Novel trends in modelling techniques of Pelton Turbine bucket for increased renewable energy production

Bhattarai, Suyesh; Vichare, Parag; Dahal, Keshav; Al Makky, Ahmed; Olabi, Abdul

Published in:
Renewable & Sustainable Energy Reviews

DOI:
10.1016/j.rser.2019.05.045

Published: 30/09/2019

Document Version
Peer reviewed version

Novel Trends in Modelling Techniques of Pelton Turbine Bucket for Increased Renewable Energy Production

BHATTARAI Suyesh[a*], VICHARE Parag[a], DAHAL Keshav[a, b], AL MAKKY Ahmed[a], OLABI Abdul[c,d]

[a]University of the West of Scotland, School of Engineering and Computing, High Street, Paisley, UK
[b]Nanjing University of Information Science and Technology, Nanjing, China
[c]Sustainable and Renewable Energy Engineering, University of Sharjah, P.O. Box: 27272, Sharjah, UAE
[d]Mechanical Engineering and Design, Aston University, School of Engineering and Applied Science, Aston Triangle, Birmingham, UK

[*]Corresponding Author: BHATTARAI Suyesh, E-mail: Suyesh.Bhattarai@uws.ac.uk, bhattarai_suyesh@hotmail.com

Abstract

Pelton turbines have been used for harvesting clean energy from water jet for over 100 years. The wide range of their applicability and the robustness with minimal onsite monitoring and carbon free energy production makes them one of the most desired hydro turbines for renewable energy production. Pelton turbine buckets are subject to complex turbulent multiphase flows with free surfaces, thus received lot of attention from Computational Fluid Dynamics (CFD) researchers to visualise the flow patterns. In addition, the bucket geometry optimisation has been a prevalent research stream using analytical and graphical methods. Both of these investigations fields have resulted in significant improvement in the performance of the Pelton Turbine system. However, the design investigations for each feature are carried out independently due to the complexity in incorporating large number of design parameters. Thus, analysing the complex fluid flow on each pre-optimized design was rather challenging due to expensive computational needs of CFD and limited manufacturing possibilities. Today, development of accurate and inexpensive CFD models and innovative manufacturing technology such as rapid prototyping has made complex freeform shape possible to simulate and manufacture. Hence, any bucket designs disregarded in optimisation on the basis of manufacturing feasibility can now be possible to manufacture and is worth investigating. This will require the combination of CFD and design optimisation field of investigation with novel analysis approach. This paper examines these fields of studies with the view to establish the background for such novel approach. This approach could be a foundation for design optimization of turbomachinery or any reaction surface leading to increased production of renewable and sustainable energy from existing resources.

Keywords: Pelton turbine; Computational fluid dynamics; Optimisation; Bucket; Design

Abbreviations:

2D	Two dimensional	MAR	Multivariate Adaptive Regression
3D	Three dimensional	MPS	Moving Particle Semi-implicit
ALE	Arbitrary Lagrangian Euler	MW	MegaWatt
ANOVA	Analysis Of Variance	NURBS	Non-Uniform Rational B-Spline
CFD	Computational Fluid Dynamics	PC	Personal Computer
FEA	Finite Element Analysis	RAM	Random Access Memory
FLS	Fast Lagrangian Solver	SPH	Smoothed Particle Hydrodynamics
FSF	Free Surface Flows	SST	Shear Stress Transport
FVPM	Finite Volume Particle Method	Twh	Terrawatthour
GA	Genetic Algorithms	UK	United Kingdom
GB	Giga Byte	VOF	Volume of Fluid
GHz	Giga Hertz		
GW	GigaWatt		
HEL	Hybrid Euler Lagrangian		

Nomenclature:

Cp	coefficient of pressure
D	diameter of jet
Di	Diameter of smoothing function
k	turbulent kinetic energy
m/s	meters per second
Q	Flow rate
Qmax	Maximum flow rate
Rj	Radius of jet
Rmax	Maximum radius of jet
T	Torque
t	time
α	angle of incidence
ϵ	rate of dissipation of turbulence energy
ω	specific rate of dissipation into internal thermal energy

1. Introduction

The most common application of the Pelton turbine in the world today is for power generation. These turbines are widely used in the areas with a steep gradient of water flow or where reservoirs can be constructed to generate the required head. Its working range varies from low to medium flow (0.1 to 11000 litres per second) and medium to high head (30 to 1000 meters) [1].There is a large potential for hydropower that has not been harnessed in many developing countries. For example, there is a potential of 43,000 MW in Nepal that is technologically and economically feasible but only 600MW is being harnessed [2]. The gradient from the Himalayas to the southern flat lands in Nepal has provided ideal conditions for distributed small to medium scale hydropower generation on local level. These small and medium scale projects are realizable without heavy investments [3] or major environmental impact [4]. Similarly, there is a potential for 1.5GW of untapped hydropower in the UK [5]. An accurate estimate for global potential for hydropower has not been agreed upon yet as detailed technical, environmental and socio-economic feasibility studies have not been conducted in global scale. A report from World Energy Council [6] estimates the unutilized global potential for hydropower at 10,000TWh/year. With the total installed capacity of 1,064 GW until 2016, the report foresees an increase to 2,000 – 2,050 GW capacity by 2050, that is, almost double the total existing capacity in next 30 years. These potential will be realized sooner as the world energy demand is shifting towards renewables following the awareness on effects of climate change caused by burning of fossil fuels [7]. Currently, the efficiency of the turbine is taken to be around 82-85% in lab conditions, but mathematically, up to 96% of the energy can be harnessed [8]. If this gap can be bridged, there will be 10% increase in the energy generated from the same resources without major changes and with added possibilities of improved irrigation, water supply or flood control [9].

The Pelton turbine is essentially an impulse type turbine that operates at the atmospheric pressure with a free jet of water striking on its buckets and the kinetic energy of the jet causing the turbine to rotate. Large reservoir dams are constructed to generate high head required for the Pelton turbine or run-off-river schemes are used for areas with large gradient. This water at high head is then brought to the powerhouse that houses the turbine through penstock pipe that is laid such that least head is lost during transportation. At the end of the penstock pipe is the nozzle which lets out the water as a fast, circular, continuous jet. This jet is directed towards the buckets on the Pelton turbine runner. This high speed jet causes the turbine to rotate which in turn rotates the generator to give electric power. The Pelton buckets are essentially two ellipsoidal cup shaped buckets joined together. A sharp ridge between the two buckets called the 'splitter' bifurcates the oncoming jet and spreads it on each bucket before the water exits the buckets from the sides. There is a cut-out at the outer end of the bucket which allows the jet to contact the splitter directly on each bucket. This ensures comparatively smoother flow and continuous torque generation. The geometry of the bucket is mainly governed by parameters such as depth, width, height, splitter angle, exit angle, shape and the cut-out. The flow of jet from the nozzle is at atmospheric pressure and the Pelton casing is filled with air. Such working principle creates conditions as free surface flows (FSF), splashing, unsteady feeding along with centrifugal and Coriolis effects. Hence, major challenges relating to modelling the flow in the Pelton turbines are highlighted in figure 1.

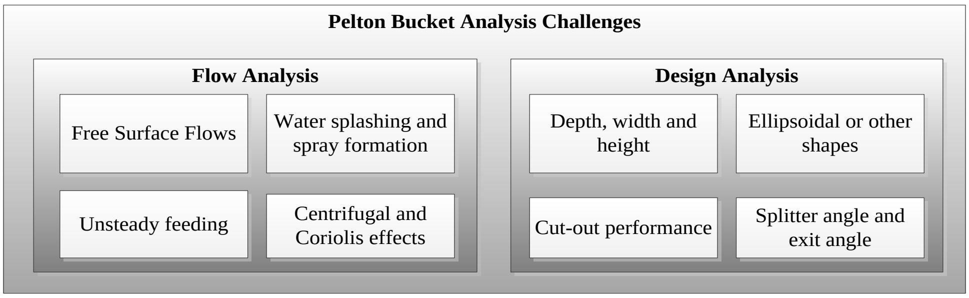

Figure 1. Challenges for Pelton Bucket Analysis

FSF involves effect of air resistance and surface tension at the surface of water exposed to air [10]. As CFD methods derive physical properties of the elements based on the neighbouring elements, the identification of an exact borderline between two fluid phases is complex and hence effect of air resistance and surface tension cannot be analysed. This results in inaccurate simulation results because of the blurred borderline.

Water splashing effects and spray formation after the jet strikes the bucket could intercept the jet or impinge on unwanted surfaces reducing the efficiency [11]. This creates disturbances in the pressure distribution and density of the water and, also increases the free surface area of water. CFD methods are unable to accurately simulate such drastic changes in the physical properties of the fluids and these splashes lose their valuation in the course of the simulation.

Unsteady feeding of the jet on the bucket is created as the jet bifurcates as it strikes on the cut-out when the turbine is rotating. This creates additional free surface and Coanda effect which partially deter the flow of the jet stream whose CFD modelling has not been accurately achieved.

Centrifugal and Coriolis effects act when water continues to exert force as it flows in the bucket while the bucket is rotating [12]. These affect the force experienced by the bucket while rotating. Accurate modelling of this phenomenon is still a challenge in CFD.

Turgo turbines are impulse turbines similar to the Pelton turbine and share the same problems in modelling as the Pelton turbines. It differs from the Pelton turbine as it has only one ellipsoidal bucket and lacks the splitter. The jet strikes the bucket at an angle allowing more flow through the bucket. It enables higher speed of the runner and can operate at head and flow rates suitable for Francis turbines as well [13]. Yet, it covers lesser range in both the head and flow rate as compared to the Pelton turbine. Research on Turgo turbines have been limited owing to the popularity of Francis turbines and the difficulty in modelling impulse turbines [14]. In this paper, we cover research on Pelton turbine in greater detail as they have been widely researched and these research address problems similar to those experienced by Turgo turbines owing to similar working conditions.

This paper examines various approaches on analysing Pelton turbine bucket design as listed in figure 1. The investigation is focused on analysing interaction between inner and reverse side of the bucket with the water jet. This paper presents a comparative table of existing literature, critical evaluation of various approaches and presents a research scope which will provide novel approach towards analysing and optimizing the performance of turbo-machinery.

The literature available on Pelton turbine performances ranges from investigating the effect of a single bucket feature such as the number of buckets [15]and the angle of attack on the bucket [16] or various other parameters such as the length, width, depth, height of the splitter, angle of the splitter, angle of exit, etc. shown in figure 2. Zidonis et. al. in [17] studied the Pelton bucket using 15 different variable parameters to define the shape of the bucket. Similarly, Anagnostopoulos et. al. in [18] and [19] studied the shape of the Pelton turbine with 19 and 18 free geometric variables respectively. Various types of hydro-turbines such us Francis turbine, Pelton turbine, and axial Kaplan turbines were analysed with varying flow conditions and corresponding rotating speeds to optimise performance [20]. Few studies have also been conducted to validate the ability of CFD codes with Pelton turbine as a case study [21]. Other studies explored alternative options such as new designs in controllers and generators [22], nozzle geometry [23]and penstock [24].

