Kathmandu University Journal of Science, Engineering and Technology, Vol. 15, No. 2, August 2021

Kathmandu University
Journal of Science, Engineering and Technology

Numerical investigation of Pelton turbine with experimental validation

Pawan Lal Bijukchhe*[a], Sailesh Chitrakar[a], Nabin Raj Chaulagain[b], and Prabin Timilsina[b]

[a]Department of Mechanical Engineering, Kathmandu University Dhulikhel, Nepal.
[b]Department of Aerospace and Mechanical engineering, Pulchowk Campus, Institute of Engineering, Pulchowk, Nepal.

Abstract

Computational Fluid Dynamics (CFD) has been established as a prominent tool for the effective analysis and optimization of hydro-turbines. Various CFD codes, both commercial type and open-source type, are being used in the industry for flow simulation and optimization. For impulse turbines like Pelton, the numerical simulation requires solving of various complex phenomena like multi-phase fluid interaction, multi-reference frame study, jet-bucket interaction and free-surface transient flows. A number of simplifications has to be made in terms of the geometry and flow physics for solving the flow problem in reasonable time and less computational cost.

The aim of this research is to use OpenFOAM to develop a numerical model for flow analysis of Pelton turbine with an existing bucket geometry, and compare the results with experimental data. A numerical model consisting of three buckets was studied, out of which the torque produced in the middle bucket during a transient flow was observed and results were compared with the experimental data from project carried out in Norwegian University of Science and Technology. The jet of water hitting the bucket is modelled as a cylinder with a constant diameter corresponding to a defined opening of the spear valve. The *snappyHexMesh* utility of OpenFOAM was used for meshing the geometry and *interFoam* solver with laminar turbulence model was used to simulate the flow over buckets. The results obtained have shown a good concurrence with the actual values from the experiment.

Keywords: CFD; OpenFOAM; Multiphase; interFoam

---

1. Introduction

Pelton turbine, invented in 1889 by American engineer Lester Allan Pelton, is widely used parallel flow impulse turbine. It is ideal for high head (from 15meters to 1,800meters) and low discharge conditions [1]. A simple Pelton turbine setup in power-house has a Pelton runner at the end of penstock nozzle. Water from a high head location is directed through a penstock to the runner and torched with a nozzle, fitted at the end of penstock. The coherent, circular free jet from nozzle impinges tangentially over the curved buckets fitted on runner rim. Buckets are double hemispherical cup with sharp dividing central edge and curves that allows impinging jet to suffer a 165° deflection (in most designs) [2]. The jet transfers its momentum to the buckets and resulting tangential force creates torque on the shaft of the wheel. The discharge to the turbine is regulated by translation motion of a needle valve, termed as spear valve.

The use of CFD is noted for the optimization of flow over the turbine in order to harness maximum energy through the turbine. CFD based works on impulse turbines are found to be more complicated than reaction turbines due to the multi-phase flow, jet-bucket interaction and unsteady nature of the flow over the fluid domain. Adding multi-reference frame setup complicates such simulation as flow becomes highly unpredictable at that moment. Number of assumptions and simplifications have to be made in terms of geometry and flow physics for solving the flow problem. These assumptions cause significant errors between experimental and numerical approach. So, practical experiences have greater impact in hydraulic design of a Pelton turbine besides applying general design rules [3].

Various approaches are made in order to solve the multiphase flow in Pelton turbine. Bucket simplification, domain simplification and symmetry assumption are some noted ideas applied in order to simplify the computational domain and reduce the computational expenses. As rotation of Pelton is periodic in nature, majority of simulations are performed by modeling only some odd number buckets (3, 5 or 7 buckets at a time and mostly 3 buckets) [4]. While taking three buckets, torque in middle bucket is measured and is used to construct the torque on the runner assuming steady state load over complete turbine [5] [6]. The other assumption is simulation over half Pelton bucket. As the buckets are symmetric, the result from half bucket is later corrected to be assumed as result from complete bucket [7]. Today, due to the availability of high computational power, there is possibility to conduct the 3D simulation of the flow over a complete turbine setup. However, reduced geometry can also approximate the results as obtained by modeling a complete turbine. Most researcher prefer study in reduced geometry so as to save computational power.

