© Universiti Tun Hussein Onn Malaysia Publisher's Office

IJIE

http://penerbit.uthm.edu.my/ojs/index.php/ijie
ISSN : 2229-838X e-ISSN : 2600-7916

The International Journal of Integrated Engineering

Numerical Simulation of Different Splitter Angles of a Pelton Bucket to Increase the Power Generated by The Pelton Wheel

Jorge Andrés Sierra Del Rio[1,2*], Alejandro Ruiz Sánchez[1]

[1]Department of Mechatronics Engineering, MATyER,
Instituto Tecnológico Metropolitano, Medellín, 050035, COLOMBIA

[2]Department of Engineering Environmental, GIIAM,
Institución Universitaria Pascual Bravo, Medellín, 050035, COLOMBIA

*Corresponding Author

DOI: https://doi.org/10.30880/ijie.2023.15.07.001
Received 9 May 2023; Accepted 3 October 2023; Available online 5 December 2023

Abstract: The efficiency of a Pelton turbine depends mostly on its geometry. A design with suitable dimensions, offers higher generation of energy. However, some of these dimensions are underappreciated because of their low influence. Therefore, this study presents through a recompilation of data, the simulation was developed in ANSYS 19®. The importance of taking in count each parameter of the geometry of a Pelton bucket, analyzing in this work the influence of the splitter angle in the force generated by a bucket. Nine geometries of a Pelton bucket are developed with splitter angles between 17 and 25°. The most suitable angle is determined, finding through the simulations that the geometry of 23° generates more force than other geometries. Simplifying the geometry and the mesh can generate faster results, however, exaggerating in the meshing process will affect the reliability of the obtained results.

Keywords: Splitter angel, energy, Pelton turbine, Pelton bucket, efficiency

1. Introduction

Non-renewable energies have supplied the highest quantity of energy to the world's population during years. Coal, fuel, and gas are used in fossil fueled plants to generate electricity, usually through a turbine that uses the steam of a combustion chamber. Every nuclear power plant generates electricity with the heat of the main process in the plant: nuclear fission. Although these two types of electricity generation have useful advantages, the environmental impact they caused can be devastating, e.g., the nuclear waste, the disaster in Chernobyl, global warming, the Ozone depletion, etc. Due to these risks, and the fact that the energies described use non-renewable resources, the investigation and use of renewable energies is justified [1]. Non-interconnected zones (NIZ), pollution, global warming, etc. are problems that need investigations about quick and possible solutions.

In search of contributing to the increase of knowledge about the latter, this project will investigate the most suitable angle for a bucket's splitter of a Pelton wheel. Improving the design of a Pelton bucket should increase the power generated by the Pelton turbine, which would help to satisfy the current electrical energy demand, especially in those places close to lakes or rivers, because a hydroelectric power station can be built there with enough output to satisfy the electricity need around it. If this investigation shows a significant improvement in the Pelton wheel, then the power plants that already exist will benefit too, as this will increase their performance. The environment will be affected too, as one of the problems this paper thinks about is the pollution. To achieve the increase described, this investigation will focus on the splitter angle of a Pelton bucket, aiming to obtain results that will help to satisfy the electrical energy demand. Solid results and justifiable conclusions should inspire more investigations about this turbine

*Corresponding author: jorgesierra@itm.edu.co

to satisfy its knowledge demand and rise the possibilities of hydric resources to replace fossil fuels and become the primary fuel of the world in the future.

The Pelton turbine is an impulse turbine that was first patented in 1880 by Lester A. Pelton. This technology consists of a rotating wheel with buckets, where the jet tangentially impacts the Pelton bucket, to be divided in two parts that flow on the surface of the bucket [2]. Figure 1 shows a schematized view of the Pelton turbine and Pelton bucket. As this technology has a significant age, it has gone through many investigations to improve its design. The differences between the original design and a state-of-the-art schematized turbine are significant and can be noticed by comparing Figure 2 with Figure 3. Currently, the Pelton turbines can considerate the use of multiple nozzles that regulate the water flow. This is advantage is noticeable in cases where the flow cannot be predicted, and the discharge usually changes during the year [4]. The geometry of a Pelton turbine has numerous parameters, especially the Pelton bucket, which due to its complex shape, the task of representing it in a CAD software requires attention and ability, and their simulations require experimental validation [5], [6] [7]. The dimensions of these parameters can be determined through numeric analyses [8], trial and error approaches, and simulations. Židdonis [9], parameterized the Pelton turbine in 11 design parameters of interest as can be seen in Figure 4.

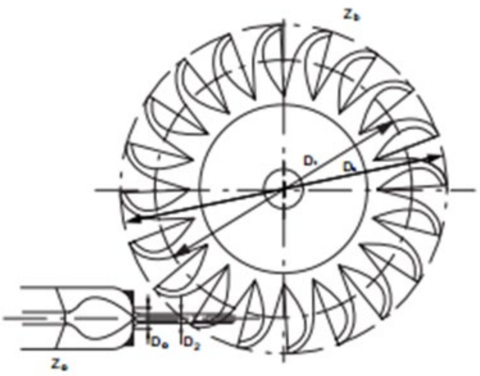


Fig. 1 - Pelton turbine and Pelton bucket [3]