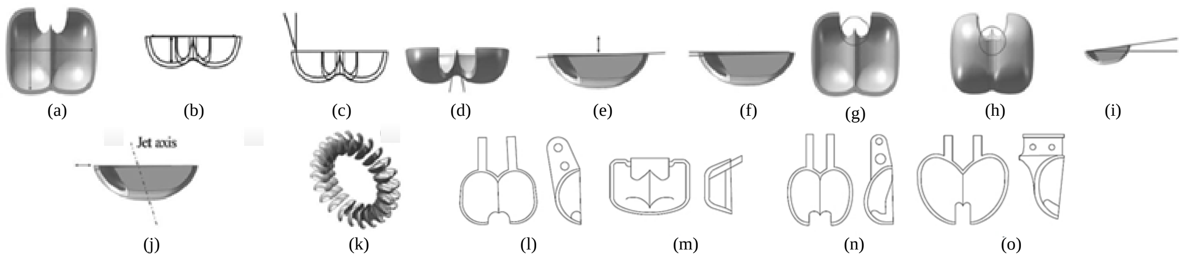

Figure 2: Bucket Features a) length to width ratio, b) depth to width ratio, c) exit angle, d) inlet angle, e) splitter level, f) tip angle, g) tip geometry, h) backside of splitter, i) inclination angle, j) radial position, k) number of buckets [17], l) circular bucket, m) ellipsoidal bucket, n) rectangular and, o) heart-shaped bucket tested by Gibson [16]

After studying the wear on the Classic Pelton buckets William Doyle [25] concluded that the turbulence in the water is the main cause of bucket abrasion. He then patented the design that is quite similar to those used today and defined it as the 'bucket formed of two side-by-side ellipsoidal cavities joined by a stream splitter, with the leading edge cut away to clear the incoming jet' [25]. Modern Pelton turbine blades are very similar to this design. Before the application of CFD, these turbines were developed using a graphical methods [16,26] or analytical method [11]. Various profiles for the bucket such as circular, ellipsoidal, rectangular etc. (shown in figure 2) were tested with a degree of performance improvement reported in the literature.

There has been a rise in research after the CFD capabilities were proven for FSF by the end of the twentieth century. Avellan et. al. [27] verified the performance of the Volume of Fluid approach used in some of the commercial CFD packages to predict the flow at injectors and bucket geometry of a Pelton turbine. Muggli et.

al. [28] conducted the numerical and experimental studies of the FSF of the jet stream later leading to the study of flow interactions in Pelton turbines [11]. Sick et. al. [12] made studies relating to challenges in study of Pelton turbines and flow analysis in the distributors and the injectors [29]. These studies were mostly based on the jet and the nozzle, but much remains to be done for the optimization of the bucket itself [14]. Some research has been done to optimize the shape of the Pelton bucket using analytical methods and computational methods but a significant improvement has not been obtained.

There are various problems relating to study of Pelton turbine buckets relating to the flow simulation and optimization that have curbed the design development of Pelton bucket. The problems can be classified as Flow analysis problems and Design analysis problems. Research is being carried out in these areas but a convincing approach to handle these problems is yet to be formulated. The methods used for study have been described in detail in the following sections.

2 Strategies for representing flow environment

2.1 Static Blade Condition (Stationery Bucket):

The first attempts to numerically study the performance of the Pelton turbine was made with stationery buckets. Janetzky et. al. [30] introduced a volume fraction for each finite element to approximate the position of the free surface and then used the Navier-Stokes equations along with the transport equations to solve for pressure distribution and flow simulation. The studies were made in three positions of the stationery bucket which were a) when the jet enters the bucket through the cut-out, b) when the jet completely enters the bucket and c) when the jet leaves the bucket. The study assumes steady flow in a stationery bucket, ignores the centrifugal and Coriolis forces (figure 3) and surface tension at the free surface is not accounted.

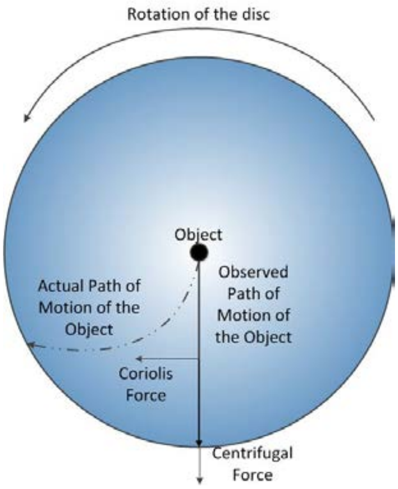

Figure 3: Coriolis Forces on a Rotating Frame

Similar study was conducted by Avellan et. al. [27] while validating the ability of a commercial CFD software for analysing free surface flows. Flow vectors of the bucket (figure 4) and the pressure field on the bucket were generated from this study. Pressure values obtained from CFD experiment were compared with physical pressure sensors installed on the buckets. The difference in the values were attributed to stationary bucket boundary conditions in CFD study. However, assumptions allowed researchers to focus on the interaction between the jet and the bucket for a given instance rather than the continuous complex interaction between the water jet and the bucket while rotating. The simulation was immensely simplified on account of these assumptions. For example; there was no unsteady feeding caused by the interference of the water jet by the succeeding bucket; the centrifugal forces and the Coriolis forces that are generated during the actual operation due to the rotation of buckets could be ignored in the stationery bucket assumption. Similarly, the interaction between the sheets of water and the rotating bucket surface has been ignored in the stationary bucket investigation.

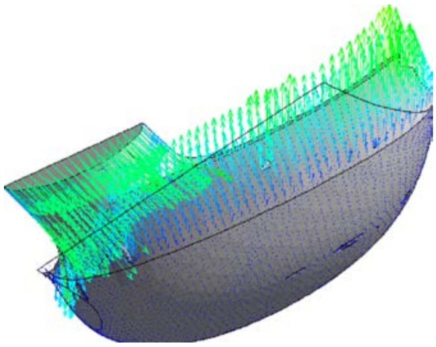

Figure 4: Side view of the calculated flow in the bucket [27]

Matthias and Promper [31] validated the CFD codes for impact of the free jet on a flat disc that was considered as the simplification of the Pelton bucket design. The results obtained at different angles of impact where simulated pressure on the plate surface is compared with the total force of the jet. Close agreement between CFD approach and practical experiments confirmed the validity of the simulation approach.

Zoppe et al. [32] made studies on stationery buckets by varying the angle of incidence (as shown in figure 5) to represent the pressure generated during the rotation of the turbine. Pressure was measured with 21 points in the wetted area of the bucket. Thrust and torque measurements were taken through the sensors placed in the bucket handle. The experimental values and the numerical calculations showed agreement except in two extreme cases of the incidence angle of 60 degrees and 120 degrees, which were associated with the flow loss through the cut-out. Gupta et. al. [33] made studies of flow inside the Pelton bucket with circular and rectangular jet stream and obtained error of 3.7% and 6.01% for circular jet stream in 50m/s and 68m/s velocity. This validates use of VOF method to analyse FSF problems in a stationery bucket.

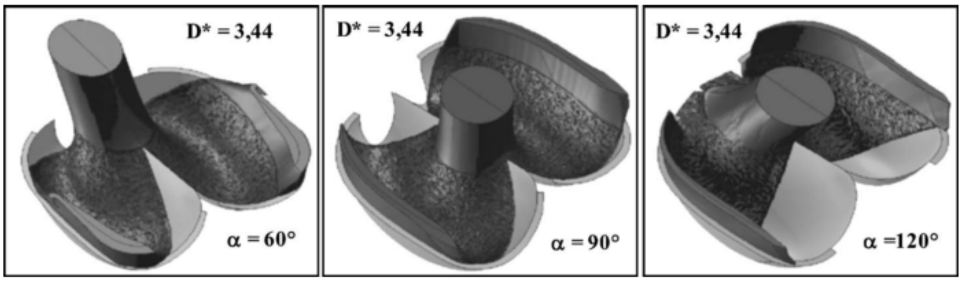

Figure 5: Flow visualization at different angle of impact [32]

Grozev et al. [34] and Kvicinsky et al. [35] and Bhattarai et al. [36] recorded pressure measurements on static buckets. Kvicinsky et al. [35] and Guilbaud et al. [37] also conducted experiments to study the water layer thickness on the static buckets. All these research made studies on individual mechanisms of Pelton-Jet impact cycle, but were unable to incorporate all of it into one single simulation. Hence, this type of analysis is not a complete representation of the actual performance of the Pelton turbine but it established the ability of the CFD software to handle the FSF problems with acceptable accuracy.

Alternative methods to the mesh based VOF method was also tried to encounter the problem of FSF. Marongiu et. al. [38] studied the application of Smoothed Particle Hydrodynamics (SPH) method for static and rotating Pelton bucket. The study suggested a new model for the treatment of solid boundary conditions and was able to simulate the complex geometry, free surface flow and interaction between runner and the casing. The experimental results had close agreement except 10% of error in initial and ending stages. The pressure prediction on sharp edges was not satisfying as the tangential components flow momentum was higher than the normal. Nakanishi et. al. [39] too employed the Moving-Particle Semi-implicit (MPS) method to a stationery Pelton bucket which reproduced the tendency of pressure distribution. The simulation results were smaller than the experimental results by up to 10% in maximum at different angular positions. The simulation predicted correctly the tendency of FSF and the corresponding pressure distribution at different angular positions of the bucket.

Table 1 explains the inherent complexities in representing the flow environment found in the literature. The rotating bucket assumption exhibits dynamic blade and water body interface, with all features listed in Table 1. The stationery bucket assumption simplifies the problem to the extent that it discounts 6 major phenomena.

Table 1: Inherent complexities in Representing Flow Environment

	Dynamic Blade Condition	**Ref**	**Comments**
Static Blade Condition	Free Surface Flow	[24,27,31]	Difficulty in distinguishing exact boundary of the fluid in two phase flows A mixed zone is created in the interface area which diminishes the accuracy
Static Blade Condition	Pressure Readings	[25,30,32]	Readings obtained with sensors closely matched simulation results Difficulty in obtaining readings for rotating bucket due to requirement of waterproof, wireless sensors
Static Blade Condition	Flow Vectors	[19]	Experimental validation of flow vectors has not been possible due to high speed flow with splashing effects
Static Blade Condition	Water Sheet Thickness	[14,29]	Cameras used to validate the simulation results but accurate measurements are still a challenge in rotating conditions
	Unsteady Feeding	[NA]	Inability to determine effect of unsteady feeding and associated Coanda effect on the jet
	Coriolis Forces	[NA]	These forces are generated with the bucket's rotation but have not been validated experimentally
	Centrifugal Forces	[NA]	These forces are generated with the bucket's rotation but have not been validated experimentally
	Splashing Effects	[NA]	The effects of splashing on other components and the backside of succeeding bucket have not been studied
	Impact of Cut-Out	[32]	The lift generated and the Coanda effect at the Cut-out have not been verified
	Traversing Jet in Bucket	[NA]	The interaction of jet with the water flowing within the bucket diverts the flow vectors whose effects have not been studied

2.2 Dynamic Blade Condition (Rotating Bucket):

Today's computational and the programming capabilities have made possible to analyse above-mentioned issues with satisfactory accuracy using the VOF method and the two-phase homogeneous method [40]. The rotating buckets approach is more accurate and comprehensive representation of the problem; however, it requires greater computational effort and time. In most studies, this computational problem is simplified by modelling only three consecutive buckets [41][42]. The bucket at the centre undergoes the entire cycle of jet entering the bucket and leaving it along with the splashing effects from the previous bucket; hence, it encompasses the entire cycle of any bucket.