The assumptions made for simpler domain cause errors while validation of numerical result with experiments directly. Experimental torque, mechanical power and efficiency of turbine are generally taken as methods for validation. In this study, validation over the torque at steady state operation is done between similar conditions in numerical and experimental approach. Torque in mid bucket is measured and extrapolated as total dynamic torque during steady state operation.

---

*Corresponding author. Email: bijukchhepaw@gmail.com

Figure 1: Pelton turbine model used for experimental test in NTNU [10].

2. Materials and method

The numerical approach was applied over a Pelton design prepared by Solemslie [8] [9]. Experimental study of the design was performed in Water Power Lab at Norwegian University of Science and Technology. The experimental setup had 23 buckets with pitch angle 10.90°. Diameter of jet was taken as 35 mm for designing of buckets. The flow over bucket was captured with high-speed camera. The experimental lab setup can be seen in Fig. 1 [10]. Torque was measured at different spear valve opening condition in the experimental setup. In this study, the constant jet diameter of 35 mm was taken as done in analytical design.

Fig. 2 shows the sketch of simplified domain taken into study. Meshing was done over the domain using built-in meshing tool of OpenFOAM, *snappyHexMesh*. The *snappyHexMesh* generated hexahedral mesh over the domain, which is shown in Fig. 2. The hex-dominant mesh was refined in important areas like buckets and jet developing areas. Two different mesh sizes were studied having number of elements of 200 thousand and 400 thousand. Each mesh was generated with the same physical setup and boundary conditions. Comparison between the results obtained in those meshes was also carried out.

The domain has two sub-domains, stationary and rotating. Jet inlet and outer atmosphere are defined in the stationary domain, while the rotating domain contained the buckets. An Arbitrary Mesh Interface (AMI) surface is defined between these two domains. The aim of using AMI surface is to project the interface patch of one domain into the other domain and interpolate [11]. The hub part does not play an important role in torque production and was removed in order to decrease the cell number. However, a complete circular domain was created for maintaining the pitch ratio 1 over the AMI surfaces. The computational domain and its patches are displayed in Fig. 3.

The numerical model has 3 buckets and a jet of diameter 35 mm. The purpose of using three buckets was to generate the realistic nature of back-splashing and cut-off jet over mid bucket. The first bucket is required to produce the back-splashing water that impacts the mid bucket, while the third bucket is required to realistically cut the jet when jet is still impacting the second bucket [1].

Pelton turbine, being an impulse turbine incorporates free jet interaction with the atmosphere, thus requires a multi-phase study. OpenFOAM has no generic solver applicable to all case. A specific solver is chosen for a class of problems to solve [11]. Most multiphase problems, that include air-water interaction, are solved using *interFoam* solver. The *interFoam* solver uses Volume Of Fluid (VOF) method to track the multi-fluid interaction.

An overview of the applied boundary condition can be found in Table 1. Flow velocity of 38.38 m/s was given at the inlet which corresponds to 70 m head as stated by Rygg in his Master's report [7]. His work also had similar approach for study of same turbine. Buckets were provided with moving wall velocity and other wall surfaces were assumed as no slip wall. The condition was defined in the 0 folder, under p_rgh, U and alpha.water for the dynamic pressure, velocity and volume fraction respectively, as per the case structure of OpenFOAM. The domain was initially filled with air, water was supplied to the domain through the inlet.

The rotational properties were provided in constant/dynamicMeshDict. The rotational speed of the runner was taken as 62.83 radian per second, corresponding to 600 rpm. Turbulence model was to be set in constant/turbulenceProperties and for simplification of study, laminar model was used. The simulation was carried out for 0.035s, after which the effect of jet on third bucket would be negligible. Simulation was performed in HPC of Kathmandu University Supercomputer Center. Each simulation was performed using 12 processors. Approximately 72 computational hours was required to simulate coarse mesh setup for 0.035 seconds.