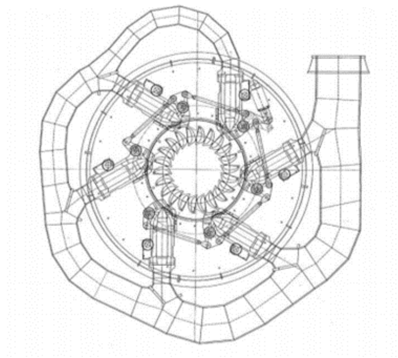


Fig. 2 - Schematized view of a Pelton turbine with 6 nozzles [4]

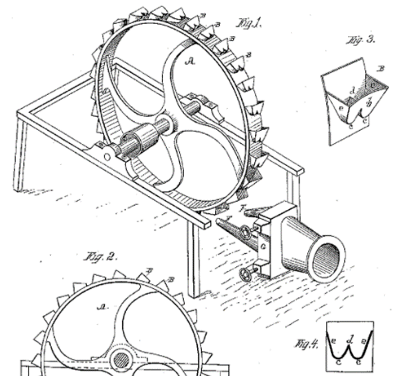


Fig. 3 - Pelton wheel from original patent [2]

These procedures of parametrization of complex geometries allows to simplify the analyses that will be done in them. Also, this decreases the computational cost that would take to simulate every dimension in a Pelton turbine. The splitter angle is a parameter of the Pelton bucket. In Figure 5.a, a bucket with a section line A-A is shown, and in Figure 5.b, the bucket is shown from another view, being noticeable the splitter angle, as it is represented as ‘2α’. Its task in the geometry is to divide the jet in two equal parts. The divided jet will flow over the surface of the bucket, transmitting its kinetic energy to the turbine, to finally leave the bucket with a residual velocity. This parameter is the first one to contact the jet, having significant importance because their interaction will determine the efficiency of the turbine. Also, this impact causes deterioration by high pressures that will be shown further in this work, being necessary to develop studies related to the protection of the splitter and other high-pressure zones of the bucket.

There are some investigations in the literature where the authors studied or considered the same topic at present paper has. Chukwuneke et al. [9] did an experimental investigation, where the power generated by a Pelton turbine was studied by changing the splitter angle of the Pelton bucket and the turbine speed; the authors found a maximum power output for a turbine with a 23° splitter angle and speed of 1700 rpm. Perrig [3], in his work about the flow in the buckets of a Pelton turbine, didn't mention the value he used for his investigation; however, he states that a splitter angle is usually never smaller than 20° to avoid rapid destruction of the splitter, providing a reference value to study this parameter. Solemslie [10] aims to develop a publicly available Pelton turbine geometry and design methodology through empirical data and simulations, and based on his references, he considered insignificant the effect of the splitter angle during the description of his geometry in his work. Zhang [11] states in his article that the half of the splitter angle is usually between 10° and 20°, which would be a range for a splitter angle that starts from 20° and 40°. Židonis et al. [12] identified 11 parameters that are believed to have influence in the Pelton turbine's performance, where the splitter angle is included, finding in their simulations a result of 20° as the most suitable angle for their geometry. García et al. [13], after performing CFD simulations in his work about the influence of the splitter angle on the torque generated by a Pelton turbine, determined that a 24° splitter angle is the most suitable for their geometries. As a summary of the information obtained in the literature, Table 1 presents the relevant information extracted from some references that are related to this work. According to literature review, the main objective of present paper is determining numerically the most effective splitter angle to improve the performance of a Pelton turbine by increasing its generated power according to previous research angles studied.

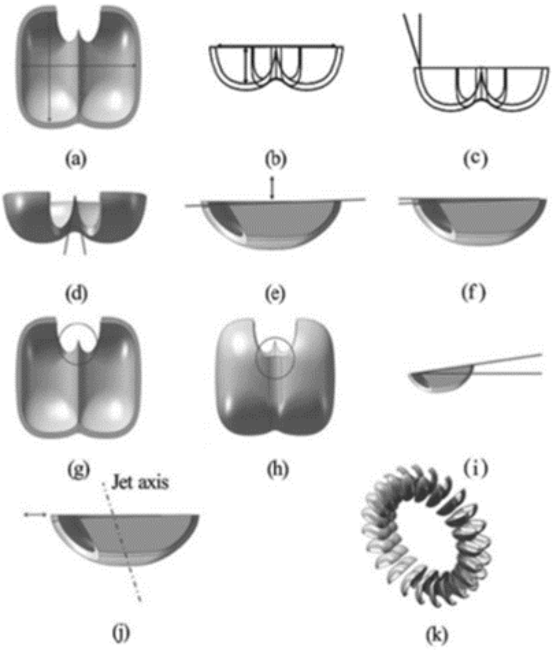
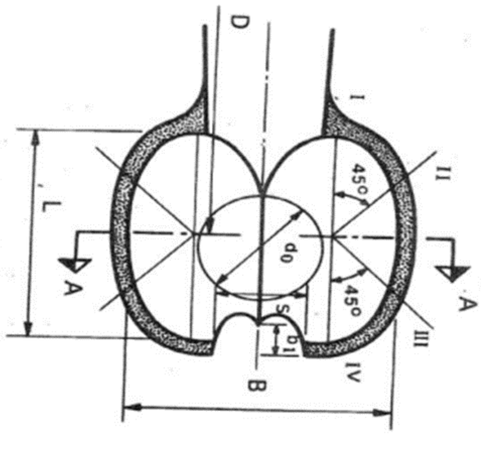
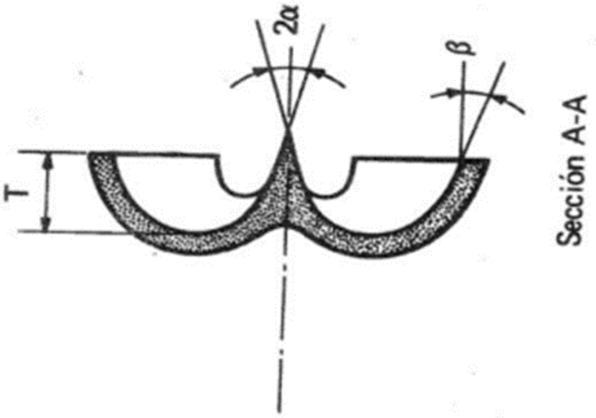