The rotation was modelled using two different approaches viz. rotating inlet and rotating buckets. Considering the grid with the buckets to be stationery while the jet was moving in the inlet [43,44] allows for reduced computational costs as fewer elements would rotate in each time frame. But it also causes a slight deviation in the jet and is not the most accurate assumption. The other method is more common and has the inlet section as stationery and the rotating zone with the buckets. This method is a much more accurate approximation of the actual phenomenon. The agreement of the pressure and torque readings between simulation and experimental results are encouraging. Similar simulations can provide valid readings for the reverse side of the bucket.

The study conducted by Perrig et al. [40] based on two phase homogenous model measures the pressure in the inner surfaces of the buckets. The resultant patterns had a qualitative agreement and variation of 7-26% was observed in experimental and numerical results. From further analysis, it was established that the region near the cut-out contributed most to the bucket power and the region at the root of the bucket was less productive (figure 6). It was also noted that the reverse side of the bucket close to the cut-out contributed to the bucket torque. The jet, due to Coanda effect at the cut-out, created lift force which generated additional angular momentum. But this effect has not been verified experimentally. The author also emphasises the necessity to study the air flow between the adjacent buckets.

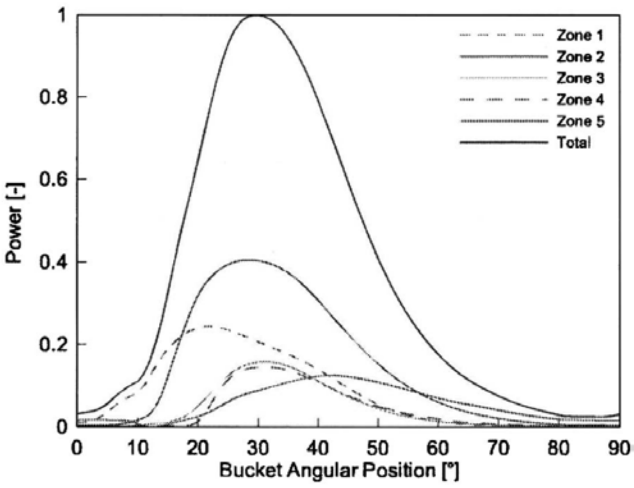

Figure 6: Normalized contribution of each zone to the total mechanical power exchange as a function of the runner rotation angle [40].

Xiao et. al. [45] made a good attempt to study the unsteady FSF patterns and torque using commercial CFX code by using the Realizable k-epsilon turbulence model with the two phase volume of fluid method. The simulation results for relative efficiency were lower by 1-2% which the author attributed to the coarse grid space discretization. Barstad [46] tested the performance of the CFX codes for the bucket design developed by DynaVec by predicting the torque applied to a non-stationary Pelton bucket by a high speed water jet. It was concluded that the model over-predicts the torque by approximately 1.5%. These studies verify the ability of the CFX codes to handle such problems with acceptable accuracy.

Zidonis et. al.[17] and Anagnostopoulos et. al. [18] made use of the Fast Lagrangian Solver (FLS) to study the flow inside the Pelton turbine bucket. This model was able to introduce additional terms into the particle motion equations to account for the various hydraulic losses and flow spreading which were regulated by the experimental data. The FLS does not solve the Navier-Stokes flow equation and hence its accuracy is restricted. Direct comparison with experimental values has not been made for these studies and hence their accuracy cannot be verified. The dynamic blade condition was utilized by Zidonis et. al. in [15] to evaluate the effect of number of buckets in the turbine wheel. Tests were made with 14, 15, 16, 17 and 18 buckets and the readings suggested that 15 buckets instead of the traditional 18 were optimal since the jet was cut by the succeeding bucket before it completely acted on the current bucket.

Hybrid Eulerian-Langrangian (HEL) method was applied by Rossetti [41] to investigate the jet-bucket interaction by means of a traditional mesh-based numerical approach. These results were then integrated using a predictor-corrector algorithm to determine the fluid particle trajectories on the rotating buckets. Using this method, discharged kinetic energy, momentum variation and total energy variation of each particle during the jet-bucket interaction can be analysed. This method provides acceptable accuracy in the calculation of torque as the mean torque presented maximum error of 2.7%. The study highlights contribution of various areas of the bucket on total torque to identify the most efficient and the least efficient areas.

Few phenomena such as secondary flows, spray formation, unsteady feeding, interaction with components and film flow are illustrated in the figure 7. Studies have shown greater proximity to the actual experimental readings when tests have been done on the rotating turbine buckets as above phenomenon can be accounted in the simulation. The study with dynamic blade condition allows accurate and realistic evaluation of the Pelton bucket. The grid based method provides greater accuracy while the particle based methods provided detailed information of the flow of each particle in the bucket. Hence, we can be concluded that the dynamic blade condition is the better strategy for flow environment replication in Pelton turbine buckets.

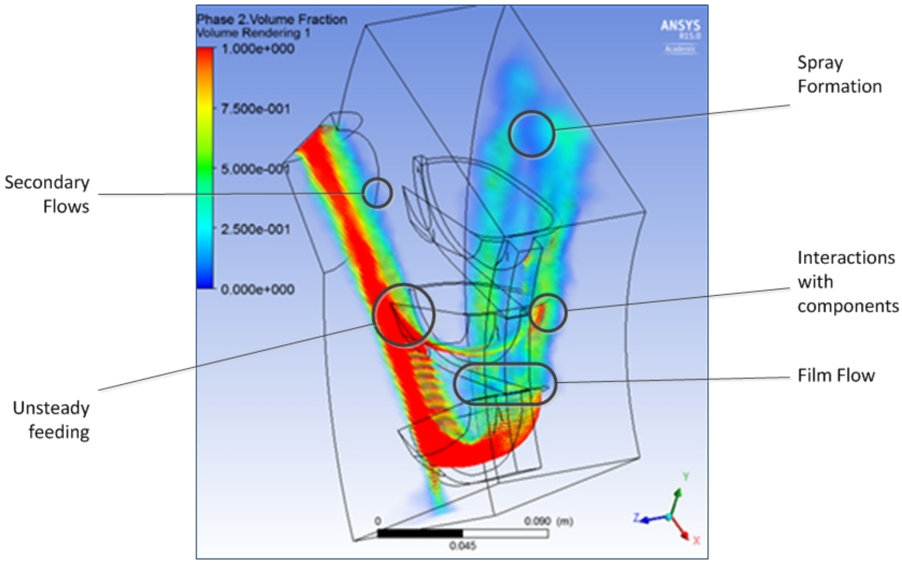

Figure 7: Illustration of various conditions during water-bucket-air interaction (original in colour)

3 Flow Simulation Strategies

Unlike reaction turbines, which operate in pressurised condition without the presence of air, impulse turbines work at atmospheric pressure and have interaction between air and water. This interaction between water and air, because of their different physical properties, have pushed the limits of CFD methods. The surface tension and viscous properties of water has to be considered when it interacts with air. These properties cause film flow, spray formation, secondary flows, pressure losses and unsteadiness and complex interactions with components [14]. In addition, energy losses due to the disturbance caused by incoming jets and interfering with water sheets affects performance of the Pelton turbine which cannot be completely analysed theoretically [47]. Thus, above aspects are analysed in the literature with flow simulation strategies such as space discretized approach or the particle representation approach. Other simplified 2D or 3D quasi state approximations [13,48] have not been considered in this study.

3.1 Space Discretized Approach (Eulerian)

Eulerian methods make use of the traditional grid-based numerical methods, where the environment is discretized into various smaller elements as shown in figure 8. The Navier-Stokes based continuity equation, momentum equation and diffusion equations are used to calculate the properties of the fluid volume within an element or a fixed space [40]. Being a discrete technique, higher discretization of space and timestep provides more accurate results. The computational requirement is in correlation with discretization and tends to increase exponentially with the increase in the number of elements and/or timesteps. The most popular commercial solvers are Fluent and CFX. There are some basic differences in the working of the two solvers [5].

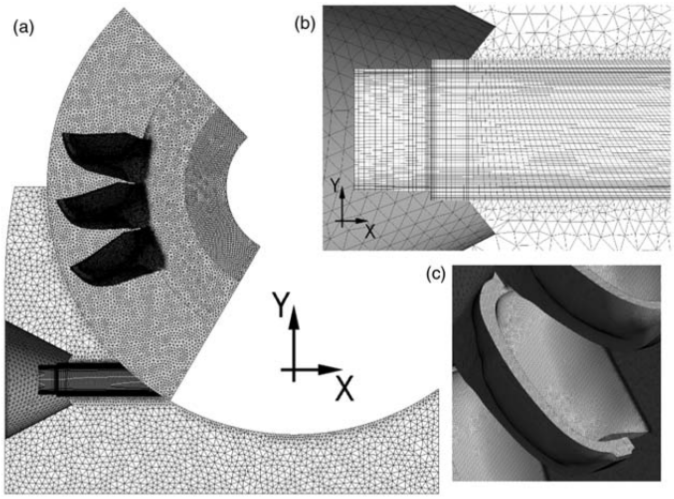

Figure 8: Mesh of the 3 buckets model (a), details of the jet (b) and the bucket (c) [41]

3.1.1 Fluent

Fluent uses VOF method for solving problems with free surfaces and allows users to choose among density, segregated pressure or coupled pressure based methods for solving the governing equations [5]. From literature it has been established that the VOF method along with k-ϵ model of turbulence generated the most accurate results from Fluent [17,32,49] for Pelton turbines while SST k-ω have shown to be more accurate for reaction turbines [50].

Zoppe et. al. [32] published experimental and numerical analysis of flow in a fixed bucket of a Pelton turbine which provided good consistency with the experimental results when Fluent code was used with two-phase flow VOF method. The pressure distribution was very well predicted for all range of studied parameters as is visible in figure 9. Detailed analysis of the torque produced, sheet of water thickness in the bucket and thrust generated enabled evaluation of losses due to the edge and the cut-out. The discretization scheme of the second order upstream was used to model the fluid advection. Free surfaces were denoted by the volume fraction value of 0.5. The piecewise linear interface calculation method was used for the geometrical reconstruction of the interface. The turbulence intensity of the jet stream was taken as 5%. The time taken to run the simulation and achieve the convergence for 4500 timesteps on a bi-processor PC AMD Athlon 2000+ was around 35 hours. Similar study was also conducted by Souari and Hassairi [49] and was compared with the experimental investigations developed by Kvicinsky et. al. [44]. Above studies confirms that the Fluent codes are capable to correctly predict the pressure coefficient and water sheet thickness on the bucket surface.

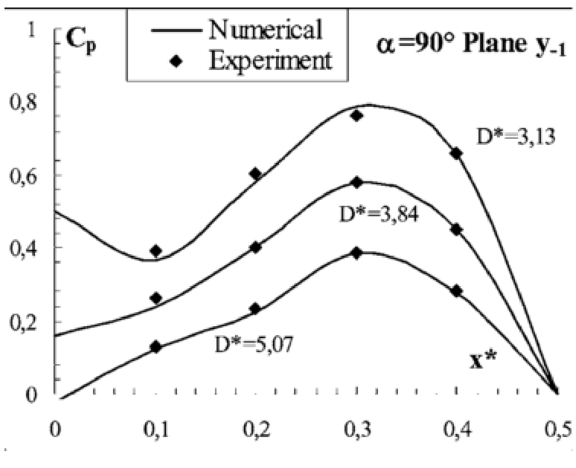

Figure 9: Pressure coefficients comparison between Fluent code and Experimental values [32]

Zidonis et. al. [17] used an Intel Xenon Computer with four cores of 3.4 GHz and 16 GB RAM and spent almost 8 days for the simulation to complete. The study modelled only two consecutive half buckets because of the heavy computational cost and the case being periodically symmetrical and the bucket being geometrically symmetrical. Later, the problem was reduced to 1.5 million elements from 2.8 million elements and the conditions of inviscid flow and with ideal jet was applied to reduce the computational time to 2 days. The error introduced by this simplification was only 0.5%.

