

Computational Fluid Dynamic Analysis of Cross Flow Turbine

Mrudang Patel, Nirav Oza, Karna Patel

U.G. Student, Department of Mechanical Engineering, NIRMA University, Ahmedabad, Gujarat, India

ABSTRACT: This report gives the basic understanding of turbine. Further the Cross-flow turbine is described in brief. CFD analysis of the flow of water inside the Cross-flow turbine with 18 curved blades is done in ANSYS CFD. The computational domain was consisted of guide vane with casing and impeller. As a turbulence model, k- ϵ model was used in the solution method for around 95, 40,000 grid elements. As boundary conditions velocity was specified at casing inlet and static pressure was defined at casing outlet. The impeller was considered in moving reference frame with rotational speed of 300 rpm. The hydraulic efficiency obtained with CFD analysis and theoretical approaches were found as 84% and 88% respectively; which shows very good agreement between the two approaches.

KEYWORDS: Cross Flow Turbine, Computational Fluid Dynamics, Efficiency, Hydraulic turbine.

I. INTRODUCTION

Hydro-power is an environmental friendly and an easy way to generate electricity. It's a renewable source of energy. Here cross-flow turbine is taken into consideration which is a low – head turbine. In small villages or hilly areas where the daily requirement is low, cross-flow turbine serves good purpose there.

In recent years there is increasing research in the field of small hydro-electric power plants. There are many sites which are suitable for low-head hydro turbines. Low head hydro turbines include Kaplan Turbine which is axial inlet and axial outlet, Cross-flow Turbine which is radial inlet and outlet of water is across the runner, etc. have been the main field of interest.

Nowadays, the Cross-flow hydraulic turbine is gaining popularity in low head and small water flow rate establishments, because of its simple structure and ease of manufacturing in the site of the power plant. Cross-flow turbine is also called Banki-Mitchell turbine as they were the inventors and early developers of this turbine. Cross-flow turbine is low head turbine which works with tremendous flow rate. Thus, it is low specific speed turbo machine. The major parts of cross-flow turbine are: the runner and the nozzle. The runner is a hollow circular section of turbine on which the curved blades are supported at the ends.

The nozzle is the component of the turbine through which emerges a high speed jet of water. This water jet strikes the runner blades imparting the potential energy which is converted into the kinetic energy. The water from the jet strikes the curved blades due to which the runner starts rotating. The water after passing through the blade passes across the runner to the other side to again strike the blades which forces the runner once again. Here again, there will be the energy transfer between the water and the runner which maximizes the efficiency of this type of turbine. The efficiency of the cross-flow turbine is a function of several design parameters. These parameters include runner diameter, runner length, runner speed, turbine power, and water jet thickness, radius of blade curvature, blade spacing, and number of blades, the blade angles, exit angles and the attack angle. There is still research going on the above mentioned parameters to maximize the efficiency further.

II. RELATED WORK

Hydro-power was considered as one of the most desirable source of electrical energy due to its environmental friendly nature and extensive potential available throughout the world. Nowadays, the cross-flow hydraulic turbine is gaining popularity in low head and small water flow rate establishments, due to its simple structure and ease of manufacturing

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 9, September 2016

in the site of the power plant. The cross-flow turbine is composed of two major parts, the runner and the nozzle. One of the major design considerations in crossflow turbine was to minimizing this undesirable uncrossed water flow in order to achieve maximum efficiency.

Researchers Chiyembeke, Cuthbert and Torbjorn [7] had researched in the field of Cross-flow turbine. They had conducted research on the topic of “A Numerical Investigation of Flow Profile and Performance of A low cost Cross Flow Turbine” in the year of 2014. They studied the flow of water inside the turbine and found efficiency at the best efficiency point. Also they studied different results at the points away from best efficiency point. They used numerical methods to find the efficiency and to study the flow of water inside the turbine. They found the maximum efficiency of the turbine as to be 82%.

III. MODELLING, MESHING AND SOLUTION TECHNIQUES

Modelling

The modelling of the Cross-flow turbine includes the obtaining of the output parameters after the input parameters are applied. Some of the parameters like the friction coefficient of the blades, inlet velocity, outlet pressure, attack angle of the blades etc. are to be assumed and the output parameters are the design parameters of the turbine based on which the design is done. These parameters are the input and output diameters of the runner, area of inlet nozzle, number of blades, RPM of runner, output power of turbine, etc.

