Numerical Analysis of Multi Jet Pelton Turbine Model

Vishal Gupta[1], Vishnu Prasad[2], Ruchi Khare[3]
[1]Dept of Mechanical Engineering, Radharaman Group of Institutes, Bhopal (M.P.) India.
[2]Dept of Civil Engg, MANIT, Bhopal (M.P.) India.
[3]Dept of Civil Engg, MANIT, Bhopal (M.P.) India.

ABSTRACT

Every turbine is designed for specific site condition (head, discharge and speed). For high heads and low discharge values, Pelton turbines are mostly used. Variation in head and discharge causes variation in performance of turbines. Earlier only experimental techniques were used to optimize the design of turbo-machine and check their performance at designed and off design conditions. With the advances in numerical techniques and computational power, Computational Fluid Dynamics (CFD) has proved to be boon in designing and predicting the performance of turbo-machine at all possible operating regimes. With the help of CFD, not only global parameters can be found out but local parameters which are not possible experimentally can be found out. In present work, flow analysis of a six jet Pelton turbine model has been done using multi phase flow. Numerical results are compared with available model test results at the best efficiency point. The effect of mesh size, turbulence model and time step is also studied.

Keywords— Multi jet, CFD, numerical techniques, Pelton turbine, multi phase flow

I INTRODUCTION

Earlier, for predicting the performance of turbo-machines model testing was the only way. In the last few decades, with the advances in computational power and numerical methods, Computational Fluid Dynamics (CFD) has emerged as a very powerful tool to optimize turbo-machine in design phase only. CFD has been extensively used for design optimization of turbo-machines involving one fluid flow since many decades. But now with advent of Volume of Fluid and 2-phase homogeneous model etc., CFD can also be used for determination of performance of Pelton turbine.

For simulating flow around a Pelton turbine runner, transient analysis needs to be performed by considering multiphase flow. Simulating flow around Pelton runner is complex and it requires a lot of computation time.

Initially, only injector design optimisation and stress calculation on Pelton runner was done and it was first carried out by Francois [1]. The most detailed Computational Fluid Dynamics (CFD) analysis of rotating Pelton turbine was done by Perrig et al. [2] by considering five buckets (one-quarter of the runner) and the computed results were compared with experimental results at best efficiency point (BEP). Zoppe et al. [3] performed flow analysis inside stationary Pelton turbine bucket using commercially available CFD code Fluent and validated the results experimentally. Gupta and Prasad [4] have presented effect of jet shape on water distribution in Pelton bucket. Parkinson et al. [5] have simulated unsteady analysis of Pelton runner. Gupta et. al.[4], Patel et al. [6], Dynampally and Rao [7] have worked on effect of time step and grid refinement. Flow in stationary flat plate was simulated by Konnur et. al. [8]. Xiao [9] and Zhang [10, 11] have studied or simulated effect of friction on Pelton buckets. Zhang[12, 13,14], Binaya et.al [15], Santolin [16] have worked for impact, flow dynamics and pressure distribution in Pelton bucket. Gupta et al.[17-23] have worked for different shapes of jets on stationary plate or Pelton bucket. They did detailed multiphase flow analysis in Pelton turbine and also found the importance of nozzle distance from bucket on the performance of Pelton turbine.

The main points of focus for analyzing a Pelton turbine are torque and efficiency and hence the value of torque and efficiency were found out at Best Efficiency Point (B.E.P.). The runner diameter to bucket width ratio is considered 3.4 in present analysis.

II GEOMETRIC MODELING

The existing Pelton model has six jets. The jet diameter and pitch circle diameter are 32.4 mm and 175.5 mm respectively. The runner of Pelton is symmetric and hence geometry of half of runner has been considered for simulation. The buckets are also symmetrical about the splitter and therefore half jet and bucket are modelled due to limitation of computational facility. The modelling has been done in Workbench. The modelled geometry of stator domain (jet from nozzles) is shown in Figure 1 and rotor domain (Pelton runner) geometry is shown in Figure 2.

