VISUALIZATION FLOW ON THEPELTON TURBINE BUCKET BY CFD

PhD. Sedat Yayla[1], Azhin Abdullah[1]
[1]Yüzüncü Yıl University Faculty of Engineering Department of Mechanical Engineering, 65080 Central, Van,
syayla@yyu.edu.tr, azhinshanya@gmail.com

Abstract*: In this study, series of CFD simulation were carried out for two phase flow (free surface), three dimensional geometry and turbulent flow through the pelton turbine. The present investigation employed on the pelton turbine bucket models various splitter angle and inlet velocity values for visualizing the flow pattern and identifying the force on the bucket. In this study, Two different bucket models were applied at different inlet velocity (20, 25, 30,35and 40m/s) and four different splitter angle (55, 75,90and 115) for finding the effect of every single parameter on the effective force on the bucket. The obtained results discovered that there is a linear relationship between force and inlet velocity on the bucket. It uncovered that relationship between splitter angle and force on the bucket is linear until 90 degree after that this relationship is a non-linear.*

Keywords: CFD, PELTON TURBINE, FREE SURFACE FLOW, VOF, BUCKET DESIGN

1.Introduction

The most commonly used turbine for high head power plant is Pelton wheel. Because of change of impulse (momentum) of moving wateras opposed to its weight like traditional overshot water wheel,The Peltonwheel extracts energy. Even though deviations of impulse turbines presented prior to Pelton design, they were less effectual than Pelton's design, the water leaving these wheels typically still had high speed, and carried away much of the energy.

Thus, all engineers,designers and researchertrytoobtain high efficiency by adjusting the bucket and nozzle. This researchapplied two different bucket models of the pelton turbine for identifying the visualization fluid flow andforce on the bucket by modifyingseveral working conditions and geometrical parameters.An example of working conditions is the inlet velocity which affects the force and visualization fluid flow. Furthermore, the instance of the geometrical parameter is the splitter angle between nozzle and bucket which impacts the value of the force on the bucket of peloton turbine. The jet of circular cross section is distributed from nozzle and transfers in air before striking the bucket in Pelton turbines. The bucket is separated into two symmetrical semi ellipsoidal cups via a sharp edge splitter. The jet strikes the bucket on the splitter. The jet is alienated into two similar sheets of water having free surface which transfers on the curved path of bucket by the splitter. The profile of curved path of bucket impacts the force, pressure and velocity distribution over bucket.For instance, only Kvicinsky ,S., 2002, compared the calculated pressure distribution experimentally. However,another researcher applied a stationary pelton turbine bucket as water impinged itexperimentallyGrozev et al 1988.The presentinvestigationimplemented computational fluid dynamic and some CFD models like VOF model and K-epsilon model.Severalresearchersexamined the steady bucket like Janetzky et al., 1998.The coming out water of the nozzle is circular in cross-section. The water jet moves liberally in air and executes on a bucket of the runner thusrotations occur. For better comprehending the flow, Visualization was helpful. The resultsdemonstrated at the end of this examination, are utilized for providing vision into the benefit we can expect of stable–state calculations of the flow throughout the universal process of pelton design optimization.

