Experimental and Numerical Computational Fluid Dynamics Analysis on the Flow at Pelton Turbine Nozzle with Various Opening Settings

Zar Chi Thaung

Department of Mechanical Engineering,

Technological University

Thanlyin, Myanmar

Theingi

Rector,

Technological University

Thanlyin, Myanmar

Kyi Kyi Swe

Department of Mechanical Engineering,

Technological University

Thanlyin, Myanmar

Abstract: This paper refers to the flow analysis of nozzle with spear for different nozzle opening of Pelton turbine. Experimental test was performed by using model turbine to find out the parameters during testing period. The flow rate, head and the rotational speed on the turbine shaft are measured for various nozzle openings covering a wide operation range. The best efficiency is 86% and output power of the model turbine is 110W. For water flow simulation of nozzle with spear by applying Computational Fluid Dynamics (CFD) software ANSYS CFX-14.5 are presented. The pressure and velocity distribution have also been obtained to study the effect of nozzle on flow pattern. The flow simulation has been carried out the standard k - ε turbulence model. And then, CFD simulation results with experimental value have been compared in this research.

Keywords: nozzle with spear, Pelton turbine, pressure, velocity, experimental test, CFD software

1. INTRODUCTION

Pelton turbine is the most commonly used impulse turbines in modern hydropower system. It is also called the constant-pressure turbine and there is no pressure drop across the buckets. It is regulated by decreasing or increasing the diameter of the jet and this is affected by means of a needle or spear which closes or opens the throat of the nozzle [1].

A nozzle transforms water under a high head into a powerful jet. The jet leaving the nozzle is surrounded by air [2]. The momentum of this jet is destroyed by striking the runner, which absorbs the resulting force. If the velocity of water leaving the runner is nearly zero, all the kinetic energy of the jet is has been transformed into mechanical energy, so the efficiency is high. The nozzle must be spaced so closely that water issued from one jet spared after striking the runner interferences with another jet. A Pelton turbine has one more nozzles discharging jets of water which strike a series of buckets mounted on the periphery of a circular disc [3, 4].

2. TESTING PROCEDURE OF MODEL PELTON TURBINE

The main purpose of this performance is to measure the power output and efficiency variation with rotational speed for different openings of the nozzle and to compare this to the CFD software ANSYS-14.5.

2.1 Experimental Set-Up

Experimental set-up consists of Pelton turbine, inlet pressure gauge, centrifugal pump, tachometer, calibrated orifice meter connected to mercury manometer, brake drum dynamometer with rope and mass loading arrangement.

2.2 Operation of Model Pelton Turbine

The model turbine is located in the Renewable laboratory at Mechanical Engineering Department which is situated in Mandalay Technological University. A schematic view of the experimental setup is shown in Figure 1. In this case, the nozzle opening position is five conditions for the whole test.

They are 20%, 40%, 60%, 80% and 100% nozzle opening position. Turbine is tested as a pump storage type of plant.
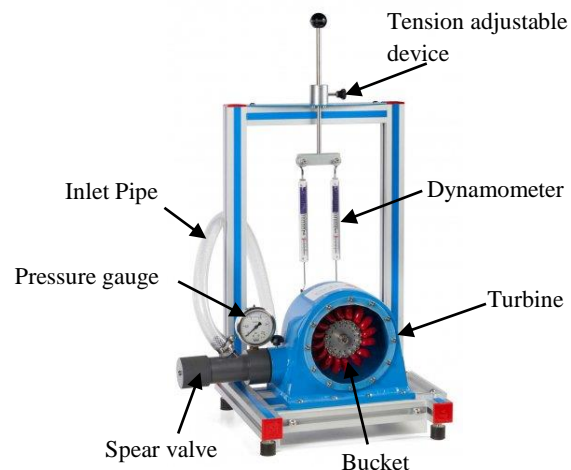

Figure 1. Schematic diagram of model turbine

The resource of required water is carried from the ground tank. It is pumped to the artificial water way by using the pump. The required jet opening is set by turning the hand wheel in anti- clockwise direction. The inlet pressure is read by using a pressure gauge. A pressure gage is attached to the water pipe entering the turbine for reading the available water head. The speed of the turbine is measured by using a tachometer. The rectangular iron box is used to measure the flow rate. The volume of the container is known and the flow rate is obtained simply by the ratio of the volume and the filling time.

