

Performance Evaluation of Pelton Turbine: A Review

Vishal Gupta, Dr. Ruchi Khare and Dr. Vishnu Prasad



Vishal Gupta

Dr. Ruchi Khare

Dr. Vishnu Prasad

Abstract: Earlier only experimental techniques were used to predict the performance of turbines. With advanced numerical techniques and increase in processing power of computers, Computational Fluid Dynamics (CFD) has emerged as an effective tool for the performance prediction of Pelton hydraulic turbine involving multi-fluid flow. Extensive work has been done for design optimization of reaction turbines using CFD. Now it is being extended for impulse turbines. The flow in reaction turbines involves only water as working medium, but in case of impulse turbines, water and air are working medium. The water jet issued from nozzle is surrounded by air and pressure around the jet and turbine is atmospheric. The performance of Pelton turbine depends upon the shape, size and quality of jet as well as shape of the buckets. In the present paper, the literature review on applications of CFD for performance prediction, design optimization of Pelton turbine have been discussed.

Key words: Computational fluid dynamics, Pelton turbine, free surface flow, multi-phase fluid flow

Introduction

In hilly areas, high head hydro power plants are common and the turbine used for such plants is Pelton turbine. In Pelton turbine, all the available energy of water is converted into kinetic energy or velocity head by passing it through a contracting nozzle provided at the end of penstock. The water jet moves freely in air and impinges on a series of buckets of the runner thus causing it to rotate. Thus, the performance of the turbine depends upon many factors like shape of jet striking the turbine bucket, shape of the bucket and exit profile of the water sheet from bucket.

Earlier model testing was the only method available for assessing performance of Pelton turbine for different nozzle and bucket shapes. But this approach was time consuming, costly and did not provide detailed flow behavior. With the advancements in computational facility and numerical techniques, the design optimization of Pelton turbine can be made in less time. The flow in Pelton turbine is a multiphase free surface flow involving water and air as flow medium.

The geometry of stator and rotor is discretized into small elements known as mesh. The basic equations are the partial differential equations given by Navier-Stokes based on conservation of mass, momentum and energy which are solved iteratively over the region of interest (at nodes of the mesh or volumetric centre of mesh). Today various commercial CFD codes are available in market and the basic theory behind all of them is to solve governing equations over the region of interest.

The literature review for flow analysis of Pelton turbine is divided into theoretical studies, experimental studies, numerical investigations and validations of results from numerical investigations with the experimental studies. The papers regarding jet shape and Pelton bucket are discussed separately in this paper.

Theoretical Studies

Zhang [2007] in his paper on flow friction theorem of Pelton turbine investigated the flow interaction between

the jet and rotating buckets and the relative flow in buckets of Pelton turbine. For evaluation of existing layout and designing new Pelton turbine, detailed calculation of cut-off processes of the jet by rotating buckets has been given. The effects of centrifugal and Coriolis (Coriolis effect is a deflection of moving objects when they are viewed in a rotating reference frame) forces in the rotating bucket have been discussed in detail. Invariance equation to describe the relative flow of water in rotating bucket using energy law has been derived. The derived equation has also been applied to all parallel jet layers. He [2007] further worked on Pelton turbine and concluded that the efficiency of Pelton turbine system was greatly influenced by flow friction in the rotating buckets. Friction leads to direct influence. The indirect influence is associated with relative flow and pressure distribution in the bucket. The losses are dominated by change in relative flow and pressure distribution. Zhang [2007] worked on inlet flow conditions and jet impact work on Pelton turbine and found out that the volume flow rate in rotating frame of reference is independent of the bucket rotational speed. He explained the percentage of total jet impact energy using calculations of the impact forces and corresponding impact work. Incorrect design of the bucket geometry could cause second part of jet to hit rear side of the bucket and may cause counteracting torque and cavitations. Since, the Pelton turbines are designed for lower specific speeds, erosion would occur at high head jet flows with rich sand particles. He [2009] further analyzed the effect of centrifugal, frictional and inertia forces on exchange of power on jet and rotating buckets in detail. Flow friction has been found out to be dominating for the hydraulic efficiency. According to him, the best available alternative to CFD can be development of calculation method in using the spreadsheet which would lead to direct simplified calculations of any complex flows. Using results from previous work, Zhang [2009] calculated the relative flow in rotating buckets of Pelton turbine with respect to the influence of centrifugal,

Cavities and impact forces. By assuming the flow to be frictionless, invariance equation has been presented which helps in calculating changeable flow velocity in rotating bucket. He concluded that the centrifugal force contribute negligible work compared to effectiveness of Cavities and impact forces. The invariance equation was applied to few examples also.

The effect of bucket friction using boundary layer theory has been analysed by Atthanayake [2010]. He concluded that the formation of boundary layer and its thickness depends on surface roughness of bucket surface. The power loss occurred due to friction in Pelton bucket and change of pressure along the flow path.

To utilize the energy of fluid coming out of Pelton wheel, Yadav [2011] modified the turbine design by adding auxiliary attachments on either side of the wheel. The water coming out of the central wheel strikes on the side buckets, making auxiliary wheel to rotate and thus increasing output power of the turbine.

Experimental Studies

Kotousov [2005] experimented for 5 shape of nozzles and found out the most efficient shape of the nozzle. He observed that negative pressure zone is created near the outlet section of nozzle causing cavitations in this zone. The velocity of the jet increased due to change in density of the jet and with variation in temperature. The reduction in absolute static pressure below atmospheric in the region of dynamic regime caused gas evolution. The mean density of water decreased by 10-20% of homogeneous water as dissolved gases mainly carbon dioxide and chlorine evolved in it. As the discharge was kept constant, the effect of decrease in density will lead to increase in the jet velocity. Swelling of flow due to gas evolution and cavitations was observed.

