# A REVIEW ON PERFORMANCE EVALUATION OF PELTON TURBINE

**L.JYOTHI**
Assistant Professor
Civil Engineering Department
Mahaveer Institute of Science & Technology
kjyothi.india@gmail.com

**G.ARUNA**
Assistant Professor
Civil Engineering Department
Mahaveer Institute of Science & Technology
arunagugilla@gmail.com

**B.BHARATH KUMAR**
Assistant Professor
Civil Engineering Department
Pragati Engineering College
bharathkumar0310@gmail.com

***Abstract:***

*Experimental methods have been used to measure fan performance. But the current generation often aims to measure using quantitative methods such as CFDs. The movement of a liquid liquid is an effective tool for measuring the performance of an electric generator. Pelton fan ship, a jet of water coming out of his nose was under air pressure. The performance of Pelton fans is highly dependent on the size, velocity, pressure of the aircraft and the size of the bucket. The current review paper mainly focuses on the use of literature for design improvement and CFD applications for the performance evaluation of Pelton fans.*

**Key words:** *Pelton Turbine, CFD, Multi phase fluid flow, surface flow, etc.*

## 1. Introduction:

The Pelton Turbine is a tangent pulse turbine that uses a high-tech Pelton hydroelectric power plant located primarily in the region. The highest head, especially in mountainous areas. In bucket turbines, water is converted into kinetic energy through a nozzle at the end of the pressure line. The pressure is constant throughout the cycle and equal to atmospheric pressure, so that energy is transferred by the action of the excitation. The flow in the scale is unstable with an unknown free area, diverging from the two-phase flow of air and rising within the moving range. The performance of the Pelton turbine depends on several factors; These turbines are bucket jet size, bucket water layer profile and bucket size, bucket water layer and bucket size.

Model tests were previously used to analyze the performance of a Pelton turbine for different bucket and nozzle sizes. But this analysis is time consuming and costly, and now that day-to-day progress has been made in CFD and numerical methods, optimizing the design of the Pelton turbine can be done in a short time frame. Turbine flow is a multiphase free surface flow consisting of air and water as the working medium.

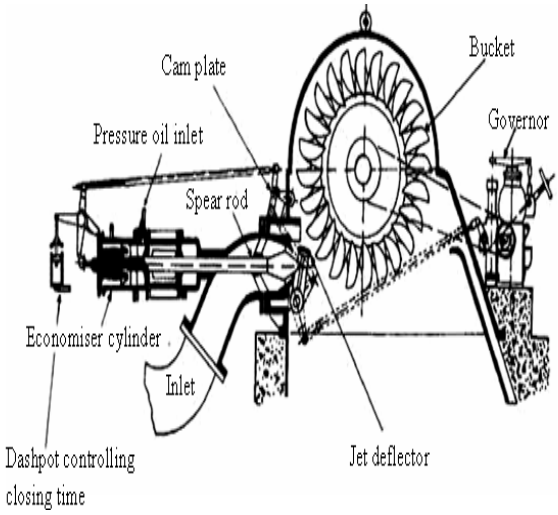


**Fig. 1 Main Components of Pelton Turbine [1].**

### 1.1 Literature Review:

BhattaraiSuyesh (2019) .et.al ... Mainly highlighted the main challenges associated with PELB turbine flow simulation.

Bucket geometry is primarily determined by parameters such as width, shape, depth, divider angle, height, exit angle and cut. The basic principle of operation of the Pelton turbine creates unstable flow conditions, including centrifugality, spatter, free surface flow and the karyol effect [2].

Zhang.z (2007) et al. This creates disturbances in the distribution of pressure and density of water and, in turn, increases the free surface area of the water. CFD analysis does not accurately simulate such abrupt changes in the physical properties of fluids, and these splashes have lost their meaning during the simulation. May Sakit M. KeckH, Perkinson (2000) et al. Study the influence of centrifugal and Coriolis effects when a jet is constantly directed into the bucket as it rotates [4]. Exact modeling of this phenomenon is still a challenge in CFDs.