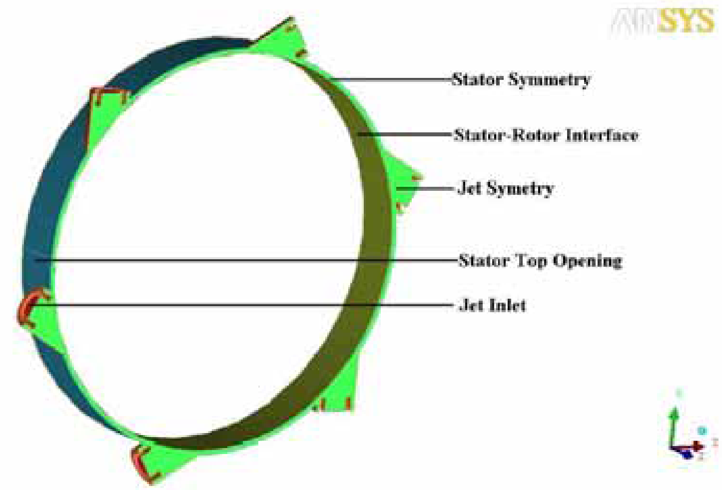

Fig No. 1 Geometry of Stator Domain

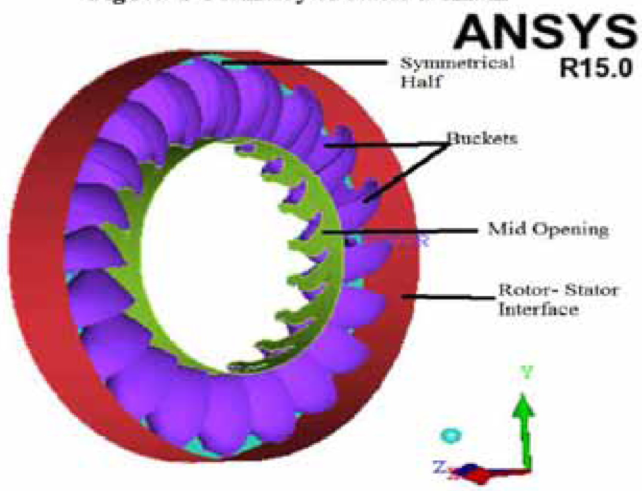

Fig No. 2 Geometry of Pelton Runner

III MESH GENERATION

The complete flow domain is discretised into tetrahedral and prismatic 3-D elements for simulation. The prismatic elements are used near to bucket surface for proper resolution of boundary layer. Meshing of both the domains has been done separately. They are connected through proper interface before defining boundary conditions. Meshing of rotor domain is shown in Figure 3.

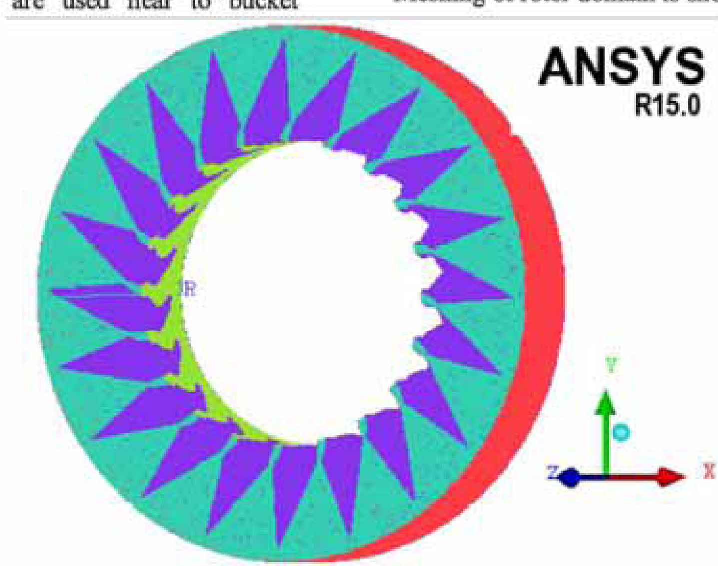

Fig No.3 Mesh of Runner

IV BOUNDARY CONDITIONS

Stator domain containing jet is kept stationary. Rotor domain has been set to speed at B.E.P. The analysis is carried out by taking SST turbulence model because of its ability to capture the flow with sharp curvatures and runner rotation.

Transient flow simulation is chosen with time step corresponding to 1° runner rotation for all the cases. Air and water are taken as working fluids with a reference pressure of 1 atmosphere. The jet inlet has been defined as inlet with water velocity corresponding to 50m head. Jet symmetry, stator symmetry and rotor symmetry have been defined as symmetry. The opening is specified all around stator and rotor. Transient rotor-stator interface is specified in between stator and rotor domains.