Zhang and Casey [2007] worked on precise shape of the jet discharged from nozzle of Pelton turbine. After experimental studies, it is found that bifurcations of the distributor affect flow development in nozzle and the interaction of the jet with surrounding air causes a turbulent free surface.

Staubli and Abgottspon [2008] have discussed the effect of location of injectors on efficiency of turbine. The authors discussed the difference between the upper and lower injector efficiencies for one and two nozzle operations with the help of results obtained at three different sites. They found that for both the cases (one and two nozzle operation), upper injector showed lower efficiencies as it had more curved bend. For one nozzle operation difference in efficiency was from 0.5% to 1.5%.

Kubota T [2010] studied jet nozzle interference for its effect on efficiency of turbine. To increase the specific speed of Pelton turbine, number of nozzles and bucket width per unit runner diameter needs to be increased. It was concluded that efficiency has decreased for high specific speed of presented 6-nozzle Pelton turbine.

Two nozzle Pelton model was designed and constructed by Stamatelos et al [2010] using standard

guidelines. The turbine could work with one injector or both the injectors. The maximum efficiency obtained was 86%. Overall efficiency and shaft power were computed using net water head, flow rate, injector characteristic curves, shaft power, rotational speed. Hill chart was also prepared for various operation modes.

Bajracharya et al [2008] studied the problem of erosion of nozzle and buckets of Pelton turbine in Himalayan Region. They have taken a case study of Chilime Hydropower Plant (Capacity 22 MW) in Nepal and performed detailed studies on erosion rate and efficiency reduction. The estimated efficiency reduction was 1.21% and it consequently resulted in loss of power generation. Quartz and Feldspar were the major mineral contents in the river water. Severe erosion was observed at partial opening condition due to additional effect of cavitations along with it. Diversion tunnels and regular de-silting were proposed.

Similar type of work was done by Padhy and Saini [2009]. They have investigated the effect of size, hardness and concentration of silt particles, velocity of flow, properties of the base material of turbine components and operating hours of turbine on phenomenon of erosion in actual conditions. Experimental studies have been carried on small scale Pelton turbine (Fig 1). Based on experimental data, correlations between wear rate of Pelton turbine bucket as function of size and concentration of silt particles and jet velocity has been developed.

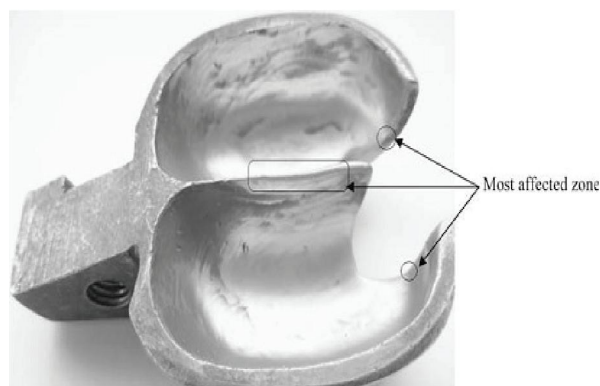


Figure 1. Surface condition of Pelton bucket after experimentation showing erosion [Padhy and Saini].

A lab experiment was conducted by Agar and Rasi [2008] to demonstrate the physics involved in hydro power plant. They constructed a laboratory scale Pelton turbine for hydroelectric generation using inexpensive components. It was observed that maximum efficiency of the turbine occurred at higher rotational speeds and after attaining peak point, the drop in efficiency could be observed. The presented turbine was found to have maximum mechanical efficiency of $47\% \pm 2\%$ for water flow rate of 0.17 l/sec.

Thermodynamic energy distribution from the sample of 25 Pelton turbines were measured by Hullas et al [2008] and found that Pelton turbine with horizontal shafts had more uniform energy distribution than that of vertical shaft turbine.

Numerical Investigations

For uniform distribution of stresses, hoops mounted on the buckets of Pelton runner were introduced by Francois et al [2002]. Stress analysis using finite element technique and CFD analysis with varying jet incidence angles have shown that risk of cracking and cost of maintenance could be reduced by implementing hoops around the runner. Stress deformation has been shown in fig 2.

His work was further extended by Chaudhari et al [2010] who have presented the stress analysis for traditional Pelton runners and hooped Pelton runners using I-DEAS software. The designed hooped runner has been compared

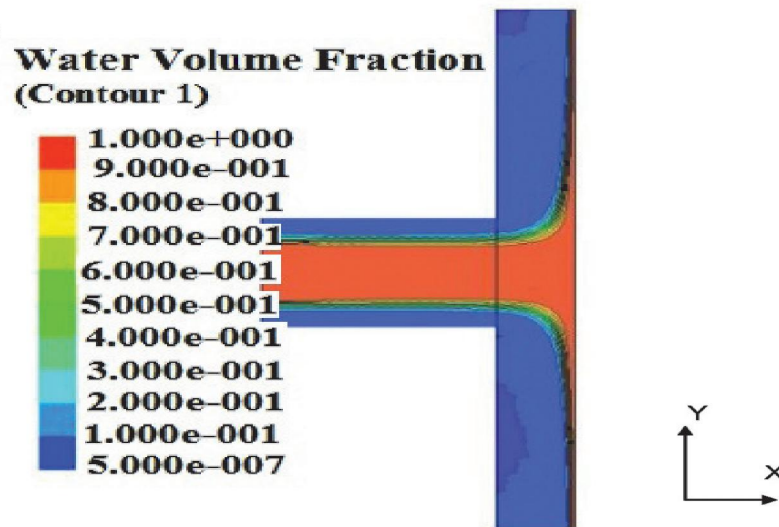


Figure 3. Water volume fraction for perpendicular jet [Konnur and Patel].

