International Journal of Research in Engineering, Science and Management
Volume-1, Issue-10, October-2018
www.ijresm.com | ISSN (Online): 2581-5792

Comparative Analysis of Hydraulic Turbine Draft Tube with Varying Geometric and Operating Parameters Using CFD

Himanshu Chaudhary[1], Vardan Singh Nayak[2]
[1]*M. Tech. Scholar, Department of Mechanical Engineering, VIST, Bhopal, India*
[2]*Assistant Professor, Department of Mechanical Engineering, VIST, Bhopal, India*

***Abstract*—The efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. The shape and velocity distribution at the inlet are, in next turn, two main factors that affects the performance of the draft tube. Traditionally, the design of this component has been based on simplified analytic methods, experimental rules of thumb and model tests. CFD are advanced models of fluid mechanics widely used in the analysis of complex in setting hydraulic systems, leading to optimal design solutions. FLUENT is a hydrodynamic model that applies the technique of finite volumes to solve the equations that describe the flow, as the continuity equation and the Euler or Navier Stokes' equations also known as Reynolds equations. This model features two types of calculation algorithms that can be solved by a system of equations. In present work 3D real flow analysis with turbulence model is done for draft tube model and characteristics of the prototype turbine predicting at the actual operating regimes. In present work we have analyze the elbow draft tube with varying cross section for mass flow rate of (500 kg/s – 2000kg/s). In optimized draft tube at mass flow rate of 2000 kg /s the velocity magnitude is high and pressure is decreasing as compare to base model draft tube. When we increasing the mass flow rate of fluid the velocity and pressure is decreasing which is required for better efficiency of the system. A CFD-based design search can further be aided with a robust and user-friendly optimization frame work theory and engineering. In this paper, the CFD analysis of draft tube has been performed and results for the same are compared with base model and modified model reading and which are found within the limit.**

***Index Terms*—CFD, Fluent, Ansys, Hydro turbine, Draft tube, elbow draft tube, convergent length**

I. Introduction

The development and application of green energy technology have become the focus of promotion in global policy as well as research and development in industry and academia. With the progress and development of such technologies, the global demand for energy has also been increasing. In recent years, in addition to an increase in environmental awareness, people have begun to pay attention to how to reduce reliance on fossil fuels, and therefore hydroelectric technology has become one of the major options for the development of clean and renewable energy. This study considers the conversion of the smallest hydroelectric power mode and attempted to develop a power generation mode from the potential energy of building water supply pipelines. Taiwan Island is a densely populated area. Its undulating changes in terrain are extremely large, the steep terrain catchment does not enable easy conservation of water or preservation of water resources and most water resources are discharged after use, without being harnessed effectively. The purpose of this study is to design ultra-fine power generation blades that are suitable for building water piping systems and place them into a building water piping system; the water flow of the pipe drives the rotation of the power generation blades, thus achieving power generation. The pipeline diameters of building water supply systems are typically about 4 to 6 inches, so the blades to be placed inside the pipe require extreme miniaturization; and considering the design of the angle and the length of blades, precise calculations and multiple attempts by multiple methods are required. This innovative research presents a certain degree of difficulty in the development of the design and its applications. The urban architecture of Taiwan is mainly in the form of apartment complexes and the average height of these complexes is between 10 to 40 floors. Most of the buildings possess a "potential energy", which is very suitable for the development of Pico-hydroelectricity. Inside apartment complexes, in addition to the water supply and drainage systems, there are also many public facilities such as public spas, indoor and outdoor swimming pools, ecological ponds, fountains and landscaped ponds and other facilities, which require a large amount of water. If it is possible to reuse these water resources effectively for power generation, the electricity converted by the water could be reused in such facilities. In summary, this study hopes that the future generating capacity could supply the electricity consumption of public facilities in apartment complexes, such as lighting of public spaces. The utilization of the hydraulic force in the domain of the electricity production has long been majority; it has started in antiquity with mills of water. The techniques permitting the exploitation of hydroelectric resources have benefited important progress during the twentieth century, in the scope of projects construction of the hydroelectric center of great speed. Through their size, their precision and their efficiency, the equipment of these hydroelectric center and especially hydraulics turbines arrived in first plan of realization. The hydraulic turbine is a mechanic dispositive which is used to transform potential energy and the kinetic energy of water, in mechanic energy. This will then be transformed into electric energy by an alternator. There exist two categories of hydraulics turbine. The turbine of action, which do not constitute a draft tube and function with the kinetic energy of water, and the turbine of reaction, which function with the pressure difference and the energy pressure. With the increasing cost of energy and the high demand of green energy, hydraulic turbine of thin height of falls such as Francis and Kaplan turbines, are those targeted as being economically profitable. They are constituted of distributor, of volute, of runner and draft tube. The draft tube permits the recuperation of excess water kinetic energy coming from the runner and converts it into energy of static pressure. Many studies on the draft tube flow have been done. Marjavaara, carried out a numerical study to show that the draft tube have an important rule on the global efficiency of a hydraulic turbine.