V GOVERNING EQUATIONS

Partial differential equations based on conservation of mass, momentum and energy are the fundamental governing equations of fluid dynamics. These are known as continuity, momentum and energy equations [20]. Flow having more than one working fluid is termed as multiphase flow. The water jet which leaves nozzle and hits the buckets of Pelton wheel is surrounded by air from all sides. When all working fluids have the same velocity, pressure, turbulence fields etc. except volume fractions, it is the limiting case of Eulerian- Eulerian multiphase flow and the flow may be termed as homogeneous multi phase flow. Free surface flow refers to the multi phase situation where two fluids are physically separated by distinct resolvable inter-phase [27]. The homogeneous multiphase flow is commonly used for free surface flows simulations.

Continuity

$$\frac{\partial \rho_m}{\partial t} + \vec{\nabla} \bullet \left(\rho_m \vec{W}_m\right) = 0 \tag{1}$$

where the mixture density and the mixture relative flow velocity are defined as

$$\rho_m = \sum_{n=1}^{2} \alpha_n \rho_n \tag{2}$$

and

$$\vec{W}_m = \frac{\sum_{n=1}^{2} \alpha_n \rho_n \vec{W}_n}{\rho_m} \tag{3}$$

the volume fraction ῳ is given by

$$\alpha_n = \frac{V_n}{\sum_{n=1}^{2} V_n} \tag{4}$$

Momentum:

$$\frac{\partial}{\partial t}\left(\rho_m \vec{W}_m\right) + \rho_m \left(\vec{W}_m \bullet \vec{\nabla}\right)\vec{W}_m = -\vec{\nabla} p_m + \vec{\nabla}\left(\bar{\bar{\tau}}_m + \bar{\bar{\tau}}_{t_m}\right) - \rho_m \vec{\omega} \times \left(\vec{\omega} \times \vec{r}_m\right) - 2\vec{\omega} \times \vec{W}_m + \vec{f}_m \tag{5}$$

VI FORMULAE USED

Water power available at nozzle outlet

$$P_I = \rho \times g \times Q \times H \tag{6}$$

Numerically calculated power is given by

$$P_O = \frac{2 \times \pi \times n \times T}{60} \tag{7}$$

Efficiency

$$\eta_n = \frac{P_O \times 100\%}{P_I} \tag{8}$$

Time step corresponding to 1° runner rotation:

$$\Delta t = \frac{60}{360 \times n} \tag{9}$$

Torque Coefficient

$$K_T = \frac{T}{\rho \times C_1^2 \times B \times L \times D} \tag{10}$$

Normalised Efficiency

$$\partial = \frac{Efficiency}{Efficiency\ at\ B.E.P.} \tag{11}$$

VII GRID INDEPENDENCY STUDY

The mesh size affects the results obtained from simulation and hence grid independency test was done for both stator and rotor.. It was found that 547940 nodes and 2589738 elements were required for stator. For studying the effect of mesh size of rotor, three cases were considered.

Table No. 1
Effect of Mesh on Efficiency

S.No	Mesh	No. of Nodes in Rotor	Efficiency	Time Duration
1.	Coarse	46936	68.02%	19 hours
2.	Fine	1141518	77.23%	52 hours
3.	Finer	1480537	77.76%	62 hours

It is found that for increase in mesh size from 1141518 to 1480537 elements, computational time increases by 10 hours and the deviation in efficiency is only 0.53% so fine mesh is chosen for further study.

(a) Study on Turbulence Model - Two equation turbulence models are mostly used for modeling turbulence. (e M with scalable wall function and SST model with automatic wall function have been considered.

Table No. 2
Effect of Turbulence Model

S.No	Turbulence model	Efficiency
1.	(- M with scalable wall functic	77.23%
2.	SST with automatic wall function	77.93%

Table 2 shows that SST model gives 0.7% higher efficiency at B.E.P. The reason for this can be that SST model accounts for the transport of shear stress accurately than (e M model. -(M model over predicts the production of turbulent kinetic energy near stagnation points and in adverse pressure gradient flows leading to lower efficiency. For further study, SST turbulent model has been used.

(b) Study on Advection Scheme - Numerical errors for a given mesh size are directly proportional to the order of discretisation scheme. The above results were obtained for first order upwind scheme. More simulations were carried for High resolution and Blend factor of 1.

