CFD based stochastic optimization of pelton turbine bucket in stationery condition

Bhattarai, Suyesh; Dahal, Keshav; Vichare, Parag; Mishra, Bhupesh

Published in:
2018 9th International Conference on Mechanical and Aerospace Engineering (ICMAE)

DOI:
10.1109/ICMAE.2018.8467607

Published: 20/09/2018

Document Version
Peer reviewed version

Bhattarai, S., Dahal, K., Vichare, P., & Mishra, B. (2018). CFD based stochastic optimization of pelton turbine bucket in stationery condition. In *2018 9th International Conference on Mechanical and Aerospace Engineering (ICMAE)* (pp. 53–57). IEEE. https://doi.org/10.1109/ICMAE.2018.8467607

CFD based stochastic optimization of Pelton turbine bucket in Stationery condition

Suyesh Bhattarai*, Keshav Dahal, Parag Vichare, Bhupesh Mishra
University of the West of Scotland
Scotland, UK
*suyesh.bhattarai@uws.ac.uk

***Abstract*—Computational fluid dynamics (CFD) and stochastic optimization are both highly computationally expensive processes. These processes may not produce the same unique result every time and demand large computing resources. The outcomes are determined as the final if the results repeat themselves for some predefined number of iterations causing convergence. Due to this expensive and non-deterministic nature, research on CFD optimization using stochastic optimization method such as Genetic Algorithm has been limited. This paper presents a noble method in which the CFD codes can be used together with genetic algorithm to optimize the shape of a responsive surface such as a Pelton turbine bucket. An existing Pelton bucket's model has been acquired and a set of random surfaces have been created as the initial population to optimize the shape of the bucket in stationery condition. The results show that an increase in efficiency by 13.21% to the normalized efficiency of existing design can be obtained by incorporating the changes suggested.**

Keywords-Pelton turbine; bucket design; Computational fluid dynamics; design optimization; genetic algorithms; turbomachinery

I. INTRODUCTION

Pelton is an impulse hydro-turbine that works at atmospheric pressure and has free surface flow in the turbine buckets. It is mostly used to generate electricity at conditions of high head and up to medium flow. The practical efficiency of the turbine is usually 82-85%, but in theory, up to 96% of the energy can be harnessed [1]. This gap in efficiency is partly due to the difficulty in analysing the flow because of the pressure losses, secondary flows, film flow etc. These problems have limited the studies to pressure measurement, visualization and water sheet thickness measurement.

In addition to the above reasons, the turbine's modelling and simulation using CFD methods has been difficult due to free surface flow, as it demands very fine mesh around the boundary layer for accurate modelling. Some studies have been made for analysis of flow within the bucket but they have been limited to accounting the pressure distribution and the thickness of the water sheet inside the bucket [2]. Due to this computationally expensive nature, the optimization attempts for Pelton buckets using traditional grid based approaches have been limited. Some attempts have been made for optimization using Lagrangian methods [3, 4] along with evolutionary algorithms but with limitation in accuracy and the number of parameters for optimization. This paper provides a noble study of combining a more accurate Smoothed Particle Hydrodynamics method with genetic algorithm to generate a more efficient Pelton bucket design at stationery bucket condition.

This paper discusses the previous studies in section 2, methodology in section 3, results in section 4 and conclusions in section 5.

II. LITERATURE

The complex nature of flow within the Pelton bucket has limited the number of optimization attempts made on it. Most attempts were experimental with trial error approach, but later a few attempts have been made with optimization algorithms.

Zidonis [3] studied the different features of the bucket such as bucket length to width ratio, bucket depth to width ratio, bucket exit angle, splitter inlet angle, splitter level, splitter tip angle, splitter tip geometry, backside of the splitter, inclination angle, and the number of buckets. He made CFD analysis of all these features individually and found out that bucket length to width ratio, bucket depth to width ratio, bucket exit angle, inclination angle, and number of buckets are sensitive parameters in the design of the a bucket. Small changes to these parameters keeping all other the same causes huge changes in the performance of the turbine. Yet the study fails to make assessment of the amount of energy transferred by each of the features of the bucket. The traditional NURBS approach applied by Zidonis [3] and Anagnostopoulos [4] limited the scope of exploration to shapes other than ellipsoidal. Also, NURBS has various parameters such as knots and weights which need to be carefully evaluated to produce valid surfaces that might not be efficient in stochastic applications. NURBS method weighs each coordinate point and forces C1 continuity, it could very easily overlook the changes suggested by the optimization process.