A band brake connected to two dynamometers allows varying the load given to the turbine by means of a connection device. The rotational shaft speed is measured by a digital tachometer counting the frequency of the marking on the surface of the brake drum on the back side of the turbine. Energy is extracted from the turbine using an assembly comprising friction plates centered on the turbine shaft [5].

3. EXPERIMENTAL TEST OF MODEL TURBINE

The pressure gauge reading, the speed of turbine and load are read during the test. After reading the data, efficiency of Pelton turbine is calculated in various nozzle opening. The specifications of model turbine are described as follows;

Table 1. The specifications of model turbine

Descriptions	Symbols	Value	Units
Diameter of runner	D_r	96	mm
Nozzle inlet pipe diameter	D_i	16	mm
Diameter of nozzle	D_n	10	mm
Center to center distance from the rotor axis	C	110	mm
Distance between jet and bucket	X_{nb}	20	mm
Number of bucket	Z	16	-
Number of jet	n_j	1	-
Coefficient of velocity	C_v	0.985	-

3.1 Calculation of Effective Head and Efficiency

Firstly, the required 20% nozzle opening is set by turning the hand wheel in anti-clockwise direction and adjust the inlet pressure (at 1.37 bar indicated by a Bourdan tube pressure gauge) by operating the bypass line valve. Keep on loading the Pelton turbine by adding masses from no load up to until the Pelton wheel stops. At each loading, note down the rotational speed of the turbine using a tachometer. After that, 40%, 60%, 80% and 100% nozzle opening for each run have been done above procedure.

The primary feature of the impulse turbine is the power production as the jet is deflected by the moving buckets. Negligible head loss at the nozzle and at the impact with the buckets (assuming that the entire available head is converted into jet velocity). Jet velocity can be calculated by using Equation 1;

$$Q = A_j V_j n_j \tag{1}$$

The velocity of the water jet is a function of the pressure difference, when viscosity is absent;

$$V_j = C_v \sqrt{\frac{2p_{out}}{\rho_w}} \tag{2}$$

The theoretical power is a function of the bucket speed and the jet speed. The bucket speed is;

$$V_b = R_r \omega \tag{3}$$

The effective head at inlet to the turbine can be measured through use of the following Bernoulli equation;

$$H_{available} = \frac{p_{in}}{\rho_w g} + \frac{V^2}{2g} + Z \tag{4}$$

The energy equation applied to the control volume provides the power extracted from the available head by the turbine;

$$P_{available} = \rho g Q H_{available} \tag{5}$$

By applying the angular momentum equation to the same control volume about the axis of the turbine shaft of the power developed by the turbine can be written as

$$P_O = T\omega \tag{6}$$

And then, the efficiency of the turbine is defined as the ratio between the power developed by the turbine to the available water power can be obtained by using the equation 7[5, 6]. The result table of different nozzle position for model turbine is illustrated in Table 2.

$$\eta_t = \frac{P_O}{P_{available}} \tag{7}$$

Table 2. Results data for model turbine

Descriptions	Nozzle Opening (%)				
	20	40	60	80	100
N (rpm)	1583	1380	1255	1200	1075
$Q \times 10^{-4}$ (m^3/s)	5.08	5.17	5.22	5.28	5.32
V_{in} (m/s)	2.6	2.57	2.52	2.49	2.47
V_{out} (m/s)	15.9	15	14.85	14.6	14.4
p_{in} (Bar)	1.37	1.33	1.23	1.2	1.1
p_{out} (Bar)	1.3	1.16	1.14	1.1	1.07
H_{avai} (m)	26.9	25	23.8	23	21.8
P_o (W)	108	110	85	68	61
η (%)	80	86	70	57	53