Table No. 3
Effect of Turbulence Model

S.No.	Turbulence model	Efficiency
1.	Upwind	77.93
2.	High Resolution	78.94
3.	Blend Factor 1	80.60

(c) Time Step Study-For any transient simulation, time step plays a vital role. Smaller is the time step, better are the results obtained but at the cost of high computational time. In present case, three time steps corresponding to 0.5°, 1° and 2° runner rotation have been considered. Table 4 shows the obtained results.

Table No. 4
Effect of Time Step

S. No.	Time Step	Normalised efficiency	Time Taken
1.	0.5°	0.91	118 hours
2.	1°	0.88	60 hours
3.	2°	0.80	31 hours

Due to limitation of time and computational power, time step lesser than 0.5° could not be used. At larger time steps, the deviation in efficiency is high. This may be due to the fact that at larger time steps, time scales may not be properly resolved.

VIII DISCUSSIONS

(a) Variation in Torque with Respect to Degree of Rotation of Runner Initially, water starts to get out of nozzle but it does not hit the runner. So the value of torque observed is zero. As water starts hitting the buckets, torque is found out to have some value and then start increasing till the buckets successively comes in front of jet and have almost uniform value of torque as shown in Figure 4.

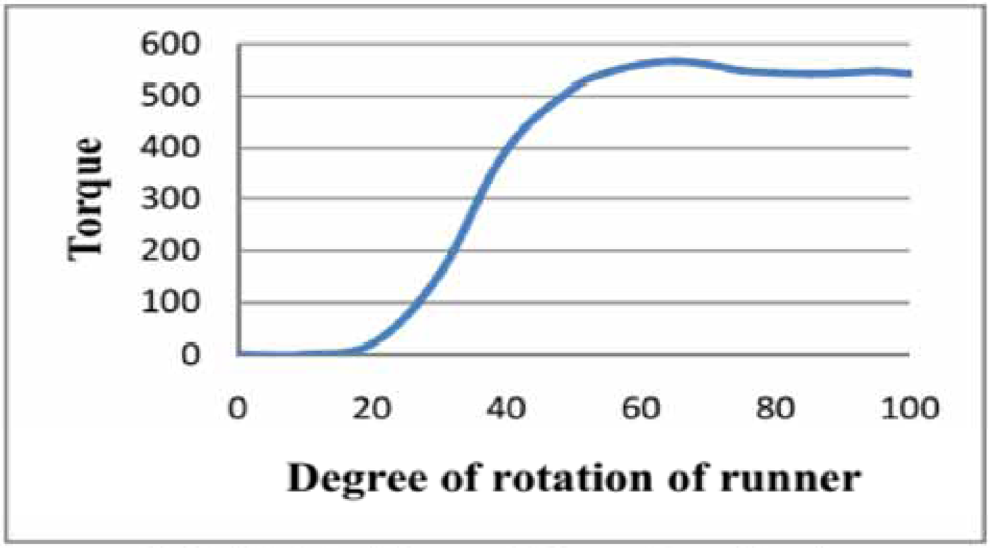

Fig No. 4 Variation in Torque with Degree of Runner Rotation

Slightly higher value of torque is observed when the water jet starts hitting the runner. During later stages, torque becomes uniform.

(b) Flow Visualisation - Figure 5 shows flow within the region of interest at BEP. It can be observed that jet strikes almost normal to all the buckets. The preceding bucket does not obstruct the water and no water is left without striking the buckets. . So the design of runner is appropriate and any modification is not required.

