International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-04, Issue-03, June 2018

Experimental and CFD Analysis of Mixed Elbow Draft Tube and Comparative Analysis with Simple Elbow Draft Tube

Chaman Pal

Assistant Professor, ***palchaman0310@gmail.com***

T. Gulshan, Shashi Singh, Daleshwar, Nisbat Ali, Chandan Mundhra

UG Students, Department of Mechanical Engineering, SER, ITM University, Naya Raipur (C.G.)

Abstract - The draft tube is one of the essential components in hydraulic reaction turbines. Its function is to create positive suction pressure utilizing unused kinetic energy at the exit of turbine runner and hence to increase the effective net head on turbine unit. The energy recovery depends on the design of draft tube. 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. Amendment of design is made by changing the hydraulic parameters of suction cone, elbow and exit diffuser. 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 preferred to ensure maximum recovery with minimum loss. In this paper, the CFD analysis of Mixed Elbow draft has been performed and results for the same are compared with experimental reading and which are found within the limit and also compare the experimental and CFD result with simple elbow draft tube.

Keywords**: - Draft tube, reaction turbine, hydraulic parameter, CFD, efficiency, energy, and elbow.**

I. 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). The increase in length of draft tube increases the frictional losses and thus reduces the draft tube efficiency. Similarly, the increase in diffuser angle will result in flow separation at the walls of draft tube and eddies may generate in the flow passage, which further increase the losses and reduce the efficiency.

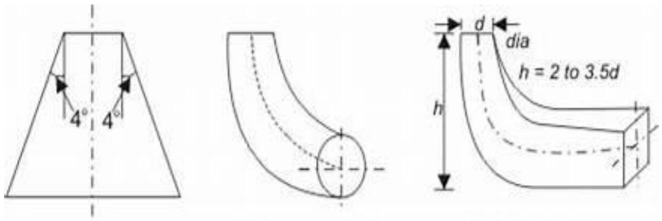
In a mixed flow reaction turbine, kinetic energy from runner is up to 15 % whereas in low head and high speed axial flow turbines, Kinetics energy leaving the runner may go up to 50 % of total energy. The recovery of kinetic energy is achieved by increasing the cross sectional area of the draft tube in flow direction. Initially, the straight conical tubes with inlet and outlet area of different cross – section were used.

Types of Draft Tube

Simple Conical Draft Tube - The draft tube has the shape of a frustum of a cone. This is generally provided for low specific speed. The cone angle is not to exceed 8^0. For greater value of the cone angle it is seen that the flowing body of water may not touch the sides of the draft tube (Leaving the boundary). This will lead to the eddy formation bringing down the efficiency of the draft tube.

Simple Elbow Draft Tube -The draft tube consists of extended elbow type tube used when turbine has to be placed close to tail race. It helps to cut down the cost of excavation and their exit diameter should be as large as possible to recover kinetic energy at the outlet of the runner. Such draft tubes are approximately 60% efficient.

Elbow Draft Tube with Varying Cross Section - This is further improvement of Simple Elbow Draft Tube. The outlet of draft tube should be situated below the tail race.



(a) Straight type (b) Simple elbow type (c) Elbow type with varying cross-section

The shape and velocity distribution at the inlet are the two main factors that affects the performance of the draft tube. Traditionally, the design of this paper, an attempt has been

 DOI : 10.18231/2454-9150.2018.0292

made for design automation of modelling of draft tube using Creo parametric and computational software. The use of computational fluid dynamics continues to grow due to is flexibility and cost-effectiveness. A CFD-based design can further be aided with a robust and user-friendly optimization framework theory and engineering.

An optimum design of draft tube is found out by using CFD as a tool. Assuming suitable dimensions for making Initial design of draft tube. 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.

Mixed elbow type draft tubes gives better efficiency in comparison to simple elbow draft tubes because of more recovery of vortex flow coming out of runner. In this paper, the numerical simulation of various draft tube is done using CFD code ANSYS CFX 14.5 for different length and diffuser shape of the draft tube. The performance of draft tube is analysed by calculating head loss, head recovery coefficients and efficiency of draft tube from simulation results. As well as the proper experimentation results have been compared by fabrication and experimentation on the different prototypes.