3. Results and discussion

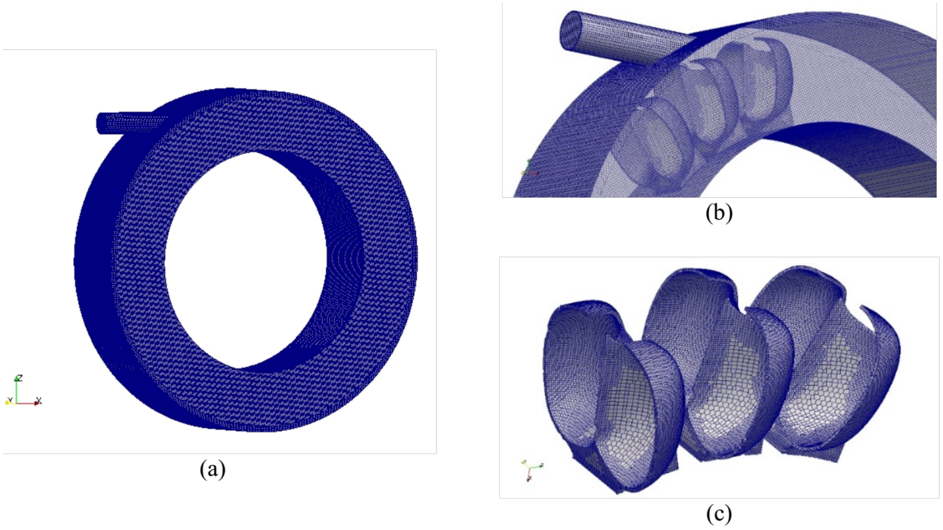
Results were extracted using built in function objects in OpenFOAM. Torque was calculated over the mid bucket using force function object available in OpenFOAM. The force function provides, force and moment over the patches (buckets in this case). The moment about y-axis is the required torque to be calculated. Other isosurfaces were produced using ParaView, which is also opensource post-processing tool.

3.1. Torque generation

As studied in the past research works [12], this study assumes that in the case of three buckets' Pelton turbine model, the torque produced from the middle bucket could be replicated for obtaining the time dependent torque. This torque has the effect of jet hitting both preceding and succeeding buckets. Thus, it can be taken as steady state torque for turbine operating at constant speed. Torque in middle bucket was measured and extrapolated as per the pitch angle and rotational speed. Then, the dynamic torque was plotted with respect to the torque in mid bucket, assuming the same loading in every bucket at steady state operation. The steady state torque for 35 mm jet was measured to be 185.94 N·m, with maximum torque of 126.14 N·m on mid bucket alone. Fig. 4 shows the nature of torque generation while operating the turbine in steady state. Some negative torque at the starting is due to the back-splashing created by the first bucket.

The torque curve of the mid bucket is replicated with a proper time shift and integrated to produce the total torque curve. The time shift was calculated as 0.0043s using number of bucket and 'omega' provided to the bucket. The average value of total torque can be taken as the total torque acting on the turbine shaft, as it is assumed every bucket produces identical torque periodically which was found to be done in different literatures [5].

During steady operation, the average torque from experimental results was measured to be 261 N·m. The total torque obtained numerically for the same case was 185.94 N·m. The reason for deviation can be the assumptions made for simplification of the case and the mesh density. However, the trend of torque generation during the time of jet striking the middle bucket was found to be similar to the past studies [12].


(a)

(b)

(c)

Figure 2: (a) Complete domain mesh with enlarged view (b) and (c).

Table 1: Boundary condition in each patch as shown in Fig. 3.

Patches	p_rgh	U	alpha.water
inlet	zeroGradient	fixedValuevalue uniform (38.38 0 0)	fixedValue 1
jetwall	zeroGradient	noSlip	zeroGradient
inwall	zeroGradient	noSlip	zeroGradient
AMI1	cyclicAMI	cyclicAMI	cyclicAMI
out_atm	totalPressurep0 uniform 0	pressureInletOutletVelocityvalue uniform (0 0 0)	inletOutlet
bucket	zeroGradient	movingwallvelocityvalue (0 0 0)	zeroGradient
AMI2	cyclicAMI	cyclicAMI	cyclicAMI
atmosphere	totalPressurep0 uniform 0	pressureInletOutletVelocityvalue uniform (0 0 0)	inletOutlet