3.1.2 CFX

CFX uses finite elements (cell vertex values) to discretize the domain, it uses coupled algebraic multigrid approach and allows choosing from homogeneous and inhomogeneous methods for solution [5]. The two-phase homogeneous method along with k-ω shear stress transport (SST) model of turbulence provided most accurate results for CFX [41,51,52].

Numerical simulations based on the generalized homogeneous multiphase flow with additional sources of momentum for Coriolis effect and centrifugal accelerations in rotating frame of reference was employed by Perrig et. al. [40]. CFX-5 simulation code with high-resolution upwind scheme and physical advection terms were used to provide a good trade-off between diffusion and dispersion. An ideal jet was assumed and second order backward Euler scheme was used for transient terms. The simulation was run with rotating 5 half buckets and a stationery injector made up of 2.4 million tetrahedral elements. The time taken to attain convergence of the simulation has not been reported. The results for pressure coefficient confirmed with experimental values in the area where the flow was not affected by the jet-bucket interaction (C12 and C22 in figure 10). The areas of cut-out (C11 and C12 in figure 10), splitter and the outflow did not match the experimental results very well.

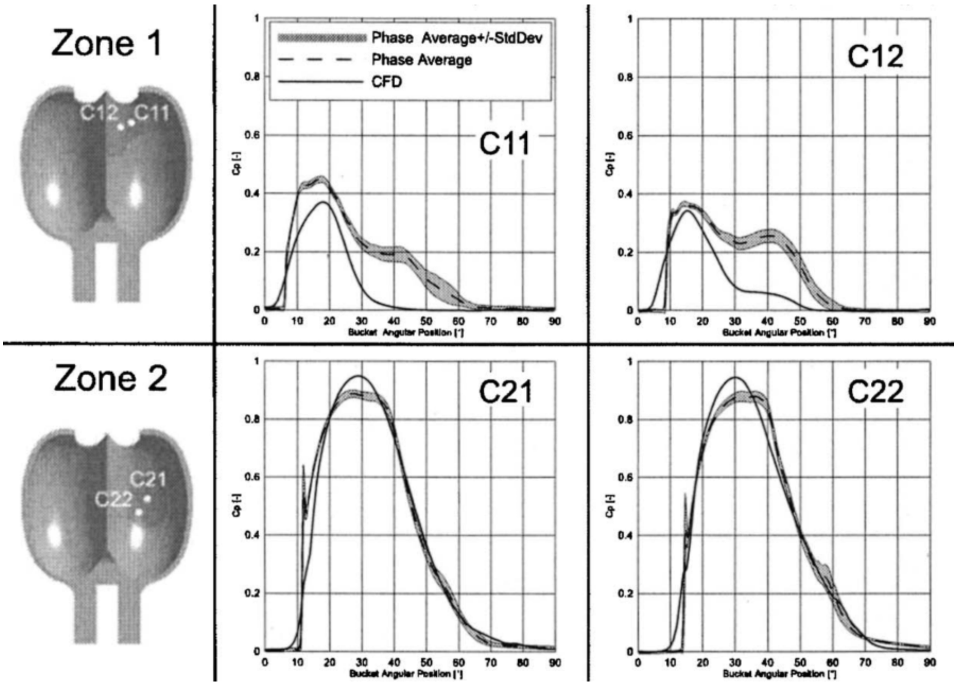

Figure 10: Comparison of experimental with CFX simulation pressure coefficient results [40]

Zidonis et. al. [17] used an Intel Xenon Computer with four cores of 3.4 GHz and 16 GB RAM and spent approximately 72 hours for the simulation to complete with $3x10^6$ mesh elements. But the studies made with 1.5x106 elements by compromising accuracy for quicker solutions followed the trend as the previous accurate solution. This allowed the simulation to be completed in 24 hours. The details of the parameters set in the solver and the various models used have not been disclosed.

The accuracy of these methods is commendable but the computational costs and the time taken for the convergence of the simulation are very high. These approaches would be useful to test the final design with high accuracy but are not feasible for testing interim designs with small variations.

3.2 Particle Representation Approach (Lagrangian)

These methods are based on the incompressible Navier-Stokes equations. The flow is modelled by following the individual particles and tracking their trajectories as shown in figure 11. This method can provide the precise readings of the water sheets by following the particles. Hence, it is possible to study the interaction of a single water particle with the interfering surface, discharged kinetic energy, momentum variation and total energy variation. This ability of the Lagrangian methods makes it very popular for application on the Pelton Turbine.

Various solvers have been developed for simulating the flow in Pelton turbines such as the Fast Lagrangian Solver (FLS) [17,18], the Smoothed Particle Hydrodynamics (SPH) method [38,53] and the hybrid methods [41] etc.

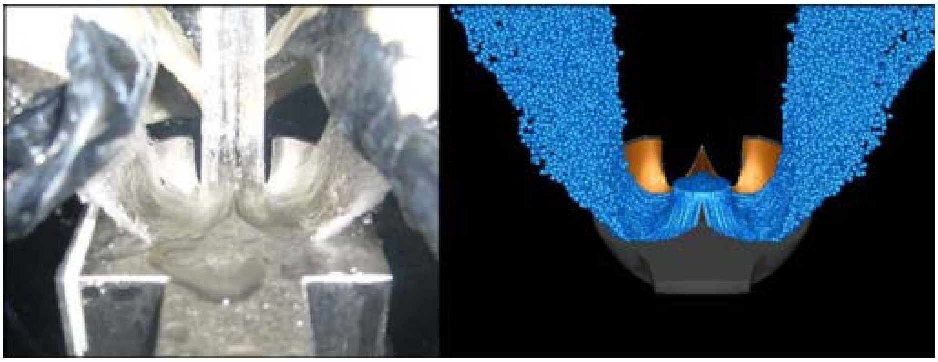

Figure 11: Experimental and Simulation visualization of particle representation method [39]

3.2.1 Fast Lagrangian Solver (FLS)

The FLS was designed to provide quick results from simulation so that stochastic optimisation software could be coupled for optimisation [18]. But its accuracy is dependent on few constants in the particle motion equation which can be determined only through experiments or more accurate CFD solvers [5]. Trajectories of an adequate number of representative fluid particles are tracked and particle motion equations are integrated with hydraulic losses and pressure effects. But this method is unable to calculate the impact of sharp edges in the cut-out and the splitter in the bucket. The FLS is a very cost effective tool for configuring preliminary design, where the performance of numerous designs needs to be simulated. Flow pattern with more than 10^4 trajectories were computed in just a few seconds with average computing capability [18]. The discrepancies in result shown in figure 12 do not exceed 3% units in most cases. For the smallest jet, the measurements show a drastic efficiency reduction due to the degradation of the jet in such small injectors.

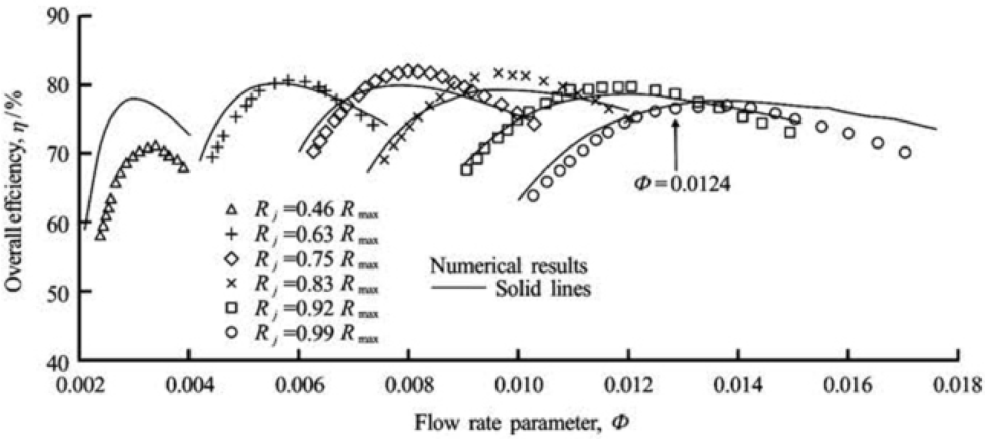

Figure 12: Comparison between FLS results and measured turbine efficiency [18]

3.2.2 Hybrid Euler Lagrangian (HEL)

Rosetti et al. [41] presented an analysis of particle flow tracks based on a HEL method to investigate the influence of bucket geometry on the efficiency of the turbine. The study evaluated the contribution of different bucket areas to the total torque of the turbine with respect to time. Initially, the jet-bucket interaction was numerically analysed by the traditional mesh-based Eulerian approach using the CFX codes. The results were then integrated into a predictor-corrector algorithm using fourth order Adams-Bashforth method as predictor and the fourth order Adams-Moulton method as corrector to determine the fluid particle trajectories in the rotating buckets. The torque on each bucket was calculated from the particle that came in contact with the same bucket. The predictor-corrector algorithm made an error of maximum 3.8% as compared to the CFX results as can be seen in the figure 13. This study established the accuracy for Lagrangian method very close to that of the Eulerian method. This study suggests that the exit angle at the side of the buckets can be reduced for better performance. There are no suggestions relating to the profile of the bucket and no studies have been made on the reverse side of the bucket.

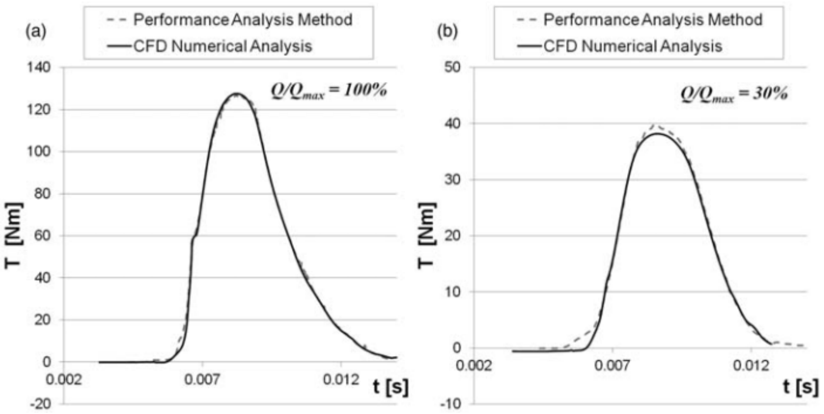

Figure 13: Torque value comparison between CFX results and predictor-corrector algorithm [41]

3.2.3 Smoothed Particle Hydrodynamics (SPH)

A recent study [53] shows that SPH method is as accurate as Eulerian methods such as CFX and Fluent. However, the pressure coefficient comparison (figure 14) shows some difference between the two curves at the beginning of the jet-plate interaction and also after the peak with unwanted oscillations in the SPH curve. Also, this method only studied the inner surface of the bucket and ignored the losses at the edges and the cut-out.