II. LITERATURE REVIEW

Performance of hydraulic component depends on the design of passage from where the fluid would be flowing. To design an optimum draft tube is a tough task since several kind of flow phenomenon takes place simultaneously inside it at BEP and at off design conditions. Gubin, 1973 [1] had carried out extensive study to examine the flow characteristic downstream of runner for reaction turbines. From this study it was observed that the flow downstream of reaction type runner is having certain amount of swirling [2], [3]. 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. 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 draft tube is one of the important components in a hydraulic mixed and reaction turbines. Without this the pressure at the outlet could drop as a result of lack of water which may adversely affect the efficiency of plant and may even fail to produced desired power. The study of flow patterns at inlet is very important for the effective knowledge about the performance of draft tube. In this study, we have discussed the principle of draft tube and its types through literature review. The parameter affecting the performance of draft tube are also discussed with the help of researches carried out earlier. The type of methods involved in the analysis of performance of draft tube are also considered from the literature available (11).

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. Generally, there is non-uniform distribution of velocity at entrance section of draft tube and the flow has certain amount of vorticity. Moreover, asymmetric flow in initial suction cone gives rise to flow separation even with cone angle 2α at which flow separation should not occur. The purpose of exit diffuser is to connect the elbow with the tail race with further recovery of pressure from flow downstream of an elbow. Since it is diffusing in shape both in plan and in elevation view, care should be taken by designer while designing this part to minimize the separation and hence the back pressure of flow. [1], [7]. Hydraulic losses in draft tube suction cone depends on several data like divergence angle 2Ø, wall roughness, relative area at the inlet (F_2) and outlet (F_{eo}) and characteristic of flow at entrance of draft tube. Generally, these days a convergent-divergent kind of elbow designs are adopted with initial part as divergent and remaining part with slight convergent to minimize the hydraulic losses induced due to separations and eddies generated in the curvatures of draft tube elbow (i.e. on inner surface) [5].

A new Reynolds-averaged Navier-Stokes (RANS) turbulence model is developed in order to correctly predict the mean flow field in a draft tube operating under partial load using 2-D axisymmetric simulations. It is shown that although 2-D axisymmetric simulations cannot model the 3-D unsteady features of the vortex rope, they can give the average location of the vortex rope in the draft tube. Nevertheless, RANS simulations under predict the turbulent kinetic energy (TKE) production and diffusion near the centre of the draft tube where the vortex rope forms, resulting in incorrect calculation of TKE profiles and, hence, poor prediction of the axial velocity. Based on this observation, a new k ε turbulence RANS model taking into account the extra production and diffusion of TKE due to coherent structures associated with the vortex rope formation is developed. The new model can successfully predict the mean flow velocity with significant improvements in comparison with the realizable k ε model. This is attributed to better prediction of TKE production and diffusion by the new model in the draft tube under partial load. Specifically, the new model calculates 31%

 DOI : 10.18231/2454-9150.2018.0292

more production and 46% more diffusion right at the shear layer when compared to the k ε model [12].

Draft tubes are used to increase performance in spouted beds. Performance of these tubes depends on its geometry and location. CFD modelling of bed pro- vide the best condition. In this work a CFD modelling technique is used to optimize draft tube geometry. First, model accuracy was assessed by comparing the results with experimental results. After it became clear that the model works, it was used to optimize the designing of spouted bed. The Eulerian–Eulerian multi-fluid modelling approach was applied to predict gas–solid flow behaviour. The results present that optimized selection of draft tubes lead to uniform distribution of particle velocity and it can increase also particles circulating [13].

A design optimization study of an elbow type draft tube based on the combined use of Computational Fluid Dynamics (CFD), design of experiments, surrogate models and multi-objective optimization is presented in this study. The geometric variables that specify the shape of the draft tube are chosen as input variables for surrogate models and the pressure recovery factor and the head loss are selected as output responses. It is determined that, pressure recovery factor, which is the main performance parameter, can be increased by 4.3%, and head loss can be reduced by 20% compared to the initial CFD aided design [14].

III. METHODOLOGY

As per the client's specification, dimensions for draft tube have been decided and 2D drawing for the same has been prepared as shown in figure. The specification of model turbine for which optimum design of draft tube is required is Tabulated from laboratory turbine specification. Various dimensions of elbow type draft tube are shown in Fig.2 and the non-dimensional parameters of initial draft tube design are tabulated in Tab.1.