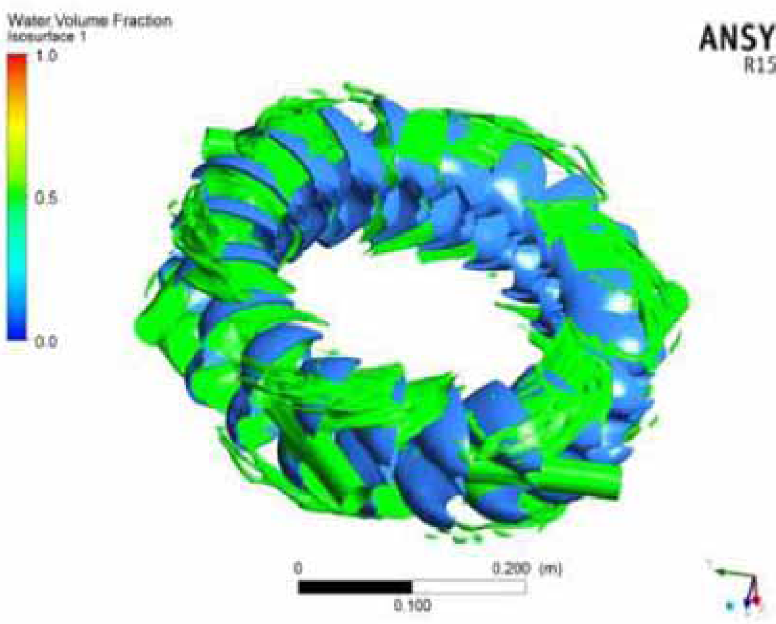

Fig No. 5 Flow Visualisation In Pelton Runner

(c) Effect of Variation in Speed - The variation in torque with runner rotation at different speed is shown in Figure 6. Higher values of torque at runner are observed at lower speed and at higher speed; runner experiences lower value of torque for given flow rate which resembles the characteristics of Pelton wheel. At lower speeds, uniform value of torque is observed whereas as the speed of runner increases, more fluctuations in value of torque are there. The torque is maximum when bucket is normal to the jet and decreases when bucket either reaching to or moving away from normal to jet position.

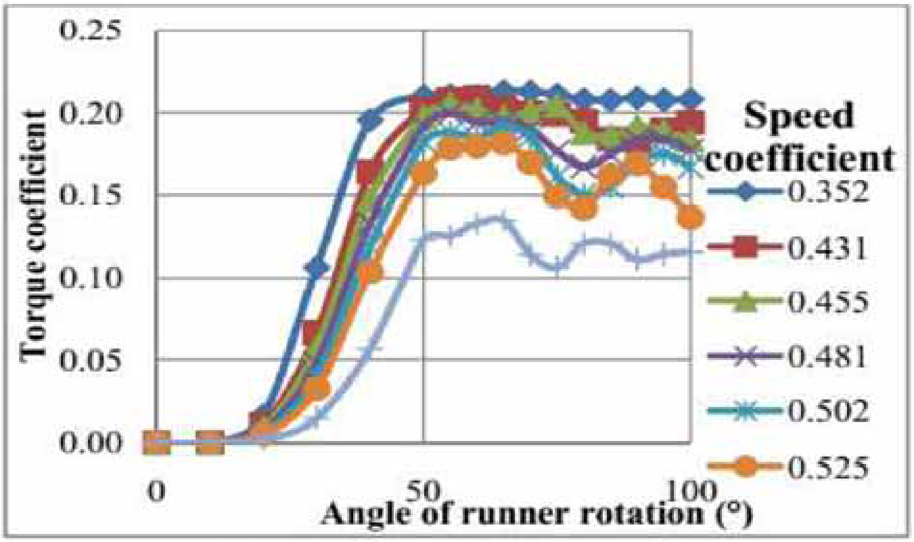

Fig No. 6 Variation in Torque Coefficient with Angle of Runner Rotation

IX CONCLUSION

Numerical simulation for predicting the efficiency, flow visualisation has been done for variation in grid size, turbulence model, advection scheme at BEP. The results are compared with available model test results. The results obtained from CFD are very useful to understand flow behaviour inside the Pelton runner. Flow can be easily visualised using volume fraction in CFD which is very difficult in experiment. The torque experienced by runner is found to be increasing with decrease in rotational speed. The fluctuations are minimum at lower speed and increase with speed.

g- acceleration due to gravity (9.8 m/s2) H- head (m) N- rotational speed of rotor (rpm) PO- numerical Power Output (Watt) PI- power input (Watt) Q- volume flow rate of fluid at jet inlet (m3/s)	T- torque on runner (N-m) (k density of water at 20°C (997 kg/m3) (bE experimental efficiency (%) (bH hydraulic efficiency (%) P speed coefficient

REFERENCES

[1] Francois M, Vuillerod G. Development and Recent Projects for Hooped Pelton Turbine. Proceedings of Hydro-2002. Turkey. 4-7 Nov 2002

[2] Perrig A, Avellan F, Kueny JL and Farhat M. Flow in a Pelton Turbine Bucket: Numerical and Experimental Investigations. Transaction of ASME. 2006. 128: 350- 358.

[3] Zoppe B, Pellone C, Maitre T and Leroy P. Flow Analysis Inside a Pelton Turbine Bucket. Transaction of ASME. 2006. 128: 500-511.

