mass in x, y, z directions respectively.

2.3 Energy Equation: Energy is the mathematical expression for the Law of Energy Equation. The physical principle stated above is nothing more than the first law of thermodynamics. It can be applied in the following format.
Energy equation is obtained by multiplying the equation of momentum by velocity components in each coordinate direction and then adding and integrating over the volume.

3. Discretization of differential equations

Discretization means breaking continuous into discrete form the flow equation concept. The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. There are many numerical techniques used for discretization of differential equation.
This is the standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. FVM recasts the PDE's (Partial Differential Equations) of the NS equation in the conservative form and then discretize this equation. This guarantees the conservation of fluxes through a particular control volume. Though the overall solution will be conservative in nature there is no guarantee that it is the actual solution. Moreover, this method is sensitive to distorted elements which can prevent convergence if such elements are in critical flow regions. This integration approach yields a method that is inherently conservative (i.e. quantities such as density remain physically meaningful). The numerical method is mostly used in commercial CFD software including AnsysCFX14.0. The most commonly used method is discussed here in brief.

3.1 Finite Element Method (FEM)

This method is popular for structural analysis of solids but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. Although in FEM conservation has to be taken care of, it is much more stable than the FVM approach. Subsequently it is the new direction in which CFD is moving. Generally stability/robustness of the solution is better in FEM though for some cases it might take more memory than FVM methods. In this method, a weighted residual equation is formed. The approximate solution is defined as

$$U=\sum_{i=1}^{i=K}N_i U_i$$

Where U_i is the value of any parameter at i^{th} node of the element and K is total number of nodes in an element. This approximate solution is substituted either in the weighted integral form or weak form of differential equations and element matrix formulation is carried out by integration. The accuracy of solution can be improved by increasing the order of interpolating polynomials. This method is again not much suited for complex geometries encountered in flow problems. The element matrices are assembled to generate the global matrix, which is solved for unknown nodal values after application of boundary conditions.

3.2 Finite Difference Method (FDM)

This method is based on the Taylor`s series expansion. It is the oldest method among the three described methods. This method has historical importance and is simple to program. It is only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid. This method is node based and has poor convergence and well suited for rectangular grids. The FDM is node based and approximation of first order differential terms using central difference method is given as

$$\frac{dU_i}{dx} \cong \frac{U_{i+1} - U_{i-1}}{2\Delta x}$$

$$\frac{dV_i}{dy} \cong \frac{V_{j+1} - V_{j-1}}{2\Delta y}$$

The FDM formulation may be either explicit or implicit. The explicit formulations are simple but become conditionally stable while implicit formulations are unconditionally stable. It is well suited for rectangular grids. The values at other locations between nodes are obtained by interpolation and type of interpolation depends on user. The accuracy can be improved further by using higher order differences.

3.3 Finite volume method (FVM)

This numerical method is mostly used in commercial CFD software including Ansys CFX14.0. In this method, each element of discretized domain is referred as cell and a grid point as a node. In 2-Dimentional can use triangular or quadrilateral cells. In 3-D problems, the cells are usually hexahedral, tetrahedral or prisms. This method is similar to sub-domain method except that no explicit introduction of approximate solution and integral form of conservation equations are applied to the control volume defined by a cell to get the discrete equation. The discretization of 2-D continuity equation for steady flow is given below:
The continuity equation for 2-D steady flow is

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0$$

where U and V are velocities in X and Y directions respectively.

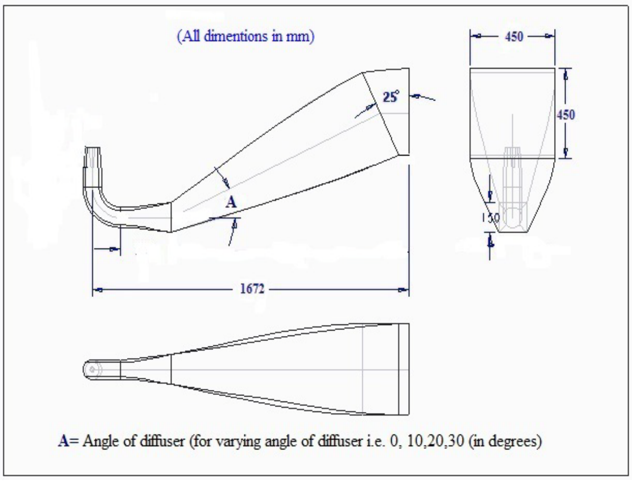

Figure 1: 2D drawing of elbow draft tube

In step 2, the meshing of elbow draft tube model is done. In meshing CFD mesh type is selected and fine meshing is done by
10 node tetrahedral elements. The reasons for selecting this element is that gives the good meshing on curvature parts meshing model of elbow draft tube as shown in fig. In meshing model have 9030 nodes and 46401 elements.

4. Results and discussions

The pressure and velocity distribution are determined by ANSYS 14.0 CFX solver in the postprocessor stage. The outcomes for the velocity and pressure contour for the Elbow draft tube as shown in figures 2 and 3 respectively.

