BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 66 (70), Numărul 3, 2020
Secţia
CONSTRUCŢII DE MAŞINI

PELTON NOZZLE PRESSURE LOSS COEFFICIENT - A CFD ANALYSIS

BY

BOGAN CIOBANU[1,*] and MĂLINA CIOBANU[2]

[1]“Gheorghe Asachi” Technical University of Iaşi,
Department of Fluid Mechanics
[2]“Gheorghe Asachi” Technical University of Iaşi,
Bachelor student

Received: July 17, 2020
Accepted for publication: September 25, 2020

Abstract. The determination of the pressure loss coefficient for the elements of installations with complex geometry is usually done experimentally. In the case of the Pelton turbine nozzle this is difficult because, by its construction, the injector is not an on-line hydraulic resistance but an end-one. The paper proposes a method for determining the pressure loss coefficient for a Pelton turbine nozzle, using CFD analysis. The method consists in determining the pressure distribution in the inlet and outlet sections of the Pelton injector and the calculation of the pressure loss coefficient in relation to the average velocity of the fluid flow in the inlet section of the nozzle. Both the pressure and the speed on each sections of the nozzle are determined by CFD analysis, performed with the FluidFlow package integrated in Solidworks. The analysis was performed for a Pelton micro-turbine nozzle.

Keywords: Pressure loss coefficient; hydraulic micro-turbine; Pelton turbine nozzle; numerical simulations (CFD).

[*]Corresponding author; *e-mail*: bogdancioban@tuiasi.ro

1. Introduction

The process of obtaining electricity based on the hydraulic energy available for a given location is an ongoing challenge. Although the hydropower potential of Romania is widely used by the existing large hydropower facilities, there is still room for the energy systems equipped with micro and pico hydraulic turbines. The Pelton turbine, which is the subject of our study, can be included into last category.

An important step in the design and use of an impulse turbine consist of flow simulation of phenomena that occur in both rotor (jet-bucket impact) and stator device (nozzle). There are researches in terms of flow phenomena both in the nozzle and in the rotor. Most of these researches are about the runner efficiency and the impact between the jet and the bucket (Catanase *et al*., 2004; Perrig *et al*., 2006; Zoppe *et al*., 2006; Santolin *et al*., 2009; Rosetti *et al*., 2013; Chukwuneke *et al*., 2014; Panthee *et al*., 2014; Zidonis *et al*., 2015; Zidonis and Aggidis, 2016; Nigussie *et al*., 2017). Many of these researches focused on high and medium head turbines. Also, there are researches regarding the flow in nozzle (Sharma *et al*., 2012; Sharma *et al*., 2013; Zhang *et al*., 2018; Han *et al*., 2019). The last one analyzes the flow in a 6-nozzle Pelton turbine.

This paper presents a numerical simulation (CFD) made for a Pelton turbine nozzle in order to estimate the pressure loss coefficient of the nozzle.

2. Simulation Case

The numerical simulations are focused on the analysis of the flow behavior inside and after the nozzle of a Pelton turbine.

The purpose of simulating the flow through the nozzle of a Pelton turbine was, on the one hand, to determine the pressure in the inlet and outlet sections of the nozzle and, on the other hand, to estimate the fluid velocity distribution in the impact zone with the bucket.

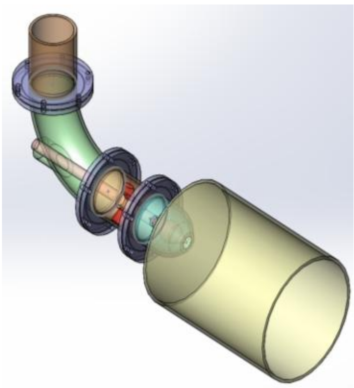


Fig. 1 – 3D-Model for the Pelton turbine nozzle and the hydraulic circuit.

The 3D-model of the nozzle and the hydraulic circuit was created using SolidWorks package programs (Fig. 1).

The analysis was performed for the particular case of the adjusting needle being placed with the tip in the outlet section of the Pelton nozzle.

