International Research Journal of Advanced Engineering and Science
ISSN (Online): 2455-9024

To Optimize Model of Elbow Draft Tube for Maximizing Efficiency

Jitendra Gupta[1], Santosh Sahu[2]
[1, 2]Mechanical Engg. Dept., Rungta College of Engineering and Technology, Raipur, Chhattisgarh, India-492001
Email address: jitg2211@gmail.com[1], santosh2707@gmail.com[2]

Abstract— *In this research our main objective is to design a elbow draft tube with varying diffuser angle and to find out the optimized model for the same or with enhanced diffuser angle. In order to enhance the design methodology as compare to the conventional draft tube design , we are adopting the simulation method to resolve the problem of draft tube design and to reduce the cost of prototyping and to get the improved results by using computational fluid analysis in ANSYS 14.0 by CFX solver.*

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 base paper experimental values are taken. Analysing of pressure and velocity contour in the previous results and our results will validate. The pressure contour and velocity contour taken are different diffuser angle.

Keywords— *Draft tube, Draft tube performance, Numerical simulation.*

I. Introduction

The purpose of the draft tube of a water turbine is to reduce the exit velocity with a minimum loss of energy. The draft tube ‘converts’ the dynamic pressure (kinetic energy) into static pressure.

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.

Draft tubes can be designed in slightly different ways, but some design variables are of less importance than others. The shape of the outlet, circular or rectangular, is often of less importance than the outlet area. However, the shaping of the elbow is one of the most intricate problems with draft tubes. The challenge is to change the shape with minor losses of energy and without risks for damaging mechanisms such as severe cavitations. Earlier, the design of the draft tube was governed by a few hydro-mechanical principles with great consideration of structural and constructional application.

In the design of a diffuser there are two major phenomena to take into account. Too rapid expansion can make wall boundary layer separate, which leads to large losses. If the expansion is too slow the diffuser must be made longer and consequently the fluid will be exposed to an excessive area of walls. This will lead to large wall friction losses, separation and a more expensive construction. The optimal rate of expansion is obviously where these losses are minimized. Many times there are spatial restrictions of the size of the diffuser that also increase the importance of an optimal design.

Geometrical Modeling and Boundary Condition

Meshing of elbow draft tube model is done. In meshing CFD mesh type is selected and fine meshing is done by ten nodes tetrahedral elements shown in fig 1.The reason 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 total no of nodes and element are 325432 and 1855473.

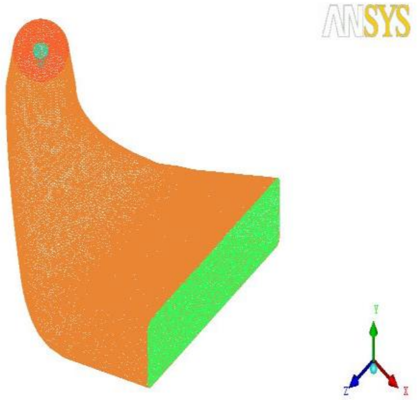

Fig. 1. Meshing of elbow draft tube.

The boundary conditions for elbow draft tube analysis have been applied in boundary condition inlet mass flow rate is given 200kg/s and outlet boundary condition is given 1 atm. Now next step is run the CFX solver to get the result. After the last step is analysis and visualize the result in post processor. Here, in the post processor, the pressure and velocity contours are visualized.

II. Result & Discussion

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.

Jitendra Gupta and Santosh Sahu, “To optimize model of elbow draft tube for maximizing efficiency,” *International Research Journal of Advanced Engineering and Science*, Volume 2, Issue 2, pp. 340-342, 2017.

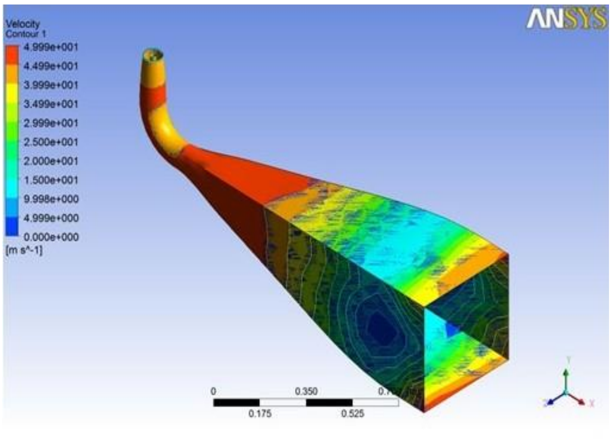

Fig. 2. Velocity distribution.

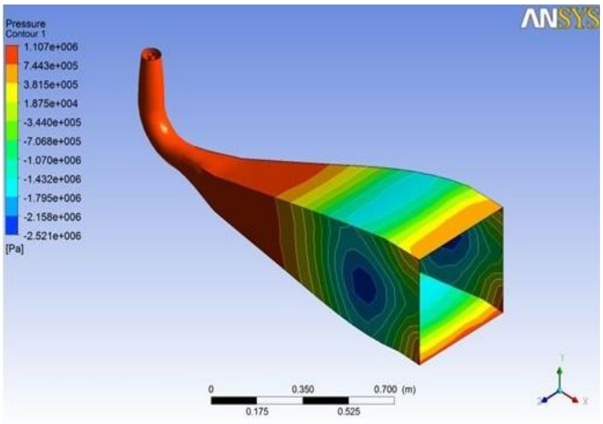

Fig. 3. Pressure distribution.

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 figure 4.

Fig. 4. Comparison between present, ANSYS (CFX) and practical reading.

