iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 Issue: VI Month of publication: June 2017

DOI:

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Numerical Modeling and Analysis of Hydro Turbine Draft Tube with Different Geometry Using Fluent

Geet Agnihotri[1] Dr. A.C. Tiwari[2]
[1]*M.Tech. Scholar, Mechanical Eng. UIT Bhopal*
[2]*HOD Mechanical Eng. Department, UIT Bhopal*

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. 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. To provide electricity in off grid areas of the country, pico-hydro power plants are very good option. Harnessing of energy from falling water, is the hydro power, such as water falling through steep mountain as a waterfall. The flowing water energy is converted into useful mechanical power by means of a water turbine.

In present work we have analyze the conical draft tube with varying cross section for mass flow rate of (1000 kg/s – 25000kg/s) and paper validation is also done with same boundary condition which is used in reference. In moody draft tube at mass flow rate of 25000 kg /s the velocity magnitude is low and pressure is increasing as compare to conical draft tube. 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.

Key words: cfd, fluent, ansys, hydro turbine, draft tube, moody draft tube, conical draft tube.

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. **5.** Danilo de Souza Braga, Newton Sure Soeiro, Marcos Antônio Feitosa da Silva, Jacques Philipe Marcel Sanz, Jacques, At Brazil, the generation of electrical energy from the available water resources is significant. This generation process can be based on the use of hydro generator, which is made up of an electric generator and turbine, and this turbine unit that converts hydraulic energy into mechanical energy, which is converted into electrical energy in the generator. This paper presents a numerical modeling, based on computational fluid dynamics (CFD) using the ANSYS CFX® for simulation hydro generator in operation at a generating unit located on north Brazil, where the hydrodynamic behavior of the set is determined, especially on volute, pre-guide vanes, turbine and duct suction. The results allow us to deduce that the phenomena arise from the hydraulic flow in each component, providing valuable information about the performance of hydro generator working in adverse

operating conditions. The model results are validated from parameter information obtained by sensors installed in the components of the hydro generator.

III. METHODOLOGY

A. Governing Equation

This section is a summary of the governing equations used in CFD to mathematically solve for fluid flow and heat transfer, based on the principles of conservation of mass, momentum, and energy.

1) *Conservation Equations:* Conservation equations used in this simulation report are as follows:

a) *Law of Conservation of Mass:* Fluid mass is always conserved or in other word we can say that, the difference in the rate of mass inlet to outlet will be equal to rate of mass storage in the system. In mathematical form it can be written as- mass =constant, d/dx (mass) = 0 ....(3.1)

Conservation of mass is responsible for the birth of the continuity equation.

Continuity equation: General 3-D continuity equation without any assumption can be written as-$\partial/\partial x\ (\ell u) + \partial/\partial y\ (\ell v) + \partial/\partial z\ (\ell w) + \partial/\partial t\ (\ell) = 0$ ....(3.2)

b) *Conservation of momentum*- Momentum is conserved in every direction of flow. Conservation of momentum helps us to find out several unknown in simple and convenient way. Equations for conservation of momentum in different direction are as follows:

1) Equation for conservation of momentum in the x-direction- $\partial/\partial t\ (\ell u) + \partial/\partial x\ (\ell uu) + \partial/\partial y\ (\ell uv) + \partial/\partial z\ (\ell uw) = \partial/\partial x\ (\sigma_{xx}) + \partial/\partial y\ (\tau_{yx}) + \partial/\partial z\ (\tau_{zx})$ ......(3.3)

2) Equation for conservation of momentum in y-direction- $\partial/\partial t\ (\ell v) + \partial/\partial x\ (\ell vu) + \partial/\partial y\ (\ell vv) + \partial/\partial z\ (\ell vw) = \partial/\partial x\ (\tau_{xy}) + \partial/\partial y\ (\sigma_{yy}) + \partial/\partial z\ (\tau_{zy})$ ......(3.4)