The 3D modeling of the turbine was done on DassaultSolidworks according to the design proposed by Oza and Patel [6]. The major specifications of the turbine are as per given in the table 3.1. Also the designed 3D fluid domain of the turbine is showed in the figure 1 (a), (b), (c), (d) & (e). The figure 1 (a) and (b) shows the fluid domains of the casing and the impeller of the turbine.

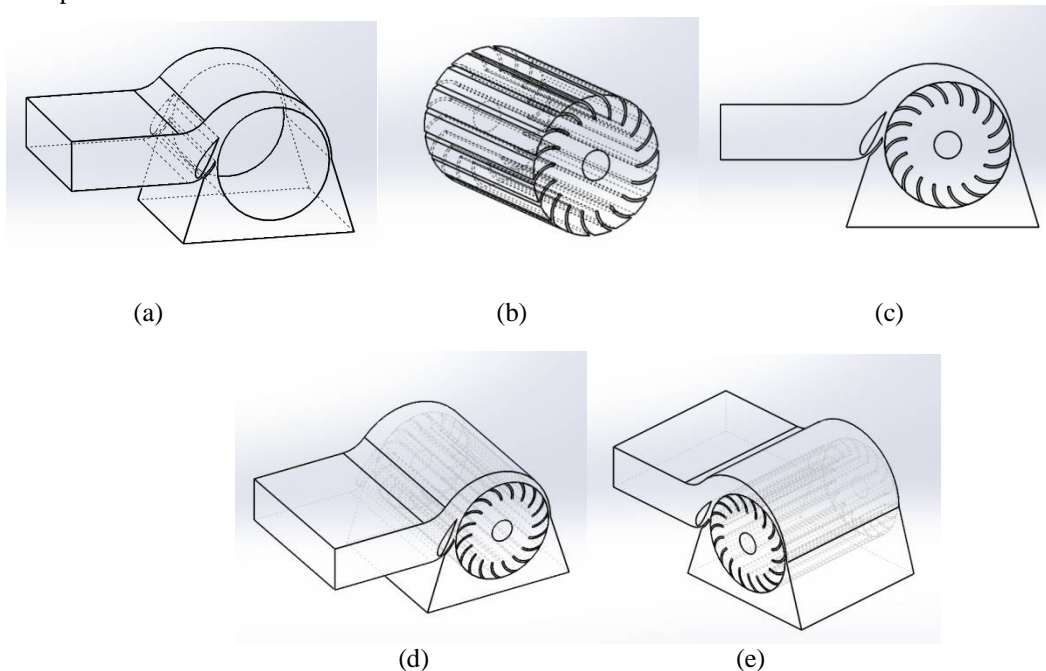


Figure 1 (a) Fluid domain of the casing of the turbine, (b) Fluid domain of the impeller of the turbine, (c) Side view of the turbine assembly of the fluid domains, (d) Isometric view of the fluid domain assembly, (e) Isometric view of the fluid domain assembly

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 9, September 2016

The figure 1 (a) shows the fluid domain of the casing. This is a stationary fluid domain in which there will be flow of water at the given input velocity. Here the inner wall of the fluid domain is defined as the interface between the impeller and the casing through which there will be transaction of water.

The figure 1 (b) shows the fluid domain of the impeller of the turbine. This is a rotational fluid domain which rotates at the given input RPM. Here the interface is the faces of the blades through which the water will transfer from the casing to the impeller fluid domain.

The figure 1 (c) shows the side view of the assembly consisting of the casing and impeller fluid domains. Here in this view we can see the 18 blades of the runner. Further in the figure 1 (d) and figure 1 (e) the isometric view is depicted from which we can get a clear idea of the whole fluid domain assembly.