4. NUMERICAL APPROACH

In general, a numerical analysis procedure involves the following steps: (i) development of governing equations for the flow physics; (ii) discretising the equations in both time and space, (iii) discretising the flow domain (meshing); (iv) solving the discretised equations using a computerised solver (or program) and (v) post-processing of the results using a computerised program. It is necessary to justify the numerical results by comparing them with experimental or theoretical results. As stated already, numerical analysis is currently the standard way of designing turbines in the industry.

Numerical analysis for predicting the performance of impulse turbine runners poses a special challenge because of complexity of the flow physics. In impulse turbines like Pelton turbine, the jet flow in the runner is intermittent and there is a movable free surface boundary between the jet and surrounding air. This means that the actual flow is unsteady and two phase with a movable free surface. Two approaches are being used to numerically analyze such a flow. These approaches are particle tracking method and a classical CFD method [7, 8].

Some previous investigators have attempted to numerically study the flow through the nozzle and bucket of Pelton turbine. Mack et al. used Eulerian CFD methods for simulating the flow at the distributor of a multi-jet Pelton turbine. Also they used the volume of fluid model to study the effects of secondary flows, due to pipe bends, on the quality of the jet formed at a Pelton turbine nozzle [9].

4.1 Governing Equation

Computational Fluid Dynamics (CFD) is concerned with numerical solution of differential equations governing transport of mass, momentum and energy in moving fluids [10]. Methods of CFD assume computation of liquid and gas flows by numerical solution of Navier – Stokes and continuity equations (for turbulent flows – Reynolds equations) which describe the most general case of movement of fluid medium. The correspondent sequence of actions from creation of a geometrical model and specifying boundary conditions to analysis of computation results is described in the paper [11].

The equations used in this numerical flow analysis are the Reynolds Averaged Navier Stokes (RANS) equation with turbulence models based on eddy viscosity concept [12]. RANS for the continuity and momentum equations are given in equation (8) and (9)

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \tag{8}$$

$$\frac{\partial \rho V}{\partial t} + \nabla \cdot (\rho V \otimes V) - \nabla \cdot (\mu_{eff} \nabla V)$$

$$= -\nabla p' + \nabla \cdot (\mu_{eff} \nabla V)^T + F \tag{9}$$

$$\mu_{eff} = \mu + \mu_t \tag{10}$$

where, F = the sum of body forces

   p = the modified pressure

   μ_{eff} = the effective viscosity accounting for turbulence

   V = vector of velocity

5. METHODOLOGY

The numerical analysis for this study was performed in ANSYS Workbench 14.5 where ANSYS CFX Project with its subprograms, namely: Geometry, Mesh, ANSYS CFX-Pre, ANSYS CFX solver and ANSYS Post-processing, is used as presented in Figure 2. As stated already, numerical analysis require coming up with a flow domain which has to be discretized (or meshed) in space and time (for unsteady state analysis) [13].
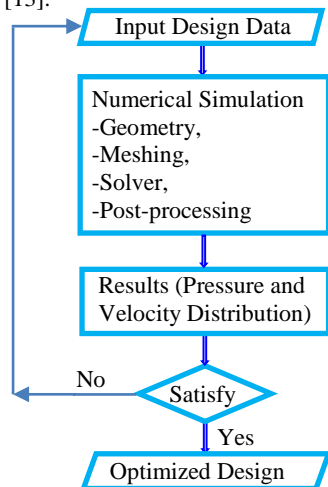