3) Equation for conservation of momentum in z-direction- $\partial/\partial t\ (\ell w) + \partial/\partial x\ (\ell wu) + \partial/\partial y\ (\ell wv) + \partial/\partial z\ (\ell ww) = \partial/\partial x\ (\tau_{xz}) + \partial/\partial y\ (\tau_{yz}) + \partial/\partial z\ (\sigma_{zz})$ ....(3.5)

c) Conservation of energy- Energy is always conserved quantity. It can neither be created nor destroyed but can change one form energy to other form of energy. Bernoulli's equation is popular equation based on energy conservation.
Bernoulli's equation are- $P/\ell g + V^2/2g + Z = \text{constant}$. ....(3.6)

General conservation equation of energy in Cartesian form is-

$\partial/\partial t\ (\ell E) + \partial/\partial x\ (\ell Eu) + \partial/\partial y\ (\ell Ev)\ \partial/\partial z\ (\ell Ew) = \partial/\partial x\ (u\sigma_{xx} + v\tau_{xy} + w\tau_{xz}) + \partial/\partial y\ (u\tau_{yx} + v\sigma_{yy} + w\tau_{yz}) + \partial/\partial z\ (u\tau_{zx} + v\tau_{zy} + w\sigma_{zz}) + \partial/\partial x\ \{k\partial/\partial x\ (T)\} + \partial/\partial y\ \{k\partial/\partial y\ (T)\} + \partial/\partial z\ \{k\partial/\partial z\ (T)\}$.........(3.7)

2) Sutherland's law- Sutherland's law is used to determine the dynamic viscosity and its general equation is: $\mu = \mu_o\ (T/T_O)\{(T_O + 120)/(T + 120)\}$ ....(3.8)

B. 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*

a) Solution Method: Choose the Solution method to solve the problem i.e. First order, second order

b) Solution Initialization: Initialized the solution to get the initial solution for the problem.

c) Run Solution: Run the solution by giving no of iteration for solution to converge.

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

Volume 5 Issue VI, June 2017
ISSN: 2321-9653

C. 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.

step 2

mesh file – to be meshed

.generated mesh model in the ansys

step 3 checks the mesh: -

various checks on the mesh and reports the progress in the console. also check the minimum volume reported and make sure this is a positive number select mesh to mm.

D. Methods

1) Pressure based

2) 3d Model is used

3) Gravity is enabling

E. MODEL

1) Energy equation is enabled.

2) K-Epsilon turbulence model used.

3) P-1 radiation model is used, since it is quicker to run. However DO radiation model can be used for more accurate results in typical models.

4) Finite rate / eddy dissipation in turbulence chemistry. Interactions are used for species model.

F. Step 4 simulation set up

1) Boundary conditions

a) Mass Flow inlet: mass flow rate is taken 10000, 15000, 20000 and 25000 kg/s,.

b) Outlet – pressure based, pressure is 1 bar.

2) *Material*

a) Fluid: Water

b) Thermal conductivity: Define two polynomial coefficients

(a) 0.0076736 (b) 5.8837*10-5

3) Polynomial coefficient for viscosity

(a) 7.6181e-06 (b) 3.2623e-8

4) For absorption coefficient taken complete domain.

5) Scattering coefficient is 1e-9.

6) Step 5 Solutions Method

a) Coupled

b) Presto model is used

Presto model is often used for buoyant flows where velocity vector near walls may not align with the wall due to assumption of uniform pressure in the boundary layer so presto can only be used with quadrilateral or hexahedral.

8) Pseudo transient is enabled

a) 0.1 time scale factor of turbulent kinetic energy and turbulent dissipation rate

b) Time scale factor of species and energy is 10

Note: - higher time scale size is used for the energy and species equation to converge the solution in less number of iterations.

Solution initialisation: - the solution is initialized

Run calculation: - start the calculation for 1000 iterations.

For fixed walls- on fixed walls, the no slip conditions are applied.