Table 1 Specifications of Cross-Flow Turbine

Head	16 m
Water Flow Rate	$0.315 \text{ m}^3/\text{s}$
Output Power	26.95 kW
Rotational Speed	300 rpm
Number of blades	18
Impeller Diameter	230 mm

Meshing

There arises a need for meshing to mesh a particular component for its analysis in the numerical software like CFD. The meshing is the differentiation of the components into very small sized elements of different shapes like triangular mesh, square mesh, hexagon mesh, tetrahedron mesh etc. which can be employed at different places of the component for the accuracy of the results. The accuracy of the results is based on the mesh quality and sizing. As the size of the mesh reduces the accuracy of the result increases but the time for reaching to that result increases. The software calculates the inlet and outlet parameters at the different faces of the mesh thereby generating accurate result.

Another quality factor of the meshing is its skewness. The skewness is defined as to how much the mesh shape is distorted. Lower the skewness higher is the quality of mesh. In this study the size of elements of the impeller is 6 mm and that in the casing is 7mm the skewness was found out to be 0.234. The detail views of the meshed components and the view of inflated surfaces are given in the figures 2 (a), (b), (c), (d), (e) & (f). Also the mesh details can be seen in the table 2. It can be seen from the figures that the size of the mesh is very small and very accurate meshes are generated on the domain. We have selected the 'proximity and curvature' option to create minute and accurate meshes at the corners where there are chances of meshes failures. Inflation is provided at the interface of the fluid domains. Inflation is provided to check on the proper transaction of the fluid at the interface from one domain to another. In this study the inflation is provided at the four places which are:

- (i) At Casing – Runner Interface
- (ii) At the inner face of Guide blade
- (iii) At the side face of the Impeller
- (iv) At the faces of the Runner blades

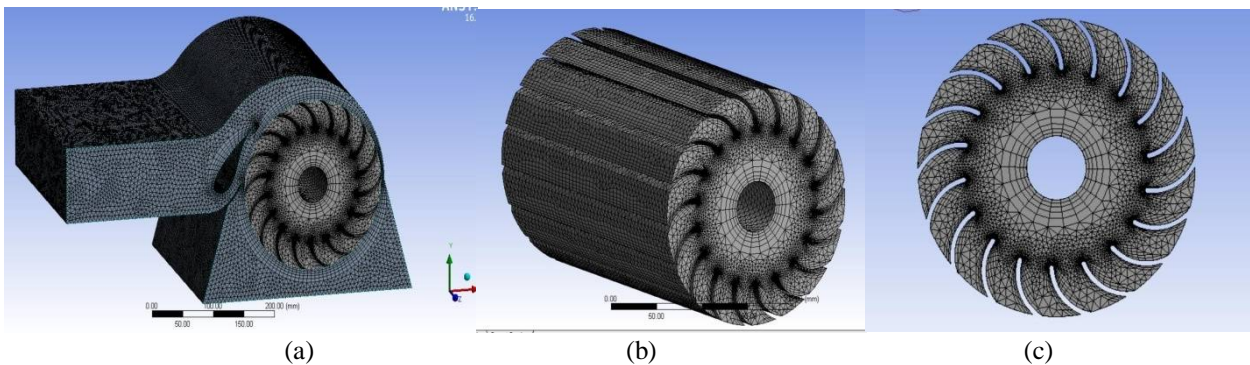
International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