II. LITERATURE REVIEW

1. Chih-Yuan Chang, Sy-Ren Huang, Yen-Huai Ma, Yin-Song Hsu, Yao-Hua Liu, with the rapid development of industry, commerce, and standards of living, the energy requirements of various nations are increasing significantly. Currently, the primary sources of global energy supply are oil, coal, and other fossil fuels. Due to the depletion of natural resources, the development of green energy technologies has become a vital topic in national development and academic research. Considering that household water from urban buildings is discarded after use, converting the energy from household water into electricity is the focus of this study.

2. Gunjanb. Bhatt, Dhaval B. Shah, Kaushik M. Pate, The efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. The shape and velocity distribution at the inlet are, in next turn, two main factors that affects the performance of the draft tube. Traditionally, the design of this component has been based on simplified analytic methods, experimental rules of thumb and model tests. In this paper, an attempt has been made for design automation of modeling of draft tube using Excel spreadsheet and Creo parametric software. In the last decade or two, the usage of computational fluid dynamics (CFD) has dramatically increased in the design process and will continue to grow due to is flexibility and cost-effectiveness. A CFD-based design search can further be aided with a robust and user-friendly optimization frame work theory and engineering. In this paper, the CFD analysis of draft tube has been performed and results for the same are compared with experimental reading and which are found within the limit.

3.R.G. Simpson, A.A. Williams, A research project is currently being undertaken in collaboration with Practical Action (ITDG) to develop a standard design procedure for Pico propeller turbines that can be manufactured locally in developing countries. A 5 kW demonstration turbine has been set up at a test site in Peru and Computational Fluid Dynamics (CFD) has been used to obtain overall performance data for the turbine and to assist in the design of a new rotor. It was found that an incorrect matching between the turbine rotor design and the available flow rate at the site significantly affected the turbine operation and in order to provide an acceptable performance it was possible to adjust just the runner design and operating speed of the turbine. The paper will present the initial CFD and field test results, and discuss the process by which computational fluid modelling has been used as an appropriate design tool.

4. Ramos, H. M., Simão, M. Borga, This paper deals with new design of low head turbines, as feasible solutions to solve the lack of energy in rural and remote areas, or to provide energy from urban water pipe systems. Propeller turbines are then the subject of this research because they are suitable for small heads, discharges with little variability, easy to manufacture and with low costs associated. Hence, the aims are the design of quite simple tubular propeller turbines and the analysis of hydrodynamic behaviour for different number and configuration of blades, based on CFD analyses and experimental tests development. An advanced hydrodynamic code based on the finite volume method, as well as blades configuration and mesh specific models are used for the impeller and the turbine design. The blade geometry is optimized using mathematical formulations and experimental results, concerning the possible range of operation under best efficiency conditions. Performance curves are obtained for typical characteristic parameters allowing comparisons between CFD and experimental results. Based on the similarity theory applied to turbo machines it is possible to evaluate the hydrodynamic behavior through a tubular propeller for different sizes, in a scale model application.