Figure 2. Flow chart for simulation procedure

5.1 Geometric Modeling

The flow domain Figure 3 is designed in "Geometry" sub-program using the ANSYS Design Modeler. The flow domain consists of the nozzle with spear and jet [14]. The jet is designed in ANSYS Design Modeler while the nozzle with spear in SolidWorks 2012. The numerical problem can be solved by using the "multiple frame of reference" as detailed in ANSYS Theory Guide of 14.5 [15].
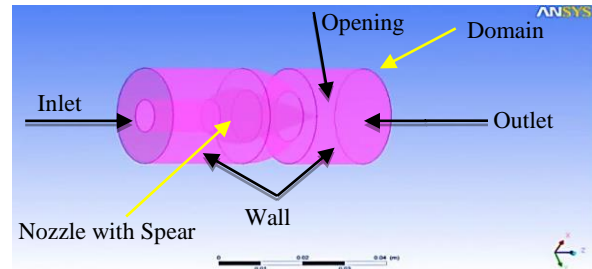

Figure 3. Three dimensional view of nozzle with spear

5.2 Mesh Generation

The geometry is divided into small sub parts for CFD analysis called mesh and the process is called meshing. The shape of the mesh elements can be triangular, quadrilateral, tetrahedral, hexahedral or prism depending upon the size and shape of the geometry. The shape and size of the mesh elements can be varied and are kept according to the dimension of the geometry, accuracy required, computational power of the system and memory [14].

The meshing of nozzle and spear domain has been done at spear shapes and the meshing of nozzle with spear domain for fillet spear shape [16]. As shown in Figure 4, the meshing of the fluid domain was divided into two standalone meshing components, one for the nozzle domain and one for the opening domain.
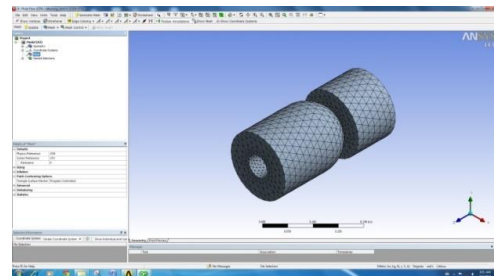

Figure 4. Meshing of flow domain for 100% nozzle opening

The main reason for this was that the nozzle domain was meshed with an advanced size function. The fluid domain was meshed with the advanced size function and consists mostly of triangular elements around the jet core and jet domain. Sweeping gives a high degree of structure in a mesh and this method was used on the jet and the surrounding domain. ANSYS Meshing 14.5 actually does a great job meshing the large part automatically. The summary of meshing data for each surface has been described in Table 3.

Table 3. Mesh information for nozzle with spear domain

Domain	Nodes	Elements
Nozzle	11181	61282

5.2.1 Domain and Interface Properties

The mesh is converted into the required format and is imported to ANSYS CFX-14.5 software. In ANSYS CFX-Pre the properties of the domains and fluid are defined along with their interface properties. The summary of the domain properties are given in Table 4.

Table 4. Domain physics for CFX in nozzle

Component	Feature	Details
Design Modeler		Geometry Creation
		Named Selection
CFX-Pre	User Mode	Quick Setup Wizard
	Location	Fluid
	Domain type	Fluid Domain
	Fluid list	Water
	Reference pressure	1 [atm]
	Buoyancy model	Not Buoyant
	Domain motion option	Stationary
	Turbulence option	k-Epsilon
	Coordinate Frame	Coord 0

5.2.2 Boundary Conditions

The flow parameters like pressure and velocity or mass flow rate are to be specified in the form of inlet and outlet boundary conditions to obtain numerical simulation and the nature of the boundary condition affects the accuracy of numerical solution. The boundary conditions were prescribed at inlet, outlet, wall and interface locations of the nozzle as shown in Table 5.