Fig. 4 - Design parameters of a Pelton wheel (a) bucket length to width ratio; (b) bucket depth to width ratio; (c) bucket exit angle; (d) splitter inlet angle; (e) splitter level; (f) splitter tip angle; (g) splitter tip geometry; (h) backside of the splitter; (i) inclination angle; (j) radial position of the bucket; (k) number of buckets. Extracted from [9].



a) Pelton bucket with section line A-A [4].



b) Section A-A of Pelton bucket in Figure 7 [4].

Fig. 5 - Pelton bucket

Table 1 - State of the art of the analysis of the splitter angle

Reference	Study	Splitter angle (°)	Suitable (°)	Software
[13]	Simulation	16, 24, 32, 40	24	Ansys
[14]	Simulation	1-25	Not mentioned	Matlab
[15]	Not mentioned	At least 15	Not mentioned	Not mentioned
[16]	Not mentioned	10, 18	18	Not mentioned
[17]	Not mentioned	21	21	Not mentioned

2. Methodology

2.1 Design of The Buckets

The shape of the bucket is determined using the graphic method. Lines, tangent arcs, and circles are drawn in this method to achieve the desired form [14]. The size of these draws is defined in function of the jet diameter, with the proportions shown in Table 2, where ‘d’ is the jet diameter, which for this study is 0.125 m. β2 is for splitter angle. L is for the total length of the hydraulic surface of the bucket in the radial direction. B is for bucket diameter. D is bucket deep. f is for space between bucket and chord. M is width notch. e is for space between notch angle and top bucket. l is for length crest. On the other hand, the spatial positioning and the geometric characteristics of every section or slice previously shown, can be seen in Figure 6 and Figure 7, where in the first one is possible to observe in detail the general characteristic of the bucket, especially in its top view [15]. When the design parameters of the bucket were determined, 10 geometries were built using the software Solid Edge ST9. These geometries are differentiated by their splitter angles, starting from 20° and ending in 25°. The files of these geometries were converted to IGES format, and were imported to ANSYS Design Modeler, where the fluid volumes were generated. These are necessary to develop the simulations. An example of the geometries created, with the fluid volume incorporated is shown in Figure 8. The fluid volumes of the other geometries were generated in the same way as the geometry of 23°.

Table 2 - Dimensions of the Pelton bucket depending on the jet diameter. Extracted from [14]

Parameter	Value depending on ‘d’	Value for the created buckets	Units
d	0,125	0,125	m
B	3d	0,375	m
L	2,8d	0,35	m
D	0,9d	0,1125	m
f	0,9d	0,1125	m
M	1d	0,125	m
e	0,45d	0,05625	m
β1	15	15	degrees
β2	-	-	degrees
l	1,6d	0,2	m
β3	5	5	degrees
β4	13	3	degrees

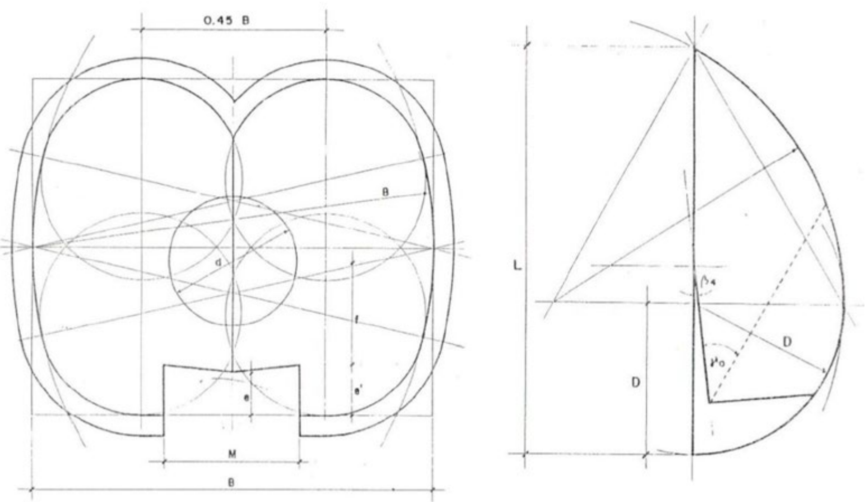


Fig. 6 - Determination of the shape of the Pelton bucket through the graphic method [14]