Different Cases for Optimized Model for Elbow Draft Tube

Four different cases are proposed for design Optimization of elbow draft tube in which 20 degree diffuser angle is selected as a base model and it is termed as Case 1, in which the maximum outlet pressure and maximum outlet velocity are achieved 1.10×10^{5}Pa and 20.58 m/s respectively. In case 2, 3 and 4 diffuser angle with horizontal section of elbow draft tube is modified by 0, 10 and 30 degree.

For four different Cases of elbow draft tube are suggested as a part of design optimization to enhance maximum pressure at outlet Using CFD analysis in ANSYS 14.0 CFX, for the same boundary conditions pressure distribution of each case are determined.

The maximum pressure for each Case is given in table I.

TABLE I. Pressure at outlet and inlet for different cases.

Pressure	Case1(Base model)	Case 2	Case 3	Case 4
Maximum (Outlet) in Pa	1.10×10^{5}	8.38×10^{5}	1.11×10^{6}	8.20×10^{5}

For maximum outlet pressure, graph has been generated to compare all concepts as shown in figure 5.

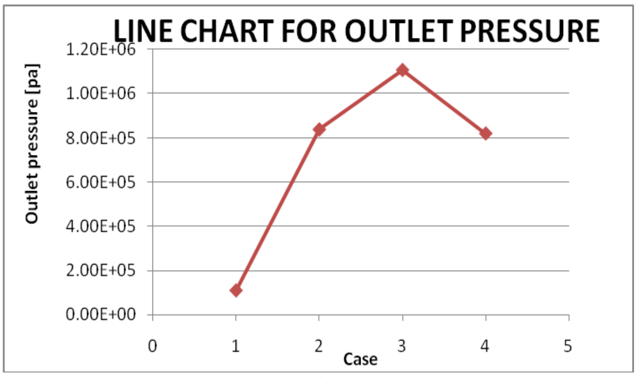

Fig. 5. Line chart for outlet pressure.

As shown in the output pressure line chart figures 5, the maximum output pressure attained in Case-3 i.e. diffuser angle $10^{0}$from the horizontal and its value is 1.11×10^{6} Pa. The maximum and minimum outlet and inlet Velocity for each Case are given in table II.

TABLE II. Velocity at outlet and inlet for different cases.

Velocity	Case 1	Case 2	Case 3	Case 4
Maximum (Outlet) in (m/s)	20.58	49.99	64.54	48.18

For maximum outlet Velocity, graph has been generated to compare all concepts as shown in figure 7.

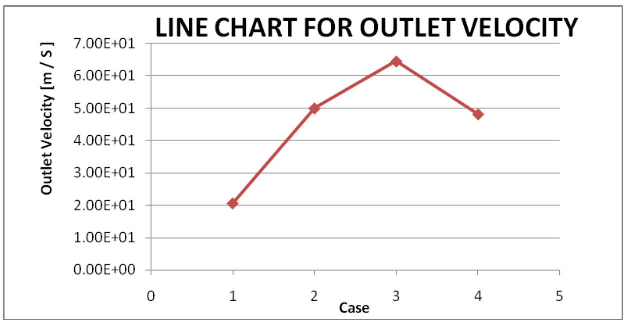

Fig. 6. Line chart for outlet velocity.

Jitendra Gupta and Santosh Sahu, “To optimize model of elbow draft tube for maximizing efficiency,” *International Research Journal of Advanced Engineering and Science*, Volume 2, Issue 2, pp. 340-342, 2017.

As shown in the output pressure line chart figures 5, the maximum output velocity attained in Case-3 i.e. diffuser angle 10^0 from the horizontal and its value is 64 m/s . Hence it is clear from the results the optimized model is obtained for maximum outlet pressure and velocity with Case-3, 10^0 diffuser angle with horizontal is achieved as compare to base model i.e. Case-1. The value of optimized maximum outlet pressure is 1.11×10^6 Pa.

III. CONCLUSION

Four cases have been proposed for draft tube to improve efficiency and pressure at outlet. The same analysis has been performed for different Cases with same boundary conditions. The enhanced value of maximum pressure and velocity is achieved in Case- 3 gives maximum outlet pressure as per analysis results compared to all other cases.

Percentage deviation in between Present research and Experimental Reading at outlet pressure 1.78 % achieved, which shows both results are in good agreement and acceptable range with each other. The value of optimized maximum outlet pressure is 1.10×10^6.

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.

The optimized model is obtained for maximum outlet pressure and velocity with Case-3, 10^0 diffuser angle with horizontal.

REFERENCES

[1] G. Bhatt, Dhaval B. Shah, and K. M. Patel, “Design automation and CFD analysis of draft tube for hydro plant,” *International Journal of Mechanical and Production Engineering*, vol. 3, issue-6, pp. 40-43, 2015.

[2] U. Anderson, “Experimental study of sharp heel draft tube.” Thesis (Phd), Lulea University of Technology, Isbn: 978-91-86233-68-6, Sweden, 2009.

[3] R. Khare, V. Prasad, and S. Kumar Mittal, “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*, pp. 2054-2059, 2012.

[4] V. Prasad, R. Khare, and A. Chincholikar, “Hydraulic perfomance of elbow draft tube for different geometric configurations using CFD,” Ighem -2010, Oct. 21 -23, Ahec, Iit Roorkee India.

[5] U. Rajak, V. Prasad, and R. Khare, “Numerical flow simulation using star Ccm+,” *IISTE*, vol. 3, no. 6, 2013.

[6] J. Gupta and S. Sahu, “A review paper on design of elbow draft tube for unsteady flow,” *International Research Journal of Advanced Engineering and Science*, vol. 2, issue 1, pp. 144-147, 2017.

Jitendra Gupta and Santosh Sahu, “To optimize model of elbow draft tube for maximizing efficiency,” *International Research Journal of Advanced Engineering and Science*, Volume 2, Issue 2, pp. 340-342, 2017.