[4] Gupta V and Prasad V. Numerical Investigations for Jet Flow Characteristics on Pelton Turbine Bucket. International Journal of Emerging Technology and Advanced Engineering 2012, 2(7): 364-370.

[5] Parkinson E, Neury C, Garcin H, Vullioud G and Weiss T. Unsteady Analysis of Pelton Runner with flow and Mechanical simulations. Hydropower & Dams. 2006.

[6] Patel T, Patel B, Yadav M and Foggia T. Development of Pelton turbine using numerical simulation. 25th IAHR Symposium on Hydraulic Machinery and Systems. IOP Conf. Series: Earth and Environmental Science 12. 2010. 0.00 0.05 0.10 0.15 0.20 0.25 0 50 100

[7] Dynampally P and Rao VS. CFD Analysis of Pelton Turbine, Proceedings of Thirty Ninth National Conference on Fluid Mechanics and Fluid Power 2012. pp 58.

[8] Konnur MS and Patel K. Numerical Analysis of Water Jet on Flat Plate. National Conference on Fluid Mechanics and Fluid Power. 2006, Raipur, India.

[9] Xiao Y X, Zeng C J, Zhang J, Yan Z G and Wang Z W. Numerical Analysis of the Bucket Surface Roughness Effects in Pelton Turbine. 6th International Conference on Pumps and Fans with Compressors and Wind Turbines. IOP Conf. Series: Materials Science and Engineering 52 (2013) 052032

[10] Zhang Zh. Flow Friction Theorem of Pelton Turbine Hydraulics. Journal of Power and Energy 2007. 221(Part A): 1173-1180.

[11] Zhang Zh. Analytical Method for Frictional Flows in a Pelton Turbine, Journal of Power and Energy 2009.223(Part A): 597-608.

[12] Zhang Zh. Inlet Flow Conditions and Jet Impact Work in a Pelton Turbine. Journal of Power and Energy 2009. 223(Part A): 589-596.

[13] Zhang Zh. Flow Dynamics of Free Surface Flow in the Rotating Buckets of a Pelton Turbine. Journal of Power and Energy 2009. 223 (Part A):609-623.

[14] Zhang Zh. Flow Interactions in a Pelton Turbines and the Hydraulic Efficiency of the Turbine System. Journal of Power and Energy 2007, 221 (Part A): 343-357.

[15] Binaya KC and Thapa Bhola. Pressure Distribution at Inner Surface of Selected Pelton Bucket for Midro Hydro. Journal of Science, Engineering and Technology 2009. 5(2): 42-50.

[16] Santolin A, Cavazzini G, Ardizzon G and Pavesi G. Numerical Investigation of the interaction between jet and bucket in a Pelton turbine. Proceedings of the Institution of Mechanical Engineers. Journal of Power and Energy 2006. 223:721-728

[17] Gupta V, Prasad V. and Rangnekar S. Performance analysis of nozzles used in impulse hydraulic turbine using CFD. Proceeding of National Conference on Fluid Mechanics and Fluid Power. College of Engineering, Pune. December 17-19, 2009

[18] Gupta V, Prasad V, Khare R. Effect of jet shape on flow and torque characteristics of Pelton turbine runner. International Journal of Engineering Research and Applications, 2014:4(1):318-23.

[19] Gupta V., Prasad V. Numerical computation of force for different shapes of jet using CFD. *Proceedings of the 38th National Conference on Fluid Mechanics and Fluid Power. December 15-17. 2011. MANIT, Bhopal*

[20] Gupta V, Prasad V. Numerical investigations for jet flow characteristics on pelton turbine bucket. Int. J. Emerg. Technol. Adv. Eng. 2012 Jul;2(7):364-70.

[21] Gupta V, Prasad V, Khare R. Numerical simulation of six jet Pelton turbine model. Energy. 2016 Jun 1;104:24-32.

[22] Gupta V, Prasad V, Khare R. Effect of jet length on the performance of pelton turbine: distance between nozzle exit and runner. ARPN Journal of Engineering and Applied Sciences. 2016; 11 (19): 11487-11494

[23] Gupta V, Khare R, Prasad V. Performance evaluation of Pelton turbine: A review. Hydro Nepal: Journal of Water, Energy and Environment. 2014 Mar 13;13:28-35.

[24] ANSYS CFX 14 Software Manual.