Table 5. Boundary physics for CFX

Domain	Boundaries	
Domain 1	Boundary - inlet	
	Location	inlet
	Flow Regime	Subsonic
	Mass Flow Rate	5.17e-01 [kg s^-1]
	Turbulence	Medium Intensity and Eddy Viscosity Ratio
	Boundary – opening	
	Location	opening
	Flow Direction	Normal to Boundary Condition
	Flow Regime	Subsonic
	Mass And Momentum	Opening Pressure and Direction
	Relative Pressure	1.0000e+00 [atm]
	Turbulence	Medium Intensity and Eddy Viscosity Ratio
	Boundary – outlet	
	Location	outlet
	Flow Regime	Subsonic
	Relative Pressure	1 [atm]
	Pressure Averaging	Average Over Whole Outlet
	Boundary - wall	
	Location	wall
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

6. GRAPHICAL PLOTS FOR NOZZLE

Pressure contours and velocity stream lines are achieved using insert contour and streamline commands of menu bar in ANSYS CFX-Post. Pressure contours and velocity stream lines for geometry at different nozzle openings have been shown below.
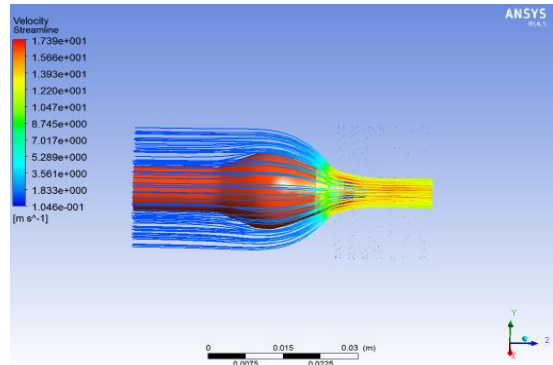

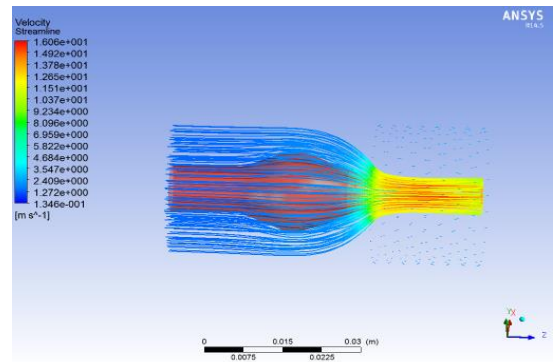Figure 5. Velocity distribution in geometry with 20 % nozzle opening

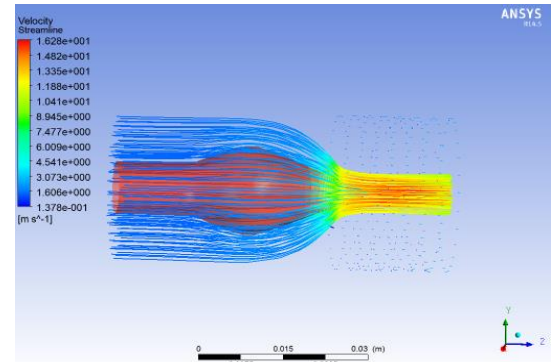Figure 6. Velocity distribution in geometry with 40 % nozzle opening

Figure 7. Velocity distribution in geometry with 60 % nozzle opening
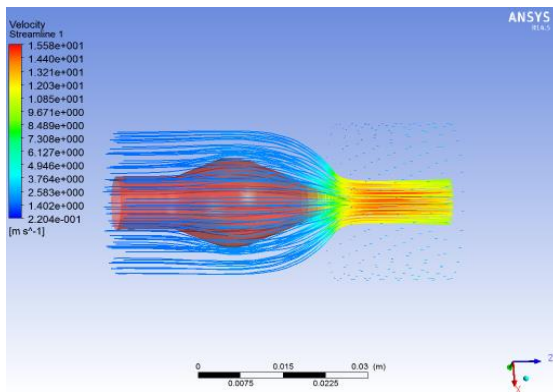