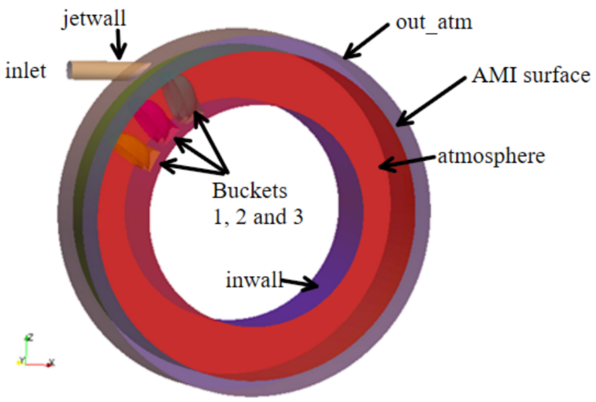


Figure 3: Boundaries in the fluid domain and their patch names.

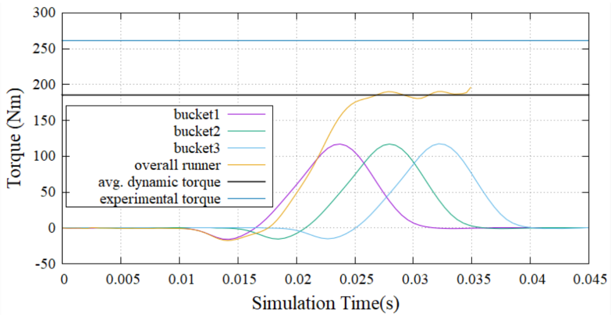


Figure 4: Torque on buckets at steady state operation.

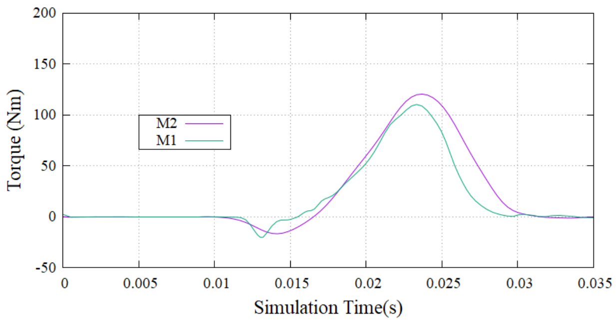
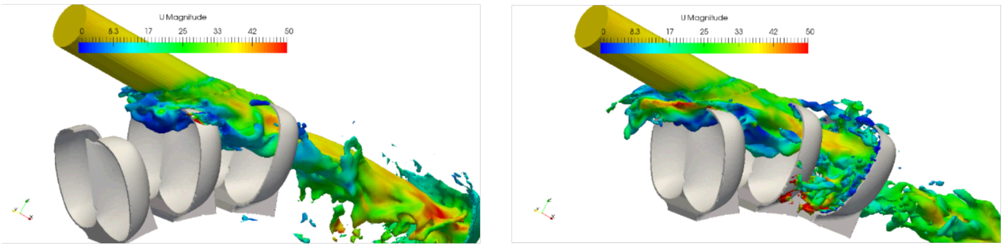


Figure 5: Torque in mid-bucket while taking different mesh sizes.

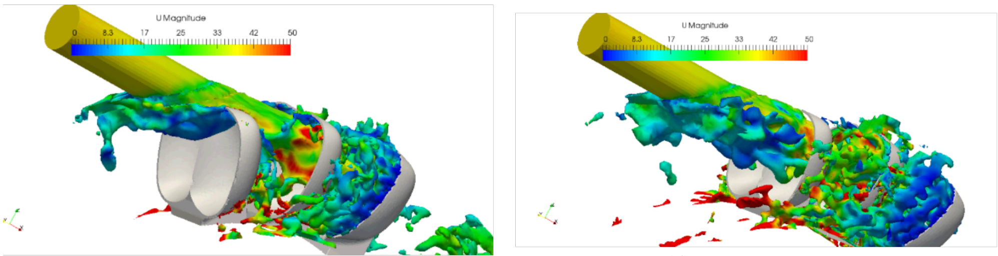
Fig. 5 shows a comparison of torque measurement in two mesh sizes. Details of the mesh and torque thus generated is tabulated in Table 2. Results were found to be directly dependent on the quality of mesh.