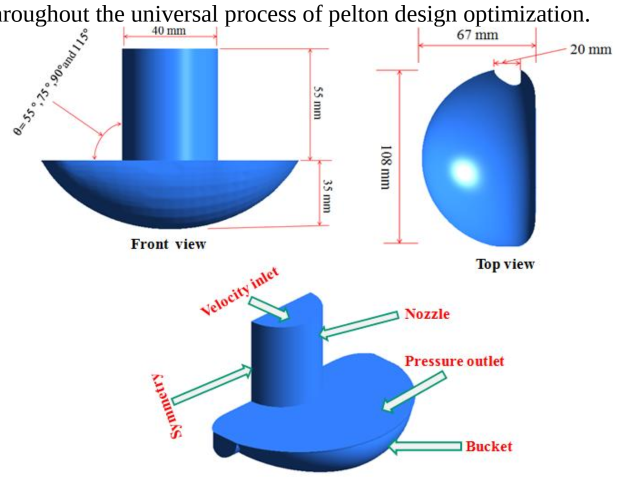

Figure 1.*Solution domain and boundary conditions of the model 1.*

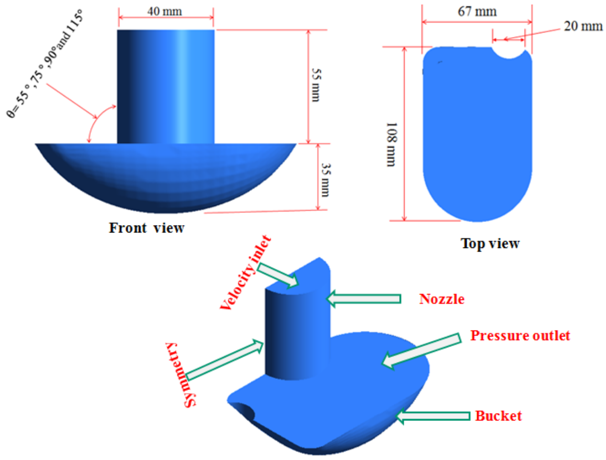

Figure 2 .*Solution domain and boundary conditions of the model 1.*

2.Materials and Methods

This investigationappliedANSYS Workbench to generate the bucket of pelton turbine geometry as indicated in Figure 1 and 2. Specifically, Design Modular module of ANSYS was utilized. Concisely, in this work two different models were investigated and both models had 50% width of the bucket and diameter of the nozzle of 67 mm and 40 mm correspondingly. The influence of force on bucket pelton turbine shapes and visualization fluid stream on the bucket was evaluated in this examination. In this study four various splitter angle (55, 75, 90, and115 degree) and different inlet velocity (20, 25, 30, 35 and 40 m/s) were employed. Grounded upon mesh independence test, the volume of ahighest mesh was determined. The minimum mesh size was set to be less than the first layer thickness close to the wall and the lowest mesh sizes were determined to be 0.000019m to avoid incorrect automatic mesh generation in the solver. Figure 3and 4 illustrated the full size of mesh elements for both models. The right and left sides of the domain were identified as pressure outlet and velocity inlet respectively. Figure 1 and 2demonstrated that when the computational domain was generated, top and bottom sides were wall. In this research, For determining force on the bucket and visualization of the flow computationally, the commercial software ANSYS FLUENT was applied.

The sweep method isa new technique in ANSYS-FLUENT utilizedfor capturing the free surface flow profile of flow. It can be reported that higher mesh gives force on the bucket to be 3319.6 N rather than escalating number elements do not affect the accuracy. By the way, the difference in the mesh independent test can be up to 4 % or5% relying on nature of the issue which demonstrated in table 1. The aim of this technique is to refine the mesh size along the direction normal to the flow direction.

***Table 1.** Denotes Investigation of mesh dependence.*

Number of element	Force on the bucket(N)	Difference (%)
331694	1310	2.6
658152	1324	1.56
966322	1335	0.74
1299188	1345	0

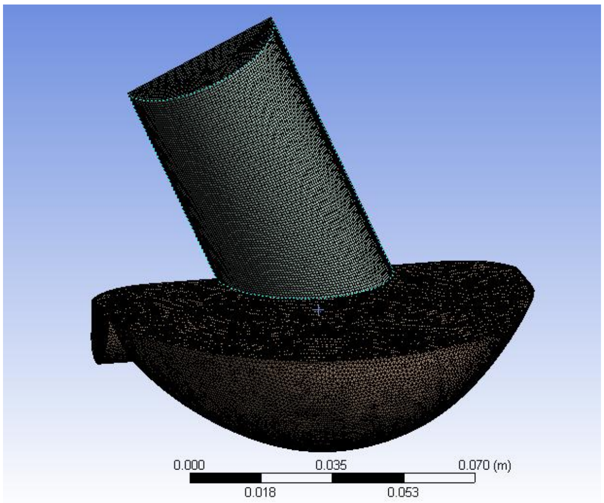

***Figure 3.** Computational mesh for Model 1.*

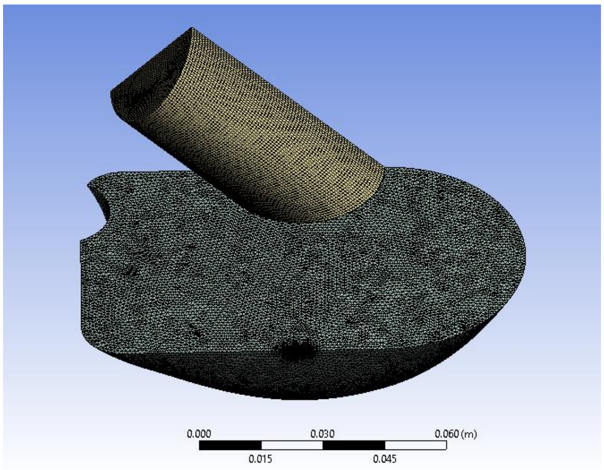

***Figure 4.** Computational mesh for Model 2.*

Mathematical model and equations utilizing of Reynolds number;

Must be known to determine the regime of flow turbulent or laminar. The characteristic length of geometry, average velocity and fluid properties are effecting the transition from laminar flow to turbulent flow. Generally, Re can be expressed as for two phase flow.