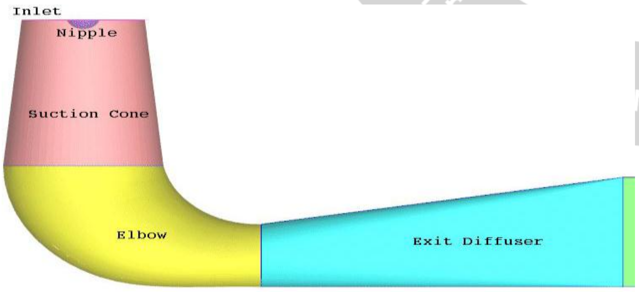


Figure 1

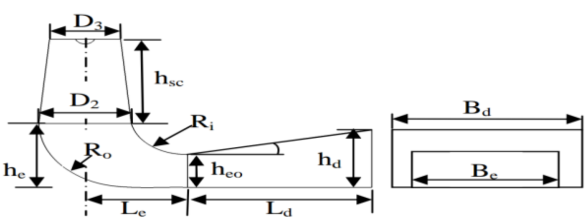


Figure 2- 2D sketch of draft tube with Nomenclatures.

Tab.1: Initial Non-Dimensional Design parameters of suction cone and exit diffuser (5).

Non-Dimensional Parameter	Value
D3/Di	0.696
D2/Di	0.904
hsc/Di	1.193
Be/Di	1.728
Bd/Di	1.904
hd/Di	0.646
Ld/Di	2.041
α	9.5^0

Design of a simple elbow was prepared by taking dimension of laboratory draft tube and mixed elbow draft tube was prepared by according to inlet diameter of simple draft tube and by using non dimensional parameters from tab.-1. Design of both draft tube taking considerations of height, length and bend condition of laboratory setup. Also a convergent-divergent hybrid elbow design adopted with an assumption of optimal law of variation of cross sectional area as function of length for an elbow. The transition from circular to rectangular cross section of an elbow is accomplished by intermediated oval cross-sections.

From [4] it is confirmed that there is specific importance of each and every component of draft tube to the performance of a turbine unit. Keeping this thing in mind, a new approach was evolved in which it was decided to modify each part individually and to check the performance of draft tube by combining them with several permutations and combinations.

GEOMETRIC MODELLING & BOUNDARY CONDITION

Design automation of modelling of draft tube using Creo parametric and ICEM CFD is used for CFD analysis of both the draft tubes. Creo model of draft tube has been converted in to STEP file and this step file has been imported in ICEM CFD. In fig-3 show the geometry of simple elbow draft tube which is similar to laboratory draft tube. In fig.-4 geometry of mixed elbow draft tube which are made in workshop by sheet metal and elbow section.

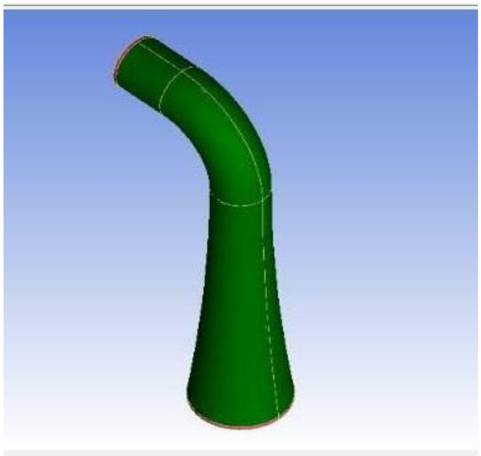


Figure 3- Geometry of simple Elbow draft tube

 DOI : 10.18231/2454-9150.2018.0292

ISSN : 2454-9150 Vol-04, Issue-03, June 2018

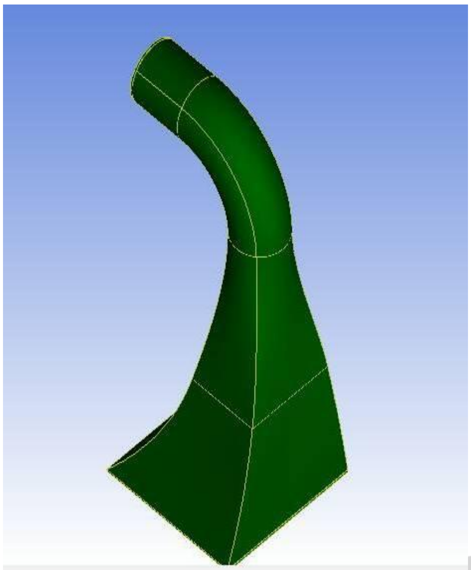


Figure 4-Geometry of mixed Elbow draft tube

MESH CREATION

In meshing CFD mesh type is selected and fine meshing is done by using one node tetrahedral elements shown in figure. The reason for selecting this element is that is gives the good meshing on curvature parts. Meshing model of draft tube is shown in figures in the meshed models 54133 nodes and 230965 elements were created for simple elbow type draft tube and 75627 nodes and 317727 elements for mixed elbow type draft tube were created.