T.R. Bajracharya et.al., (2008) discusses the problem of cutting a Pelton turbine nozzle in the Himalayan region. The study was conducted by the authors with analysis. Flow trap diagrams were compiled and the wear rates of needles and buckets were analyzed [5].

B. Zopee et.al .. (2006) presented a detailed experimental and numerical analysis of the flow in a stationary tank of an operating Pelton turbine. Experimental analysis provides flow presentation along with pressure and torque measurements. Numerical analysis was performed in the FLUENT program using a two-stage volumetric flow of the liquid method [6].

## 2. Methodology:

### 2.1 Steps Involved in Design of the Turbine:

To begin the initial design, a calculation is made for the size of the turbine components. The theory behind this is largely based on the Tack Guide to the Micro-Hydropelton Turbine [7]. Various assumptions are made during the design process using design guidelines and literature.

### 2.2. Calculation of the Net Head ($Hn$).

The net head at the nozzle exits can be expressed by the following formula:

$H_n = H_g - H_l$,

Let $H_g$ =Gross head

$H_l$ = Total head losses due to open channel, trash rack, intake, penstock, and gate or valve. These losses.

$H_l \approx 5\%$ of $H_g$

The method is used to analyze the CFD of a Pelton turbine. 3D model of Pelton Hydroelectric Plant. The rotating domain and fixed domain for CFD analysis have been created using PRO / Engineer and ANSYS design models by several researchers, respectively. Numerical methods and boundary conditions are defined in ANSYS CFX, and numerical results are verified statistically and analytically. Export calculated torque calculation for calculated allowable pressure distribution fatigue analysis [8].

The geometry of a Pelton turbine bucket is mainly determined by parameters such as width, height, exit angle, depth, splitter angle, size, and cut. The jet stream from the nozzle is at atmospheric pressure and the pelton body is filled with air. This principle of operation produces centrifugal and creole effects as well as free surface currents (FSFs), splashing, unstable power supply conditions. Thus, the main problems associated with modeling flow in Pelton turbines are shown in fig. 2.

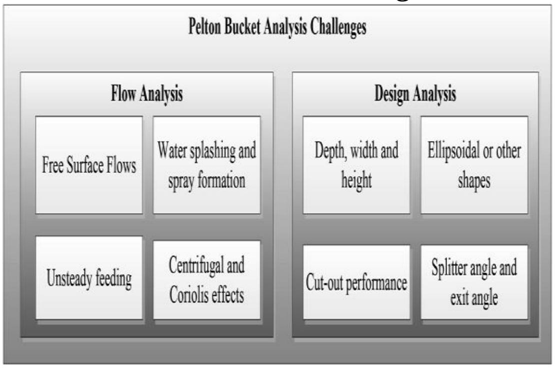

**Fig 2: Challenges for Pelton Turbine Analysis**

## 3.Numerical Investigations & Results:

For an equal distribution of the pressures and hoops applied to the bucket, Franois et al discussed the pressures applied to the piston impeller bucket and the uniform distribution of the hoops. (2002) CFD analysis of finite element method and conversion of jets with event. The angle indicates that cracks and maintenance costs can be reduced by applying hoops to the slider. In figs. 3 shows the pressure distribution [9].

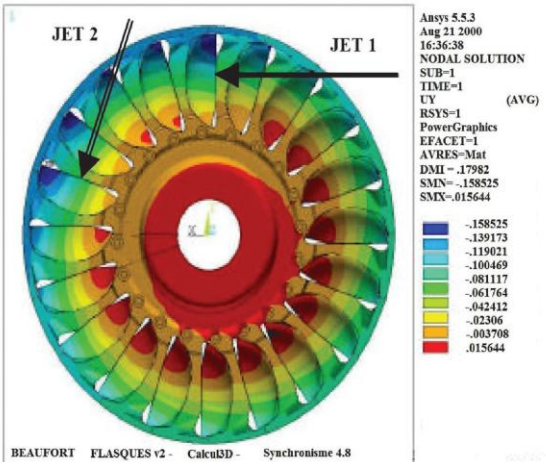


**Fig 3. Tangential displacement from FEA on 3D model**.

Chaudhary et al (2010) worked on stress analysis for traditional Pelton runners and Hope Pelton runners using I-DEAS software. The designed hoop runner is comparable in performance to a traditional runner. The model is designed to rotate 18 working buckets, a jet and 1000 rpm. Finally, Hoop concludes that the use of pressure is more evenly distributed [10].