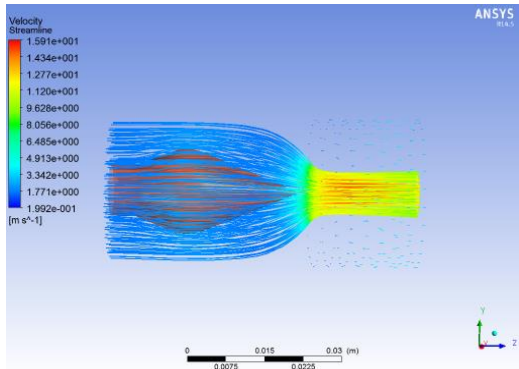Figure 8. Velocity distribution in geometry with 80 % nozzle opening

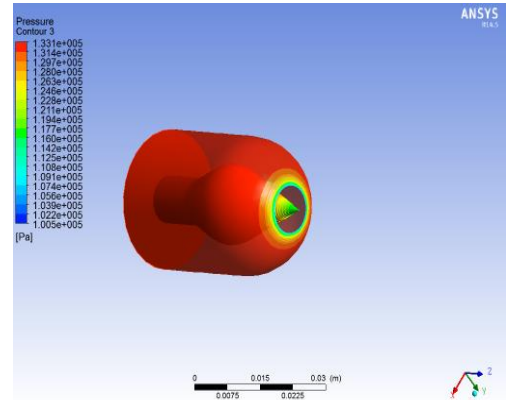Figure 9. Velocity distribution in geometry with 100 % nozzle opening
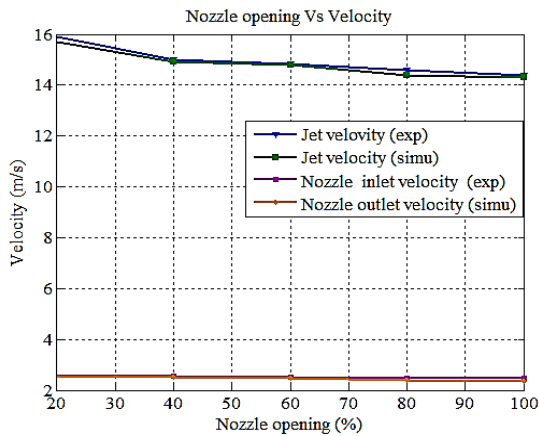

Figure 10. Relationship between nozzle opening and velocities
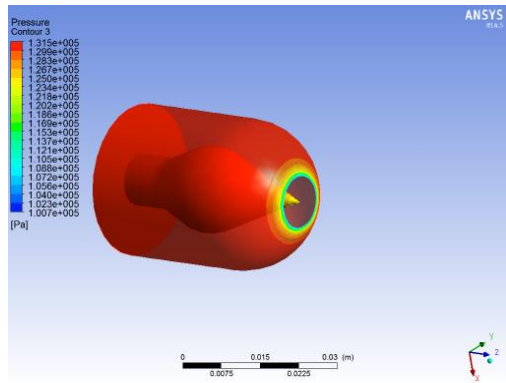

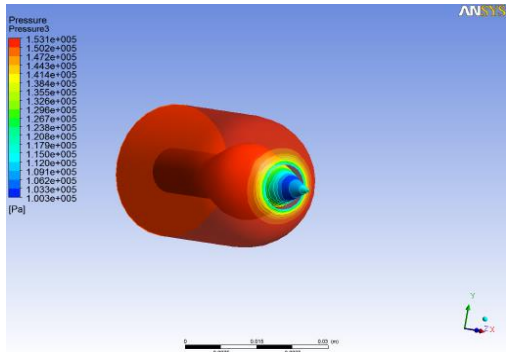Figure 11. Pressure distribution in geometry with 20 % nozzle opening