$Re=(Inertial\ forces)/(viscous\ forces)=VD/v=\rho vl/\mu$ (1)

μ Viscosity of fluid.
ρ Density of fluid.
L length
V velocity of fluid

The governing equations of viscous flow are based on conservation of mass, momentum and energy which are langrangian in nature. The governing equations are expressed using equations .

Conservation of mass $\partial p/\partial t+\rho \nabla V=0$ (2)

Conservation of Momentum $\rho\ Dv/Dt=\rho g+\nabla \cdot \tau i\acute{j}-\nabla p$ (3)
Conservation of Energy $\rho\ Dh/Dt=Dp/Dt+\nabla(K \nabla T)+\Phi$ (4)

3.Result and Discussion

The Effect of outlet Pressure

Hydraulic design in both a single flow and two-phase flow systemsis extremely impacted by pressure. In the presentexamination, pressure gauge is a substantial elementfor determining outlet boundary pressure particularly in the pelton turbinePressure is an important element in characterizing the flow condition and designing the bucket. At the beginning pressure was high then it declined progressively at the edge of the bucket. Pressure was generated on the bucket as the water crushed from the nozzle to the bucket. The impact of high jet resulted in the pressure distribution of in the bucket. Jet strike was increased at the inlet velocity because of the pressure distributions on the bucket. The water volume fraction contour clearlydemonstrate the water strike on the bucket. As illustrated in figure (5), once the Reynolds number was 1194017.9 and splitter angle was 115 of the model 1 highest force and pressure on the bucket was 2230.7N and 922141.4 Pascal correspondinglyThese outcomes are similar tofindings of (Gupta and Prasad, 2012) and (pellone and maitre 2006).

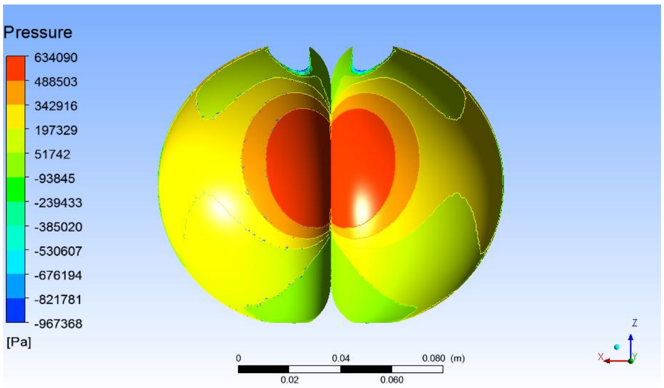

Figure 5.\ *Variation of pressure when the splitter angle = 115degree and velocity =30 m/s of the model 1.*

Figure (6) displays that by applying the Reynolds number was1194017.9 and splitter angle was 115, the highest force and pressure on the bucket was 2442N and 899829.6 Pascal correspondingly in the model 2. Pressure is an important component for designing and characterizing the flow condition of the bucket. The water crush on the bucket is evidently demonstrated in water volume fraction. Owing to the pressure distributions on the bucket, Jet strike was escalated at the inlet velocity. At the beginning pressure was high then it declined progressively at the edge of the bucket. The effect of high jet caused the pressure distribution in the bucket. Pressure was generated on the bucket as the water crushed from the nozzle to the bucket.These findings agree with (Gupta and Prasad, 2012) and (pellone and maitre 2006).