III. OBJECTIVE OF THE STUDY

- In present our main objective is to investigate the various type of operating conditions as well as the geometrical parameter for elbow type of draft tube and comparison will be done by CFD Simulation results.
- In the second part of study we will evaluate the results with various turbulence model and comparison will be done as per streamline flow in 3d Simulation. In present work we are going to apply mass flow rate for elbow draft tube and see the effect on pressure drop and velocity streamline flow and stability of the whole system.

IV. METHODOLOGY

A. *Basic Steps to Perform CFD Analysis*

1) *Pre-processing: CAD Modeling*

Creation of CAD Model by using CAD modeling tools for creating the geometry of the part/assembly of which you want to perform FEA.CAD model may be 2D or 3d.

2) Meshing:

Meshing is a critical operation in CFD. In this operation, the CAD geometry is discretized into large numbers of small Element and nodes. The arrangement of nodes and element in space in a proper manner is called mesh. The analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size (increasing no. of element), the CFD analysis speed decrease but the accuracy increases.

3) Type of Solver

Choose the solver for the problem from Pressure Based and density based solver.

4) Physical model

Choose the required physical model for the problem i.e. laminar, turbulent, energy, multi-phase, etc.

5) Material Property

Choose the Material property of flowing fluid.

6) Boundary Condition

Define the desired boundary condition for the problem i.e. temperature, velocity, mass flow rate, heat flux etc.

7) Solution

- *Solution Method:* Choose the Solution method to solve the problem i.e. First order, second order
- *Solution Initialization:* Initialized the solution to get the initial solution for the problem.
- *Run Solution:* Run the solution by giving no of iteration for solution to converge.

8) Post Processing

For viewing and interpretation of Result. The result can be viewed in various formats: graph, value, animation etc.

B. CFD Method Applied

The model was simulated and the required geometry configurations were pre-processed in ANSYS 14.5. This following section illustrates the method used in the CFD simulations in this particular study.

1) For fixed walls: On fixed walls, the no slip conditions are applied.