Zidonis in his same study [3] used the Genetic Algorithm for optimization of a Pelton turbine runner. For the maximization of the hydraulic efficiency of the runner, he used Fast Langrangian Solver, the parameterization tool and the Evolutionary Algorithms System (EASY). There were 15 free design parameters that were separated into three groups and investigated separately. The first group had 3 basic dimensions, second had remaining 10 parameters related with the shape and the inner surface and the third group had 2 parameters that defined the exact position of the bucket. The results show the enhanced performance of the optimized design by 6.8%. The performance was cross checked using the popular Fluent software.

The EASY software has the advantage of easy introduction of any number of design variables and is suitable for the complex non-linear and multi-parametric problems. Apart from the regular selection, crossover and mutation, Anagnostopoulos et. al. in his study [4], incorporated a penalty value on the fitness function to reduce the probability of further processing of unacceptable or lower fitness surface design encountered. From FLS based tests carried out individually for each parameter, it was found that the hydraulic efficiency of the runner depends more on its main bucket dimensions of length, width and depth than on the exact shape of the rim or the lateral surface pattern. The final optimization required all the design parameters to be analysed simultaneously due to the cost function discontinuities that were introduced by the geometric constraints. The results converged after 2000 simulations. The optimized design obtained 81% overall efficiency as compared to the 78% of the standard design.

An alternative method known as the Smoothed Particle Hydrodynamics has also been tested for accuracy in CFD applications. This method solves the system of Euler equations in Lagrangian formalism, treating the medium as weakly compressible by means of a discrete number of particles [5]. This method uses the physical properties of the particles lying in the surroundings to infer properties of each particle. A publication on modelling of a Turgo turbine, using this method claims that it has produced similar results as compared to Fluent faster and is hence a good alternative to the Eulerian methods [6].

III. Methodology

A HP Z230 computer with 3.4 GHz Intel Xeon E1245 processor and 16GB RAM was used as the workstation. A 2 GB NVIDIA Quadro 4000 GPU unit with 256 cores was used for the flow simulations using the CUDA enabled DualSPHysics solver. This ensured faster processing times for the simulations. The time to run one single simulation was less than one minute.

A. *Bucket surface parameterization*

The bucket surface parameterization is one of the most important aspects of this optimization attempt as its limitations and benefits could have big impact on the search space of the optimization engine. The ideal method would be able to generate surface from any random set of points such that any kind of surface could be tested. Also it should require the least number of parameters for reduced complexity and faster convergence towards the results. Along with the desire to incorporate every generated data point in the surface, the bucket was parametrized with the aid of Adapted Delaunay Triangulation (ADT) method [7]. This method requires only the coordinate information of all the points that form the surface to parameterize the surface. It easily forms a surface from any set of random points generated. The set of points to form a surface are first projected into a plane perpendicular to the direction of flow of the jet. This 2 dimensional set is then triangulated using the Delaunay triangulation algorithm and this triangulation information is used to form the surface in 3 dimensions on the original plane. This process is robust and fast and allows for smooth surfaces as well as sharp ridges in the surface.