Volume 5 Issue VI, June 2017
ISSN: 2321-9653

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

IV. RESULTS & DISCUSSION

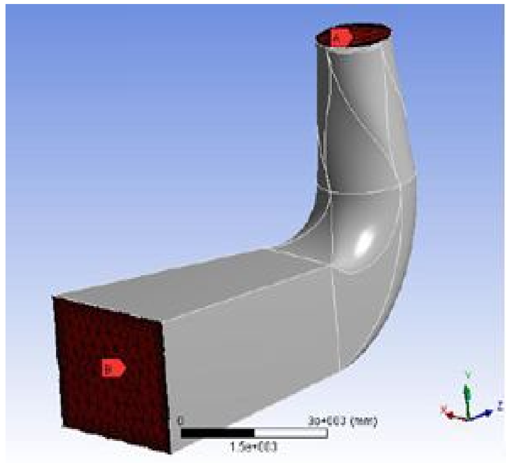

Figure 1 Base Model

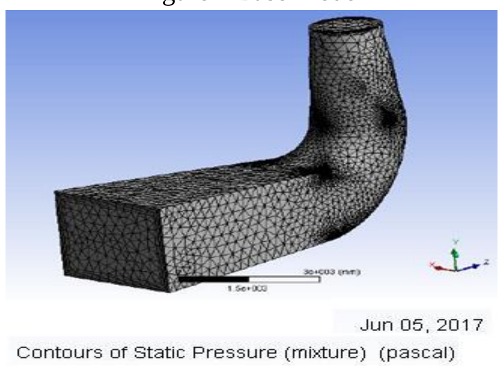

Figure 2 Mesh Model

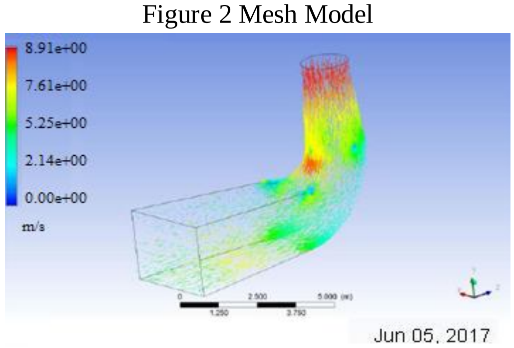

Figure 3 Velocity Contour at Mass Flow Rate 10000 kg/s

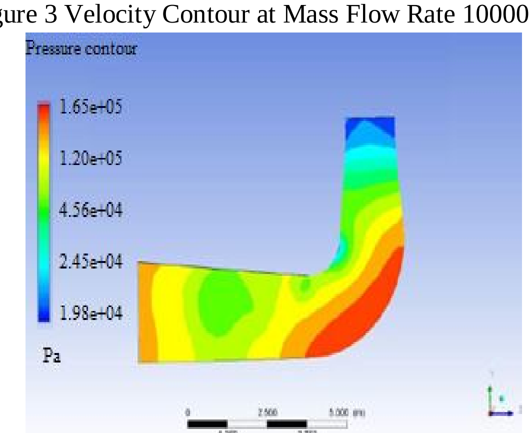

Figure 4 Pressure Contour at Mass Flow Rate 10000 kg/s

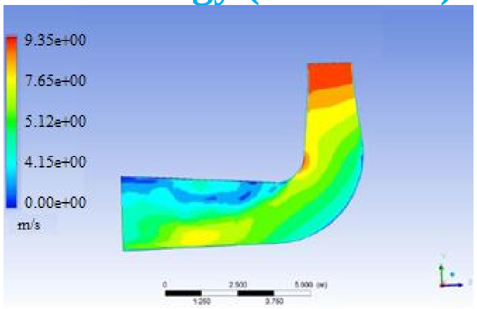

Figure 5 Velocity Contour at Mass Flow Rate 15000 kg/s

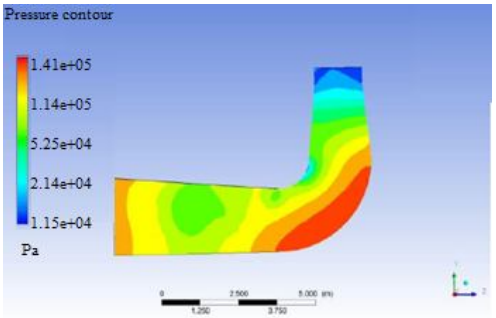

Figure 6 Pressure Contour at Mass Flow Rate 15000 kg/s

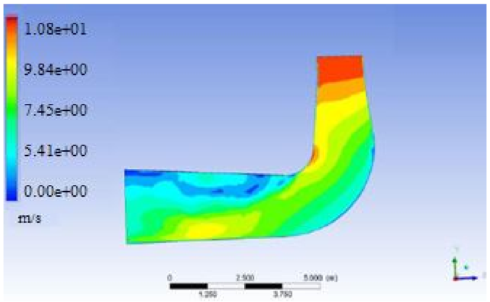

Figure 7 Velocity Contour at Mass Flow Rate 20000 kg/s

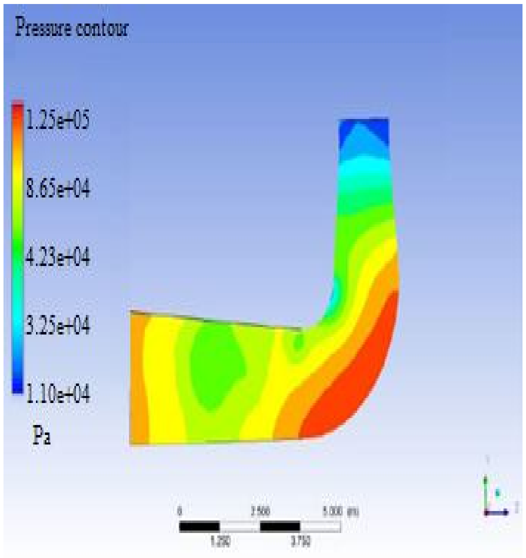

Figure 8 Pressure Contour at Mass Flow Rate 20000 kg/s

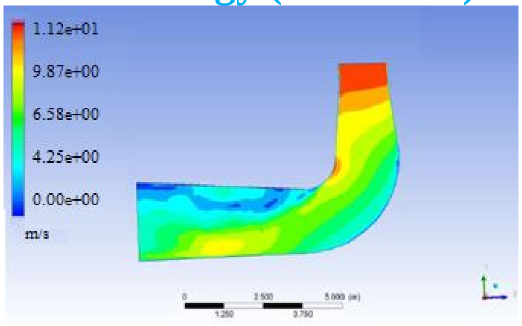

Figure 9 Velocity Contour at Mass Flow Rate 25000 kg/s

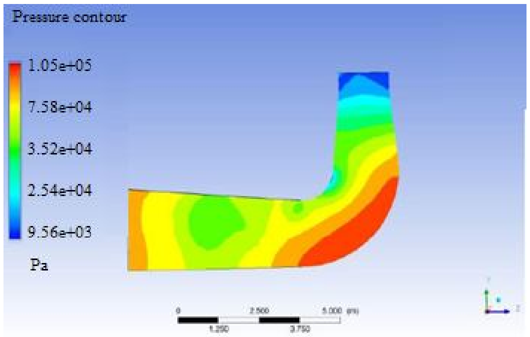

Figure 10 Pressure Contour at Mass Flow Rate 25000 kg/s

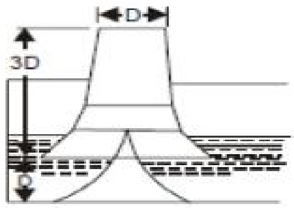

Figure 11 Schematic Diagram of Moody Draft Tube

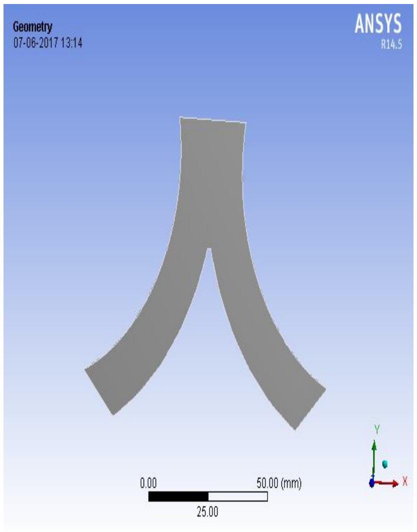

Figure 12 CAD Model of Moody Draft Tube

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

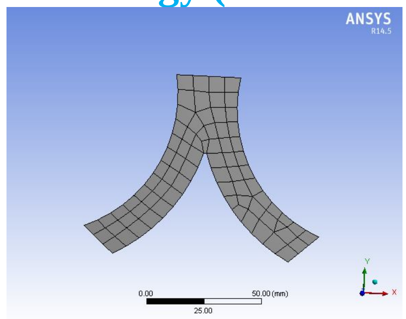

Figure 13 Mesh Model of Moody Draft Tube

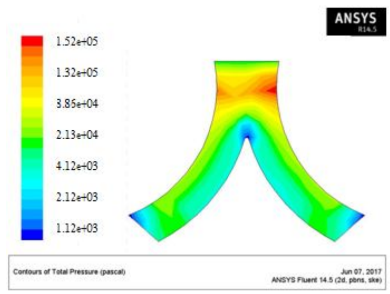