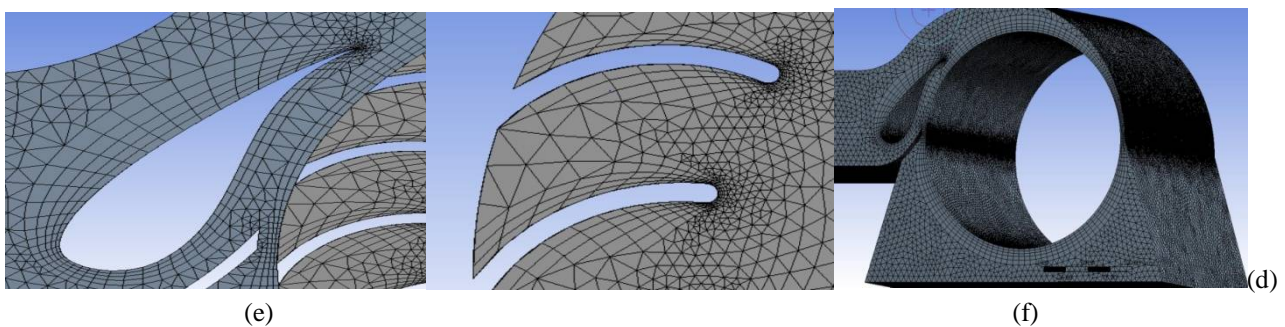
Vol. 5, Issue 9, September 2016

Table 2 Mesh details

Domain	Nodes	Elements
Casing	274505	1034249
Impeller	2178724	7853296
All Domains	2453229	8887545
INFLATION	5	
GROWTH RATE	1.2	
TRANSITION RATIO	0.77	
MINIMUM EDGE LENGTH	0.060345mm	



The figure 2 (a) shows the 3D view of the meshed assembly. Here we can see that a very fine mesh is generated. Also the inflation can be seen at the given faces. The figure 2 (b) shows the 3D view of the meshed impeller. Here the inflation is given on the side face of the impeller which can be seen in the figure 2 (c). Figure 2 (c) shows the 2D view of the impeller and the inflation on the face. Figure 2 (d) shows the inflation at the inner face of the guide blade. Figure 2 (e) show the inflation at the faces of the runner blades and the figure 2 (f) shows the inflation at the inner face of the casing.



International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 9, September 2016

Figure 2 (a) view of the meshed assembly, (b) 3D view of the meshed impeller, (c) 2D view of the side face of the impeller, (d) Inflation at the face of the guide blade, (e) Inflation at the runner blades, (f) Inflation at the inner face of the casing

Solution Technique After the successful meshing of the modelled component, different inlet and outlet boundary conditions are given on the inlet and the outlet faces of the fluid domain. Also the interfaces are generated wherever needed. The interfaces are needed to be generated wherever there is transaction of fluid from one domain to another domain. Boundary conditions are also needed to be provided at the interface. The boundary conditions are the initially assumed conditions of the fluid. The boundary conditions at inlet and outlet are shown in the Tables 3. The Table 4 shows the solution technique.

Table 3 Boundary conditions at outlet and inlet

Boundary – Outlet	
Type	OPENING
Location	Outlet
Settings	
Flow Direction	Normal to Boundary Condition
Flow Regime	Subsonic
Mass And Momentum	Opening Pressure and Direction
Relative Pressure	1.0000e+05 [Pa]
Turbulence	Zero Gradient

Boundary – Inlet	
Type	INLET
Location	Inlet
Settings	
Flow Regime	Subsonic
Mass And Momentum	Normal Speed
Normal Speed	9.0060e+00 [m s ⁻¹]
Turbulence	Medium Intensity and Eddy Viscosity Ratio

Table 4 Solution Technique

Turbulence Model – K Epsilon method	
solver control settings	
Advection Scheme	High Resolution
Turbulence Numeric	First Order
Convergence Control	
Min. Iteration	1
Max. Iteration	1000
Fluid Timescale Control	
Timescale Control	Auto timescale
Length Scale Option	Conservative
Time Scale Factor	1
Convergence Criteria	
Residual Type	RMS
Residual Target	0.001

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 9, September 2016

IV. RESULTS & DISCUSSION

General

After the successful input of the boundary conditions at the required faces, the solver is started for the calculations which calculate the momentum and mass flow at each and every faces of the mesh generated earlier. For the quality of accuracy, a convergence ratio is defined which is the quantity which governs the maximum allowable tolerance in the final results. In our case we have given the convergence to be 0.001 which gives very accurate final results. The numbers of iterations were given to be 1000. The above mentioned parameters can be varied from the solver panel.

Grid Independent Study

This is the very important part of the CFD analysis of any component. Here the final results are checked for different values of the number of mesh elements. When by varying the number of elements, the final results doesn't show variation then that number of elements is considered best for the calculation of the results i.e. by further increasing the number of elements the results would be independent of them. Thus, it is called grid independent study. In this study we have calculated the efficiency by varying the number of elements and the graph was plotted of efficiency – number of elements which can be seen in the figure 3.