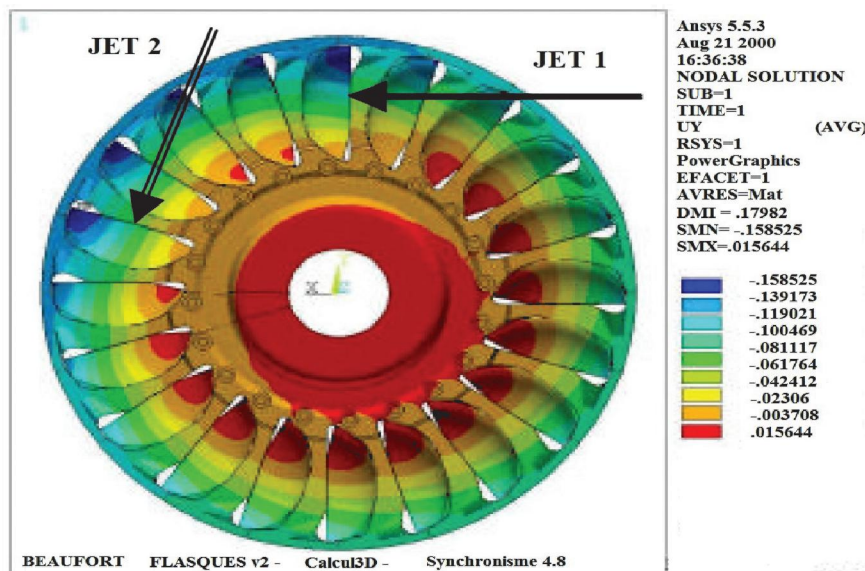


Figure 2. Tangential Displacement from FEA on 3D model [Francois et al].

to traditional runner on the basis of performance parameter. The Pelton model used had 18 runner buckets with one jet and rotating at 1000 rpm. It was concluded that the use of hoops allow stress to be minimized and more uniformly distributed.

The procedure to generate the 3D solid geometry has been defined and finite element method (FEM) has been used by Nedelcu et al [2008] to calculate deformation and stress values for Pelton turbine blade. Geometry was made using Autodesk Inventor software and linear static analysis was done on single blade with Cosmos Design Star software. The geometry was discretized into tetrahedral solid elements. The maximum values of stress were found in the area of the taper shank hole where the buckets are bolted to runner (fig 3).

Sadlo et al [2004] used CFX-TASC flow solver for simulating the scaled model components of Pelton

turbine. They considered steady flow for manifolds and injectors and time dependent two phase flow was considered for simulating jet and flow in buckets. The authors presented field line placement algorithm for investigation of vertical flow. Two visualization tools were also developed for the explorative analysis of velocity (vorticity) fields. It was concluded that stream tubes gave better visualization of the results than stream lines.

Sick et al [2009] have analyzed three different phases of interaction process of jet and bucket: jet loading, maximum torque and water

evacuation and found that accuracy of torque prediction is not yet satisfying. For getting the torque accurately, more refined modeling of two phase flow, detachment and cavitations zones was required. Water sheets have been used to visualize the flow and its impact on the bucket.

The main problems (calculations for turbulent flow jets, bucket flow, two phase mixtures, amount of time to carry out calculations) concerning a flow in the bucket of Pelton turbine have been discussed by Matthias and Promper [2004]. They simulated the impact of the jet on the disk at different angles using FLUENT. The results were compared experimentally as well as analytically. It was also concluded that the simulation of free jets in rotating system is possible.

Numerical simulation of the losses in the inlet piping system, casing and quality of jet was done by Marongiu

et al [2005]. They developed a Smoothed Particle Hydrodynamics [SPH] tool (Code Name: NEMO) and discretized the water jet to calculate the impact of jet on flat plate and validated the results. After that they used the code to simulate the jet on Pelton turbine bucket. Marongui et al [2007] used the above developed tool: meshless smoothed particle hydrodynamics (SPH) method which uses Lagrangian approach, for numerically finding the jet velocity. They also found that the results obtained were in agreement with experimental and with CFX (volume of fluid) results. For a Pelton turbine bucket, it was observed that the maximum load was at the area where the curvature is maximal. Marongui et al [2010] further modified SPH method and introduced Arbitrary Lagrange Euler (ALE) description of fluid flow with meshless numerical method Smoothed Particle Hydrodynamics (SPH) to simulate free surface flows. The hybrid method using ALE and SPH was adopted to overcome the drawbacks of SPH. The results of SPH-ALE have been compared with CFX results and the results are in good agreement.

ANSYS-CFX has been used by Konnur and Patel [2006] for numerically investigating water jet on flat plate at three different positions of jet i.e. at 0° , 30° and 60° incidence angles. Steady state numerical analysis has been performed. For wall pressure distribution and flow visualization, two phase homogeneous model for free surface flow with k-epsilon turbulence model were performed. Theoretical and numerical comparison of force on flat plate has been covered. The results were found to be quiet encouraging. Fig 4 shows the water volume fraction for one of the case i.e., with perpendicular jet.