Popular methods have been used to calculate the deformation and pressure analysis of Pelton turbine blades to create 3D finite elements of solid geometry. Geometry is divided into tetrahedral solid elements. Maximum pressure values were found in the area of the taper shank bore attached to the bucket for the runner (Figure 4)

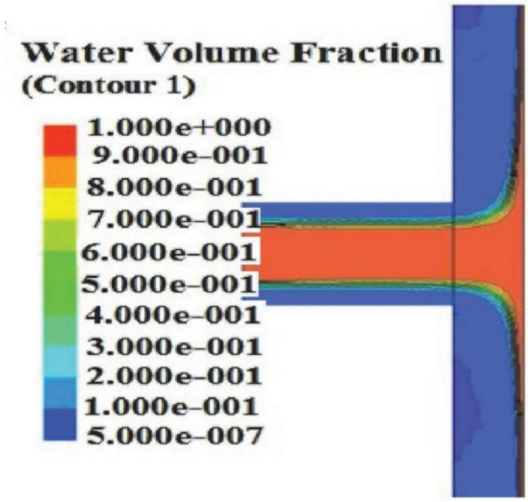


**Fig 4: water volume fraction for perpendicular jet [11]**

sick et.al., (2009) analyzed three different stages of the reactive and cup interaction process; It was found that current load, water discharge and maximum torque did not achieve maximum accuracy of torque estimation. Accurate torque correction requires more precise two-phase flow changes, cavitation area, and segmentation. Layers of water are used to visualize the flow and its effect on the roof [12].

ANSYS-CFX is used for digital analysis of the flow of water in a flat plate at three different current locations at 0, 30 and 60 angles. Continuous numerical analysis of the situation was performed by Gupta [13]. Then Gupta et al. (2013) extend water jet modeling on curved panels with different flow angles at the outlet. They performed a steady state analysis based on a homogeneous two-stage model with a K – E turbulence model using Ansis – CFX. Theoretical and numerical changes in pressure, velocity and current strength along the plate profile were investigated. It has been found that as the outlet angle increases, the thickness of the water layer decreases, as the water has to move over a larger area [14]. The highest pressure at the center of the plate was also observed and was the same in all cases (Fig. 5).

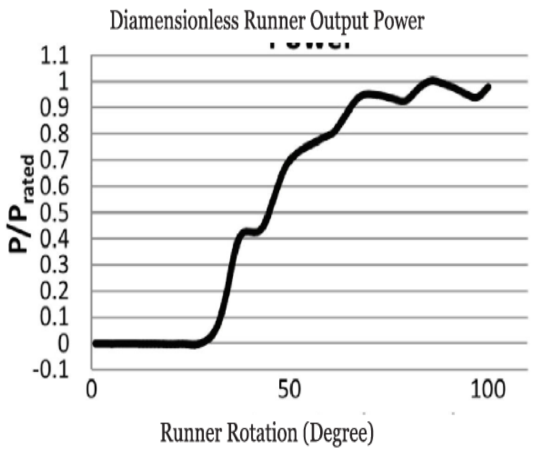


**Fig 5: Power variation with runner rotation (Desai et.al[15])**

Results of 3D modeling performed by Jianfeng U (2015) et al. The maximum running speed was found to be 1.78 times during the maximum running phase. Acceleration is an important feature in the early stages of walking, and the nozzle cracks and strikes the reindeer's lips and back. And the last phase [16].

Tilah Nigussi (2017) et al. His case study gives an example of how turbine size affects turbine efficiency. They are discussed in detail to understand the key issues of dynamic design, which will help improve the efficiency of future research and reduce production costs [17].

