Simplified simulation of a small Pelton turbine using OpenFOAM

V Hidalgo1,2,4, C Díaz3, J Erazo2, S Simbaña2, D Márquez2, D Puga2,5, R Velasco2, C Mafla4, G Barragán1, C Parra2, E Valencia1 and X Luo5

1 Departamento de Ingeniería Mecánica, Escuela Politécnica Nacional, Quito 170525, Ecuador
2 Laboratorio Informática–Mecánica, Escuela Politécnica Nacional, Quito 170525, Ecuador
3 Laboratorio de Materiales, Escuela Politécnica Nacional, Quito 170525, Ecuador
4 Carrera de Ingeniería Automotriz, Facultad de Ingeniería y Ciencias Aplicadas, Universidad Técnica Norte, Ibarra 100105, Ecuador
5 State Key Laboratory of Hydro Science & Engineering, Tsinghua University, Beijing 100084, China

Correspondence: victor.hidalgo@epn.edu.ec; Tel.: +593-982-491-193

Abstract. In Ecuador, the implementation of hydroelectric plants has had remarkable growth in the energy sector due to its high efficiency, low environmental impact and opportunities to generate employment. One of the sectors with the greatest benefits due to this type of energy has been the rural sector, where several small power plants with a generating capacity between 500KW and 1MW have been installed, which are usually Pelton turbines. In the present research it is necessary to use reverse engineering followed by a numerical study to performance of small Pelton turbines due to the fact that geometry dimensions have not been provided. For this purpose, the geometry has been obtained by applying reverse engineering using a technique based on the solidification of polymers. A silicon elastomer is spread over the buckets to obtain a rubber-type shell and it is then cover with plaster to maintain the shape of the rubber. A 3D scanner is used to obtain a cloud of points which is then processed to generate a faceted surface model. The three-dimensional model is edited and improved using CAD software. A basic simulation is presented to show that this reverse engineering process can be used to perform numerical analysis of Pelton turbines.

1. Introduction

Since it was first patented in 1889, the Pelton turbine has been subject of research due to its importance and wide use in the power generation industry. Furthermore, the designs of hydraulic turbines are usually kept secret. The runner geometry has a determinant role in the efficiency of the turbine. An increase of 0.1% of efficiency in the Ecuador’s largest hydroelectric plant, the “Coca Codo Sinclair” which generated 5953 GWh in 2016 [1], would correspond to 5.953GWh. The average annual Ecuadorian household was 1659.6 kWh in 2012 [2]. Therefore, the extra power generated would be enough for 3587 households.

Through Pelton turbine, the kinetic energy of the moving water is transformed into mechanical energy for electricity generation. One or more injectors are used to eject a high velocity water jet which impacts the buckets perpendicularly, so that maximum torque is applied to the rotor [3]. Apart from generating renewable energy, Pelton turbines have the advantage to regulate its power capacity from 10% to 96%,
depending the electricity demand. The upper limit is the maximum efficiency that can be achieve for a Pelton turbine [4][5]. Typical damages on the Pelton turbines consist in erosion of the inner face of the bucket. This is due to cavitation and particles traveling in the high velocity water jet. Additionally, the pulsating forces, to which the runner is subjected, generate fatigue and weaken locations on the runner which may lead to fatigue fracture [3][6]. Small Scale Hydropower plants have a generation capacity of 30MW or less. One of the most criticized aspects of traditional plants is the local impact on fishery sources and riverine ecosystems [7]. Additionally, high capital cost, resettlement of people and geographical disturbance are common drawbacks of medium-large plants. Small Scale plants do not present these complications and provide a win-win situation.

The present study is based on the reverse engineering of the bucket of a Pelton turbine of Empresa Eléctrica Provincial Cotopaxi, ELEPCO. The small hydropower plant Illuchi 2 uses two Pelton turbines which generate a power of 7MW distributed in two groups of 3250kVA and 720rpm [8]. The buckets present erosion damage and the turbines are constantly replaced as the buckets are welded to the runner. Experimental visualization of the working conditions of the water flow on a Pelton turbine is difficult due to the violent water splashing [9]. Therefore, Computational Fluid Dynamics CFD is commonly used for observing the flow and designing bucket geometries. Plenty of studies have been performed regarding Pelton turbine geometries analysis and optimization methods such as Stochastic and combined Lagrangian-Eulerian [10].

2. Methodology
2.1. Geometry Extraction

A silicone rubber was used to recover the geometry of the Pelton turbine bucket. The product consists of two parts, a silicone viscous fluid and a catalyst that allows the silicone rubber to cure once the mixture has been placed on the Pelton’s bucket. The mix ratio of the product is 1:1 by volume and weight, it means that the silicone rubber and the catalyst must be mixed in equal parts to obtain the desired result. It has a pot life of five minutes, so the mixing and application processes must be quick. The mixture must be spread evenly until kept in place, see Figure 1. It is important that the surface is clean before applying the silicone rubber, for which a damp wipe is used to remove dust and other undesirable particles from the bucket’s surface. Then, it is necessary to remove the excess of water using a dry wipe to guarantee a completely dry surface. Finally, the entire bucket surface is covered with the silicone rubber, and it is necessary to maintain a coating layer of at least 5mm to ensure that the mold does not deform or break.