Figure 14 Pressure Contour at Mass Flow Rate 25000 kg/s

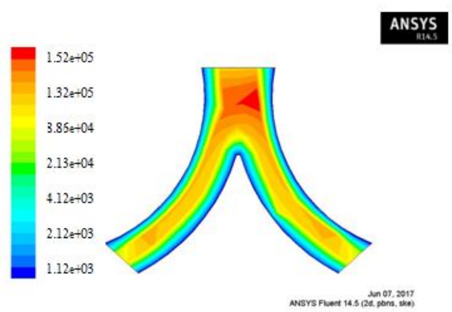

Figure 15 Dynamic Pressure Contour at Mass Flow Rate 25000 kg/s

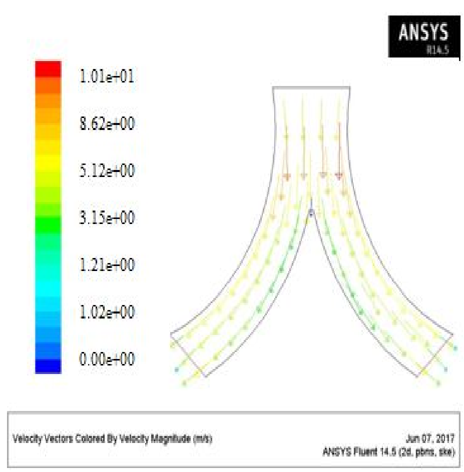

Figure 16 Velocity Contour at Mass Flow Rate 25000 kg/s

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

V. CONCUSION

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 conical draft tube with varying cross section and moody draft tube type geometry for simulation through CFD. Numerical investigation by CFD of draft tube with different mass flow rate shows a excellent results 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 conical draft tube with varying cross section for mass flow rate of (1000 kg/s – 25000kg/s) and paper validation is also done with same boundary condition which is used in reference. In moody draft tube at mass flow rate of 25000 kg /s the velocity magnitude is low and pressure is increasing as compare to conical 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
[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, Dr. Ruchi khare, Dr. Vishnu Prasad, Dr. Sushil Kumar,
[3] Lovisa Nöid
[4] Sumeet J. Wadibhasme, Shubham Peshne, Pravin Barapatre, Santosh Barade, Saurabh Dangore5, Shubham Harde6, Prof. Shailendra Daf,
[5] Spandan Chakrabarty, Bikash Kr. Sarkar, Subhendu MaityVishnu Prasad, Ruchi Khare, Abhas Chincholikar,