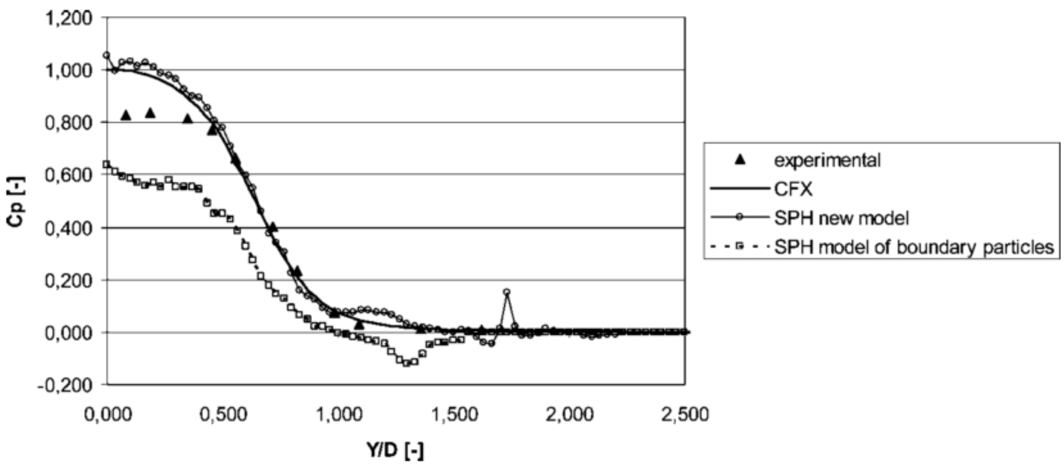

Figure 14: Comparison of pressure coefficient values on a solid plate from experimental results, CFX, SPH new model and SPH model of boundary particles [38].

The kernel function in SPH is the convolution product between the field function and the regularizing function. This kernel function can provide second-order accurate approximation in space if its integral is unity and if the function is symmetric. This gives a technique to reconstruct a continuous field using discrete values on a set of disordered points. SPH has two discretization parameters viz. mean distance between calculation points and the smoothing length. In order to get an accurate scheme, the ratio of smoothing length to the mean distance between calculation points should tend to infinity while both the parameters tend to zero. Practically it is not possible to have the ratio tend to infinity hence a value for the ratio has to be taken such that the interpolation domain contains enough points to ensure accuracy and stability [38]. The SPH method solves the Euler equations in Lagrangian formalism allowing for better accuracy as compared to other Lagrangian methods.

The inherent characteristics of the different approach discussed above are briefed in Table 2.

Table 2: Advantages and Disadvantages of Popular Flow Simulation Approaches

	Package	**Precision**	**Computational Cost**	**Ref.**	**Advantages**	**Disadvantages**

Space Discretized Approach (Eulerian)	Fluent	High	High	[17,32,44,49]	High accuracy	High computing time
					Used for stationery and rotating simulations	Large computer memory required
					Suitable for final design studies	Commercial package
	CFX	High	High	[15,17,40,54]	High accuracy	High computing time
					Used mostly for rotating simulations	Large computer memory required
					Suitable for final design studies	Commercial package
Particle Representation Approach (Lagrangian)	FLS	Moderate	Very Low	[17–19]	Very quick solution	Not accurate
					Suitable for initial design studies	
					Used for optimization studies	
	HEL	Moderate	High	[41]	Moderate accuracy	Requires data from Eulerian approach
					Fairly fast solution	Very few research work
	SPH	Moderate	Moderate	[38,53,55]	Open Source	Moderately accurate
					Moderately quick solutions	
					Suitable for initial design studies	
					Accounts for neighbouring particles with help of kernel function	

Table 3 provides a summary of analysis modes and computational models found in the literature. SPH methods have been used three times to study stationery and rotating blade conditions but optimization attempts have not been carried out. It can also be seen that there have been only 3 attempts to optimize the bucket design using artificial intelligence methods.

Table 3: Overview of Analysis Modes and Computational Models on Pelton Bucket

Publication		**Analysis Modes**				**Computational Models**								**Optimization**
						Eulerian			**Lagrangian**					
Paper	**Year**	**Stationery**	**Rotating**	**No. of Buckets**	**Experimental**	**CFX**	**Fluent**	**Others**	**FLS**	**SPH**	**HEL**	**ALE**	**Others**	**Type**
Zeng et al [56]	2018		Yes	19	No	Yes								No
Budiarso et al [57]	2018	Yes				Yes								Analytical
Vessaz et al [58]	2016		Yes	22	No								FVPM	Stochastic
Panthee et al [54]	2014		Yes	3	Yes	Yes								No
Solemslie and Dahlhaug [59]	2014		Yes	1	Yes									Analytical
Vessaz et al [21]	2014	Yes	Yes	1/5	Yes		Yes					Yes		No
Zidonis and Aggidis [15]	2014		Yes	All	Yes	Yes								No
Zidonis et al [17]	2014	Yes		1	Yes	Yes	Yes		Yes					Stochastic
Furnes [55]	2013	Yes		1	Yes					Yes				No
Rossetti [41]	2013		Yes	3	No	Yes					Yes			No
Souari and Hassairi [49]	2013		Yes	3	No		Yes							No
Xiao et al [60]	2013		Yes	All	No		Yes							No
Anagnostopoulos and Papatonis [18]	2012		Yes	1	Yes				Yes					Stochastic

Barstad [46]	2012		Yes	3	Yes	Yes								No
Gupta and Prasad [33]	2012	Yes		1	No	Yes								No
Xiao et al [45]	2012		Yes	All	No		Yes							No
Jost et al [61]	2010		Yes	All	Yes	Yes								No
Patel et al [62]	2010		Yes	All	No		Yes							Experimental
Marongiu et al [53]	2010		Yes	1	No	Yes				Yes				No
Stamatelos et al [63]	2010		Yes	All	Yes									No
Nakanishi et al [39]	2009	Yes		1	Yes								MPS	No
Santolin et al [51]	2009		Yes	All	Yes	Yes								No
Jost et al [64]	2008		Yes	4	Yes	Yes								No
Marongiu et al [38]	2007	Yes		1	Yes	Yes				Yes				No
Perrig [10]	2007		Yes	5	Yes		Yes							No
Anagnostopoulos and Papatonis [19]	2006		Yes	1	Yes				Yes					Stochastic
Perrig et al [40]	2006		Yes	5	Yes		Yes							No
Zoppe et al [32]	2006	Yes		1	Yes		Yes							No
Matthias and Promper [31]	2004	Yes	Yes	5	Yes		Yes							No
Kvicinsky [44]	2002		Yes	1	Yes	Yes								No
Vesely and Varner [65]	2001		Yes	1	Yes		Yes							Experimental
Avellan et al [27]	1998	Yes		1	Yes		Yes							No
Janetzky et al [30]	1998	Yes		1	No			Yes						No

4 Design Optimisation Strategies

There have been plenty of studies on the flow mechanism within a Pelton turbine bucket as described in the previous sections. These studies have been able to numerically model the interaction and the transfer of energy from the water to the turbine satisfactorily. There have been studies that concentrated on other aspects of hydroelectricity production optimization such as affordability [66], institutional barriers [7] and wear [67,68]. Nirmal et. al. [69] numerically analysed the flow in a cross-flow hydro turbine to optimize its performance by geometrically modifying the several parameters such as the nozzle shape, the guide vane angle, the number of runner blades where simulations were carried out individually while Jiyun et. al. [70] concentrated only on the attack angle. Pujol et. al. [71] suggested and tested an improved design for the ancient Spanish watermill with an improvement of over 44%. Also, effects of nozzle diameter, jet inlet angle, number of blades and the rotational speed of the turbine were analysed by Gaiser et. al. [72] experimentally to identify the optimal combinations of these parameters to improve efficiency of Turgo turbine. Williamson et al [13] studied the optimal inclination of the jet for a low head pico-hydro Turgo turbine using MATLAB for analytical calculations. Sutherland et. al. [73] examined the effect of lateral and stream-wise turbine spacing and concluded staggered array with decreased streamwise spacing as the optimal setting for a crossflow turbine. Mustafayev et.al. [20] concluded that the performance of Francis turbine and Pelton turbine decreased by 19% and 9% respectively when turbine RPM was adjusted to the flow rate. On the contrary, Kaplan turbine performance increased upto 6% under similar conditions. All these attempts were based on the experiences and preconception of the experts and some extrapolation on the ideas they held. Similar trial and error based methods for optimisation of the bucket profile to extract more energy might be time consuming and not able to produce the best results; hence, we require a more exhaustive but efficient approach.

Most optimisation problems have a random solution and the necessity is to find the global optimal solutions at the shortest time period. These methods can be classified as traditional interpolation methods and artificial intelligence methods. Traditional methods include analytical methods such as Simplex methods, Conjugate gradient methods [74], Metamodeling methods [75], Integer Programming [76] etc , whereas artificial intelligence methods include application of Neural networks [77] and Genetic Algorithms (GA) [78].

4.1 Traditional methods in design optimization

The analytical methods used are entirely numerical methods based on theories and formulae of fluid dynamics and mechanics. Researchers have made use of the computational power of modern day computers to solve tedious design optimization problems using classical numerical methods. These methods have mostly been used for structural optimization problems as they have regular geometric elements and proven mathematical formulae that can provide simple and accurate results. Simplex methods and Conjugate gradient methods are the simplest forms of optimization for linear functions. These methods can be used for problems with multiple constraints and parameters. These methods usually search for the optimal results in a linear fashion, limiting themselves to local optima. For nonlinear optimization problems, this approach requires diverse samples and approximation to generate best results. Tapia et. al. [76] used integer programming to minimize the costs for civil work relating to a microhydro-power plant. The environmental conditions were taken as the constraints and generated electric power, water flow, excavations and supports, piping costs were optimized using established mathematical functions. The method is simple handles only 12 functions. It would not be able to handle design optimization problems of continuous and large domain. In real life design optimization scenarios, the solutions are usually non-linear with multiple variables and functions hence, more robust methods are desired.

For structural design, the analysis codes such as FEA and CFD pose a complex problem. These problems cannot be solved with the simple linear solvers owing to their complexity and computational demands. Metamodelling methods are simplified approximations of such complex problems. This approach approximates the complex computational functions into simple analytical models known as metamodels. In this approach, low fidelity models are corrected using response values from both high and low fidelity models with minimum number of function calls. This correction can be adapted for optimization process [79]. The benefits of metamodel based optimization have been listed as: 1) ease to connect simulation codes, 2) parallel computation for same simulation of design points, 3) filters numerical noise better than gradient-based methods, 4) renders entire design space and 5) is easier to detect errors [75]. These models have been used for crashworthiness design with 2, 11 and 20 variables [80], oil tanker design with 6 inputs, 14 outputs and 50 function evaluations [81], intake scoop design of helicopter's engine cooling bay with 5 inputs and 45 function evaluations [82] etc. The issues concerning this technique can be listed as [75]: First, computation expenses are exponential to the number of design variables in metamodelling. 'The curse of dimensionality' makes this approach unattractive for problems with large variables. Second, the outcome of metamodelling depends largely on the initial sampling of the problem. If the functions are considered 'black-box', the best sample and the sample size cannot be determined. Hence, progressive and intelligent sampling techniques need to be developed to achieve the best samples. Last, since it is an approximation technique, there is always some uncertainity in the results. Hence, a lot of work still remains to be done to make this method viable and attractive to desing optimization problems with higher number of variables.

Vessaz et al [58] presented a strategy for optimization for performance of Pelton turbine bucket. The parametric model of the bucket was developed with four bicubic Bezier patches defined by 21 free parameters. Finite volume particle method was employed for CFD simulations for the advantage of Lagrangian Eulerian formulation. The optimization problem classified as High-dimension with Expensive Black-box function was optimized with cubic multivariate adaptive regression (MAR) spline surrogate model. 2000 sample buckets were developed by the aid of Halton sequence which would explore wide range of possibilities. Clustering of the parameters into four groups was attempted to decompose the problem to smaller dimensions for simpler and faster outcomes. The work focused more on efficient exploration of the design space and does not explicitly mention the improvement achieved in the performance as compared to the existing designs.