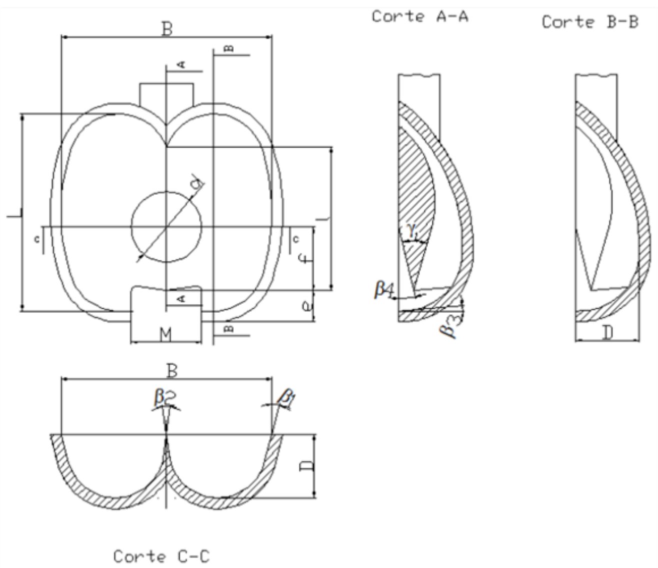


Fig. 7 - Geometry of a Pelton bucket. Adapted from [13]

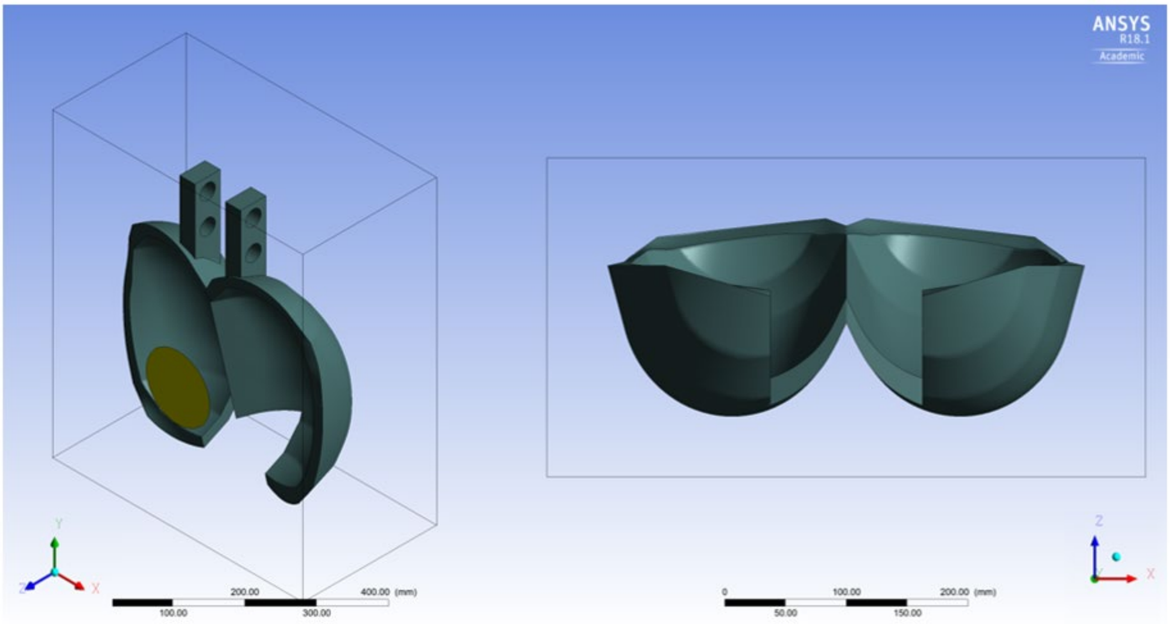


Fig. 8 - Fluid volume for the bucket with splitter angle of 23°

2.2 Meshing

One of the required steps to develop a fluid-dynamic analysis using numerical methods, is the discretization of the domains, also called meshing [17]. As the accuracy and reliability of the results obtained through the numerical simulations are highly influenced by the quality of the discretization done, is necessary to evaluate some parameters of the mesh to guarantee the reliability of the results. For this, a mesh study was performed where the quantity of elements was linked with the result of the studied parameter (force applied by the water on the surface of the bucket), with the goal of finding a suitable range of elements where the variation of the results is less than 2%, obtaining that the minimum number of elements required to guarantee this condition is 3.5×106 elements [18]. The discretization of the turbine control volume was done in the "Meshing" module of ANSYS software. In the discretization process to achieve grid independence with respect to force five meshes were configurated for each splitter angle to reach grid independence. Figure 9 Shows mesh independence for each splitter angle. Additionally, other four parameters were evaluated: obliquity, orthogonal quality, aspect ratio and element quality. Their maximum, minimum and average values obtained in each configuration can be appreciated in Table 3. This table allows to guarantee propitious conditions of the domain discretization to perform the fluid-dynamic simulations.