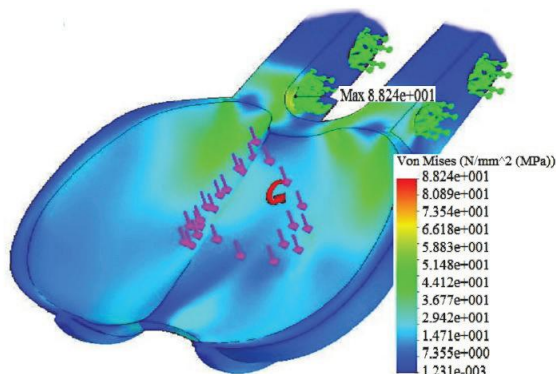


Figure 4. The Von Mises Plot on Pelton Bucket [Nedelcu et al].

κ - ω SST turbulent model have been used by Jost et al [2010] to find out energy losses in the distributor and the shape and velocity of jets for several needle strokes. Unsteady flow analysis of runners with jets was also done. Numerical results found to be deviated from the

experimental results by 4%.

High speed turbulent water jets in air have been simulated by Ghua et al [2010]. The results have been compared with existing experimental results. Mesh generation has been done in GAMBIT software and simulation has been done using FLUENT code. The Eulerian multi phase and standard k- ϵ turbulence model with standard wall functions have been used for the simulation. It was concluded that the simulation under predicts the velocity and over-predicts the volume fraction distribution near the jet edge.

Gupta and Prasad [2012] have compared flow characteristics of circular and rectangular jets for two jet velocities keeping other parameters (jet area, jet distance from bucket) constant on static Pelton bucket using multi-phase flow simulation. ANSYS ICEM CFD-13.0 has been used for geometric modelling and the flow simulation has been carried out using ANSYS CFX. The results were compared with theoretical values. It has been found that the pressure distribution for circular jet as compared to rectangular jet over bucket surface is more uniform.

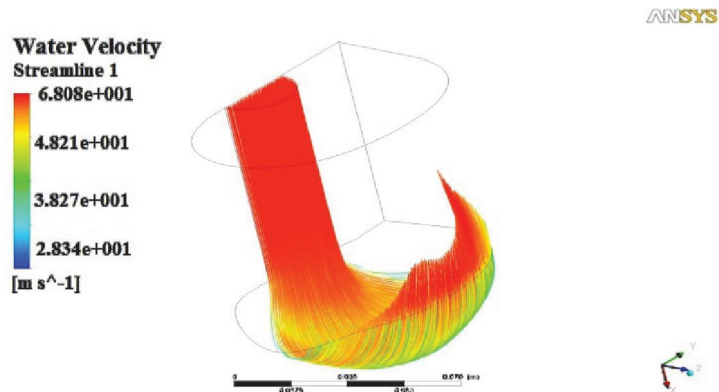


Figure 5. Velocity streamlines for semi-circular jet and jet velocity 68 m/s [Gupta and Prasad].

Further, Gupta et al [2013] have extended the work for simulating water jet on curved plates with different flow angles at outlet. The authors have done steady state analysis based on two phase homogeneous model with standard k- ϵ turbulence model using ANSYS-CFX. The theoretical and numerical variations in pressure, velocity and jet force with plate profile have been studied. It was found that with the increase in outlet angle, the thickness of water sheet decreased due to the fact that water had to travel larger area. Maximum pressure was also observed at the centre of the plate and was same for all the cases (Fig 6).

The dynamic performance of the Pelton turbine using method of animated cartoon frames has been predicted by Ye-xiang et al [2007]. The jet was discretized into multi layers. It was found that the efficiency of jet decreased slightly at low unit speeds due to small expansion of jet towards downstream whereas at higher unit speeds, the rate of decrease in efficiency is very fast.

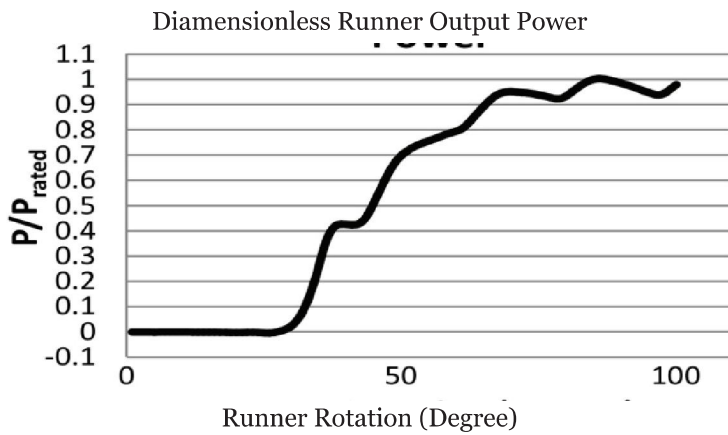


Figure 6. Power variation with runner rotation [Desai et al].

Multiphase analysis (containing water and air) of twin jet Pelton turbine using CFD at the best efficiency point (BEP) and 10% overload point has been done by Desai et al [2012] and found out that the results were quiet satisfactory. Pro-E software has been used for geometric modeling. Tetra mesh with 5491k nodes has been used for meshing. To capture the boundary layer effect, prism layer has been applied at wall boundaries. CFD analysis for 90° runner rotation in step of 1° has been performed. In this case, the water leaving out of the first bucket touched the outer surface of next bucket and imparts pressure on it. So the outer angle of bucket was changed. It has been concluded that water and air volume fractions in the runner bucket could be approximated by using CFD. Fig 7 shows the power variation with runner rotation.