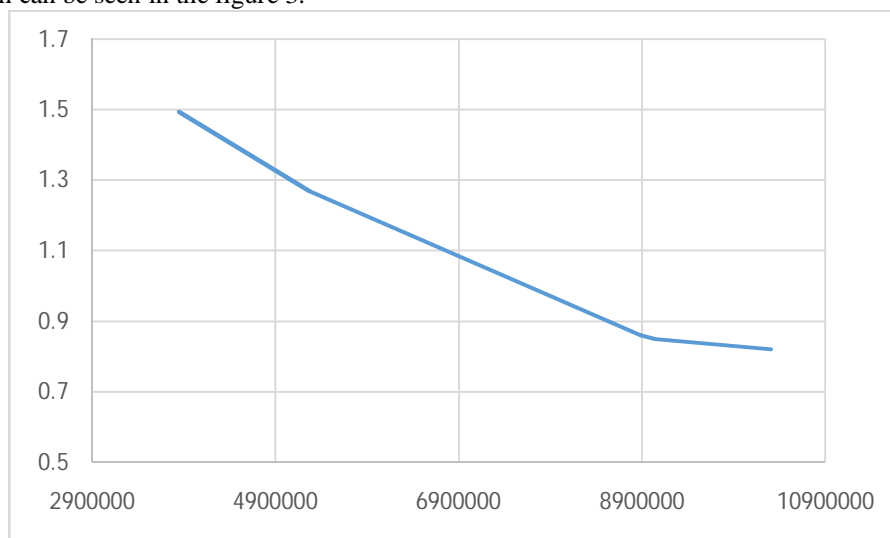


Figure 3 Graph of efficiency Vs Number of elements

Selection of Turbulence Model

The flow analysis of different turbulence model to work with like k- ϵ model, SSG model, k- ω model, Reynold stresses model etc. The solution methods vary accordingly with the model selected. The selection of the model is also considered important factor as the accuracy of the result is directly affected by it. As discussed in the literature review mostly all of the researchers had preferred the k- ϵ model we have employed the same in our flow analysis. By selecting that model we got very near approximation for the theoretical efficiency which supports the selection of the above mentioned k- ϵ model.

Flow Investigations in Different Components

After having the graph converged according to the input convergence, we can see the different contours for the results like pressure contour, velocity contour, velocity vector, streamlines etc. We can see the variation of pressure and velocity at different places in the fluid domain. For different flow rates we have computed the results and the pressure and the velocity contours are found which are displayed here. We can see that the figure 4 (a), (b) and (c) shows the velocity vectors in the casing at flow rates of $8.7m^3/s$, $9.0m^3/s$ and $9.3m^3/s$ respectively. It is seen that the velocity gradually increases from the mouth to the runner and then decreases which shows its loss of kinetic energy.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 9, September 2016

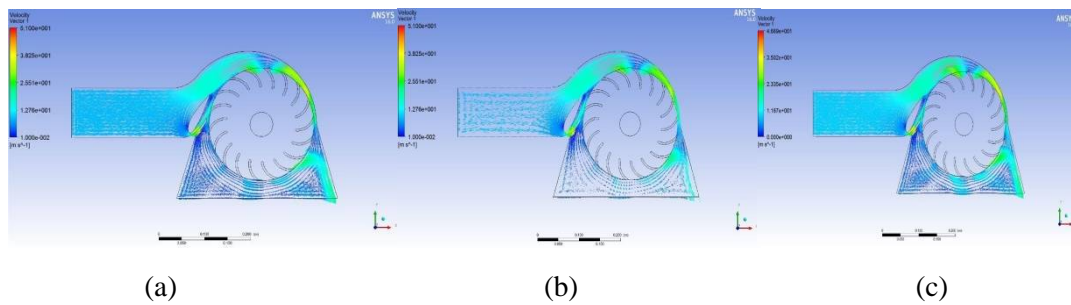


Figure 4 (a) Velocity vectors in casing at the flow rate of $8.7m^3/s$, (b) Velocity vectors in casing at flow rate of $9.01m^3/s$, (c) Velocity vectors in casing at flow rate of $9.3m^3/s$

Figure 5 (a), (b) and (c) show the pressure contour inside the casing which shows the pressure variation at different points. We can see that the figure 5 (a), (b) and (c) show the pressure contour in the casing at flow rates of $8.7m^3/s$, $9.0m^3/s$ and $9.3m^3/s$ respectively for the comparisons between them. We can see that the pressure reduces from mouth to the runner which supports the Bernoulli's equation. Also it is seen that at the places where there is very high velocity, pressure reduces drastically.