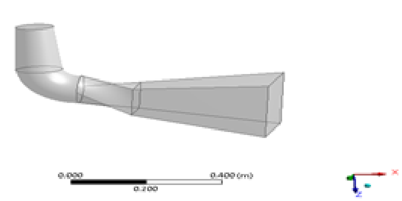

Fig. 1. CAD model

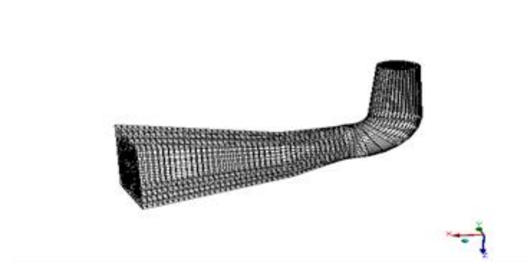

Fig. 2. Mesh model

V. Results and Discussion

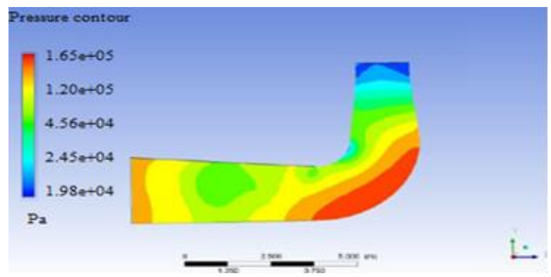

Fig. 3. Pressure contour in base model at mass flow rate 500 kg/s

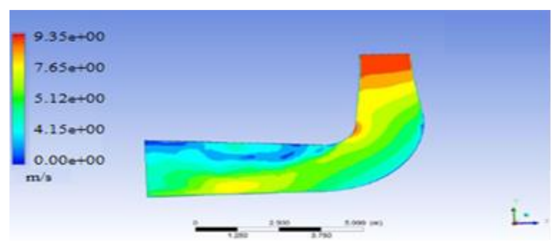

Fig. 4. Velocity contour in base model at mass flow rate 500 kg/s

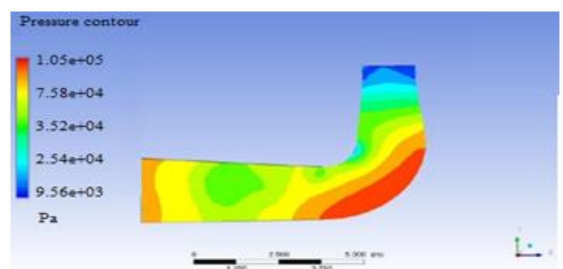

Fig. 5. Pressure contour in base model at mass flow rate 2000 kg/s

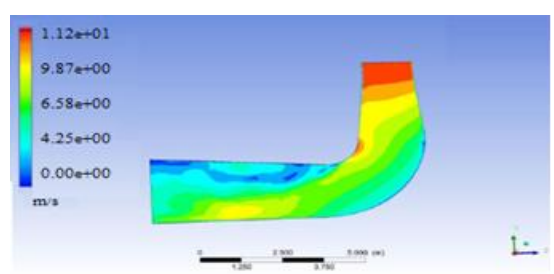

Fig. 6. Velocity contour in base model at mass flow rate 2000 kg/s

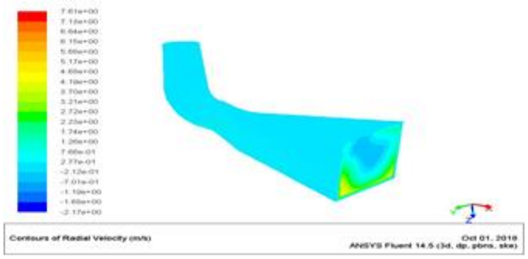

Fig. 7. Contour of radial velocity in modified model at mass flow rate 2000 kg/s

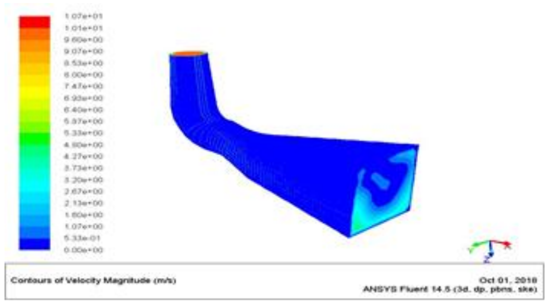

Fig. 8. Contour of velocity magnitude in modified model at mass flow rate 2000 kg/s

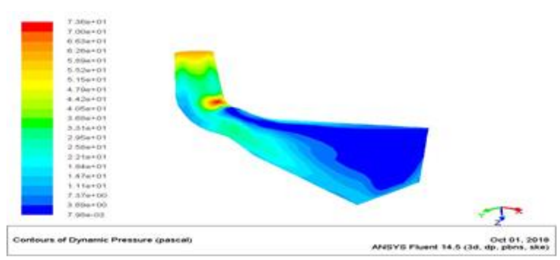

Fig. 9. Contour of dynamic pressure in modified model at mass flow rate 2000 kg/s

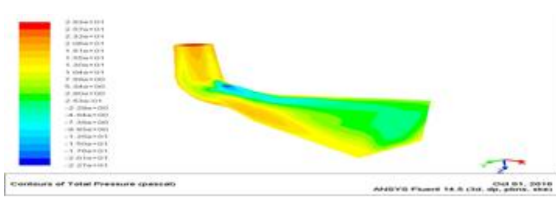

Fig. 10. Contour of total pressure in modified model at mass flow rate 2000 kg/s

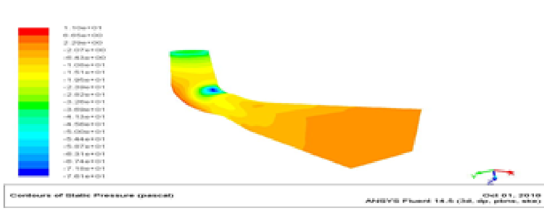

Fig. 11. Contour of static pressure in modified model at mass flow rate 2000 kg/s