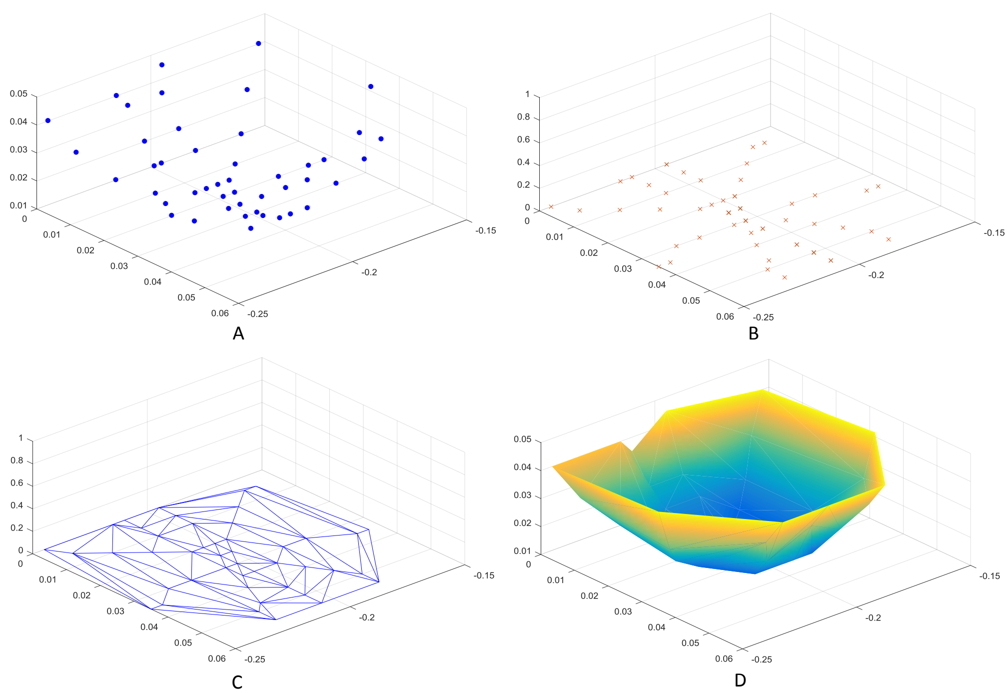


Figure 1. A) 3D points on the surface of the bucket, B) 2D projection of the points on z-plane, C) Delaunay triangulation of the 2D points, and D) the parameterized bucket surface

B. Flow simulation

SPH method was chosen over the FLS for better accuracy and over traditional Eulerian methods owing to its faster solution rates. DualSPHysics 4.0 codes developed by the John Hopkins University, Universida de Vigo and The University of Manchester were used to conduct the simulations. This method takes into consideration only the neighbouring particles defined by the kernel function to calculate the physical properties of individual fluid particles. This provides scope for artificial viscosity and handles free surface flow problems with lesser computational expenses.

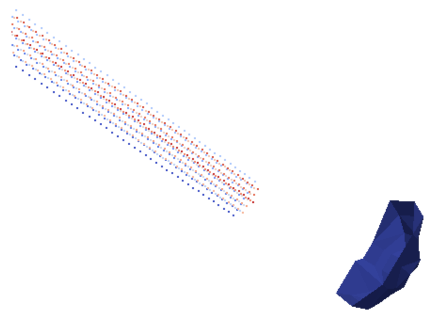


Figure 2. DualSPHysics solution environment

The case definition files, that define the environment, boundary conditions and the parameters for the simulation are created in XML which are then fed into the SPH solver which calculates the position, velocity, pressure, forces etc. for every particle in every time step. The outputs are generated in the form of .vtk, .bi4, and .csv files. The VTK and bi4 files help for visualization while the CSV files provide the numerical data for variables such as force and pressure. This information can be used to visualize and calculate the overall trajectory and energy transfer during the entire simulation cycle. Paraview is open-source software that can be used for visualization of the simulation using the output files. The parameter settings as below to replicate the properties of water flow were chosen for the simulation.