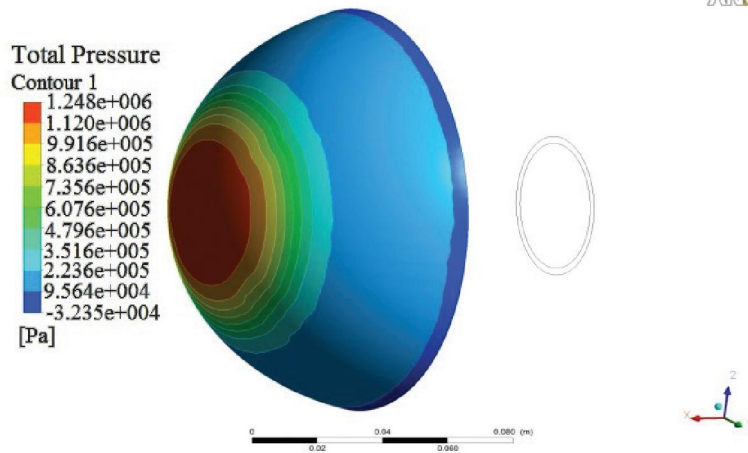


Figure 7. Pressure contour for 165° curved plate [Gupta et al].

Validation of Numerical Results

Experimental and numerical simulation was done Catanase et al [2004] using MATLAB to investigate the velocity distribution profile of the jet for different needle tip. It was concluded that the effect of head is more than that of needle stroke but kinetic energy losses depend on

needle stroke only.

Peron et al [2008] discussed the importance of water jet on the performance of Pelton turbine with the help of two rehabilitation projects: Bordogna HPP in Italy and FIONNAY HPP in Switzerland. They have used CFD as well as prototype model for checking the nozzle and distributor geometry influence on circumferential shape, velocity distribution and dispersion of jet.

The nozzles with straight inlet as well as the configuration with a 90° elbow have been analyzed by Fiereder et al [2010]. The authors have shown the effect (slight deformation of jet) of secondary flow due to 90° bend in upstream pipe using in-house finite volume 3D solver.

Perrig et al [2006] presented the results of free surface flow of Pelton Turbine model bucket using unsteady numerical simulation. They experimented and numerically investigated 5 adjacent rotating Pelton turbine bucket. Numerical simulation of flow carried out using CFX-5 code and lead to the conclusion that the outer region contributed the most to bucket power. These areas interact early with the jet and thus receive water particles with the highest momentum.

The experimental and numerical work was also done by Zoppe et al [2006] on stationary Pelton bucket but using different software- Fluent. The authors carried out detailed experimental and numerical analysis of the flow in a fixed bucket of a Pelton turbine with FLUENT code using the two phase flow volume of fluid method. To cover the turbine functioning points, the head, jet incidence and flow rate have been varied. After experimenting and numerical simulation it was concluded that for a fixed angle of incidence, with the increase in jet diameter, wetted surface increased. On increasing angle of incidence and jet diameter, a leakage flow through cut-out rapidly increases. The highest pressure part is the deepest part inside the bucket.

Detailed flow and pressure field were analysed by Parkinson et al [2006] which helped in optimisation of bucket hydraulic profile. Finite element method approach was used to perform mechanical analysis of the bucket response to unsteady hydraulic loading. Validation of the results for unsteady pressure on both inner and outer surfaces of the buckets was also presented.

Anagnostopoulos and Papanonis [2006] experimentally and numerically studied Pelton model turbine to develop computational tool for design

improvement of impulse type runners. The numerical methodology was based on the Lagrangian approach. Tracking of large number of representative fluid particles could give the idea of flow in turbine bucket. For bucket surface parameterization, modern regression techniques were implemented.

The interaction between the jet and bucket of a Pelton turbine has been numerically investigated by Santolin et al [2009]. Unsteady numerical flow analyses were carried out on a single jet Pelton turbine taking two cases using CFX-11 code. The influence of the shape of water jet on the turbine losses was computed numerically and compared with the experimental results which led to 1% less in hydraulic efficiency.

Beucher et al [2010] presented friction losses in 24 bucket aluminum Pelton wheel. It was experimentally found out that wheel friction losses are the important part of total losses in turbine when the turbine is shrouded and operated in high density atmosphere. CFD model of Pelton wheel has been analyzed to develop efficient turbine. It was found out that lowering the number and size of buckets, friction losses can be decreased by 30% of the losses.

The design of Pelton turbine bucket and inlet manifold were successfully optimized by Patel et al [2010]. They have done multiphase transient analysis to understand the flow physics and predict the performance of Pelton turbine at full load and part load. CFD and test results have been compared (Fig 8). They concluded that flow distribution inside bucket is very difficult to observe experimentally but

The work on validation of numerical results was carried out by Dynampally and Rao [2012]. The authors did CFD analysis of 6 jet vertical Pelton turbine using ANSYS CFX 13.0 software. It has been observed that the results obtained from CFD were sensitive to grid refinement, time step and turbulence model taken. To

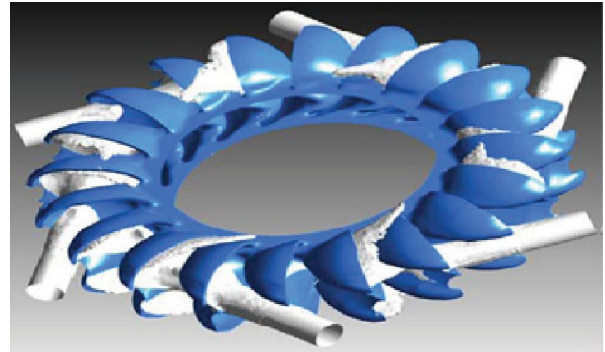


Figure 9. Flow Visualization in Pelton Turbine Runner [Dynampally and Rao].