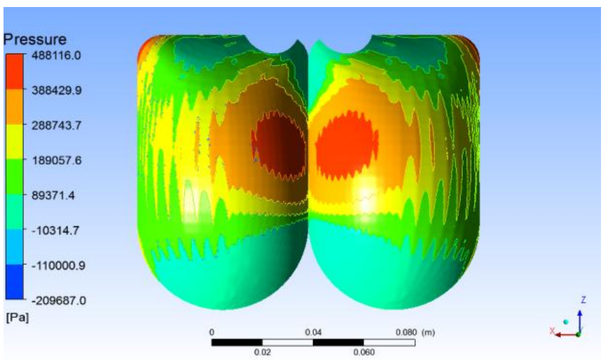

Figure 6.\ *Variation of pressure when the splitter angle = 115degree and velocity =30 m/sof the model 2.*

Water volume fraction

Water volume fraction contour illustrates the water phase and air phase on the bucket. As the flow is a turbulent in this examination, the water volume fraction is highly significant for the boundary impact of the pressure and force on the bucket. Therefore, the modification of the splitter angle resulted in change of the shape of the boundary. In general, red color represents a water phase, when the boundaries effective of highest pressure and big force on the bucket. Furthermore, the rain bow color demonstrates water and air phase as mixture. Moreover, the blue color displays an air phase which is not under pressure and force. It is observed from figure (7) when Reynolds number was 1194017.9 and splitter angle was 115 of the model 1 the highest force on the bucket was 220.7 N and pressure was 922141.4Pascal.

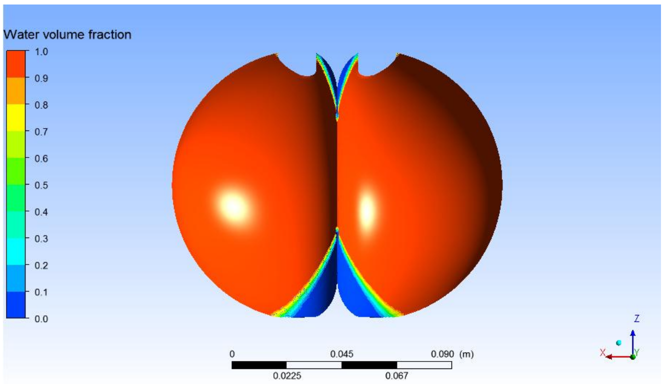

Figure 7.*Water volume fraction when the splitter angle = 115degree and velocity =30 m/sof the model 1.*

Generally, the blue color indicates an air phase which is not under pressure and force. Furthermore, when the boundary effective of highest pressure and big force on the bucket, red color demonstrates a water phase. Besides, the rain bow color demonstrate water and air phase as mixture. The water volume fraction is extremely important for the boundary effective of the force and pressure on the bucket because the flow in this study is turbulent. In the model 2, as illustrated in figure (8), When Reynolds number was 1194017.9 and splitter angle was 115 the highest force and pressure on the bucket were 2442N and 899829.6Pascal correspondingly. The Water volume fraction contour demonstrates the air phase and water phase on the bucket. Consequently, the change of the form of the boundary is resulted from alteration of the splitter angle. These findings agree with (Gupta and Prasad, 2012).

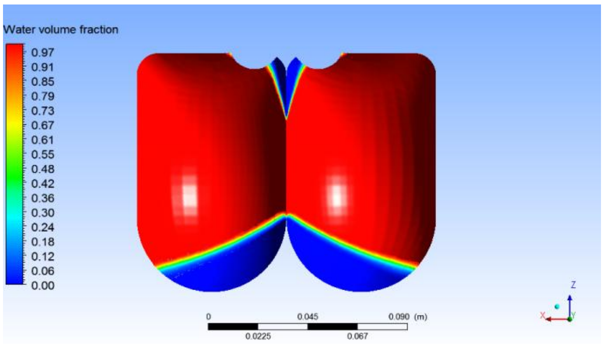

Figure 8.*Water volume fraction when the splitter angle = 115degree and velocity =30 m/sof the model 2.*

Velocity vector

It is the direction of the velocity similar as the velocity magnitude. It demonstrates the proportion of the change of the fluid position. Velocity vector is the maximum compared to the velocity in all position in the bucket of the pelton turbine.

As demonstratedinvector pattern, the jet has parallel vector linesbe force crushing the bucket. Owing to deflection of jet, the vector lines spread near the bucket. The jet spread, after striking the bucket. It can be seen that the velocity at the inlet was high then decreased gradually, as displayed in figure (9). Once splitter angle was 115 Reynolds number was 1194017.9 and of the model 1, the biggest force on the bucket was 2442N.These findingsagree with (Gupta and Prasad, 2012). Without the participation to impulse force, it was monitored that higher amounts of water leave throughout the cut-out which result in wastage.