Figure 12. Pressure distribution in geometry with 40 % nozzle opening

Figure 13. Pressure distribution in geometry with 60 % nozzle opening
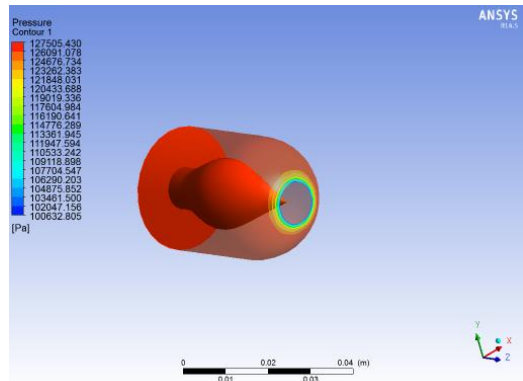

Figure 14. Pressure distribution in geometry with 80 % nozzle opening

Figure 15. Pressure distribution in geometry with 100 % nozzle opening
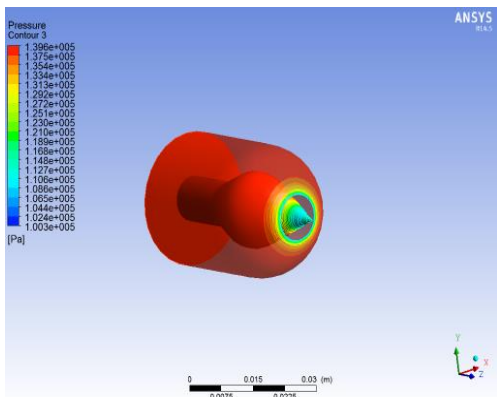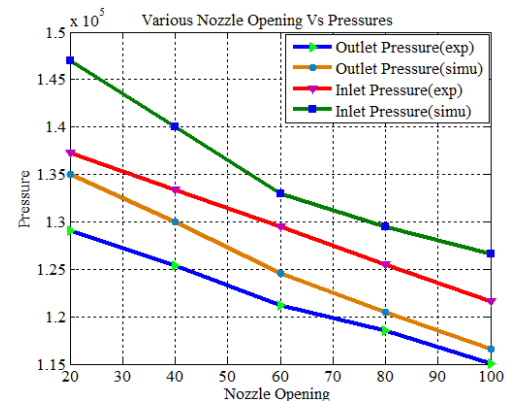

Figure 16. Relationship between nozzle Opening and pressures

7. SIMULATION RESULTS FOR NOZZLE

The flow simulation has been carried out for constant cross-sectional area of the jet and mass flow rate. The velocity obtained from numerical simulation for model turbine is close to experimental value. The streamlines in jet are parallel before striking the plate and spreads after strike of jet to the plate. As the outlet angle of the plate increases, the thickness of the water sheet decreases due to flow over the larger area of the plate before exit. Also this leads to reduction in water jet velocity at the outlet. The results are analyzed for water volume fraction iso-surfaces, pressure distribution within the flow domains. The distance between jet and bucket (20 mm) is also kept same in all cases for model turbine. The root mean square (RMS) residual is set to 10^{-7} for the termination of iterations. The solver was run for 200 iterations. The comparison between the theoretical and numerical results is given in Table 6.

Table 6. Comparison results between experimental with simulation of model turbine

Description	Experimental Results	Simulation Results	Units
Inlet Velocity	2.53	2.598	m/s
Outlet Velocity	15.84	16.793	m/s
Inlet Pressure	137293.1	142800	Pa
Outlet Pressure	125400	130900	Pa

8. CONCLUSION

In this research, the experiment of Model Pelton turbine has been carried out. Five position of nozzle are placed with the various speeds during the test. Relationship between speed and efficiency curve are obtained from the experiment results of model turbine. According to experimental results, the maximum efficiency occurs 40% nozzle opening. At 20% nozzle opening position, the efficiency decreases immediately because the area of nozzle is too small. At 60%, 80% and fully open position of nozzle, the efficiency decrease because of the cross-sectional area of nozzle is inversely proportional to the pressure. The best operating point is obtained at the maximum efficiency of turbine. In this point, the efficiency of model turbine is 86 %, speed ratio is 0.46 and the flow rate is 5.17×10^{-4} m^3 /sec at the speed of the turbine is 1380 rpm.