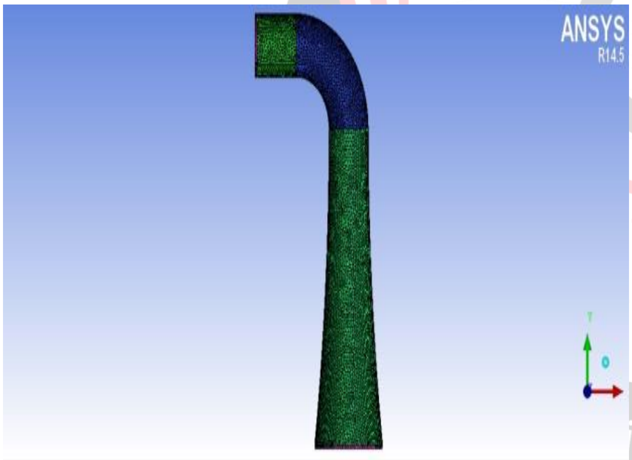


Figure 5-Meshing of simple Elbow type draft tube

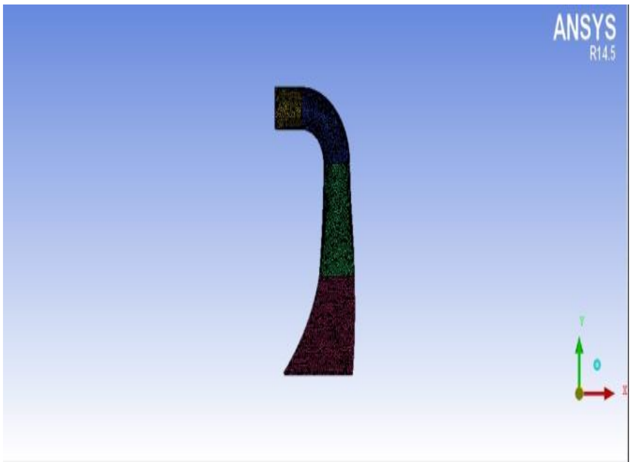


Figure 6- Meshing of mixed Elbow type draft tube

BOUNDARY CONDITION

After meshing the meshed geometry is imported in the ANSYS CFX pre. In boundary condition inlet mass flow rate is given 1000 kg/sec and outlet boundary condition is given 1atm for both the prototypes. The pressure and velocity distribution have been visualized and are as shown in figures.

VELOCITY STREAM LINES

In fig.7 and fig.8 velocity distribution are shown and we find out that velocity at outlet in mixed elbow type draft tube is very less as compared to simple elbow type of draft tube.