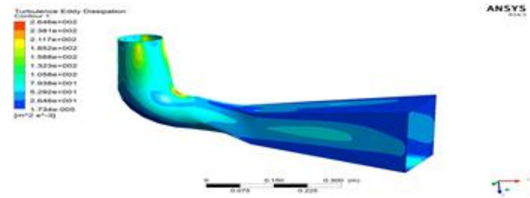

Fig. 12. Contour of Turbulance eddy dissipation in modified model at mass flow rate 2000 kg/s

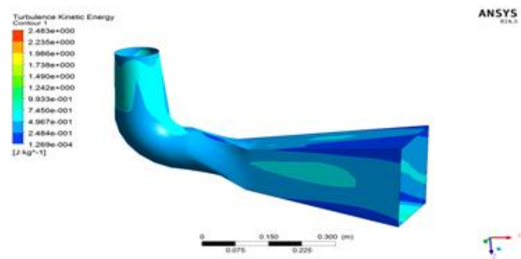

Fig. 13. Contour of Turbulance Kinetic energy in modified model at mass flow rate 2000 kg/s

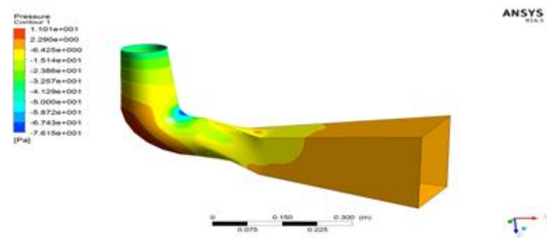

Fig. 14. Pressure contour in stream flow in modified model at mass flow rate 2000 kg/s

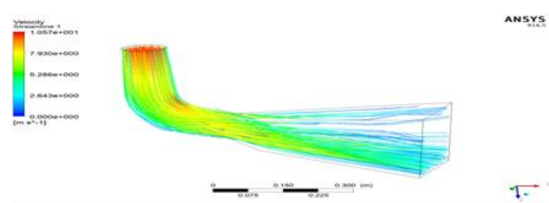

Fig. 15. Velocity streamline flow in modified model at mass flow rate 2000 kg/s

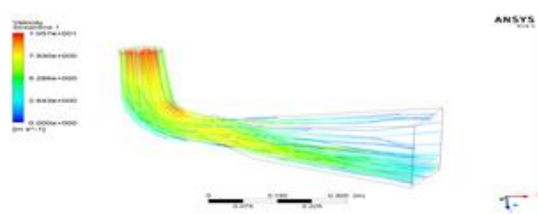

Fig. 16. Velocity streamline path flow in modified model at mass flow rate 2000 kg/s

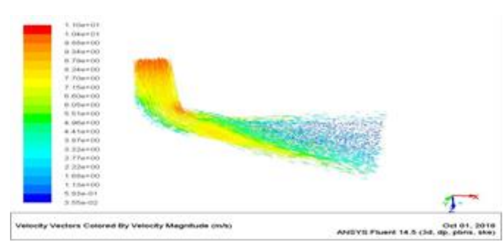

Fig. 17. Velocity vector flow of velocity magnitude in modified model at mass flow rate 2000 kg/s

TABLE I
RESULTS

Case Study	Mass flow rate	Pressure (Pa)	Velocity (m/s)
Base model (Elbow type)	500	1.65e+05	9.35e+00
Base model (Elbow type)	1000	1.41e+05	1.08e+01
Base model (Elbow type)	1500	1.25e+05	1.12e+01
Base model (Elbow type)	2000	1.05e+05	1.17e+01
Modified model Improved Convergent length	2000	2.83e+03	1.52+01

VI. CONCLUSION

With the growth of computational mechanics, the virtual hydraulic machines are becoming more and more realistic to get minor details of the flow, which are not possible in model testing. In present work, 3D turbulent real flow analyses in hydraulic Francis turbine have been carried out at three guide vane opening and different rotation speed using Ansys CFX computational fluid dynamics (CFD) software. The average values of flow parameters like velocities and flow angles at the inlet and outlet of runner, guide vane and stay vane of turbine are computed to derive flow characteristics.