## Conclusions:

Several experiments for performance testing and estimation of Pelton turbines can be seen from the analysis data of various researchers. Most researchers use Ansis-CFX and it gives good results with experiments. Some researchers are working on other things, such as the design, measurement and shear of components of the Pelton turbine. As experimental work on pelton turbines, harvesting problems often occur in barrels and cutouts. An operation on the ear cannot be caused by radiation. There are several options for assessing pelton strength for cuts and cavities.

Experiments vary depending on many factors and include factors such as time period, rotation pattern, and pressure.

## References:

1. *Modi P.N and Seth.S.M. "Hydraulics and Fluid Mechanics Including Hydraulics Machines (Paperback Edition). Standard Book House, Delhi.*
2. *Bhattarai Suyesha,⁎, Vichare Paraga, Dahal Keshava, "Novel trends in modelling techniques of Pelton Turbine bucket for increasedrenewable energy production" Renewable and Sustainable Energy Reviews 112 (2019) 87–101.*
3. *Zhang Z. Flow interactions in Pelton Turbines and the hydraulic efficiency of the Turbine system. Proc Inst Mech Eng A J Power Energy 2007;221:343–55.*
4. *Sick M, Keck H, Parkinson E, Vullioud G. New challenges in pelton research. Bern: Proc. HYDRO 2000 Conf.; 2000.*
5. *T.R. Bajra charya, B.Acharya, C.B. Joshi, R.P. Saini and O.G. Dahlhaug (2008) "Sand Erosion of Pelton Turbine Nozzles and Buckets: A Case Study of Chilime Hydropower Plant", Transaction of Sciencedirect, WEAR 264, pp. 177-184.*
6. *B.Zopee, "Flow analysis inside a pelton turbine bucket" Journal of Turbo machinery, Transactions of the ASME, Vol. 128, July 2006.*
7. *J.Thake,The Micro-Hydro Pelton Turbine Manual: Design, Manufactureand Installation for Small-Scale Hydro-Power, ITDG, publishing, London, UK, 2000.*
8. *Zidonis A., Aggidis G. "A State of the art in numerical modelling of Pelton turbines. Renewable and Sus-tainable Energy Reviews, 2015, 45: 135-144.*
9. *Froncois, M.Lowys, development and recent projects for hooped pelton turbine,*

*Proceedings of Hydro 2002, Turkey,Nov 2002.*

10. *Choudari ,Garangkumar comparative assessment of the developed stresses in the both Traditional and Hooped pelton runners, Proceedings of ICFD 10, conference of Fluid dynamics, dec 2010.Egypt.*
11. *Kannur and M.S.Patel Numerical analysis of water jet on flat plate, National conference on Fluid mechanics and Fluid power, 2006, Raipur,India.*
12. *Sick M, Michler W, Weiss T" Recent developments in dynamic analysis of water turbines, Journal of Power and energy, Volume 223, PP 215-227.*
13. *Gupta v, and Prasad v numerical investigations for jet flow characteristics on pelton turbine bucket, International journal of Emerging technology and advanced Engineering,Volume 2, july 2012,PP 364-370.*
14. *Gupta V, Prasad v and Kishore "Numerical modelling for Hydro energy converter: Impulse Turbine, Proceedings on international conference on Global scenario in energy & Environment, march 2013, PP14-16.*
15. *Desai, Jaymin D, Soni and Chauhan "Evaluation of Twin Jet Pelton Turbine using CFD" Proceedings of the 39th National Conference on Fluid Mechanics and Fluid Power,FMFP 2012.PP 194.*
16. *Jianfeng You, Xu Lai, Wei Zhou and Yongguang Cheng "3D CFD simulation of the runaway process of a Pelton turbine" ProcIMechEPart A: J Power and Energy 2016, Vol. 230(2) 234–244. IMechE 2015.*
17. *Tilahun Nigussie,1 Abraham Engeda,2 and Edessa Dribssa1 "Design, Modeling, and CFD Analysis of a Micro Hydro Pelton Turbine Runner: For the Case of Selected Site in Ethiopia" HindawiInternational Journal of Rotating MachineryVolume 2017, 17 pages.*