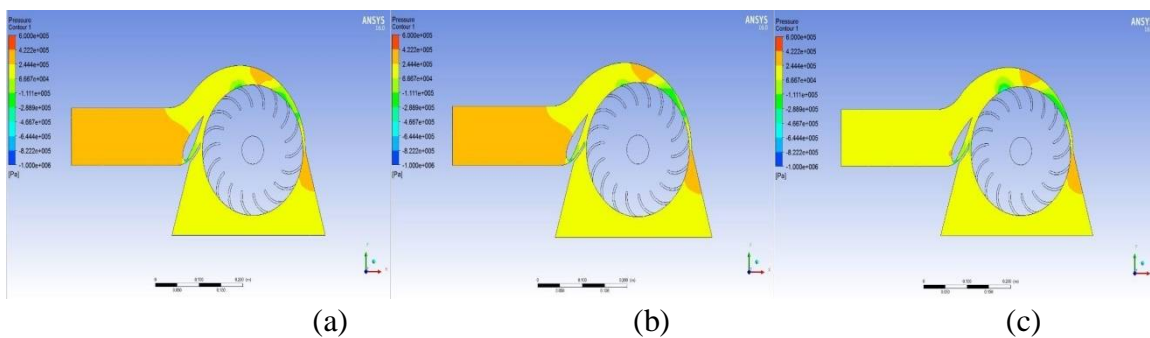


Figure 5 (a) Pressure contour in casing at flow rate of $8.7m^3/s$, (b) Pressure contour in casing at flow rate of $9.01m^3/s$, (c) Pressure contour in casing at flow rate of $9.3m^3/s$

Figure 6 (a), (b) and (c) shows the streamlines inside the casing from 1000 different points. We can see that the figure 6 (a), (b) and (c) shows the streamlines in the casing at flow rates of $8.7m^3/s$, $9.0m^3/s$ and $9.3m^3/s$ respectively for the comparison between them.

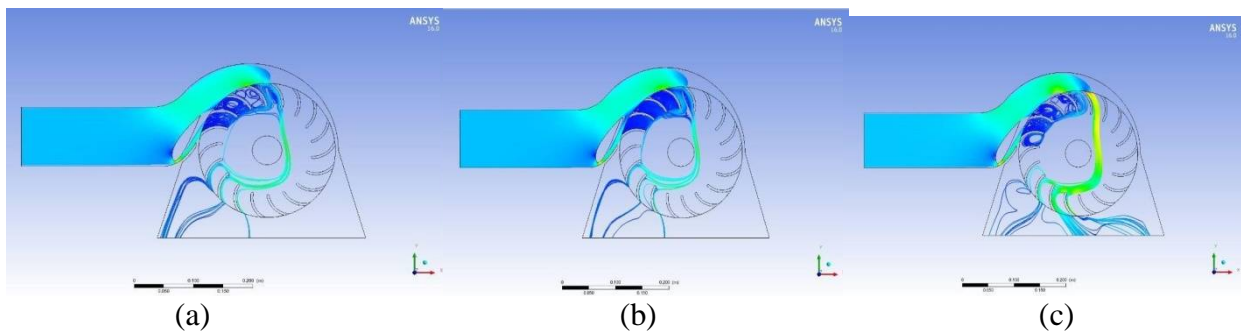


Figure 6 (a) streamlines in casing at flow rate of $8.7m^3/s$, (b) streamlines in casing at flow rate of $9.01m^3/s$, (c) streamlines in casing at flow rate of $9.3m^3/s$

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 9, September 2016

Evaluation of performance characteristics of the turbine

The performance curve shows the variation of net head, output power and the efficiency with the variation of the flow rate. Such curves can be seen in the Figures 7 (a) and (b). It can be seen that the net head and the efficiency increases with the increase in the flow rate. We can see from figure 7 (a) that on reducing the head till a certain value the flow rate increases. Further the flow rate increases non-linearly by increasing the head. Thus we can get an optimum value of head for which we can get proper flow rate. From figure 7 (b) it is seen that the efficiency increases with the increasing of flow rate. We get 84% efficiency at the flow rate of $0.315\text{m}^3/\text{s}$ which is optimum point. Further the efficiency reduces by increasing the flowrate.