In present study we have taken elbow draft tube with varying cross section and modified convergent length type geometry for simulation through CFD. Numerical investigation by CFD of draft tube with different mass flow rate shows an excellent result of flow phenomenon and stable in nature. When we increasing the mass flow rate of fluid the velocity is increasing and pressure is decreasing which is required for better efficiency of the system. In present work we have analyze the elbowl draft tube with varying cross section for mass flow rate of (500 kg/s – 2000kg/s). In optimized draft tube at mass flow rate of 2000 kg /s the velocity magnitude is high and pressure is decreasing as compare to base model draft tube. In recent years different type of draft tube is used in Industry for increasing the efficiency of hydro turbine but stability of flow phenomenon is the major concern of whole system at the time of working model. CFD simulation results shows that fluid flow steam is stable in nature at maximum mass flow rate in conical draft tube with varying cross section and turbulence effect is negligible at sharp edges due to curved effect.

REFERENCES

[1] Gunjanb. Bhatt, Dhaval B. Shah, Kaushik M. Pate, R.G. Simpson, A.A. Williams “design automation and CFD analysis of draft tube for hydro power” International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-6, June-2015.

[2] Ramos, H. M., Simão, M. Borga, Danilo de Souza Braga, Newton Sure Soeiro, Marcos Antônio Feitosa da Silva, Jacques Philipe Marcel Sanz, Jacques, Mohd Izzat B Kamarulzaman, , “CFD and Experimental Study in the Optimization of an Energy Converter for Low Heads” Energy Science and Technology Vol. 4, No. 2, 2012, pp. 69-84.

[3] Lovisa Nöid “CFD Computations of Hydropower plant Intake flow using Unsteady RANS” KTH School of Industrial Engineering and Management Energy Technology EGI-2015-019MSC EKV1083 Division of Heat and Power Technology SE-100 44 STOCKHOLM.

[4] Sumeet J. Wadibhasme, Shubham Peshne, Pravin Barapatre, Santosh Barade, Saurabh Dangore5, Shubham Harde6, Prof. Shailendra Daf,” hydraulic turbine draft tube a literature review” International Journal of Science, Engineering and Technology Research (IJSETR), Volume 5, Issue 3, March 2016.

[5] Spandan Chakrabarty, Bikash Kr. Sarkar, Subhendu MaityVishnu Prasad, Ruchi Khare, Abhas Chincholikar, “CFD Analysis of The Hydraulic Turbine Draft Tube to Improve System Efficiency”.

[6] B. Daniel Marjavaara, “CFD Driven Optimization of Hydraulic Turbine Draft Tubes using Surrogate Models Luleå University of Technology Department of Applied Physics and Mechanical Engineering Division of Fluid Mechanics 2006:41.

[7] Tiberiu Ciocan, Romeo Susan-Resiga, Sebastian Muntein, “Improving Draft Tube Hydrodynamics Over A Wide Operating Range” Proceedings Of The Romanian Academy, Series A, of the Romanian Academy Volume 15, Number 2/2014, pp. 182–190.

[8] Umashankar Nema, Rohit Rajvaidya, “Design and evaluation of performance of conical type draft tube with variation in length to diameter ratio”.

[9] François Avellan, “Flow Investigation in a Francis Draft Tube” The Flindt Project", Paper DY03, 18 pages. Proceedings of the 20th IAHR Symposium on Hydraulic Machinery and Systems, Charlotte NC-USA, August 6-9, 2000.

[10] Ali Abbas and Arun Kumar, “Development of draft tube in hydro-turbine: a review” International Journal of Ambient Energy Volume 38, 2017 - Issue 3.

[11] Dheeraj Sagar, Tarang Agarwal, Shubham Bhatnagar, Ruchi Khare, Vishnu Prasad, Mitrasen Verma, Fernando Casanova García, Carlos Alberto Mantilla Viveros,” CFD Simulation and Analysis of Conical Draft Tube of Experimental Setup of Hydraulic Turbine International Journal of Innovative Research in Science, Engineering and Technology, Vol. 5, Issue 4, April 2016.