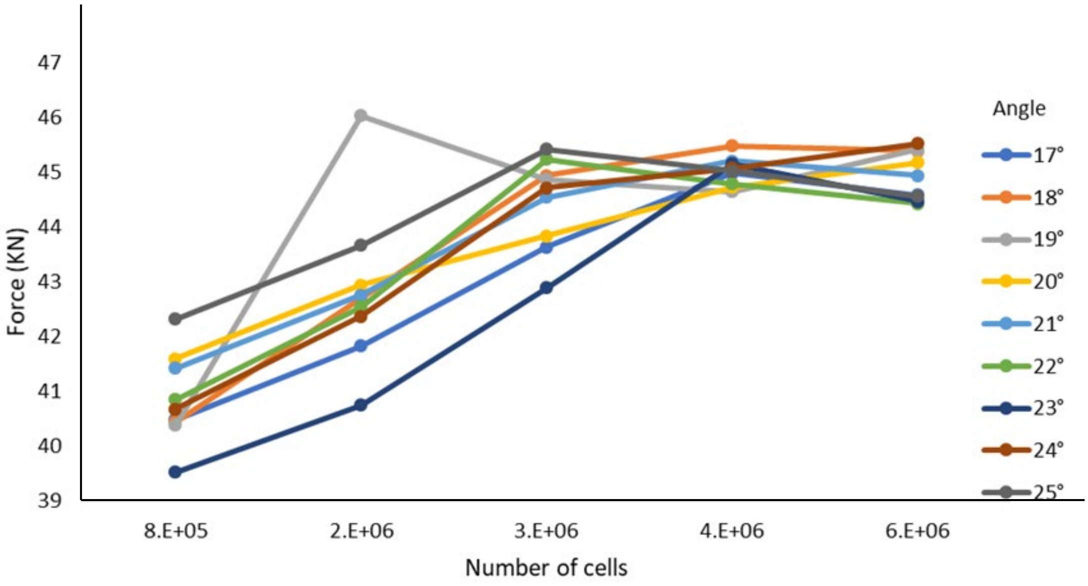


Fig. 9 - Mesh independence

Table 3 - Mesh metrics

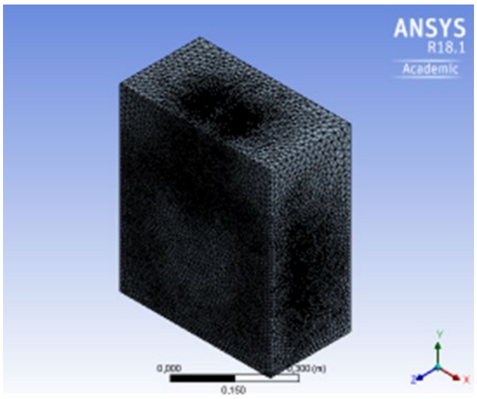
		Geometry (Splitter Angle)								
Parameter		**17°**	**18°**	**19°**	**20°**	**21°**	**22°**	**23°**	**24°**	**25°**
Elements		4.2×10^6	4.8×10^6	3.2×10^6	4.6×10^6	4.6×10^6	4.5×10^6	4.8×10^6	4.5×10^6	3.9×10^6
Nodes		0.7×10^6	0.8×10^6	0.6×10^6	0.8×10^6	0.8×10^6	0.7×10^6	0.8×10^6	0.7×10^6	0.6×10^6
Obliquity	**Min**	3.8×10^{-4}	4.1×10^{-4}	5.1×10^{-4}	1.2×10^{-4}	3.6×10^{-4}	4.3×10^{-4}	2.2×10^{-4}	1.9×10^{-4}	2.6×10^{-4}
	Max	0.7	0.8	0.7	0.9	0.7	0.7	0.7	0.7	0.7
	Ave	0.2	0.2	0.2	0.2	0.2	0.2	0.2	0.2	0.2
Orthogonal Quality	**Min**	0.2	0.1	0.2	6.8×10^{-2}	0.2	0.2	0.3	0.2	0.3
	Max	0.9	0.9	0.9	0.9	0.9	0.9	0.9	0.9	0.9
	Ave	0.7	0.7	0.7	0.7	0.7	0.7	0.8	0.7	0.8
Aspect Ratio	**Min**	1.2	1.1	1.2	1.2	1.2	1.2	1.1	1.1	1.1
	Max	6.3	6.5	5.6	7.5	5.9	5.5	5.4	6.6	5.4
	Ave	1.8	1.8	1.8	1.8	1.8	1.8	1.8	1.8	1.7
Element Quality	**Min**	0.2	0.2	0.3	0.2	0.3	0.3	0.3	0.2	0.4
	Max	0.9	0.9	0.9	0.9	0.9	0.9	0.9	0.9	0.9
	Ave	0.8	0.8	0.8	0.8	0.8	0.8	0.8	0.8	0.8

An example of discretization is shown in Figure 10, where the geometry of 23° was meshed as can be seen in Figure 10.a. The shape of the bucket is printed on the surface of the fluid volume and is noticeable in Figure 10.a and Figure 10.b. This is due to the quantity of elements used in this study. In Figure 10.c, a section plane was created and applied to observe the behavior meshing inside, as can be seen in Figure 10.d. The configuration parameters of Ansys Mesh that were used to obtain the results represented in Figure 10 are:

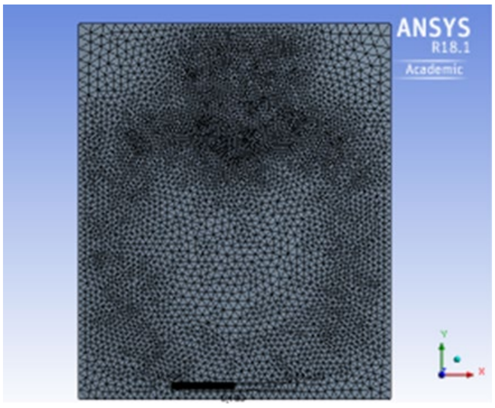
- Solver Preference: CFX
- Size Function: Configured as Proximity and Curvature.
- Curvature Normal Angle: 12°
- Num Cells Across Gap: 12
- Min Size: 0.1 mm
- Proximity Min Size: 0.1 mm
- Growth Rate: 1.1

- Target Skewness: 0.8
- Smoothing: High

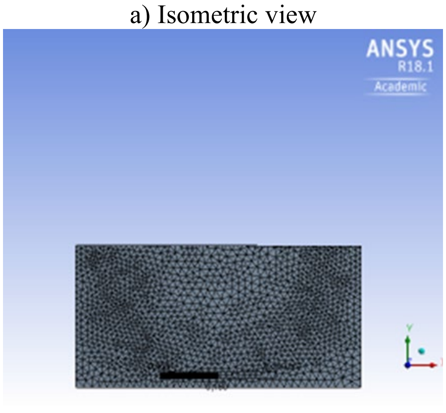
These values are not the same that were used in each geometry. The splitter angle highly influenced the mesh quality in this work, being necessary to adapt some values to obtain the results shown in Table 3.