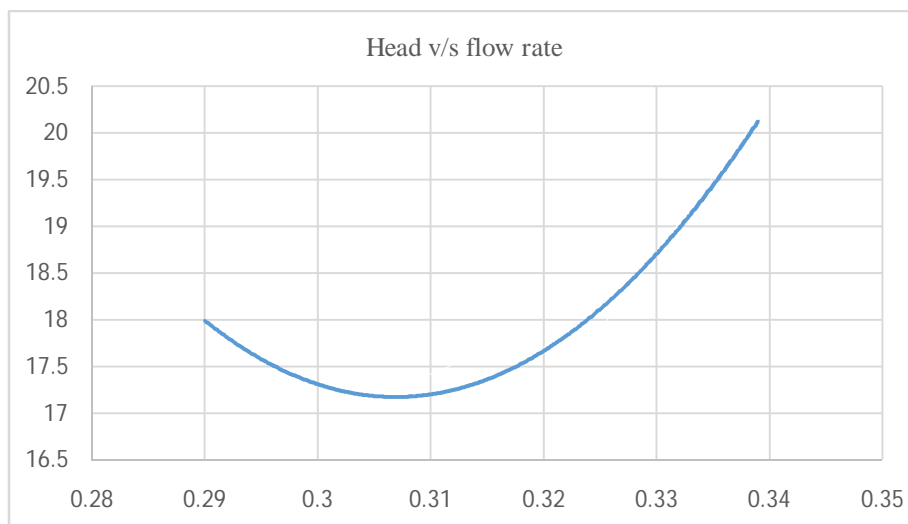


Figure 7 (a) Net Head vs. Flow Rate

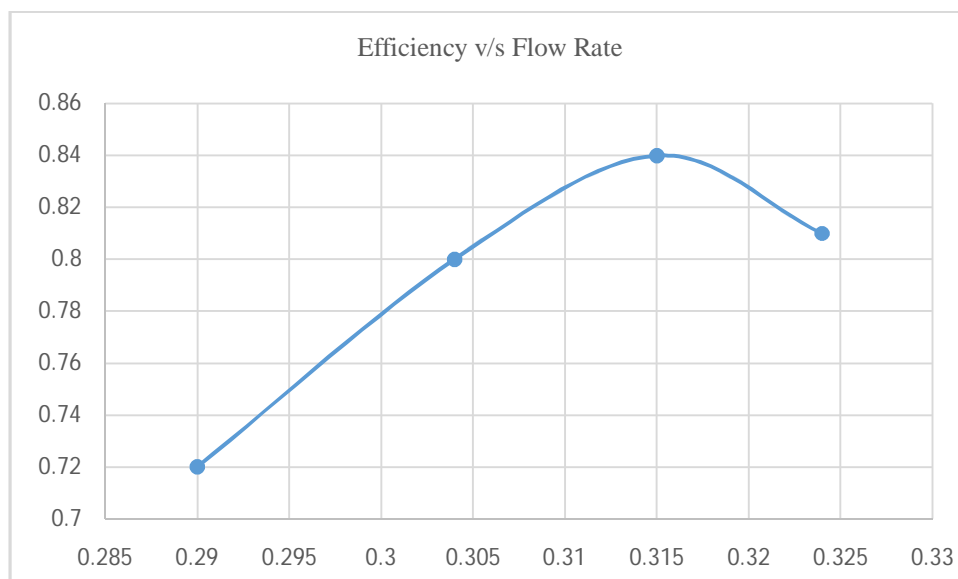


Figure 7 (b) Efficiency vs. Flow Rate

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 9, September 2016

V. CONCLUSIONS

From the present study, we get the basic knowledge of the Computational Fluid Dynamics numerical method for the flow analysis. From the Grid Independent study it is seen that further after 84% the variation in the efficiency is less than 1% on increasing the number of grid elements further from 95,40,000. The k- ϵ model employed here is the best model to reach up to the accurate results.