![Figure 1. Silicone model obtention. a) Silicone being spread and b) Applying a layer of plaster.](image-url)
The product recommendations indicate that for a temperature of 23°C the cure time is 20 minutes. In the workplace the humidity was above 80% and the temperature around 15°C. Due to the lower temperature and higher humidity the cure time increased, so the demold time was 45 minutes.

Once the silicone was completely cured, a layer of plaster bandages was placed on the surface of the silicone to ensure that the shape is maintained. In order to achieve this, a silicone rubber with an A hardness (ASTM D2240) of 25 was used. This allowed a simple removal of the mold without risk of deformation during demolding.

In order to digitize accurately the bucket geometry, two molds were obtained, one from the inside of the bucket and the other from the outer face. The next step is to polish the molds, which involves removing the parts that are not necessary or that can prevent the 3D scanner from digitizing the concave areas of the mold. The silicone rubber excess was removed using a cutter. Similarly, the unnecessary plaster parts were cut off. Furthermore, the scanner must capture only the silicone rubber and ignore the plaster so that the digitization has less imperfections.

The digitization of the model was done with “Creaform” scanner which requires reference points that are placed in the mold to form triangles with separations ranging from 3 to 5 cm. Such separations between reference points are known to generate an accurate cloud of points for a mobile scanner [11], as shown in Figure 2. The mold was placed on a translucent surface to prevent the scanner from capturing the base. The scanning took approximately 10 minutes per mold to ensure that the concave surfaces were digitized correctly.

![Figure 2](image)

**Figure 2.** Scanning process. a) Reference points used and the 3D scanner, b) The model generated by the scanner.

The scanned geometry comprises a cloud of approximately 3 million points, which correspond to a concave faceted surface of one model. Subsequently, the faceted surface model was pre-processed before performing the simulation. The pre-process of the cloud of points consists in a point reduction using the command “delete mesh”, and the surface generation from the optimized cloud of points using the command “patch”. As a result of the pre-process, a faceted surface model of the bucket was obtained. The pre-processing was developed in the commercial software Rhinoceros 6, as recommended by the scanner manufacturer. A simplification of the procedure is shown in Figure 3.
2.2. Numerical Methodology

For the sake of the study, a dynamic Mesh was developed taking in consideration some field measurements. The geometry was the first item to be treated due to the positioning process for each element. Only one side of the bucket was simulated in order to optimize computational cost. The methodology has been divided in sections, which are Geometry, Mesh, Solver, Boundary and Initial Conditions.

2.2.1. Geometry modelling

The faceted surface model of bucket was obtained with a 3D-Scan process. This file was taken into a CAD software for the pre-processing stage, where the whole computational domain was defined as follows. First, the stationary part known as the casing was modeled, considering that the water injector is placed at 40° from the z-axis, in a clockwise direction. Then, the rotatory part known as the Pelton wheel was generated. Previous studies have shown that it is enough to use three buckets in the wheel to obtain accurate results [12][13]. These buckets were positioned using a tangential linear reference to a 32 [cm] circumference at the center of the computational domain. As shown in Figure 4, the two geometry parts were assembled to obtain the final computational domain. It is necessary to generate a surface between both, since it is the main element which allows the data transfer from the master to the slave domain.
2.2.2. Mesh

The computational domain has two parts, the rotatory and stationary domain, as depicted in Figure 5. For each domain a mesh was generated, taking as first step the creation of a base mesh using blockMesh tool. The internal mesh is generated using SnappyHexMesh, which allows to attach the mesh with the complex details of the geometry. The aforementioned tool creates a hybrid mesh of hexahedral and tetrahedral elements.

Figure 5. a) Bucket mesh, b) Computational domain mesh and c) Final merged mesh.
In order to generate the moving cell zones of the rotor it was necessary to use the OpenFOAM tools topoSet. To obtain a suitable moving mesh, both meshes are merged using mergeMeshes. These tools are used to achieve a suitable case for the simulation on the dynamic solver interDyMFoam.

2.2.3. Solvers

To achieve the phenomena reproduction, attention must be paid to the following effects: unsteady free surface flow in the buckets, high fluid velocity within the open surface domain and two-phase flow (water – air). It was necessary to use a solver that could handle conditions such as: biphasic flow, rotational movement of the mesh, sliding interface between rotating and stationary mesh. Thus, the interDyMFoam solver was selected for the case study. To reduce the calculation time, the simulation was carried out only using 3 buckets and cutting in half the computational domain at the symmetry plane XY.

Inside interDyMFoam, the AMI method was used for the data transfer between moving mesh and stationary mesh. Additionally, a cyclicAMI condition was set with the function rotatingMotion from the dynamic mesh solver solidBodyMotionFvMesh. On the other hand, the turbulence model k-ω SST was chosen, due to the importance of the flow behavior analysis in the inner walls of the bucket and its high accuracy.