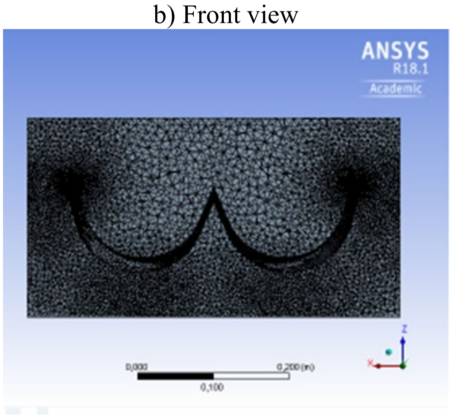
a) Isometric view



b) Front view



c) Top view

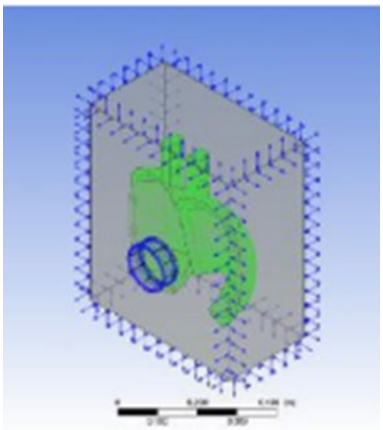


d) Bottom view

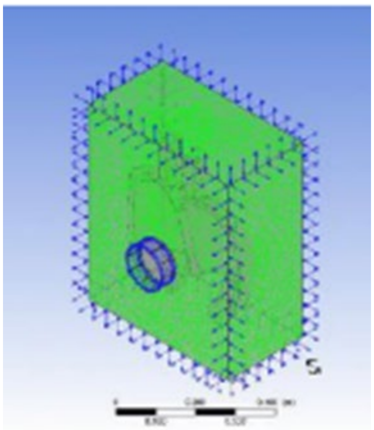
Fig. 10 - Fluid volume discretization of bucket

2.3 Boundary Conditions

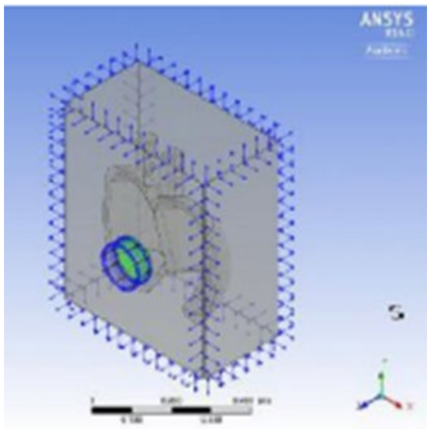
Multiphasic, transitory, and non-homogenous simulations are performed with a total simulation time of 4 seconds and a timestep of 0.01 seconds [19]. The phases used are air at 25 °C and water, which are configured as continuous fluids. The reference pressure was 1 atm. The free surface model is configured as 'Standard' to use the surface tension coefficient, which value was 0.074 N/m [18]. The turbulence model was k-ε. In the initialization, the proportions of water-air were defined as 0.01% and 99.9%, respectively. The three boundaries for the simulation are shown in Figure 11.



a) Walls



b) Opening



c) Inlet flow

Fig. 11 - Boundaries

The gravity is set towards the -Z axis, with a magnitude of 9.8 m/s2. The Buoyance Reference Density had a value of 1.2 kg/m3. At the inlet of the fluid, a speed of 47 m/s [20] and the proportions water-air of 99.9% and 0.01% are set, respectively. The walls of the bucket are considered as 'No Slip Wall', and an atmospheric pressure at the opening is set, with a relation water-air defined by '*Zero Gradient'*

3. Results

The simulations were performed in a WorkStation with 8 cores working in parallel, with a RAM memory of 12 GB and operative system of 64 bits. These simulations took around 120 hours to successfully finish in the described WorkStation. The flow behavior can be appreciated in Figure 12. The bucket is not rotating, but the behavior of the simulated system (streamlines and flow on the surface of the bucket) is like reality [3]. The geometry with splitter angle of 23° generated a higher force than the others, which coincides with the results described by Vélez et al. [13] and Chukwuneke et al. [9] in their works, demonstrating that for splitter angles higher than 25°, the force will start to decrease, so increasing the splitter angle is not necessary. Figure 13 shows the behavior of the force depending on the geometry. Between the angles 17 and 21, a decrease in the force can be appreciated, but for the studied range, the peak force is reached by the geometry with splitter angle of 23°. This behavior suggests an experimental validation, if possible, with the same geometries to check the generated changes in the CAD models during the design process and how they affect the results.