In this study a steady state CFD analysis of a 3-D model of forward curved 18 blades cross flow turbine is carried out. The contour and vector plot of pressure and velocity distributions in the flow passage are displayed. And the operating characteristics of the turbine are also computed from fluent numerical results.

From the study it was observed that there is a high pressure area inside the nozzle and near to the first stage runner blades. Then it is going to decrease at the second stage and out let of the turbine. And even it can be negative at the region where there is no cross flow.

The flow velocity at the inlet of the nozzle is constant and it is going to increase inside the nozzle nearer to the first stage runner blades and then going to decrease at the second stage outlet of the turbine.

The hydraulic efficiency obtained with CFD analysis and theoretical approaches were found as 84% and 88% respectively; which shows very good agreement between the two approaches.

REFERENCES

- [1] Javed, A. C. et al: "Design of a cross-flow turbine for micro-hydro power application", proceeding of ASME 2010 Power Conference, July, Chicago, Illinois, USA, 2010.
- [2] MACIEJ KANIECKI, (2002). Modernization of the Outflow System of Cross Flow Turbine, Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszerka 1480952 Gdansk, Poland kaniecki@imp.gda.pl. and Industrial Engineering, University of Dar es Salaam, P.O. Box 35091, Dar es Salaam, Tanzania. 2014.
- [3] Chiyembeke, Cuthbert and Torbjorn. Experimental study on a simplified cross flow turbine, Waterpower Laboratory, Department of Energy and Process Engineering, Norwegian University of Science and Technology, Trondheim 7491, Norway. Department of Mechanical
- [4] JusufHaurissa, SlametWahyudi, Yudy Surya Irawan, RudySoenoko. The Cross Flow Turbine Behavior towards the Turbine Rotation Quality, Efficiency, and Generated Power. Mechanical Engineering Doctorate Program, University of Brawijaya and Mechanical Engineering, Jayapura Science and Technology University, Jl. Raya Sentani Padang Bulan-Jayapura. Mechanical Engineering, University of Brawijaya, Jl. MT. Haryono 167, Malang (INDONESIA).
- [5] Mrudang Patel and NiravOza:" Design and Analysis of High Efficiency CrossFlow Turbine for Hydro-Power Plant", International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-5, Issue-4, April 2016.
- [6] Chiyembeke, Cuthbert and Torbjorn. Experimental study on a simplified cross flow turbine, Waterpower Laboratory, Department of Energy and Process Engineering, Norwegian University of Science and Technology, Trondheim 7491, Norway. Department of Mechanical.
- [7] Chattha, J.A., M.S. Khan, S.T. Wasif, O.A. Ghani, M.O. Zia et al., 2010. Design of a cross-flow turbine for micro-hydro power application. Proceeding of ASME 2010 Power Conference. Chicago, Illinois, USA, July.
- [8] Mockmore, C.A.; Merryfield, F. The Banki Water Turbine. In Bulletin Series, Engineering Experiment Station; Oregon State System of Higher Education, Oregon State College: Corvallis, OR, USA, 1949.
- [9] Carravetta, A.; delGiudice, G.; Fecarotta, O.; Ramos, H. Energy production in water distribution networks: A PAT design strategy. Water Resour. Manag. 2012, 26, 3947–3959.
- [10] Desai, V.R.; Aziz, N.M. An experimental investigation of cross-flow turbine efficiency. J. Fluid Eng. 1994, 116, 45–550.
- [11] Aziz, N.M.; Totapally, H.G.S. Design Parameter Refinement for Improved Cross-Flow Turbine Performance; In Engineering Report; Departement of Civil Engineering, Clemson University, Clemson, SC, USA, 1994.
- [12] Choi, Y.; Lim, J.; Kim, Y.; Lee, Y. Performance and internal flow characteristics of a cross-flow hydro turbine by the shapes of nozzle and runner blade. J. Fluid Sci. Technol. 2008, 3, 398–409.