TABLE I. PARAMETERS FOR SMOOTHED PARTICLES HYDRODYNAMICS SIMULATIONS

Parameters	**Values**	**Units**
Density of the Fluid	1000	Kg/m^3
Coefficient for smoothing length	1	
Precision	Double	
Timestepping	Symplectic Euler	
Kernel	Wendland	
Viscosity	Artificial	
Gravity	0	m^2/s
Rigid Algorithm	SPH	
Time of simulation	0.075	sec
Time steps	0.005	sec

C. Optimization engine

The engine was created using R programming codes taking genetic algorithms as the basic theory as illustrated in figure 3. Each gene contains three numbers that represents the coordinates of the Euclidean space that defines the point on the bucket surface. A set of chromosome contains 61 such points that will define an entire bucket. The initial population of 44 individuals was developed from random points. The population also included one individual with the points of the existing model and three more with only minor changes. This would ensure that we arrive to the feasible solution sooner without discarding the possibility of a totally new design.

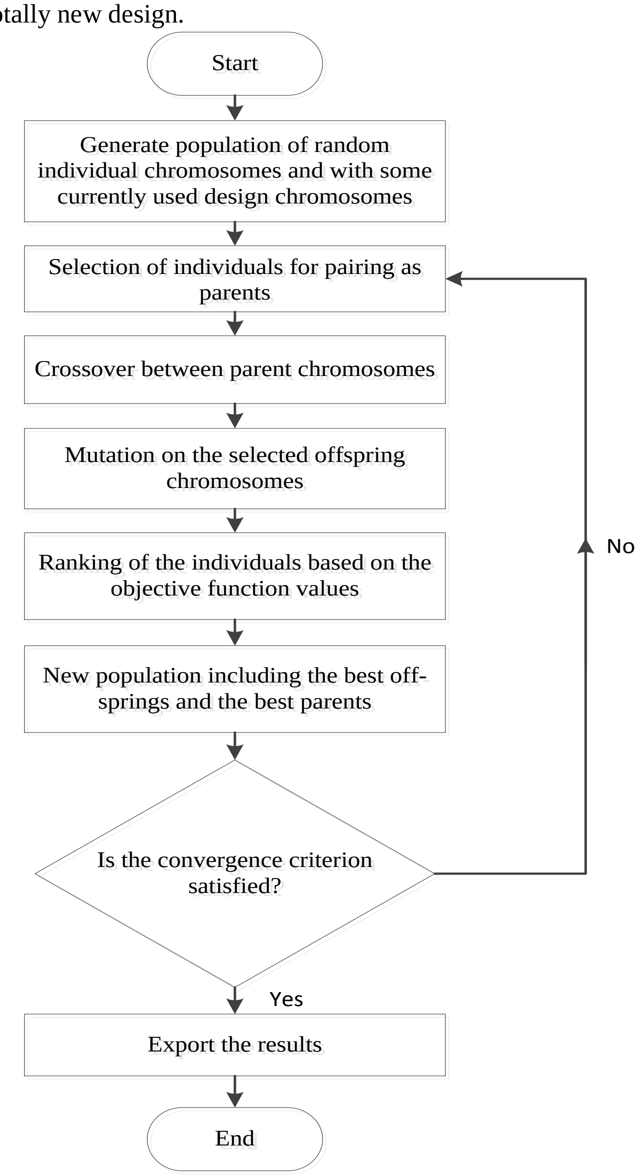


Figure 3. Optimization engine flowchart

The parents were chosen from the available population list at random. Two point cross-over was used as reproduction operator. The two points for crossover were also chosen at random. This particular crossover allows for partial crossover which aids in preserving the most efficient features while discarding the less efficient features. Mutation operator was applied to the selected off-springs generated after the crossover. Taking 0.5% mutation rate, one gene in each chromosome was mutated with a value from within the acceptable domain. This would aid to create a gene entirely different from the existing gene pool.

The fitness function in this case was defined by the overall force experienced by the turbine bucket in the direction of rotation. To obtain the force value, we used the computeforces tool from DualSPHysics and process them. This final force value acts as the fitness value as maximum force helps generate maximum torque and power from the system. Based on this fitness value, selection of the best individuals will be made as the population of the next generation. Elitism has been used as a measure to preserve the best solution from the previous generation. In its absence, the crossover and mutation could produce a batch of inferior off-springs and the already obtained best solution could be lost. This new population would then again be paired and operations would be conducted as shown in the flow diagram in figure 3.