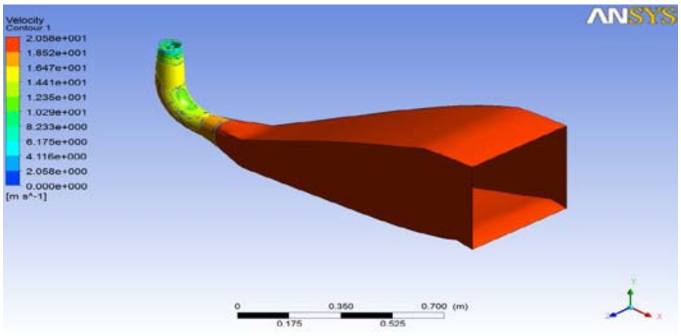

Figure 2 Velocity Contour of base model Case 1 (diffuser angle 20^0) Elbow Draft Tube

Present work in ANSYS (CFX) With Elbow and diffuser angle 20^0 is to be selected as Base model for our study and for Elbow Draft Tube, it is denoted by Case 1. For Elbow Draft Tubeminimum Inlet Pressure-1.21×10^{5} Pa and maximum Outlet Pressure1.10×10^{5}Paare obtained by pressure Contour and the value of maximum outlet velocity 20.58 m/s is obtained by Velocity Contour in Base model Case 1 for elbow draft tube with diffuser angle 20^0 .

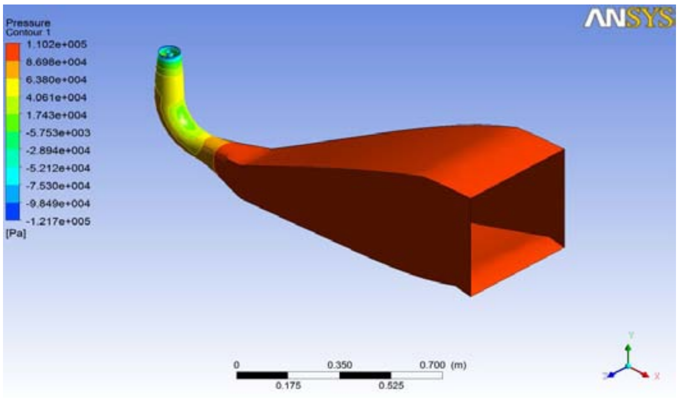

Figure 3 Pressure Contour of base model Case 1(diffuser angle 20^0) Elbow Draft Tube

The pressure distribution at inlet and outlet of draft tube has been measured by experimental procedure and ANSYS work by referring [Gunjan B. Bhatt et.al]. The same results have been compared with Present work in ANSYS (CFX) for Elbow draft tube and %Deviation in between Present and Experimental Reading has been found as given in Table 1, which shows both results are in good agreement and acceptable range with each other hence the design of elbow draft tube is validate as given in Table 1.

Table 1.Comparison between Present, ANSYS (CFX) and Practical Reading [Gunjan B. Bhatt et.al]

Comparison	Inlet Pressure	Outlet Pressure
Present work in ANSYS (CFX) With Elbow	-1.21×10^{5} Pa	1.10×10^{5} Pa
Gunjan B. Bhatt et.al ANSYS work (Without Elbow)	-2.100×10^{5}Pa	1.071×10^{5}Pa
Experimental Reading [Gunjan B. Bhatt et.al]	-1.99×10^{5} Pa	1.12×10^{5} Pa
%Deviation in between Present and Experimental Reading	2.51	1.78

5. Conclusion

- The Optimization and CFD (ANSYS 14.0 CFX) analysis has been performed for elbow draft tube to determine pressure and velocity profile at inlet and outlet condition.
- Percentage deviation in between Present work (ANSYS 14.0 CFX) and Experimental Reading is inlet pressure 2.51% and outlet pressure 1.78 % achieved, which shows both results are in good agreement and acceptable range with each other.
- The analysis for the pressure distribution at inlet and outlet of draft tube has been measured by experimental procedure and ANSYS work by referring [Gunjan B. Bhatt et.al]. The same results have been compared with Present work in ANSYS (CFX) for Elbow draft tube which shows both results are in good agreement and acceptable range with each other and this analysis may be used to reduce higher cost experimentation.

References

[1] GunjanBhatt ,Dhaval B. Shah, Kaushik M. Patel, "Design Automation And Cfd Analysis of Draft Tube For Hydro Plant." International Journal Of Mechanical And Production Engineering ,ISSN-2320-2092 Volume 3 Issue-6 2015.

[2] Anderson U., 2009. "Experimental study of Sharp Heel Draft Tube." Thesis (PhD), Lulea University of Technology,ISBN: 978-91-86233-68-6, Sweden.
]RuchiKhare, Vishnu Prasad, Sushil Kumar Mittal, 2012. "Effect of runner solidity on performance of elbow draft tube" Proceedings of the 2nd International conference on advances in energy engineering (ICAEE), Energy procedia 14, page 2054-2059.

[3] Vishnu Prasad, RuchiKhare, AbhasChincholikar , 2010. "Hydraulic Perfomance of Elbow Draft Tube for Different Geometric Configurations Using Cfd". IGHEM -2010,Oct.21 -23,AHEC,IIT Roorkee India.

[4] Upendra Rajak, Vishnu Prasad ,Ruchi Khare,2013."Numerical flow simulation Using Star CCM+." IISTE Vol.3 ,No.-6,2013.

[5] Siake A , Koueni-Toko C ,Djemako B.,Tcheukam-Toko,2014.Hydrodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine.IJHE-2014:103-114.

[6] Jiri Obrovsk, 2013. Development of high specific speed Francis turbine for low head, Engineering Mechanics, Vol. 20, No. 2, p. 139–148.

[7] Christopher, B. Cook, Marshall C. Richmond, John A. Serkowski, (2007),"Observations of Velocity Conditions Near a Hydroelectric Turbine Draft Tube Exit Using ADCP