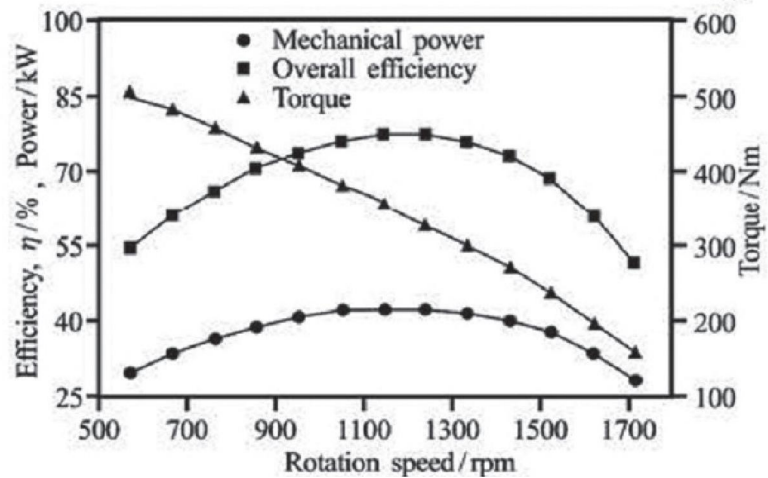


Figure 10. Effect of Rotational Speed on mechanical and overall efficiency and torque [Anagnostopoulos and Papantonis].

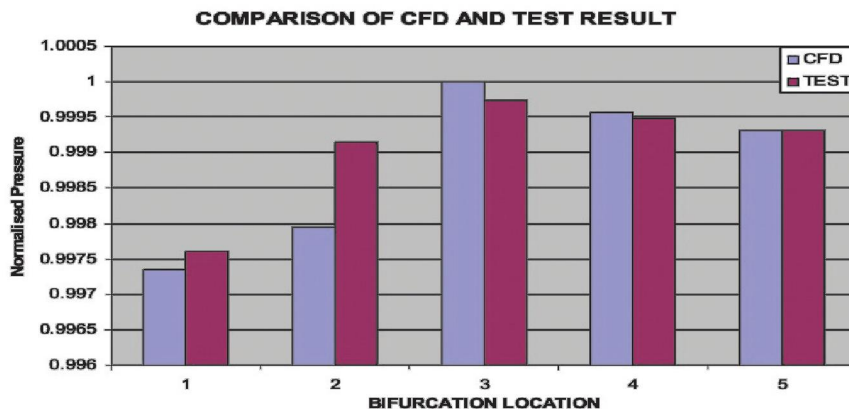


Figure 8. Comparison model test & CFD result for bifurcation pressure measurement [Patel et al.]

numerically interference of jet can be easily understood. Performance of the machine improved after optimizing bucket design

resolve the free surface fully transient CFD simulation with multiphase model has been taken. To compare the various parameters, torque and efficiency were found out. It was recommended that SST turbulence model with high resolution scheme with time step corresponding to 1 degree of runner rotation and grid size should be used. Fig 9 shows flow pattern in Pelton turbine runner.

Anagnostopoulos and Papantonis [2012] developed an alternative numerical methodology known as Fast Lagrangian Simulation (FLS) for fast and effective simulation and analysis of complex flows based on Lagrangian approach. Based on numerical results, new

Pelton bucket has been designed. It is concluded that efficiency of numerical and model test results are very close but FLS results cannot provide accurate picture of real flow field during jet and bucket interaction (Fig 10).

Conclusions

It is found from literature review that many researchers used numerical simulation for the performance analysis of Pelton turbine and its distributor and validated the results with experimental data. The most of the simulation work has been done using ANSYS CFX and its results are in good agreement with experimental results. Some work on other problems like scale formation, erosion and stress analysis on material of the turbine has also been done. It was observed from the experimental work on Pelton turbine that the problem of erosion occurs mostly at the bucket splitter and cut-outs. For nozzles, erosion is observed at its outlet. Scope of more comprehensive and detailed study on analysis of erosion, cavitations etc may be studied in detail of Pelton turbine is still possible.

The numerical simulation results depend on many factors and selection of parameters like turbulence model, time step, surface tension. These factors still need to be studied in order to improve the accuracy of multi-phase simulations.

- -

Vishal Gupta, M.Tech (Energy), 2009, from MANIT, Bhopal (India) is presently pursuing PhD in Hydro Turbines from MANIT, Bhopal. He has published four papers in International/National journals/conferences. He is life member of National Society of Fluid Mechanics and Fluid Power and Solar Energy Society of India. Corresponding address: vishalgupta.manit@gmail.com

Dr. Ruchi Khare, M.Tech (Hydro), 1992 from MANIT, Bhopal (India), Ph.D. in hydraulic Machines from MANIT, Bhopal in 2011. She is presently pursuing research and consultancy in hydraulic turbines. She has published more than 37 papers in various National and International Journal and Conferences. She is the life member of ISTE and National Society of Fluid Mechanics and Fluid Power, Fellow member India society of Hydrologist. Corresponding address: ruchif4@rediffmail.com

Dr. Vishnu Prasad, M. Tech (Water Resources Engineering), 1988, from IIT, Bombay, PhD in Hydro Turbines from RGPV, Bhopal in 2009. He is presently providing consultancy and research in hydraulic turbines and water resources. He had been to UK for training on model testing and CFD for six months. He has published 60 papers in International/National journals/conferences. He is member of Institution of Engineers (India) and life member of Indian Society of Water Resources and National Society of Fluid Mechanics and Fluid Power.