3.2. Jet-bucket interaction visualization

The nature of jet impinging over buckets at different angle of rotation can be seen through iso-surfaces as in Fig. 6. The iso-surface is generated from cells with volume fraction over 30% water. The nature of water splash after striking with buckets is shown in Fig. 6 (a), (b), (c) and (d). It can be seen from the figures that the jet-bucket interaction obtained from this CFD model shows good agreement with the flow in real turbines. After striking the buckets, water is evenly distributed between the two sides of the splitter. The outlet angle of the bucket being less than 180°, the back-splash striking the succeeding bucket is avoided. It can also be seen that the water velocity after striking the bucket reduces due to the



(a) 0.01299s= 48.90° (b) 0.019s= 71.59°



(c) 0.0210s = 79.12° (d) 0.023s = 86.66°

Figure 6: Interaction of jet with bucket at different degree of rotation.

Table 2: Mesh dependent test for tested mesh structures.

Mesh type	M1	M2
Total number of elements	233,219	389,368
Calculated average total torque (N·m)	163.62(111.16 single bucket torque)	185.94 (126.14 single bucket torque)
Standard torque (N·m)	261	261
Torque error percent (%)	37.30	28.75

transfer of momentum from the kinetic energy of the jets to the buckets. However, it can be inferred from the figure that the water splash and the velocity distribution can be refined further by increasing the mesh density.

4. Conclusion

The result from the numerical approach was found to be in a good agreement with the experiment. The total dynamic torque throughout the steady state operation was calculated to be 185.94 N·m. However, it can be concluded from this study that the result of numerical analysis is dependent on the mesh size, time-step of solution, as well as various assumptions made for the numerical approach.

The use of opensource tool for CFD was emphasized in this work. It was found that the application of OpenFOAM in the work of CFD of Pelton turbines is equally promising as commercial codes.

References

[1] Nigussie T, Engeda A & Dribssa E, Design, Modeling and CFD Analysis of a Micro Hydro Pelton, *Int. J. Rotating Mach.*, 2017 (2017).

[2] Reddy R, Pelton Wheel: Setting, Diagrams and Problem and Solutions, https://www.engineeringenotes.com/fluids/pelton-wheel/pelton-wheel-setting-diagrams-and-problem-and-solutions-hydraulics-turbines-fluid-mechanics/48165., 28 November 2020.

[3] Zhang Z, *Pelton Turbines Book*, Springer International Publishing, AG, Switzerland (2016).

[4] Zidonis A, Panagiotopoulos A, Aggidis G A, Anagnostopoulos J S & Papantonis D E, Parametric optimisation of two Pelton turbine runner deisngs using CFD, *J. Hydrodyn.*, 27 3 (2015) 840.

[5] Panthee A, Neopane H P & B. T, CFD Analysis of Pelton Runner, *Int. J. Sci. Res. Publ.*, 4 8 (2014).

[6] Kumashiro T, Fukuhara H & Tani K, Unsteady CFD Simulation for Bucket Design Optimization of Pelton Turbine Runner, *IOP Conf. Ser. Earth Environ. Sci.*, (2016).

[7] Rygg J R, *CFD Analysis of a Pelton Turbine in OpenFOAM*, Norwegian University of Science and Technology (NTNU), Trondheim (2013).

[8] Solemslie B W, *Experimental Methods and Design of a Pelton Bucket*, Norwegian University of Science and Technology, Trondheim (2016).

[9] Solemslie B W & Dahlhung O G, A reference Pelton Turbine - Design and Efficiency measurements, *IOP Conf. Ser. Earth Environ. Sci.*, 22 1 (2014).

[10] Wessel M C S, *Laboratory Test of a Pelton Turbine*, Norwegian Institute of Science and Technology (NTNU), Trondheim.

[11] Foundation T O, The OpenFOAM Foundation Website, http://openfoam.org/, 5 December 2020.

[12] Chitrakar S, Solemslie B W, Neopane H P & Dahlhung O G, Review on Numerical Techniques applied in impulse hydro turbines, *Renew. Energy*, (2020).