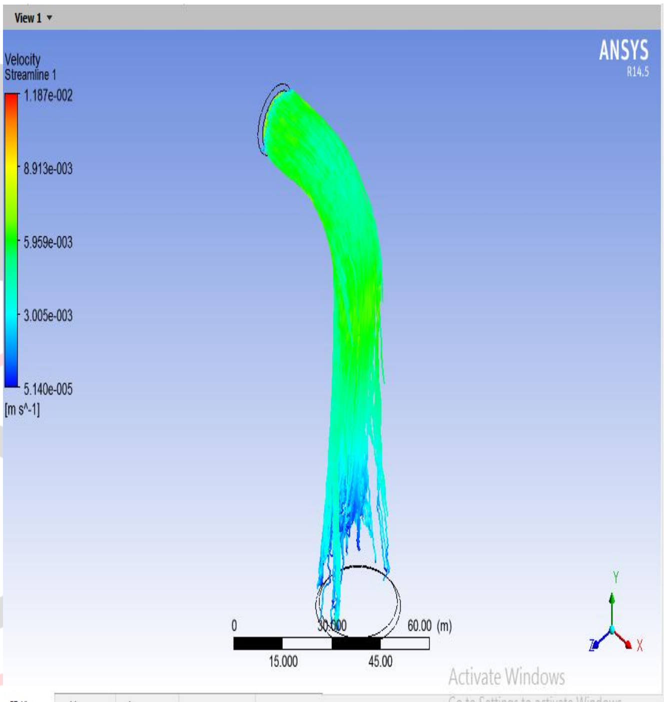


Figure 7- Velocity contour of simple elbow type draft tube

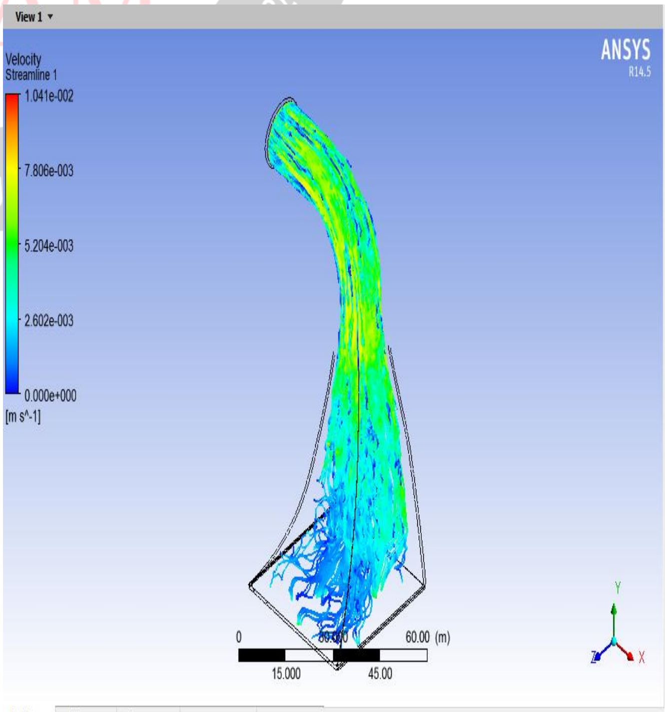


Figure 8- velocity contour of mixed elbow type draft tube

 DOI : 10.18231/2454-9150.2018.0292

PRESSURE CONTOURS

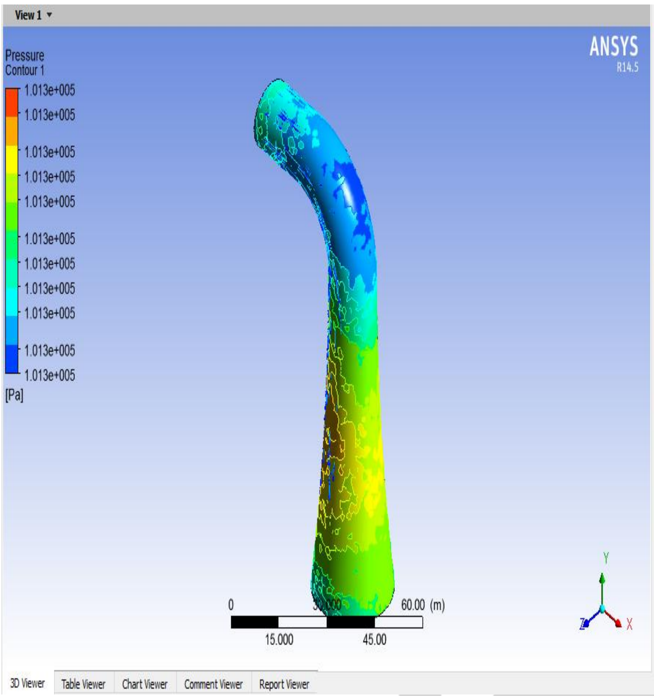


Figure 9- Pressure contour of simple Elbow type draft tube

According to fig.9 and fig.10 we checked that how pressure variation importance at every section of draft tube and how to optimize the area of draft tube.