Corresponding address: vpp7@yahoo.com

References

- Francois, M., Lowys, Pierre-Y. and Vuillerod, G., 2002, Development and Recent projects for hooped Pelton Turbine, Proceedings of Hydro 2002, Turkey, 4-7 Nov 2002.
- Sadlo, F., Peikert, R. and Parkinson, E., 2004, Vorticity Based Flow Analysis and Visualization for Pelton Turbine Design Optimization, Transaction of IEEE, October 2004, pp. 179-186.
- Catanase, A., Barglazan, M. and Hora, C., 2004, Numerical Simulation of a Free jet in Pelton Turbine, 6th International Conference on Hydraulic Machinery and Hydrodynamics Timisoara, Romania, pp. 79-84.
- Matthias, H.B. and Promper, O., 2004, Numerical Simulation of the Free Surface Flow in Pelton Turbines, 6th International Conference on Hydraulic Machinery and Hydrodynamics Timisoara, Romania, pp. 119-124.
- Marongiu, J.C., Maruzewski, P. and Parkinsin, E., 2005, Modelling the flow in a Pelton Turbine, SPH European Research Interest Community Newsletter, pp.8-9.
- Kotousov, L.S., 2005, Measurement of the Water Jet Velocity at the Outlet of Nozzles with Different Profiles, Transaction of Technical Physics, Vol. 50, No.9, pp. 1112-1118.
- Perrig, A., Avellan, F., Kueny, J-L. and Farhat M., 2006, "Flow in a Pelton Turbine Bucket: Numerical and Experimental Investigations, Transaction of ASME, Vol.128, pp.350-358.
- Zoppe, B., Pellone, C., Maitre, T. and Leroy P., 2006, Flow Analysis Inside a Pelton Turbine Bucket, Transaction of ASME, Vol. 128, pp.500-511.
- Konnur, M.S. and Patel, K., 2006, Numerical Analysis of Water Jet on Flat Plate, National Conference on Fluid Mechanics and Fluid Power- 2006, Raipur, India.
- Parkinson, E., Neury, C., Garcin, H., Vulllioud G. and Weiss, T., 2006, Unsteady Analysis of Pelton Runner with flow and Mechanical simulations, Hydropower & Dams, Issue three.
- Anagnostopoulos, J.S. and Papantonis, D.E., 2006, Experimental and Numerical Studies on Runner Design of Pelton Turbine, HIDROENERGIA, Crieff, Scotland, UK 7-9 June 2006
- Zhang, Zh., 2007, Flow Interactions in a Pelton Turbines and the Hydraulic Efficiency of the Turbine System, Journal of Power and Energy, Vol.-221, Part A, pp. 343-357.
- Marongiu, J C., Leboeuf, F. and Parkinson E., 2007, Numerical Simulation of the Flow in a Pelton Turbine using the Meshless Method Smoothed Particle Hydrodynamics: A new simple solid boundary treatment, Journal of Power and Energy, Vol.-221, Part A, pp.849-856.
- Zhang, Zh., 2007, Flow Friction Theorem of Pelton Turbine Hydraulics, Journal of Power and Energy, Vol.-221, Part A, pp.1173-1180.
- Zhang, Zh. and Casey M., 2007, Experimental Studies of the jet of a Pelton Turbine, Journal of Power and Energy, Vol.-221, Part A, pp.1181-1192.
- Ye-xiang, X., Feng-qin, H., Zhou, J-L. and Kubota, T., 2007,