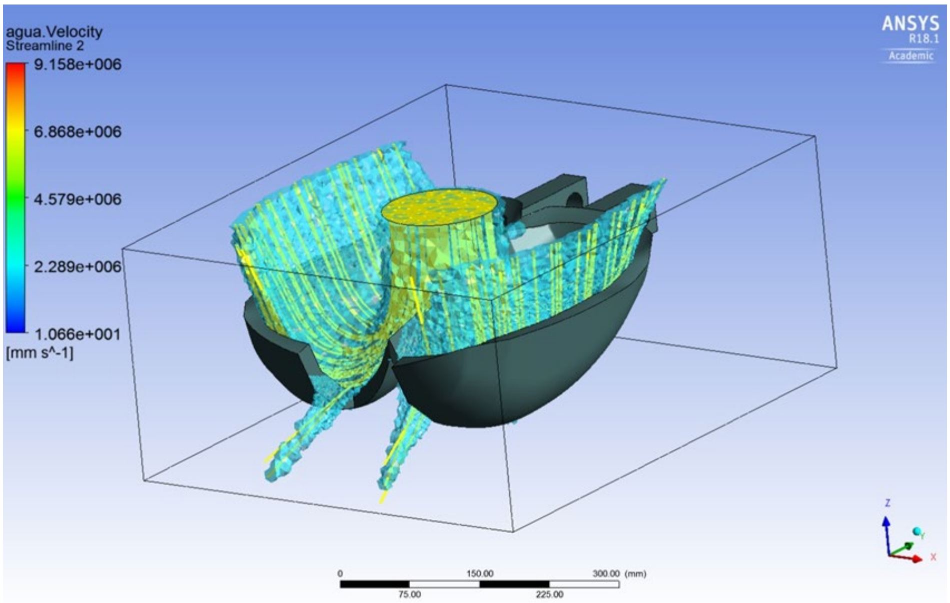


Fig. 12 - Water flow in the bucket

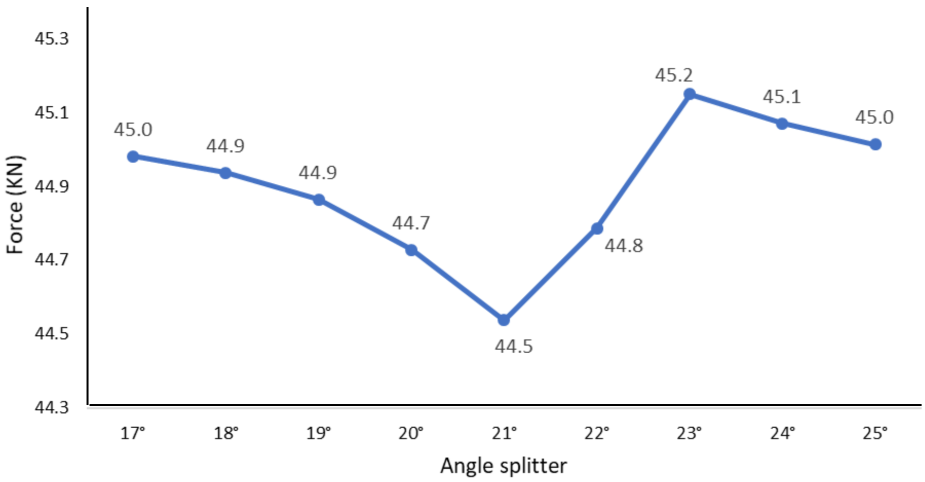


Fig. 13 - Force generated

Figure 14 shows the generated pressure on the surface of a bucket. The distribution of the pressure was similar in all the geometries, but of course, the values were different. Several regions deal with high pressures, due to the high speed of the water at the impact moment. If a splitter angle below 17° generates a higher force than the simulated geometries, then the deterioration in the bucket must be studied to determine if the result is acceptable, according to a balance-cost study. Currently, there is no general or global geometry of the Pelton bucket, so the investigators use their own designs to perform their studies. This means that the most effective configuration in one work may won't match with the results in another one, being necessary to study independent parameters (like the splitter angle) that affect the efficiency of the turbine, to determine a recommended range of sizes for each dimension and start developing parametrizations of the Pelton turbine to determine an ideal geometry [3] [12].