The variation of pressure and velocity distribution using pressure contours and velocity streamlines on the surface of the nozzle have been observed. According to numerical results, the pressure at the jet outlet is nearly equal to atmospheric pressure because the jet issued freely in air. The outlet velocity of the jet is higher than the inlet velocity of the fluid which shows that the pressure energy is being converted into kinetic energy. Finally, experimental results are verified by comparing with simulation results.

NOMENCLATURE

Q	-	Discharge of the Incoming Jet	[m^3/s]
N	-	Rotational Speed	[rpm]
T	-	Torque acting on the turbine Shaft	[N-m]
V_j	-	Velocity of Jet	[m/s]
H_{avai}	-	Available Head	[m]
p_{in}	-	Nozzle Inlet Pressure	[Pa]
p_{out}	-	Nozzle Outlet Pressure	[Pa]
P_{avai}	-	Available Power	[W]
P_o	-	Power Output of the Turbine	[W]
η_t	-	Efficiency of the Turbine	[%]

10. REFERENCES

[1] Anil Kumar, Numerical Simulation of Pelton Turbine Nozzle for Different Shapes of Spear, Maulana Azad National Institute of Technology, Bhopal, June 2011.

[2] Vishal Gupta, Vishnu Prasad, Numerical Modeling for Hydro Energy Convertor, Impulse Turbine, ICGSEE, 2013.

[3] Bansal R.K, A Text Book of Fluid Mechanics and Hydraulic Machines, 2009.

[4] Anonymous, Micro - Hydropower Systems. A Buyer's guide, 2005.

[5] Robertson, J.A. and Crowe, C.T, Engineering Fluid Mechanics, 5th edition, Houghton Mifflin, Boston, MA, 1993.

[6] White, F.M., Fluid Mechanics, 3rd edition, McGraw-Hill, Inc., New York, NY, 1994.

[7] Patel K, Patel B, Yadav V, and Foggia T.: Development of Pelton turbine using numerical simulation, 25th IAHR Symposium on Hydraulic Machinery and Systems IOP Publishing IOP Conf. Series: Earth and Environmental Science 12 ,2010.

[8] Alexandre Perrig, Hydrodynamics of the Free Surface Flow in Pelton Turbine Buckets. PhD thesis, University of Lausanne, 2007.

[9] Angehrn, R., Rettich, J., and Sch¨arer, C.: Pelton runner design based on measured unsteady pressure distributions in the bucket, Hydropower & Dams, 1999.

[10] Anil W. Date, Introduction to Computational Fluid Dynamics, Cambridge University Press, 2005.

[11] Kochevsky A. N., Nenya V. G. Contemporary Approach for Simulation and Computation of Fluid Flows in Centrifugal Hydro machines // Bulletin of Sumy State University, 2003.

[12] Chiyembekezo S. Kaunda, Cuthbert Z. Kimambo,A numerical investigation of flow profile and performance of a low cost Crossflow turbine, Volume 5, Issue 3, 2014.

[13] Xirouchakis Prof. P., president du jury Prof. F. Avelan: Hydrodynamics of The Free Surface Flow in Pelton Turbine Buckets, Lausanne, EPFL 2007.

[14] Abhishek Sharma, Prashant Sharna, Numerical Simulation for Pressure Distribution in Pelton Turbine Nozzle for the Different Shapes of Spear, IJIET,Vol.1, Issue 4, 2012.

[15] ANSYS. CFX Modeling / Theory Guide 14.5, (2010).

[16] Abhishek Sharma, Numerical Simulation and CFD Analysis for Energy Loss Computation in Fully Open Geometry of Pelton Turbine, 2013.