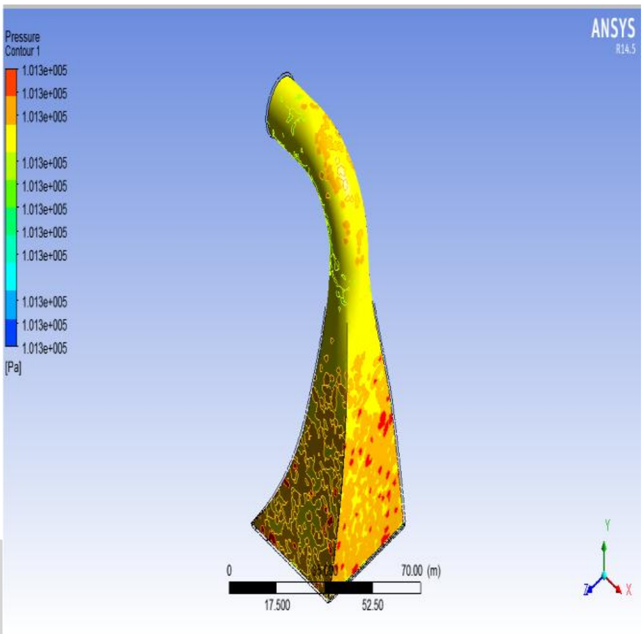


Figure 10- Pressure contour of mixed Elbow type draft tube

IV. RESULT & OBSERVATIONS

After simulation we found that difference between pressure and velocity at inlet and outlet of simple and mixed elbow type draft tube are given under tab.2. After scrutinize simulation result we fabricate a mixed elbow type draft tube with rectangular outlet cross section and experiment the model in laboratory. In experiment result, we observe that significant changes the result of performance of Francis turbine. Result show the importance of draft tube in Francis turbine and how vital in performance of particular turbine.

Tab.2-Simulation results

Type of draft tube	Average velocity(ms^{-1})		Average pressure(bar)
	Inlet	Outlet	
Simple elbow type	7.44×10^{-3}	2.35×10^{-3}	1.03×10^{5}
Mixed elbow type with varying cross section	6.54×10^{-3}	2.26×10^{-3}	1.013×10^{5}

Tab 3-Experimentation on lab apparatus (simple elbow type draft tube)

Sr. no.	Head (Bar)	Orifice meter pressure(bar)		Discharge	Load on turbine(kg)		Speed rpm	Power kw		efficiency
	H	P1	P2	Q	W1	W2	N	I/P	O/P	%
1	0.55	0.92	0.42	0.028	0	0	1454	1.51	0.084	55.4
2	0.55	0.92	0.42	0.028	1.24	0.50	1300	1.51	0.075	49.6
3	0.55	0.92	0.42	0.028	2.03	0.75	1222	1.51	0.070	46.6

 DOI : 10.18231/2454-9150.2018.0292

ISSN : 2454-9150 Vol-04, Issue-03, June 2018

Tab 4-Experimentation on prototype (mixed elbow type draft tube with rectangular cross section)

Sr. no.	Head (Bar)	Orifice meter pressure(bar)		Discharge	Load on turbine(kg)		Speed rpm	Power kw		efficiency
	H	P1	P2	Q	W1	W2	N	I/P	O/P	%
1	0.55	0.92	0.45	0.027	0	0	1414	1.45	0.082	56.2
2	0.55	0.92	0.45	0.027	1.24	0.50	1254	1.45	0.072	49.7
3	0.55	0.92	0.45	0.027	2.026	.75	1190	1.45	0.069	47.2

V. TURBINE SPECIFICATIONS

- Runner outside diameter = 150mm = 0.150 m
- Hub diameter = 78mm = 0.078 m
- No. Of runner blades = 4
- No. Of guide vanes = 10
- Guide vanes = 230mm = 0.230 m
- Brake drum diameter = 200mm = 0.200 m
- Rope diameter = 13mm = 0.013 m
- Size of orifice meter = 100mm = 0.100 m
- Area ratio = 0.56
- Orifice diameter = 75mm = 0.075 m
- Discharge Q = $k\sqrt{h1 - h2}$