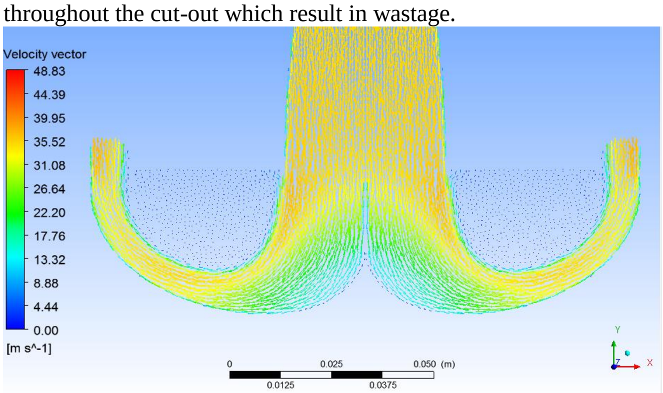

Figure 9, *Shows velocity vector when the splitter angle = 115degree and velocity =30 m/sof the model 1.*

Due to deflection of jet, the vector lines spread nearby the bucket.High velocity was observed at the inlet however it declined progressively in figure (10). The jet has parallel vector linesbeforecrushing the buckets displayedinvector pattern. Without the participation to impulse force, it was monitored that higher amounts of water leave throughout the cut-out which result in wastage The jet spread, after striking the bucket.In the model 2, themaximum force on the bucket was 2442Nas splitter angle was 115 and Reynolds number was 1194017.9.These outcomes are similar to (Gupta and Prasad, 2012).

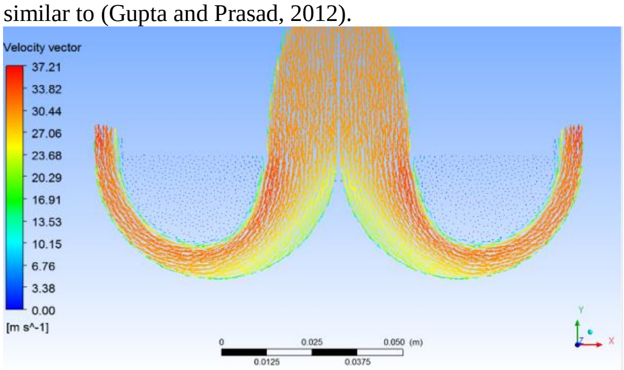

Figure 10, *Shows velocity vector when the splitter angle = 115degree and velocity =30 m/sof the model 2.*

The figure (11) displays that the study's outcomes are very close to the journal result. This is true when the fluid, model and geometries were the same. For finding the correct model for this research, this comparison has conducted. This journal has been chosen for discovering the accurate models for this research because it is close to our investigation. The results of both researches are close to each other. Thus, it can be stated that the model of this research is accurate.

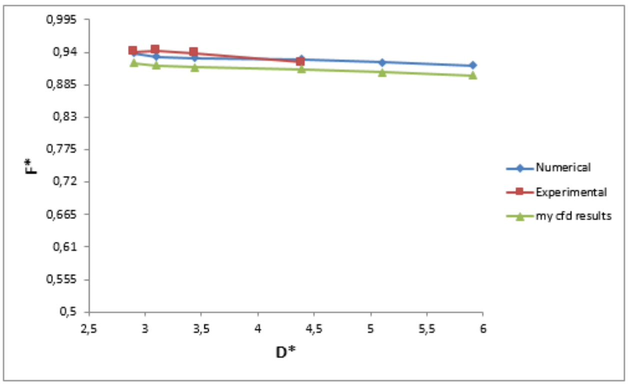

Figure 11.*Shows the difference between my CFD result and journal papers result (zoppe et al., 2006).*

When the splitter angle is 115 degree, the correlation between Reynolds number and the maximum Force on the bucket isdisplayed in the Figure (12). It is visible from the figure that relationship between the maximum force on the bucket and Reynolds numbersis positive. This means that the maximum force on the bucket increased steadily by increasing the Reynolds number. It can be seen thata Reynolds number1592023.9 results in a maximum force on the bucket of 3972 N. In addition, when the Reynolds number is 796011.96, the force on the bucket is 992.8.