Budiarso et al [57] tested 4 different types of cut-outs namely w, v, u and ω. Simulations on these cut-out types were run with a popular CFD solver at different flow velocities and different bucket positions. Torque, rotational velocity and power generated at different angles at different flow rates were analysed via ANOVA block design. Qualitative studies were made based on Coanda effect, backpressure and ease of manufacture. U type cut-out was evaluated as the best design. This research studies the cut-out designs in stationery condition and hence is not able to provide real operating performance of the cut-out.

4.2 Artificial Intelligence in Pelton bucket Design

Traditional methods [75,83–85] and the Neural network [75,77,78] methods utilise the gradient function that relates the objective function with design variables and finds the best solution by interpolating the values in the gradient function. The gradient function is generated from earlier experiments and previous data are used to train the learning engine of these systems. The neural network is particularly fast in generating solutions but it requires enough training and also is limited to the scope of the gradient function and is unable to extrapolate for results with the same accuracy.

Although GA requires more computing effort than gradient projection methods or other local search methods, it does not require gradients of the objective function and constraints with respect to the set of design variables. Actual calculations or simulations are carried out for each set of design variables. The efficiency of the GA can be improved by monitoring previously analysed design data and avoiding re-computing for the same set of variables. Various modifications to the population size, mutation probability and selection of individuals can be made to further improve the efficiency [86]. Plenty of other mechanical design experiments have been conducted for antenna design [87,88], vehicle suspension design [89], structural optimisation [78,88] etc. using GA. Hence, GA is one of the best tools for optimisation of the design of Pelton turbine bucket.

Recently, development of computing capabilities has encouraged studies on optimising the manifold, nozzles and the jet numerically [24,62,90]. Some studies have used computational methods to optimise the bucket profile of the Pelton turbine [17,18]. These studies were based on FLS for tracking the flow during a water-bucket interaction at a very low computational cost. Evolutionary genetic algorithms were used for optimisation as it allows introduction of more design variables as compared to traditional optimisation methods. The genetic algorithms are able to solve complex non-linear and multi-parametric problems [91], as required for optimisation of a bucket. The algorithm has the freedom to select the values of the design parameters within the prescribed range and then search for the set of parameters that maximises the efficiency [86].

Anagnostopoulos et. al. [18] used Non-Uniform Rational B-Splines (NURBS) method for the parameterization of the bucket inner surface. They used 15 parameters to design the bucket rim and an additional 3 parameters to develop the lateral bucket surface and one more parameter was introduced to control the radial location of the bucket illustrated in figure 15. The study concluded that the hydraulic efficiency of the turbine depends more on its main bucket dimensions (length, width and depth) than on the shape of the rim or the lateral surface pattern. The genetic optimisation algorithm provided a design that generated 3% higher efficiency.

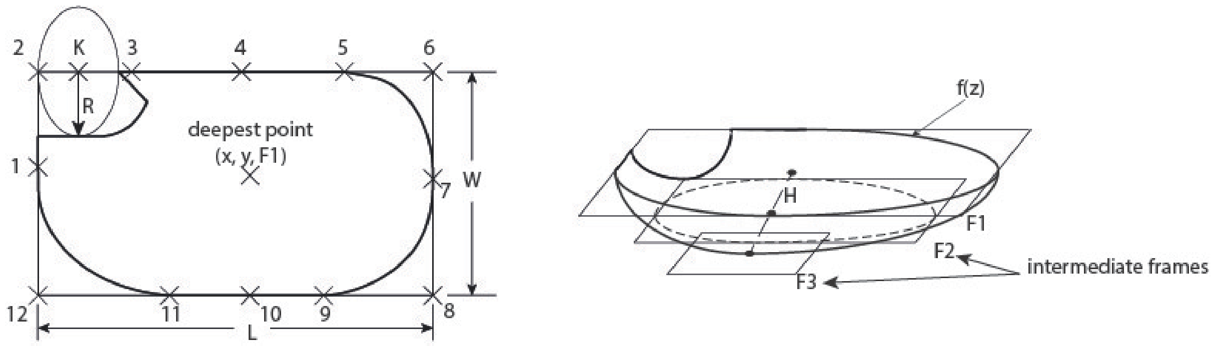

Figure 15: Design parameters and control points on bucket rim (a), parameterization of the lateral bucket surface (b) [18]

Zidonis et al [17] identified 15 design parameters that were divided into groups as parameters defining the basic dimensions, parameters defining the exact position and parameters related to the shape of the inner surface of the bucket. The final design obtained 6.8% improvement in the total efficiency. The author also identified 11 other parameters of interest that could be studied individually to analyse their effect on the total efficiency of the turbine. Some parameters such as bucket length to width ratio, bucket depth to width ratio, exit angle, inclination angle and pitch diameter were found to have a greater effect on the efficiency of the turbine. He suggested the outcome could be improved with a Eulerian method to confirm the performance of the bucket and to finalise the design. Table 4 provides the summary of the attempts that have been made to optimize the Pelton bucket.

Table 4: Comments on Bucket Optimization Attempts

Paper	**Year**	**Optimization**	**Bucket Parameters**	**Reverse Side**	**Efficiency**	**Remarks**
Budiarso et al [57]	2018	Analytical	Cut-out	No	1.3Nm increase	4 different types of cut-out were tested at different incident angles for torque and u type cut-out performed the best.

Vessaz et al [58]	2016	Analytical	Studied	No	10.3Nm increase	4 NURBS patches were combined to generate a bucket which were clustered to reduce dimensions for MAR based optimization with 2000 samples
Solemsile and Dahlhaug [59]	2014	Analytical	Studied	No	77.75%	Studies the deepest surface on the bucket profile using analytical mathematical formulas
Zidonis et al [17]	2014	Stochastic	Studied	No	6.8% increase	Studied each design parameter's sensitivity and separately applied genetic algorithms for optimisation of inner bucket profile maintaining the ellipsoidal shape
Anagnostopoulos and Papatonis [18]	2012	Stochastic	Studied	Partial	3% increase	Applied genetic algorithms for optimisation to inner bucket profile maintaining the ellipsoidal shape
Anagnostopoulos and Papatonis [19]	2006	Stochastic	Studied	No	3% increase	Applied evolutionary genetic algorithms for optimisation to inner bucket profile
Patel et al [62]	2010	Experimental	No	No	NA	Tested design changes based on experimental observations
Vesely and Varner [65]	2001	Experimental	No	No	1.4% increase	Tested design changes based on numerical calculations

5 Discussion

The basic working principle of the Pelton turbine exhibits bucket reverse side splashing, atomization of the jet, interference of water layers, secondary flow within the jet and gravity deviation of the water etc. These aspects play a major role in estimating efficiency of the turbine. CFD methods utilise the processing capabilities of the computers that can analyse and simulate water-bucket interaction with significant accuracy. It is evident in Table 3 and Table 4 that analysis of water bucket interface plays very important role in estimating performance of the turbine. So far, inner surface of the bucket has been analysed by many researchers. However, few attempts have been made to study the interaction of inner bucket surface and the reverse side surface of the bucket with the water aiming to improve the overall efficiency of the turbine.

5.1 Flow environment representation

As an angular momentum variation due to the water-bucket interaction determines the turbine energy exchange, the key aspect for improvement of the Pelton turbine performance is the analysis of the flow on the bucket surface. But due to the presence of unsteady flows developing in the rotating frame of reference, moving source and FSF simultaneously, it is a great challenge to accurately evaluate energy transfer from the jet to the turbine bucket [41]. Although stationery bucket assumptions were developed for simplicity, it does not provide the complete picture of the complex interactions among the fluid particles and the bucket surface in an operational Pelton turbine. To account for the Coanda and centrifugal effects along with unsteady feeding and splashing, rotating bucket simulations are a must. Some of these issues are investigated in stationery condition at certain angular positions; however collective effects could shed light on the overall efficiency of the Pelton system.

5.2 Flow simulation

The CFD methods itself are developing and much needs to be done in the area of FSF as the fluids show volume based properties as well as particle based properties. The Eulerian method alone has difficulties in calculating the free surface boundary due to loss of consistency of the interpolation domain and hence limits the study of their influence on bucket surface. The Lagrangian method views fluid flow as the flow of particles and hence can easily define a free surface. This helps to identify the trajectory of the fluid particles and calculate the angular momentum variation at the water-bucket interaction at the cost of overall accuracy.

Some authors have made use of combined Eulerian and Lagrangian methods to obtain satisfactorily accurate results. SPH is one such method that solves the system of Euler equations in Lagrangian formalism, binding the neighbouring particles with a user defined kernel function. This kernel function allows the neighbouring particles to infer the properties of the central particle thus allowing accounting for artificial viscosity as shown in figure 16. This adds to the accuracy of the SPH method as compared to the traditional Lagrangian methods with the computational time that is acceptable for optimisation engine.

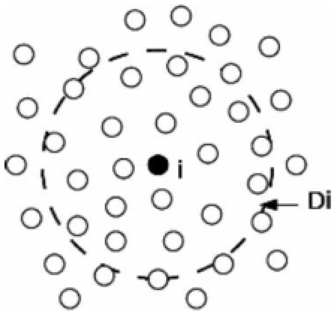

Figure 16: Interpolation domain for SPH where i is the central particle and Di is the diameter of the of the smoothing function [38]

5.3 Optimization strategies

The analytical methods were used with well-established formulae and determined the best combinations of various aspects of the hydropower system to improve the overall efficiency. These methods work with the conventional designs and do not provide radical improvement to the overall design of the system. Such strategies are not equipped to suggest changes to the bucket profile design. While analysing bucket surface interactions and associated efficiency loss, it is possible to optimise the surface parameters and determine relations between most responsive parameters and the efficiency of the Pelton system. Analytical methods lack this study of bucket surface interaction with the water jet. The experimental optimization attempts capitalized on the experience of the researchers and their understanding of the jet bucket interaction. Their trial design backed by some experiments and some mathematical logic were tested. This 'trial and error' method has the potential to provide radical improvements to the existing design of the bucket profile but, coming up with a radical change in design that would immensely improve the performance is very difficult. An improvement of 1.4% [65] was observed from the 'trial and error' optimization method. Lately, the study by Vessaz et al [58] has attempted to study the jet bucket interaction but has failed to provide an improved design for the bucket.

The literature review revealed the stochastic optimisation studies used the same CFD technique of FLS and have limited the study to the ellipsoidal inner surface of the bucket and the bucket's relative position. In addition, the profile of the bucket has been analysed with limited shape governing variables such as the length, width, depth, pitch radius, tip radius etc. as discussed by Thake [92] and Solemslie [59]. The reason being the difficulty for addressing the constraints necessary for a Pelton bucket geometry while using a NURBS surface as highlighted by Michalkova and Bastl [93] stating that ' for a given B-spline curve, the exact solution exists only in very special cases.' This reserves scope for designing a random responsive surface in order to establish a link between surface design variables and resulting efficiency of the turbine. Yet, this approach has been able to provide maximum improvement of 6.8% [17]. This shows that stochastic optimization strategy can provide greater improvement in the performance of the Pelton buckets.