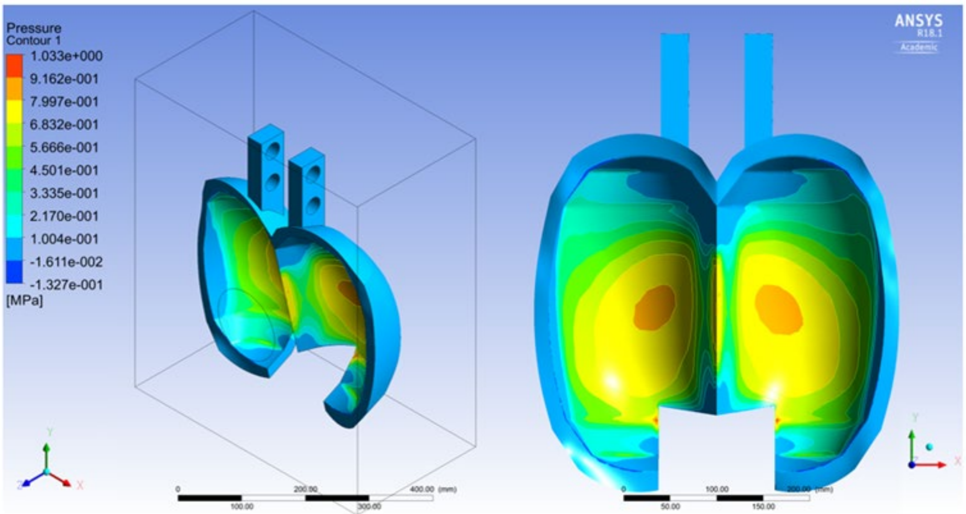


Fig. 14 - Distribution of pressure on the bucket

4. Conclusions

The splitter angle of 23° was the most suitable for the geometry used in this work. The obtained results can be corroborated with the literature, which allowed the construction of Table 1 that eased the beginning of this investigation, offering reference values for the splitter angle and showing that ANSYS CFX determines reliable values. This study verified if the splitter angle is higher than 23° in this geometry, then the force will begin to decrease. Thirteen CAD models were successfully developed. Their design time was short, so the production of four geometries that were not simulated did not affect this work. Nine geometries were successfully meshed, showing decent mesh statistics which made the geometries suitable to simulate. Simplifying the geometry and the mesh can generate faster results, however, exaggerating in the meshing process will affect the reliability of the obtained results.

Acknowledgement

The ITM deserves high praise, as this institution invests in databases where its researchers can search for useful information to develop their work.

References

[1] A. Ruiz Sánchez, J. A. Sierra Del Rio, A. J. Guevara Muñoz and J. A. Posada Montoya, "Numerical and Experimental Evaluation of Concave and Convex Designs for Gravitational Water Vortex Turbine," Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, vol. 64, no. 1, pp. 160-172, 2019.

[2] L. A. Pelton, "Water-Wheel. Patent". USA Patent 409,865, 27 Aug. 1889.

[3] A. Perring, "Hydrodynamics of the free surface flow in Pelton turbine buckets," Lausann, EPFL, Vols. DOI: 10.5075/epfl-thesis-3715, no. 1, 2007.

[4] L. Ferrada, "Diseño de Rodete de Turbina Hidráulica tipo Pelton para Microgeneración.," in Universidad de Chle, Chile, Facultad de Ciencias Físicas y Matemáticas, October, 2012.

[5] V. Gupta and V. Prasad, "Numerical Investigations for Jet Flow Characteristics on Pelton Turbine Bucket," International Journal of Emerging Technology and Advanced Engineering, vol. 2, no. 7, 2012.

[6] T. R. Bajracharya, A. Pachhain, A. Sapkota and A. Babu Timilsina, "NUMERICAL STUDY ON SEDIMENT EROSION OF MICRO PELTON TURBINE BUCKETS.," International Research Journal of Modernization in Egineering Techonology and Science, vol. 2, no. 07, July, 2020.

[7] V. Gupta, V. Prasad and R. Khare, “Numerical simulation of six jet Pelton turbine model.,” Energy, vol. 104, no. 1, 2016.

[8] A. KumarRai, A. Kumar and T. Staubli, “Effect of concentration and size of sediments on hydro-abrasive erosion of Pelton turbine.,” Renewable Energy, vol. 145, no. 1, January 2020.

[9] Chukwuneke, “Analysis and Simulation on Effect of Head and Bucket Splitter Angle on the Power Output of a Pelton Turbine,” International Journal of Engineering and Applied Sciences, vol. 5, no. 3, 2014.

[10] B. Solemslie and O. Dahlhaug, “Experimental methods and design of a Pelton bucket.,” Norwegian University of Science and Technology, https://www.researchgate.net/publication/304657563_Experimental_methods_and_design_of_a_Pelton_bucket, 2016.

[11] Z. Zhang, “Inlet flow conditions and the jet impact work in a Pelton turbine.,” Proceeding of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, vol. 223, no. 5, 2009.

[12] A. Židonis, A. Panagiotopoulos, G. A. Aggidi, J. S. Anagnostopoulos And D. E. Papantonis, “Parametric optimisation of two Pelton turbine runner designs using CFD.,” Journal of Hydrodynamics, Ser. B., vol. 27, no. 3, pp. 403-412, 2015.

[13] S. Velez, J. A. Sierra and D. A. Hincapié, “Análisis numérico de la influencia del ángulo de apertura de la cresta del cangilón en el torque generado por una turbina Peltón.,” CIBIM 2015, Cartagena, 2015.

[14] J. Chukwuneke, C. Achebe and M. NWOSU, “Analysis and Simulation on Effect of Head and Bucket Splitter Angle on the Power Output of a Pelton Turbine.,” International Journal of Engineering and Applied Sciences, vol. 5, no. 3, 2014.

[15] S. Yadav, “Some Aspects of Performance Improvement of Pelton Wheel Turbine with Reengineered Blade and Auxiliary Attachments,” International Journal of Scientific & Engineering Research, vol. 2, no. 9, pp. 1-4, 2011.

[16] R. Kumar and R. Praveen, “CFD Analysis of Pelton Bucket,” International Journal of Mechanical and Production Engineering and Development (IJMPERD), vol. 8, no. 1, 2018.

[17] K. Shogenji, “On the inclination of the Ridge of Buckets on a Pelton Wheel.” http://www.frenchriverland.com/design_of_pelton_buckets.htm, March 1927.

[18] C. Ansys, “ANSYS CFX-solver theory guide,” ANSYS CFX Release, 2009.

[19] A. Panthee, B. Thapa and H. P. Neopane, "CFD Analysis of Pelton runner," International Journal of Scientific and Research Publications (IJSRP), vol. 4, no

[20] A. Sharma, P. Sharma, A. Kothari, Numerical Simulation of Pressure Distribution in Pelton Turbine Nozzle for the Different Shapes of Spear, International Journal of Innovations in Engineering and Technology, Vol 1, Issue 4, December 2012.