FORMULAS USED

Q = Discharge in m³/sec

$Q = k\sqrt{p1 - p2}$

Where,

$K = C_d \times (p / 4) \times d^2 \times \sqrt{2 \; x \; g \; x \; 10}$

K = 0.68 x (p / 4) x 0.0752 2 x 9.81 x 10

K = 0. 0396

I/P = Input Power in KW

I / P = g QH

Where,

g = Specific weight of water = 9.81 kN/m^3

H = Inlet pressure head reading in meter

H= (Pressure gauge reading) x 10 meter

H= (1.25) x 10 meter

H = 12.5 meter

Output Power in KW

O /P power = $\frac{2\pi NT \times 9.81}{60000}$

Where,

T = Torque = Torque in N – m

T = Net load x effective brake drum radius

Where,

n = net loading in Kg

Efficiency of turbine in % =$\frac{OUTPUT\ POWER}{INPUT\ POWER}$

VI. CONCLUSION

It is seen from the simulation in draft tube that both pressure and velocity of draft tube has significant change on draft tube. From experimental calculation also observe that efficiency of both draft tube are changes. From the above observations and results it can be concluded that the mixed elbow type draft tube poses the improved efficiencies as compared to conventional simple elbow type draft tube.

The efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. This study also discussed the parameters which affects the performance of draft tube and substantially the turbine and the power plant. Through the reviews it is found that the CFD can be used as a tool for analysing the performance and the flow pattern inside the draft tube.

VII. NOMENCLATURE

Ø - Divergence angle of exit diffuser

B_d- Width of exit diffuser

BEP- Best Efficiency Point

Be_o -Width of elbow outlet

D_2- Diameter of suction cone outlet

D_3- Diameter of suction cone inlet

H -Net head

he -Height of elbow

heo -Height of elbow outlet

hloss -Hydraulic losses in draft tube

hs -Suction head

hsc -Height of suction cone

Lax -Axial length of an elbow

Ld- Length of exit diffuser

Le- Length of elbow from axis

Nsq- Specific speed

Pa- Atmospheric pressure

P_3- Static pressure at draft.

REFERENCES

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

 DOI : 10.18231/2454-9150.2018.0292

[2] Susan-Resiga, R., Muntean, S., Stein, P.Avellan, F.," Axisymmetric Swirling Flow Simulation of the Draft tube Vortex in Francis Turbines at Partial Discharge" In 24th IAHR Symposium on Hydraulic Machinery and systems, October 27-31, Foz Do Iguassu, Brazil.

[3] Susan-Resiga, R., Ciocan, G.D., Anton,L., Avellan F., " Analysis of Swirling Flow Downstream a Francis Turbine Runner" Journal of Fluids Engineering. 177-189. Vol. 128, January 2006.

[4] Barlit, V.V.," Hydraulic Turbines (Hydraulic Theory, Computations and Experimental Investigations)" Vol-1, Vol-2. Maulana Azad National Institute of Technology, Bhopal, 2007.

[5] Vishal Soni, "Design development of optimum draft tube for High head Francis turbine using CFD "Proceedings of the 37th International & 4th National Conference on FMFP, IIT Madras, 2010.

[6] Vishnu Prasad, Ruchi Khare and Abhas Chincholikar, "hydraulic performance of elbow draft tube for Different geometric configurations using cfd." Proceedings of 8th International Conference by IGHEM, IIT, Roorkee,2010.

[7] Dr. Ruchi Khare, Dr. Vishnu Prasad, Mitrasen Verma" design optimization of conical Draft tube of hydraulic Turbine",IJAEST, March 2012.

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

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

[10] Avellan, F.," Flow investigation in a Francis Draft tube: The Flindt Project" In 20th IAHR Symposium on Hydraulic Machinery and systems, 6-9 August, Charlotte,2001.

[11] Sumeet J. Wadibhasme, Shubham Peshne, Pravin Barapatre, Santosh Barde, Saurabh Dangore, Shubham Harde, Prof. Shailendra Daf, "Hydraulic Turbine Draft Tube: Literature Review", International Journal of Science, Engineering and Technology Research (IJSETR), Volume 5, Issue 3, March 2016.

[12] Hosein Foroutan, Savas Yavuzkurt "An axisymmetric model for draft tube flow at partial load" Journal of Hydrodynamics,2015.

[13] S. Moradi, A. Yeganeh, M. Salimi ," CFD-modeling of effects of draft tubes on operating condition in spouted beds" Applied Mathematical Modelling, Elsevier Publication 2013.

[14] Gizem Demirel, Erdem Acar , Kutay Celebioglu, Selin Aradag,"CFD-driven surrogate-based multi-objective shape optimization of an elbow type draft tube " Hydrogen Energy Publications LLC. Published by Elsevier 2017.

 DOI : 10.18231/2454-9150.2018.0292