6 Conclusion

This paper explored the potential to increase renewable energy production from hydro turbines. We examined the technological development in the fields of computing that has made the CFD more capable of handling problems associated with FSF, splashing effects, Coriolis effects and unsteady feeding that are essential to analyse the flow environment in Pelton Turbine. Alternative approaches that can be used to simulate the flow in the turbine bucket were discussed. Attempts made to optimise the bucket profile were analysed as summarized in table 3 and 4. An in-depth comparison of the performance of Eulerian volume-based approaches and Lagrangian trajectory-based methods were made in order to study the water-bucket interaction and were summarized in table 2. This comparison revealed compelling trend for adopting Hybrid methods such as SPH for tracking the particles and studying their energy transfer. The popular Eulerian methods are best to study the volumetric effects and are computationally very expensive. This paper dissected the achievements and contributions available in literature for flow in a Pelton turbine bucket to identify the scope for improvement and further research in the area. The segregated analysis of strategies adopted for flow environment modelling, flow simulation, optimization strategies and the research gaps on these strategies as summarized in table 3 provides a complete picture for further research in improving the performance of the turbine with existing resources.

One of the identified areas for the study is bucket profiles other than ellipsoidal as seen in table 4. The three attempts made to redesign the bucket profile have tried to change the depth, width and the length of the bucket but have not considered changing the bucket profile to circular, parabolic or any other than ellipsoidal. They have obtained an increase in efficiency up to 6.8%. The other study area is the surface on the reverse side of the bucket as it affects the overall efficiency due to the splashing and windage effects. With the use of novel trends captured in this paper for modelling CFD problems, it is now possible to explore above areas for enhancing Pelton turbine efficiency. Even an increase in 1% in the efficiency of Pelton turbines would lead to hundreds of megawatts of added renewable energy from the existing resources worldwide without large capital costs.

7 Outlook and Summary

After this extensive study, few recommendations can be made for aspiring engineers and researchers. Stochastic optimization methods have proven to be very successful even though they are computationally expensive. Combined Lagrangian-Eulerian methods have been developed that are relatively accurate and less computationally expensive. These methods are capable of providing simulation results for rotating bucket conditions with reasonable accuracy in acceptable time duration. The studies conducted have focused on ‘trial and error’ design improvement with analytical methods or with very limited shape governing parameters for stochastic optimization methods. The way forward for optimization of Pelton turbine buckets is with stochastic optimization methods with shape governing variables that are capable of accommodating varied shapes and angles.

Acknowledgement

The first author likes to acknowledge the support provided by the EU Erasmus Mundus project SmartLink (552077-EM-1-2014-1-UK-ERA) to carry out this research at the University of the West of Scotland, UK.

References

[1] Aggidis GA. Performance Envelopes of Hydro Turbines 2010. http://www.engineering.lancs.ac.uk/lureg/nwhrm/engineering/ (accessed December 15, 2015).

[2] Bajracharya TR, Acharya B, Joshi CB, Saini RP, Dahlhaug OG. Sand Erosion of Peltion Turbine Nozzles and Buckets: A Case Study if Chilime Hydropower Plant. Wear 2008;264:177–84. doi:10.1016/j.wear.2007.02.021.

[3] Okot DK. Review of small hydropower technology. Renew Sustain Energy Rev 2013;26:515–20.

[4] Jawahar CPP, Michael PA. A review on turbines for micro hydro power plant. Renew Sustain Energy Rev 2017;72:882–7. doi:https://doi.org/10.1016/j.rser.2017.01.133.

[5] Zidonis A, Aggidis GA. State of the Art in Numerical Modelling of Pelton Turbines. Renew Sustain Energy Rev 2015;45:10. doi:10.1016/j.rser.2015.01.037.

[6] World Energy Council. World Energy Resources: Hydropower. World Energy Council; 2016.

[7] Yah NF, Oumer AN, Idris MS. Small scale hydro-power as a source of renewable energy in Malaysia: A review. Renew Sustain Energy Rev 2017;72:228–39. doi:10.1016/j.rser.2017.01.068.

[8] Nasir BA. Design of High Efficiency Pelton Turbine for MicroHydro Power Plant. Int J Electr Eng Technol 2013;4:171–84.

[9] Bakis R. Electricity production opportunities from multipurpose dams (case study). Renew Energy 2007;32:1723–38. doi:https://doi.org/10.1016/j.renene.2006.08.008.

[10] Perrig A, Farhat PFADM. Hydrodynamics of the Free Surface Flow in Pelton Turbine Buckets. Ecole Polytechnique Federale De Lausanne, 2007.

[11] 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. doi:10.1243/09576509JPE294.

[12] Sick M, Keck H, Parkinson E, Vullioud G. New Challenges in Pelton Research. Proc. HYDRO 2000 Conf., Bern: 2000.

[13] Williamson SJ, Stark BH, Booker JD. Performance of a low-head pico-hydro Turgo turbine. Appl Energy 2013;102:1114–26. doi:https://doi.org/10.1016/j.apenergy.2012.06.029.

[14] Zidonis A, Benzon DS, Aggidis GA. Development of Hydro Impulse Turbines and new Opportunities. Renew Sustain Energy Rev 2015;51:12. doi:10.1016/j.rser.2015.07.007.

[15] Židonis A, Aggidis GA. Pelton turbine: Identifying the optimum number of buckets using CFD. J Hydrodyn 2016;28:75–83. doi:10.1016/S1001-6058(16)60609-1.

[16] Shogenji K, Inada T. On the Inclination of the Ridge of Buckets on a Pelton Wheel. Mem Coll Eng 1927;4:13.

[17] Zidonis A, Panagiotopoulos A, Aggidis GA, Anagnostopoulos JS. Parametric Optimization of Two pelton Turbine Runner Designs Using CFD. J Hydrodyn 2014;27:10. doi:10.1016/S1001-6058(15)60498-X.

[18] Anagnostopoulos JS, Papantonis DE. A Fast Lagrangian Simulation Method for Flow Analysis And Runner Design in Pelton Turbines. J Hydrodyn 2012;24:12. doi:10.1016/S1001-6058(11)60321-1.

[19] Anagnostopoulos JS, Papantonis DE. A numerical methodology for design optimization of Pelton turbine runners. HYDRO 2006 2006:25–7.

[20] Mustafayev R. I.and Hasanova LH and MMM. Using Regulated Electrical Machines in Small Hydropower Plants Operating in a Power Network. Russ Electr Eng 2018;89:322–7. doi:10.3103/S1068371218050061.

[21] Vessaz C, Jahanbakhsh E, Avellan F. Flow simulation of a Pelton bucket using finite volume particle method. IOP Conf. Ser. Earth Environ. Sci., vol. 22, Ljubljana: IOP Publishing; 2014.

[22] Laghari JA, Mokhlis H, Bakar AHA, Mohammad H. A comprehensive overview of new designs in the hydraulic, electrical equipments and controllers of mini hydro power plants making it cost effective technology. Renew Sustain Energy Rev 2013;20:279–93. doi:https://doi.org/10.1016/j.rser.2012.12.002.

[23] Catanese A, Barglazan M, Hora C. Numerical simulation of a free jet in pelton turbine. 6th Int Conf Hydraul Mach Hydrodyn 2004:6.

[24] Sadlo F, Parkinson E. Vorticity Based Flow Analysis and Visualization for Pelton Turbine Design Optimization. IEEE Vis 2004:8. doi:0-7803-8788-0/04/$20.00.

[25] Shortridge RW. Lestor Pelton and His Water Wheel. Hydro Rev 1989:4.

[26] Brekke H. A general study on the design of vertical Pelton turbines. Turboinstitut, 1984.

[27] Avellan F, Dupont P, Kvicinsky S, Chapuis L, Parkinson E, Vullioud G. Flow calculations in Pelton Turbines, Part 2: Free Surface Flows. Proc. 19th IAHR Symp., Singapore: 1998.

[28] Muggli F, Zhang Z, Schärer C, Geppert L. Numerical and Experimental Analysis of Pelton Turbine Flow, Part 2 the Free Surface Jet Flow. XX IAHR Symp. Charlotte, NC, 2000.

[29] Sick M, Schindler M, Drtina P, Scharer C, Keck H. Numerical and Experimental Analysis of Pelton Turbine Flow Part 1: Distributor and Injector. Proc. XX IAHR Symp., Charlotte: 2000.

[30] Janetzky B, Göde E, Ruprecht A, Keck H, Schärer C. Numerical simulation of the flow in a Pelton bucket. Proc. 19th IAHR Symp., 1998, p. 276–83.

[31] Matthias H-B, Promper O. Numerical simulation of the free surface flow in Pelton turbines. 6th Int. Conf. Hydraul. Mach. Hydrodyn., vol. 6, Timisoara, Romania: University of Timisoara; 2004, p. 119–24.

[32] Zoppe B, Pellone C, Maitre T, Leroy P. Flow Analysis Inside a Pelton Turbine Bucket. J Turbomach 2006;128:12. doi:10.1115/1.2184350.

[33] Gupta V, Prasad V. Numerical investigations for jet flow characteristics on pelton turbine bucket. Int J Emerg Technol Adv Eng 2012;2:364–70.

[34] Grozev G, Obretenov V, Trifonov T. Investigation of the Distribution of Pressure Over the Buckets of a Pelton Turbine. Proc. Conf. Hydraul. Mach. Flow Meas., Turboinstitut, Ljubljana, Yugoslavia: 1988, p. 119–25.

[35] Kvicinsky S, Kueny J-L, Avellan F. Numerical and Experimental Analysis of free Surface Flow in

a 3D non Rotating Pelton Bucket. Proc XXIst IAHR Symp Hydraul Mach Syst 2002.

[36] Bhattarai S, Vichare P, Mishra B. CFD based stochastic optimization of Pelton Turbine bucket in stationery condition. 9th Int Conf Mech Aerosp Eng 2018:53–7.

[37] Guilbaud M, Houdeline JB, Philibert R. Study of the Flow in the Various Sections of Pelton Turbine. Proc. 16th IAHR Symp. Hydraul. Mach. Cavitation, Associacao Brasileira de Recursos Hidricos, Sao Paulo: 1992, p. 819–31.

[38] Marongiu JC, Leboeuf F, Parkinson E. Numerical simulation of the flow in a Pelton turbine using the meshless method smoothed particle hydrodynamics: a new simple solid boundary treatment. Proc Inst Mech Eng Part A J Power Energy 2007;221:849–56.

[39] Nakanishi Y, Fujii T, Kawaguchi S. Numerical and experimental investigations of the flow in a stationary Pelton bucket. J Fluid Sci Technol 2009;4:490–9.

[40] Perrig A, Avellan F, Kueny J-L, Farhat M, Parkinson E. Flow in a Pelton Turbine Bucket: Numerical and Experimental Investigations. J Fluids Eng 2006;128:9. doi:10.1115/1.2170120.

[41] Rossetti A, Pavesi G, Cavazzini G, Santolin A, Ardizzon G. Influence of the Bucket Geometry on the Pleton Performance. J Power Energy - Proc Inst Mech Eng 2014;228:13. doi:10.1177/0957650913506589.

[42] Bhattarai S, Vichare P, Dahal K. Pelton turbine bucket flow analysis and visualization for evaluation of area-wise contribution. 8th Int Conf Sustain Energy Environ Prot 2018;3:367–72.

[43] Hana M. A Discussion on Numerical Simulation in Pelton Turbines. Proc. 19th IAHR Symp., World Scientific, Singapore: 1998, p. 306–15.