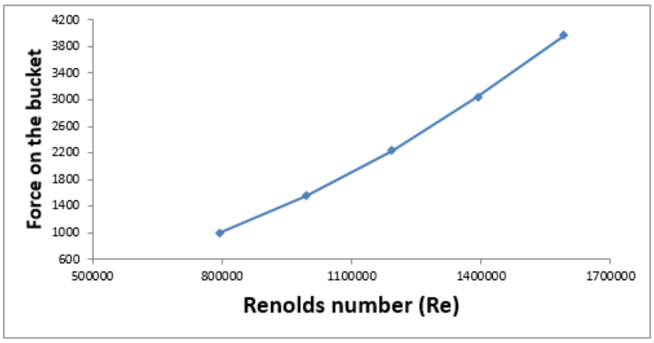

Figure 12.*Influenceof the Reynolds number on force when the splitter angle 115 deg of model1.*

4. Results

In high head and low water stream, thePelton turbine was performed because it is simple construction and ease of manufacturing. The turbine parameters have to be included in the designing processfor obtaining a Pelton hydraulic turbine with the maximumefficiency different working circumstances. The ANSYS-FLUENT was applied for the simulations of liquid flow. The VOF CFD models were employed for the two different models of the bucket. The results of pressure contour, stream line velocity, and force on the bucket, water volume of fraction and velocity vector were obtained from CFD simulations models. The simulation of visualized stream on the bucket and determined a force on the bucket in pelton turbine for four different inlet velocity (20, 25, 30, 35, and40m/s) for both models. Moreover, fourdifferent splitter angles(55, 75, 90 and 115degree), were utilized for both models. In the present study, two diverse bucket models were implemented at different inlet velocity (20, 25, 30,35and 40m/s) and four diverse splitter angle (55, 75,90and 115)for discovering the effects of every parameter on force on the bucket. It was revealed there is a linear relationship between inlet velocity and the force on the bucket. The examination revealed that until 90 degree, the relationship between splitter angle and force on the bucket is linear. However, subsequent the 90 degree, there was a non-linear relationship.

5. Resources

ANSYS fluent guide (ANSYS FLUIENT), theory and guide in (2009)

ANSYS CFX Modeling Theory, Version 14.0, 2011.

B.zoppe ,C.Pellone,T.Maitre,P.Leroy "Flow Analysis Inside a PeltonTurbine Bucket" Vol.128,July 2006.

V.Gupta,V.Prasad "Numerical Investigations for Jet Flow Characteristics on Pelton

Turbine Bucket"ISSN 2250-2459, Volume 2, Issue 7, July 2012

Ujamj.A, Chukwunekej.L, Achebe.C.H, IkwuG.O.''Modeling the effect of head and bucket Splitter angle on the power output of a pelton turbine.Aug. 2014. Vol. 5.No. 03.

NkhilJ.G,Sebin.S,Kevin.R.J,Ashwin.C.A,"Static Analysis On Pelton Wheel BucketStructural Deformation and Material Property Study"Vol. 3 Issue 3, March – 2014.

Sanam P,Hari.Prasad ,Amod P,AnujP,andBhojBahadur C,"Computational Fluid Dynamics (CFD) analysis of Peltonrunner of Khimti Hydro-power Project of Nepal"Rentech Symposium Compendium, Volume 4, September 2014

Kvicinsky, S., 2002, "Méthoded'Analyse des Écoulements 3D à Surface Libre: Application aux Turbines Pelton," Thèse No. 2526, EcolePolytechnique

Fédérale de Lausanne.

Janetzky, B., Göde, E., Ruprecht, A., Keck, H., and Schärer, C., 1998, "Numerical Simulation of the Flow in a Pelton Bucket," Proceedings of 19th IAHRSymposium, Singapore, pp. 276–283.

Grozev, G., Obretenov, V., and Trifonov, T., 1988, "Investigation of the Distribution of Pressure Over the Buckets of a Pelton Turbine," Proceedings of the

Conference on Hydraulic Machinery and Flow Measurements, Sept. 13–15,

Ljubljana, Yugoslavia, Turboinštitut, Ljubljana, pp. 119–125.

Abhishek Sharma,Prashant Sharma,Anil Kothari (2012)"Numerical Simulation for Pressure Distribution in Pelton TurbineNozzle for the Different Shapes of Spear"InternationalJournal of Innovations in Engineering and Technology (IJIET)

Dr. SedatYayla is an associate professor and head director of mechanical engineering at the University of YuzuncuYil University Van-Turkey.

Dr. Ali BahadirOlcay is an assistant professor of mechanical engineering department at the Yeditepe University in Istanbul-Turkey.

Azhin Abdullah graduated from the University Of Salahadding the department of mechanical engineering in Erbil Iraq. Currently he is a master student at the YuzuncuYil University Van-Turkey