The optimization cycles were conducted for 50 generations for each of the 3 runs. Each run had different sets of 44 random chromosomes and 4 sets of predefined chromosomes to start with. The best solution had converged mostly after the 38th generation.

IV. RESULTS

The existing bucket design was taken from the study conducted in [8]. Its bucket was considered to be in stationery condition at for the optimization attempt. Table 2 provides the turbine model operation data.

These values provide the benchmark values to compare the performance of the existing bucket. The force experienced on the bucket surface was analysed as the objective function for optimization. The simulation results for the existing bucket generated the sum of 20053.67N for 15 timesteps which averages at 1336.91N. Comparing this value with the operation data listed in table 2, we can see that the bucket is 71.3% efficient in stationery condition. This reading has been taken as benchmark for this study.

TABLE II. TURBINE MODEL OPERATION DATA

Turbine Model Operation Data		
Flow rate	37.945	kg/s
Net Head	129.6	m
Jet Diameter	31	mm
Jet velocity through nozzle	49.42	m/s
Power	1875.14	N

Figure 3 plots the performance evolution during all three optimization runs and provides the performance valuation of the existing design as a benchmark. The results obtained showed an expected pattern. The force values for the initial generations were low and showed dynamic increment. Later, increments were small and the solution finally converged.

The highest increment was seen in the first generation itself when the entire bucket moved closer to the jet source. Then, gradual improvements were seen in the readings owing to the crossover and mutation operators. Run 1 generated the final best output of 1513.56N while run 2 and run 3 generated 1477.53N and 1483.99N. The output from the first run improved the performance by 13.21% while run 2 and run 3 improved the performance by 10.52% and 11% respectively.