[44] Kvicinsky S, Kueny J-L, Avellan F, Parkinson E. Experimental and Numerical Analysis of Free Surface Flows in a Rotating Bucket. Proc. XXIst IAHR Symp. Hydraul. Mach. Syst., Laboratory for Hydraulic Machines, Lausanne: 2002, p. 359–64.

[45] Xiao YX, Cui T, Wang ZW, Yan ZG. Numerical simulation of unsteady free surface flow and dynamic performance for a Pelton turbine. IOP Conf. Ser. Earth Environ. Sci., vol. 15, Ljubljana: IOP Publishing; 2012.

[46] Barstad LF, Dahlhaug OG. CFD analysis of a Pelton turbine. Norwegian University of Science and Technology, 2012.

[47] Liu X, Luo Y, Karney BW, Wang W. A selected literature review of efficiency improvements in hydraulic turbines. Renew Sustain Energy Rev 2015;51:18–28. doi:https://doi.org/10.1016/j.rser.2015.06.023.

[48] Zhang J, Cai S, Li Y, Zhou X, Zhang Y. Optimization design of multiphase pump impeller based on combined genetic algorithm and boundary vortex flux diagnosis. J Hydrodyn Ser B 2017;29:1023–34. doi:https://doi.org/10.1016/S1001-6058(16)60816-8.

[49] Souari L, Hassairi M. Numerical Simulation of the Flow Into a Rotating Pelton Bucket. Int J Emerg Technol Adv Eng 2013;3:67–75.

[50] Du J, Yang H, Shen Z, Chen J. Micro hydro power generation from water supply system in high rise buildings using pump as turbines. Energy 2017;137:431–40. doi:https://doi.org/10.1016/j.energy.2017.03.023.

[51] Santolin A, Cavazzini G, Ardizzon G, Pavesi G. Numerical Investigation of the Interaction Between Jet and Bucket in a Pelton Turbine. J Power Energy - Proc Inst Mech Eng 2009;223:8. doi:10.1243/09576509JPE824.

[52] Wang T, Wang C, Kong F, Gou Q, Yang S. Theoretical, experimental, and numerical study of special impeller used in turbine mode of centrifugal pump as turbine. Energy 2017;130:473–85. doi:https://doi.org/10.1016/j.energy.2017.04.156.

[53] Marongiu J-C, Leboeuf F, Caro J, Parkinson E. Free surface flows simulations in Pelton turbines using an hybrid SPH-ALE method. J Hydraul Reseearch 2010;48:10. doi:10.3826/jhr.2010.0002.

[54] Panthee A, Neopane HP, Thapa B. CFD Analysis of Pelton Runner. Int J Sci Res Publ 2014;4:1–6.

[55] Furnes K. Flow in Pelton turbines. Norwegian University of Science and Technology, 2013.

[56] Zeng C, Xiao Y, Luo Y, Zhang J, Wang Z, Fan H, et al. Hydraulic performance prediction of a prototype four-nozzle Pelton turbine by entire flow path simulation. Renew Energy 2018;125:270–82. doi:https://doi.org/10.1016/j.renene.2018.02.075.

[57] Budiarso B, Warjito W, Adanta D, Syah N, Vohra H. Cutout Types Analysis on Pico Hydro Pelton Turbine. Int J Adv Sci Eng Inf Technol 2018;8:2024–30.

[58] Vessaz C, Andolfatto L, Avellan F, Tournier C. Toward design optimization of a Pelton turbine runner. Struct Multidiscip Optim 2016;55:37–51. doi:10.1007/s00158-016-1465-7.

[59] Solemslie BW, Dahlhaug OG. A reference pelton turbine - design and efficiency measurements. 27th IAHR Symp Hydraul Mach Syst 2014:10. doi:10.1088/1755-1315/22/1/012004.

[60] Ye-xiang X, Feng-qin H, Jing-lin Z, Takashi K. Numerical prediction of dynamic performance of Pelton turbine . J Hydrodyn 2007;19:9.

[61] Jošt D, Mežnar P, Lipej A. Numerical prediction of Pelton turbine efficiency. IOP Conf Ser Earth Environ Sci 2010;12.

[62] Patel K, Patel B, Yadav M, Foggia T. Development of Pelton Turbine Using Numerical Simulation. IOP Conf Ser Earth Environ Sci 2010;12.

[63] Stamatelos FG, Anagnostopoulos JS, Papantonis DE. Performance Measurements on a Pelton Turbine Model. J Power Energy - Proc Inst Mech Eng 2010:12. doi:10.1177/2041296710394260.

[64] Jost D, Lipej A, Meznar P. Numerical prediction of efficiency, cavitation and unsteady phenomena in water Turbines. ASME 2008 9th Bienn. Conf. Eng. Syst. Des. Anal., American Society of Mechanical Engineers; 2008, p. 157–66.

[65] Veselý J, Varner M. A case study of upgrading of 62.5 MW Pelton turbine. Proc. Int. Conf. IAHR, 2001.

[66] Kadier A, Kalil MS, Pudukudy M, Hasan HA, Mohamed A, Hamid AA. Pico hydropower (PHP) development in Malaysia: Potential, present status, barriers and future perspectives. Renew Sustain Energy Rev 2018;81:2796–805. doi:https://doi.org/10.1016/j.rser.2017.06.084.

[67] Padhy MK, Saini RP. A review on silt erosion in hydro turbines. Renew Sustain Energy Rev

2008;12:1974–87. doi:https://doi.org/10.1016/j.rser.2007.01.025.

[68] Zhang Y, Zhang Y, Qian Z, Ji B, Wu Y. A review of microscopic interactions between cavitation bubbles and particles in silt-laden flow. Renew Sustain Energy Rev 2016;56:303–18. doi:https://doi.org/10.1016/j.rser.2015.11.052.

[69] Acharya N, Kim C-G, Thapa B, Lee Y-H. Numerical analysis and performance enhancement of a cross-flow hydro turbine. Renew Energy 2015;80:819–26. doi:https://doi.org/10.1016/j.renene.2015.01.064.

[70] Jiyun D, Zhicheng S, Hongxing Y. Numerical study on the impact of runner inlet arc angle on the performance of inline cross-flow turbine used in urban water mains. Energy 2018;158:228–37. doi:https://doi.org/10.1016/j.energy.2018.06.033.

[71] Pujol T, Solà J, Montoro L, Pelegrí M. Hydraulic performance of an ancient Spanish watermill. Renew Energy 2010;35:387–96. doi:https://doi.org/10.1016/j.renene.2009.03.033.

[72] Gaiser K, Erickson P, Stroeve P, Delplanque J-P. An experimental investigation of design parameters for pico-hydro Turgo turbines using a response surface methodology. Renew Energy 2016;85:406–18. doi:https://doi.org/10.1016/j.renene.2015.06.049.

[73] Sutherland D, Ordonez-Sanchez S, Belmont MR, Moon I, Steynor J, Davey T, et al. Experimental optimisation of power for large arrays of cross-flow tidal turbines. Renew Energy 2018;116:685–96. doi:https://doi.org/10.1016/j.renene.2017.10.011.

[74] Kannan BK, Kramer SN. An Augmented Lagrange Multiplier Based Method for Mixed Integer Discrete Continuous Optimization and Its Applications to Mechanical Design. J Mech Des 1994;116:405–11. doi:10.1115/1.2919393.

[75] Wang GG, Shan S. Review of Metamodeling Techniques in Support of Engineering Design Optimization. J Mech Des 2006;129:370–80. doi:10.1115/1.2429697.

[76] Tapia A, Millán P, Gómez-Estern F. Integer programming to optimize Micro-Hydro Power Plants for generic river profiles. Renew Energy 2018;126:905–14. doi:https://doi.org/10.1016/j.renene.2018.04.003.

[77] Madsen JI, Shyy W, Haftka RT. Response surface techniques for diffuser shape optimization. AIAA J 2000;38:1512–8. doi:10.2514/2.1160.

[78] Papadrakakis M, Lagaros ND, Tsompanakis Y. Structural optimization using evolution strategies and neural networks. Comput Methods Appl Mech Eng 1998;156:309–33. doi:http://dx.doi.org/10.1016/S0045-7825(97)00215-6.

[79] Timothy S, Vasilli T, Vladimir B, Felipe V. Design and Analysis of Computer Experiments in Multidisciplinary Design Optimization: A Review of How Far We Have Come - Or Not. 12th AIAA/ISSMO Multidiscip. Anal. Optim. Conf., American Institute of Aeronautics and Astronautics; 2008. doi:doi:10.2514/6.2008-580210.2514/6.2008-5802.

[80] Redhe M, Giger M, Nilsson L. An investigation of structural optimization in crashworthiness design using a stochastic approach. Struct Multidiscip Optim 2004;27:446–59.

[81] Golovidov O, Kodiyalam S, Marineau P, Wang L, Rohl P. A flexible, object-based implementation of approximation models in an MDO framework. Des Optim Int J Prod Process Improv 1999;1:388–404.

[82] Wang D, Naterer GF, Wang G. Thermofluid optimization of a heated helicopter engine cooling-bay surface. Can Aeronaut Sp J 2003;49:73–86.

[83] Youn BD, Choi KK, Park YH. Hybrid Analysis Method for Reliability-Based Design Optimization. J Mech Des 2003;125:221–32. doi:10.1115/1.1561042.

[84] Sobieszczanski-Sobieski J, Haftka RT. Multidisciplinary aerospace design optimization: survey of recent developments. Struct Optim 1997;14:1–23. doi:10.1007/bf01197554.

[85] Simpson TW, Mauery TM, Korte JJ, Mistree F. Kriging Models for Global Approximation in Simulation-Based Multidisciplinary Design Optimization. AIAA J 2001;39:2233–41. doi:10.2514/2.1234.

[86] Guo P, Wang X, Han Y. The enhanced genetic algorithms for the optimization design. 3rd Int Conf Biomed Eng Informatics 2010;3:5.

[87] Linden DS. Antenna Design Using Genetic Algorithm. GECCO, vol. 2, 2002, p. 1133–40.

[88] Robinson J, Sinton S, Rahmat-Samii Y. Particle swarm, genetic algorithm, and their hybrids: optimization of a profiled corrugated horn antenna. Antennas Propag. Soc. Int. Symp. 2002. IEEE, vol. 1, 2002, p. 314–7 vol.1. doi:10.1109/APS.2002.1016311.

[89] Baumal AE, McPhee JJ, Calamai PH. Application of genetic algorithms to the design optimization of an active vehicle suspension system. Comput Methods Appl Mech Eng 1998;163:87–94. doi:http://dx.doi.org/10.1016/S0045-7825(98)00004-8.

[90] Benzon D, Židonis A, Panagiotopoulos A, Aggidis GA, Anagnostopoulos JS, Papantonis DE. Impulse Turbine Injector Design Improvement Using Computational Fluid Dynamics. J Fluids Eng 2015;137. doi:10.1115/1.4029310.

[91] Gkoutioudi K, Karatza HD. A simulation study of multi-criteria scheduling in grid based genetic algorithms. 10 Th IEEE Int Symp Parallel Distrib Process with Appl 2012:8. doi:10.1109/ISPA.2012.48.

[92] Thake J. The micro-hydro Pelton turbine manual, design, manufacture and installation for small-scale hydro power. UK: ITDG Publishing; 2000.

[93] Michálková K, Bastl B. Imposing angle boundary conditions on B-spline/NURBS surfaces. Comput Des 2015;62:1–9. doi:https://doi.org/10.1016/j.cad.2014.10.002.