[12] Philippe Gouin, Claire Deschenes, Monica Iliescu, Gabriel Dan Ciocan, Experimental Investigation of Draft Tube Flow of an Axial Turbine by Laser Doppler Velocimetry IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, October 14-16, 2009, Brno, Czech Republic.

[13] Vishnu Prasad, Ruchi khare, Abhas chincholikar, Numerical Simulation for Performance of Elbow draft tube at different geometric configurations, continuum mechanics, fluids, heat, 178-182.

[14] Avellan F. Flow investigation in a Francis draft tube: the FLINDT Project. In: Proceedings of 20th IAHR symposium on hydraulic machinery and system, Charlotte, NC, US; (2000).

[15] Nilsson H. Numerical investigation of turbulent flow in water turbines. Doctoral thesis, Chalmers University of Technology, Gothenbourg, Sweden; (2002).

[16] Mauri S. Numerical simulation and flow analysis of an elbow diffuser. Doctoral thesis, E´ cole Politechnique Fe´de´rale de Lausanne, Lausanne, Switzerland; (2002)

[17] Mohammad Hasan Shojaeefard, Ammar Mirzaei, Ali Babaei, Shape optimization of draft tubes for Agnew microhydro turbines, Energy Conversion and Management,79:681-689 (2014).

[18] Date Ab, Date Ash, Akbarzadeh A. Investigating the potential for using a simple water reaction turbine for power production from low head hydro resources. Energy Conversion Manage;66:257–70 (2013)

[19] Fluid Mechanics Fundamental and Applications, Second Edition, Yunus A. Cengel & John M. Cimbala, Tata Mc GrawHill Publication, Page 816.

[20] Ruchi Khare, Dr. Vishnu Prasad, Mitrasen Verma, “Design Optimization of Conical Draft Tube of Hydraulic Turbine”, International Journal of Advances in Engineering, Science and Technology, Vol. 2 No. 1 Mar-May 2012.

[21] Gunjan B. Bhatt, Dhaval B. Shah, Kaushik M. Patel,” Design Automation and CFD Analysis of Draft Tube For Hydro Power Plant”, International Journal of Mechanical and Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-6, June-2015.

[22] Vishnu Prasad, Ruchi Khare, Abhas Chincholikar, “Numerical Simulation for Performance of Elbow Draft Tube at Different Geometric Configurations”

[23] Vishal Soni, Amit Roghelia, Jaymin Desai, Vishal Chauhan, “Design Development of Optimum Draft Tube For High Head Francis Turbine Using CFD”, 37th International & 4th National Conference On Fluid Mechanics And Fluid Power, FMFP2010, December 16-18, 2010, IIT Madras, Chennai, India.
[24] Z. Čarija, Z. Mrša, L.Dragović, “Turbulent Flow Simulation In Kaplan Draft Tube”, 5th International Congress Of Croatian Society Of Mechanics (ICCSM), September, 21-23, 2006, Trogir/Split, Croatia.
[25] Shake A, Koueni-Toko C, Djeumako B, Tcheukam-Toko D, Soh-Fotsing B, Kitchen A., “Hydrodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine”, International Journal of Hydraulic Engineering 2014.
[26] Dr. Vishnu Prasad, Dr. Ruchi Khare, “CFD: An Effective Tool for Flow Simulation in Hydraulic Reaction Turbines”, International Journal of Engineering Research and Application (IJERA), Vol. 2, Issue 4, July-August 2012, pp. 1029-1035.
[27] V De Henau, F A Payette, M Sabourin, C Deschenes, J M Gagnon, P Gouin, “Computational Study of A Low Head Draft Tube And Validation With Experimental Data”, 25th IAHR Symposium on Hydraulic Machinery and
[28] International Journal of Science, Engineering and Technology Research (IJSETR), Volume 5, Issue 3, March 2016 Systems, IOP Conf. Series: Earth and Environmental Science 12(2010).