[6] B. Daniel Marjavaara,
[7] Tiberiu Ciocan, Romeo Susan-Resiga, Sebastian Muntein,
[8] Umashankar Nema*, Dr. Rohit Rajvaidya,
[9] François Avellan,.
[10] Ali Abbas & Arun Kumar,
[11] Dheeraj Sagar1, Tarang Agarwal, Shubham BhatnagarDr. Ruchi Khare, Dr. Vishnu Prasad, Mitrasen Verma, Fernando Casanova García*, Carlos Alberto Mantilla Viveros,
[12] Philippe Gouin, Claire Deschenes, Monica Iliescu, Gabriel Dan Ciocan,
[13] 1. 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] 2. 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] 3. 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] 5. 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] 6. 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] Dr. 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 (IJAEST), ISSN: 2249-913x, 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), ISSN: 2248-9622, Vol. 2, Issue 4, July-August 2012, pp. 1029-1035.

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

[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) 012084, DOI: 10.1088/1755-1315/12/1/012084.
[29] CFD Driven Optimization of Hydraulic Turbine Draft Tubes using Surrogate Models, Doctoral Thesis, B. Daniel Marjavaara, Luleå University of Technology Department of Applied Physics and Mechanical Engineering Division of Fluid Mechanics, 2006:4I, ISSN: I402-I544.