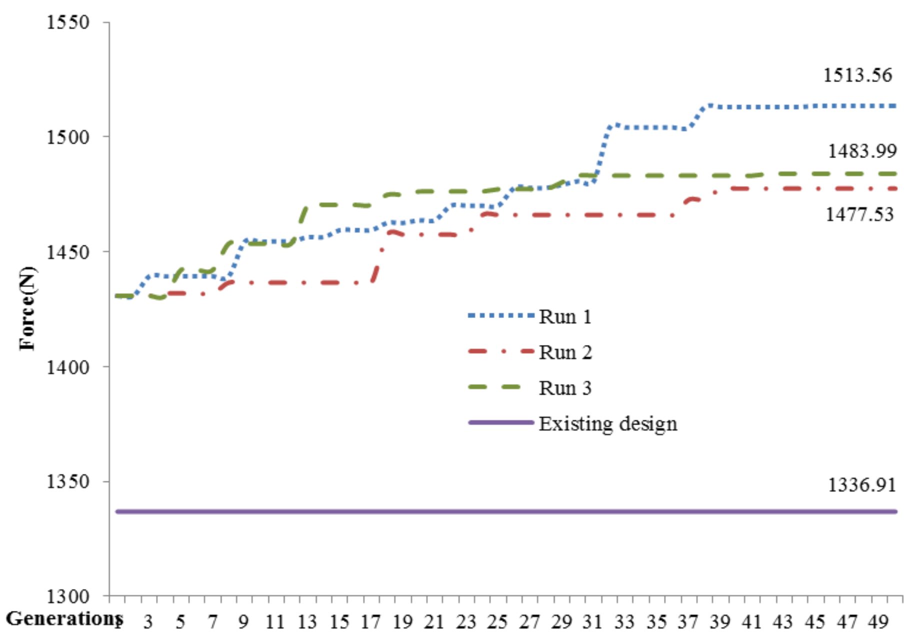
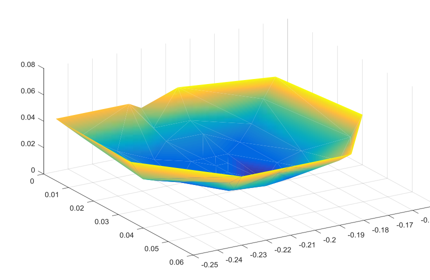
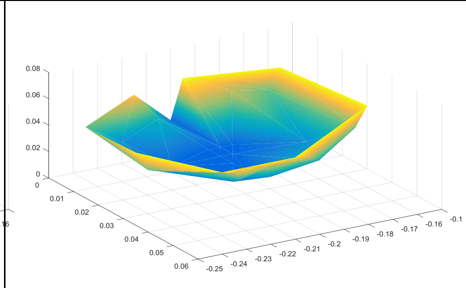
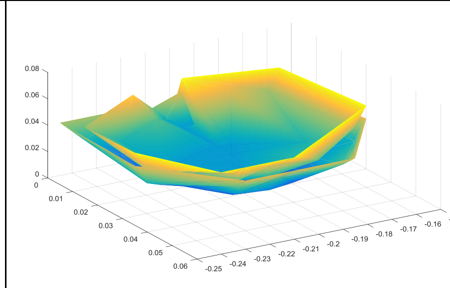
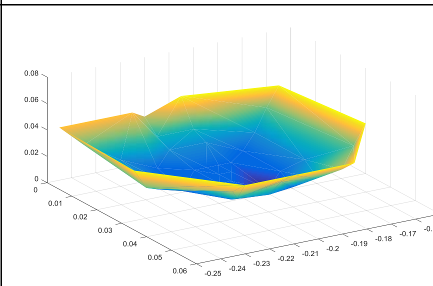
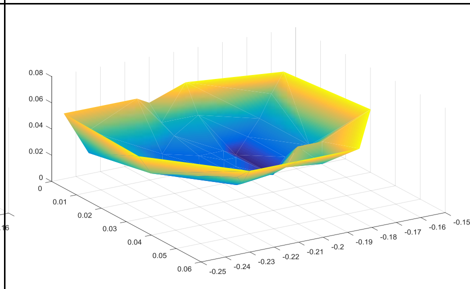
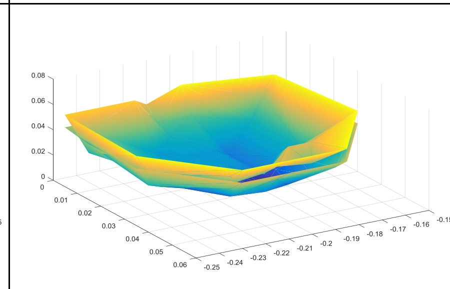
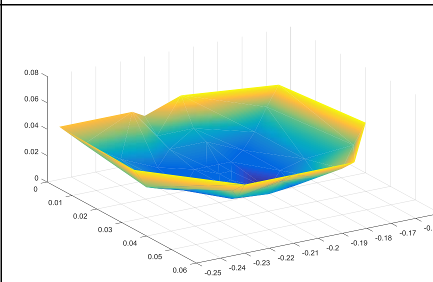
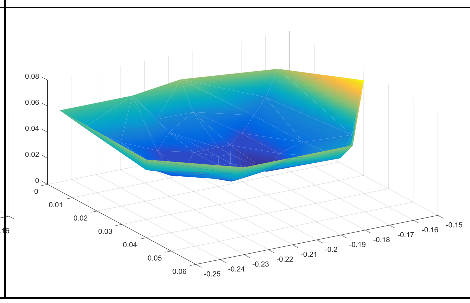
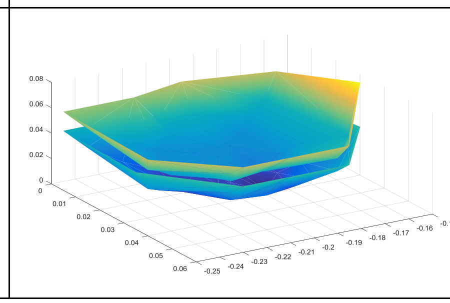


Figure 4. Performance evolution of optimized design

TABLE III. OPTIMIZED BUCKET COMPARISON WITH EXISTING BUCKET

	Existing bucket design	Optimized bucket design	Overlapping view
Run 1			
Run 2			
Run 3			

The above table clearly illustrates the differences in the design among the four bucket designs. The differences are listed as:

- The existing design has a much sharper splitter angle which is ideal for splitting the jet when the bucket is rotating. In this study, the optimized design has flatter splitter angles because the bucket is stationery and maximum force will be absorbed when the jet hits normal to the face of the bucket.
- The optimized bucket in run 1 has a notch in the splitter which must have improved the force absorption of the bucket.
- The bottom of the buckets is flatter and only the exits (outer edge) of the buckets have sharper gradients whereas the existing bucket has smooth gradient resembling an ellipsoidal cavity.
- All the optimized buckets have moved closer towards the jet as the jet carries more force when it is closer to the nozzle.

The results obtained from running the simulations have shown distinct changes in the design and the improvement in performance. The suggested designs would provide better output in stationery conditions than compared to the existing bucket designs.

V. CONCLUSIONS AND DISCUSSIONS

The study conducted shows that by using light weight surface parameterization and CFD simulation tools, stochastic optimization of complex reaction surfaces can be achieved. The optimization engine employed took 183 parameters (61 coordinate points times 3 axis values) to define each bucket surface and run the simulation of 0.075 seconds within 1 minute. 50 generations with 48 individuals each took a total of 2400 minutes to complete which is very fast solution rate as compared to the traditional optimization attempts made with CFD which would take 50 hours to complete analysing only 50 parameters [9]. The resulting bucket surface showed a 13.21 percentage improvement in efficiency. It can be seen from the comparative analysis of the designs that flatter buckets provide higher efficiency in stationery conditions.

The simulation conducted in this study was a simplified study of the complex rotating Pelton bucket study. These results should not be taken as the optimized Pelton turbine design. But the study has proven the capability of the proposed method. The suggested design is visually acceptable and can be manufactured and implemented without major changes to the existing manufacturing facilities. The same method can now be applied to the rotating bucket condition with higher number of variables to generate more optimal bucket geometry. This will require more time and computational power than compared to the current study but this method would still be the most feasible option.

This method can be used as preliminary design optimization stage to test numerous designs in less time. The final optimized design can then be tested with the more accurate Eulerian methods before experimentation or manufacture.

Acknowledgment

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] B.A. Nasir, Design of High Efficiency Pelton Turbine for MicroHydro Power Plant, Publisher, City, 2013.
[2] B.W. Solemslie, O.G. Dahlhaug, A reference pelton turbine - design and efficiency measurements, in: 27th IAHR Symposium on Hydraulic Machinery and Systems, IOP 2014.
[3] A. Zidonis, A. Panagiotopoulos, G.A. Aggidis, J.S. Anagnostopoulos, D.E. Papantonis, Parametric Optimization of Two pelton Turbine Runner Designs Using CFD, Publisher, City, 2014.
[4] S.J. Anagnostopoulos, D.E. Papantonis, A Fast Lagrangian Simulation Method for Flow Analysis And Runner Design in Pelton Turbines, Publisher, City, 2012.
[5] A. Rossetti, G. Pavesi, G. Cavazzini, A. Santolin, G. Ardizzon, Influence of the Bucket Geometry on the Pelton Performance, Publisher, City, 2014.
[6] P. Koukouvinis, J. Anagnostopoulos, D. Papantonis, Flow analysis inside a Pelton turbine bucket using smoothed particle hydrodynamics, in: HYDRO International Conference, 2010.
[7] S. Bhattarai, P. Vichare, K. Dahal, Performance Comparison of Adapted Delaunay Triangulation Method over NURBS for Surface Optimization Problems, in: P.J.S. Goncalves (Ed.) The 31st Annual European Simulation and Modelling Conference, EUROSIS-ETI, Lisbon, 2017, pp. 76-80.
[8] J. Anagnostopoulos, D. Papantonis, A numerical methodology for design optimization of Pelton turbine runners, Publisher, City, 2006.
[9] A. Khandge, S. Savani, Scaling New Heights in Aerodynamics Optimization: The 50:50:50 Method, in, ANSYS, Inc., Canonsburg, PA, 2012, pp. 7.