2.2.4. Boundary and initial conditions

The simulation was performed taking in consideration a free-surface flow within the Pelton wheel buckets. Another consideration was the non-slip condition in the walls of the buckets. Table 1 shows initial conditions and some important parameters for the simulation. These parameters are based on real working conditions of the turbines at the Illuchi 2 plant.

| Symbol | Unit   | Value   |
|--------|--------|---------|
| Flow rate | Q      | m³/s    | 0.3    |
| Jet Diameter | D     | m       | 0.07   |
| Number of buckets | Z    | U       | 20     |
| Angular velocity | W   | Rad/s   | 75.39  |
| Pressure | P      | Pa      | 0      |
| Turbulent energy | k    | m²/s²   | 91.15  |
| Turbulent intensity | I    | -       | 10%    |
| Specific turbulent dissipation rate | ω    | s⁻¹     | 3557.30 |
3. Results

The main purpose of the present simulation is to prove the data collected by reverse engineering is useful to perform an optimization study of the hydropower station turbines. The simulation was carried out with a total of 1.9 million grid elements, which have suitable quality parameters. It is evident at a mean mesh quality level the profile characteristics are captured quite easy despite of the complex geometry. Furthermore, the profile did not present problems during the dynamic stage. The results show the mirrored images for a better understanding. Figure 7 depicts how the flow clearly recognizes the profile of the bucket and trace a proper flow behavior.

![Figure 7. a) Velocity profile in a single bucket for alpha water contours and b) Velocity profile in the three buckets arrange for alpha water contours.](image)

The output dependent time variable fields and profiles of velocity, pressure, k, w, nut volumetric fraction, and others were obtained using the solver mentioned above. Some of these variables are used to compute specific values such as torque, power, efficiency, vorticity which are useful at the post-processing stage for optimization and comparison.

4. Conclusions

The numerical methodology showed a similar flow behavior to the Pelton turbine of the Illuchi 2 Hydropower station. Hence, the simulation results can be used in future optimization studies. The model captures main trends of performance and enables to assess different configurations with great accuracy.
Furthermore, the opensource nature of the simulation opens the possibilities for low cost optimization schemes and operational improvements for small Pelton Turbines. On the other hand, the methodology proposed in this work present high accuracy when capturing the intricate Pelton geometry despite of the disadvantages. The drawbacks of the method are the large amount of time during pre-process that it is needed to obtain an acceptable surface from the cloud of points and the high computational power required to pre-process the point cloud to obtain the surface. The study showed that three buckets are enough to capture the flow near the bucket during first impact and after the water jet passes over it.

Acknowledgements

The authors gratefully acknowledge the financial support provided by Escuela Politécnica Nacional for the development of this study, which is part of the research projects PIJ 17-13 and PII-DIM-2019-06.

References

[1] Armijos O 2017 Rendicion de cuentas: Unidad de Negocio Coca Codo Sinclair CELEC EP
[2] Chavez A 2016 Estadística Anual y multianual del sector eléctrico ecuatoriano Agencia de Regulación y Control de Electricidad
[3] Egusquiza M, Egusquiza E, Valentin D, Valero C and Presas A 2017 Failure investigation of a Pelton turbine runner Engineering Failure Analysis. 81 234-244
[4] Egusquiza M, Egusquiza E, Valero C, Presas A, Valentin D and Bossio M 2018 Advanced condition monitoring of Pelton turbines Measurement 119 46-55
[5] Jost D and et al. 2010 Numerical prediction of Pelton turbine efficiency 5th, IAHR Symposium on Hydraulic Machinery and Systems, IOP Conf. 12
[6] Vitorio G, Mandelli S and Malavasi S 2019 Hydro-abrasive erosion in Pelton turbine injectors: A numerical study Renewable Energy 130 474-488
[7] Kosnik L 2010 The potential for small scale hydropower development in the US Energy Policy 38 5512-5519
[8] Llamba W 2014 Elaboración del plan de mantenimiento centrado en confiabilidad (rcm) de la central hidráulica illuchi nº 2 Universidad de las Fuerzas Armadas - ESPE
[9] Alimirzazadeh S, Kumashiro Y, Leguizamón S, Ebrahim, J, Maertens A, Vessaz C, Tani K. and Avellan F 2020 GPU-accelerated numerical analysis of jet interference in a six-jet Renewable Energy 148 234-246
[10] Suyesh B, Parag V, Keshav D, Ahmed A and Abdul-Ghani O 2019 Novel trends in modelling techniques of Pelton Turbine bucket for increased renewable energy production Renewable and Sustainable Energy Reviews 112 87-101
[11] Fryskowska A 2019 An Improvement in the Identification of the Centres of Checkerboard Targets in Point Clouds Using Terrestrial Laser Scanning Sensors 19 938
[12] Rygg J R 2013 CFD Analysis of a Pelton Turbine in OpenFOAM Norwegian University of Science and Technology
[13] Solemslie B and Dahhaug 2012 A reference Pelton turbine design IOP Conference Series Earth and Environmental Science 15