- Numerical Prediction of Dynamic Performance of Pelton Turbine, *Journal of Hydrodynamics*, Ser. B, 19(3), pp.356-364.
- Agar, D., Rasi, M., 2008, On the use of a laboratory-scale Pelton wheel water turbine in renewable energy education, *Journal of Renewable Energy*, pp.1517-1522.
- Peron, M., Parkinson, E., Geppert, L. and Staubli, T., 2008, Importance of Jet Quality on Pelton Efficiency and Cavitation, *International Conference on Hydraulic Efficiency Measurements*, IGHEM 2008- Milano.
- Staubli, T. and Abgttsson, A., 2008, Efficiency Differences of upper and lower injectors and scan of tailrace temperature distributions, *International Conference on Hydraulic Efficiency Measurements*, IGHEM 2008- Milano.
- Hulass, H., Nilsen, E. and Vinnogg, L., 2008, Thermodynamic Efficiency Measurement of Pelton Turbines. Experience from investigation of Energy/Temperature distribution in the Discharge Canal Measuring Section, IGHEM 2008, pp 1-11, Milan, Italy.
- Nedelcu, D., Campian, Constantin, V. and Padurean, I., 2008, Stress and Deformations on Pelton Blade Turbine Calculated by Finite Elements Method, *The International Conference on Hydraulic Machinery and Equipments Timisoara*, October 16-17, 2008, Romania.
- Bajracharya, T.R., Acharya, B., Joshi, C.B., Saini, R.P. and Dahlhaug, O.G., 2008, Sand Erosion of Pelton Turbine Nozzles and Buckets: A Case Study of Chilime Hydropower Plant, *Transaction of Sciencedirect*, WEAR 264, pp. 177-184.
- Sick, M., Michler, W., Weiss, T. and Keck, H., 2009, Recent Developments in dynamic analysis of water turbines, *Journal of Power and Energy*, Volume 223, pp 415-427.
- Zhang, Zh., 2009, Inlet flow conditions and Jet Impact Work in a Pelton Turbine, *Journal of Power and Energy*, Vol. 223 Part A, pp. 589-596.
- Zhang, Zh., 2009, Analytical Method for Frictional Flows in a Pelton Turbine, *Journal of Power and Energy*, Vol. 223 Part A, pp. 597-608.
- Zhang, Zh., 2009, Flow Dynamics of Free Surface Flow in the rotating Buckets of a Pelton Turbine, *Journal of Power and Energy*, Vol 223 Part A, pp. 609-623.
- Santolin, A., Cavazzini, G., Ardizzon, G. and Pavesi, G., 2009, Numerical Investigation of the Interaction between the Jet and Bucket in a Pelton Turbine, *Journal of Power and Energy*, Vol. 223 Part A, pp.721-728.
- Padhy, M.K. and Saini, R.P., 2009, Effect of Size and concentration of Silt Particles on Erosion of Pelton Turbine Bckets, *Journal of Energy*, Volume 34, pp 1477-1483.
- Beucher, Y., Ksayer, Voulawz, E. and Clodic, D., 2010, Characterization of Frictional Loss in Pelton Turbine, *International Refrigeration and Air Conditioning Conference*, Purdue University, Purdue, 2274 page 1-8.
- Patel, K., Patel, B., Yadav, M. and Foggia, T., 2010, Development of Pelton Turbine Using Numerical Simulation, 25th IAHR symposium on Hydraulic Machinery and Systems, Timisoara, Romania.
- Fiederer, R., Riemann, S. and Schilling, R., 2010, Numerical and Experimental Investigation of the 3D Free Surface Flow in a Model Pelton Turbine", 25th IAHR symposium on Hydraulic Machinery and Systems, Timisoara, Romania.
- Jost, D., Meznar, P. and Lipez, A., 2010, Numerical Prediction of Pelton Turbine Efficiency, 25th IAHR symposium on Hydraulic Machinery and Systems, Timisoara, Romania.
- Kubota, T., 2010, Observation of jet interference in 6-nozzle Pelton Turbine, *Journal of Hydraulic Research*, Vol. 27, No.6, pp. 753-767.
- Stamatelos, F.G. and Anagnostopoulos, P., 2010, Performance measurements on a Pelton Turbine Model, *Journal of Power and Energy*, Vol. 225 part A, pp.351-362.
- Marongui, J-C., Leboeuf, F., Caro, J. and Parkinson, E., 2010, Free-surface flows simulations in Pelton Turbines using an hybrid SPH-ALE method, *Journal of Hydraulic Research*, Vol 48, Extra Issue, pp. 40-49.
- Atthanayake, I.U., 2010, Analytical study on Flow through a Pelton Turbine Bucket using Boundary Layer Theory, *International Journal of Engineering and Technology*, Vol.9, No.9, pp 241-245.
- Chaudhari, Gaurangkumar C., Channiwala, S.A., Shah and Samiq, P., 2010, Comparative Assessment of the Developed Stresses in both Traditional and Hooped Pelton Runners, *Proceedings of ICFD 10, Tenth International Congress of Fluid Dynamics*, December 16-19 2010, Red Sea, Egypt.
- Guha, N., Barron, Ronald, M., Balachandar, R., 2010, Numerical Simulation of High Speed Turbulent Water Jets in Air, *Journal of Hydraulic Research*, Vol.28, No. 1, pp 119-124.
- Yadav, S., 2011, Some Aspects of Performance Improvement of Pelton Wheel Turbine with Re-engineered Blade and Auxiliary Attachments, *International Journal of Scientific and Engineering Research*, Volume 2, Issue 9, September 2011, pp 1-4.
- Gupta, V. and Prasad, V., 2012, Numerical Investigations for Jet Flow Characteristics on Pelton Turbine Bucket, *International Journal of Emerging Technology and Advanced Engineering*, Volume 2, Issue 7, July 2012, pp 364-370.
- Dynampally, P. and Rao, V.S., 2012, CFD Analysis of Pelton Turbine, *Proceedings of Thirty Ninth National Conference on Fluid Mechanics and Fluid Power*, SVNIT, Surat, Gujarat, India, pp. FMFP2012- 58.
- Desai, Jaymin D., Soni, Vishal A., Chauhan, Vishal K., Charania, Shahil S. and Patel, Kiran C., 2012, Evaluation of Twin Jet Pelton Turbine Using CFD, *Proceedings of Thirty Ninth National Conference on Fluid Mechanics and Fluid Power*, SVNIT, Surat, Gujarat, India, pp. FMFP2012- 194.
- Anagnostopoulos, J.S. and Papanonis, D.E., 2012, A Fast Lagrangian Simulation Method for Flow Analysis and Runner Design in Pelton Turbines, *Journal of Hydrodynamics*, Vol. 24, pp 930-941.
- Gupta, V., Prasad, V. and Khare, R., 2013, Numerical Modeling for Hydro Energy Convertor: Impulse Turbine, *Proceedings of International Conference on Global Scenario in Energy & Environment*, March 14-16, 2013, MANIT, Bhopal, India.