The main parameters of the computational domain are shown as below:

Computational domain size:

X min	-0.358 m	Y min	-0.115 m	Z min	-0.101 m
X max	0.324 m	Y max	0.235 m	Z max	0.111 m

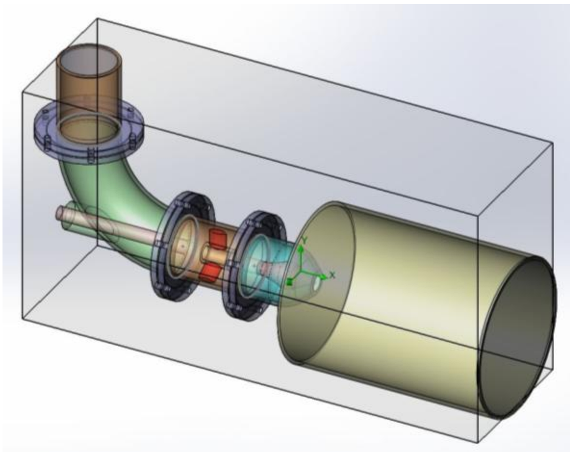


Fig. 2 – Computational domain size.

Basic Mesh Dimensions:

Number of cells in X	100	Number of cells in Y	75	Number of cells in Z	70

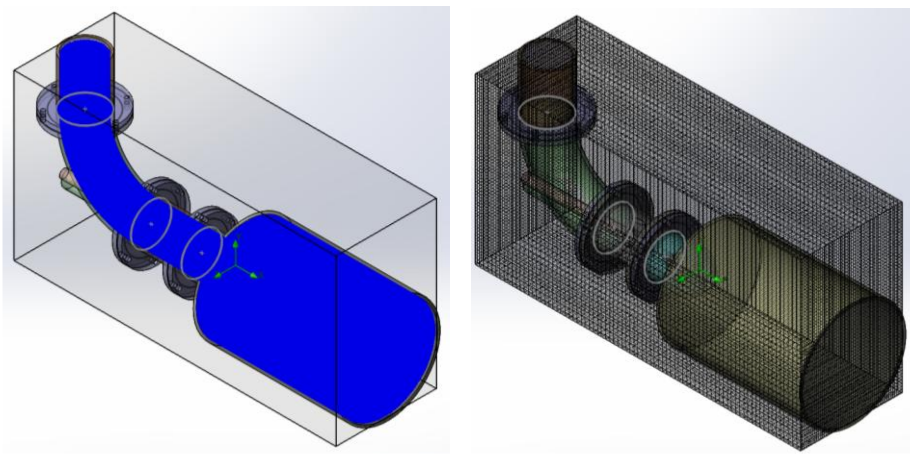


Fig. 3 – Flow domain and global mesh.

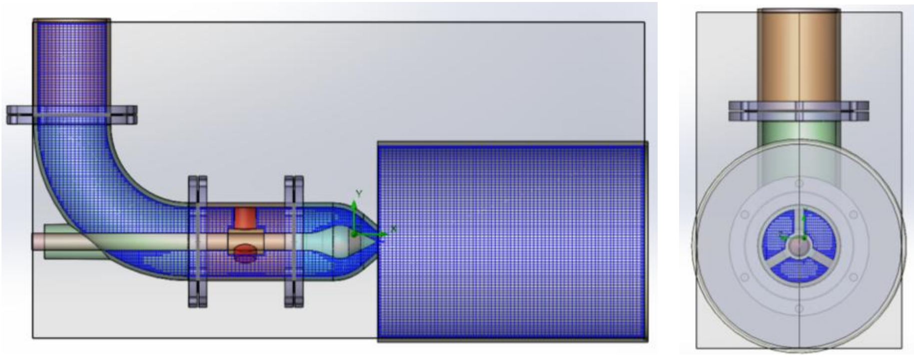


Fig. 4 – Fluid cells mesh in the main longitudinal and a cross-sections.

Number of cells:

Total cells	217806
Fluid cells	428827
Solid cells	76852
Partial cells	0
Irregular cells	0
Total	723485

Solid/Fluid Interface

Small solid features refinement level	5
Curvature refinement level	1
Curvature refinement criterion	0.505 rad
Tolerance refinement level	3
Tolerance refinement criterion	0.0005 m

Initial and boundary conditions

Initial conditions:

Static Pressure: 101325.00 Pa
Temperature: 293.20 K
Turbulence model: k-ε standard

Turbulence parameters	Turbulence intensity and length Intensity: 3.50 % Length: 0.0028956 m

Gravitational effects: On

X component	0 m/s^2
Y component	-9.81 m/s^2
Z component	0 m/s^2

Working fluid: Water

Boundary conditions:
Inlet Volume Flow parameters:
  Flow vectors direction: Normal to face
  Volume flow rate normal to face: 0.0100 m^3/s
  Fully developed flow: Yes
Boundary layer parameters:
  Boundary layer type: Turbulent
Static pressure at the outlet: 101325.00 Pa

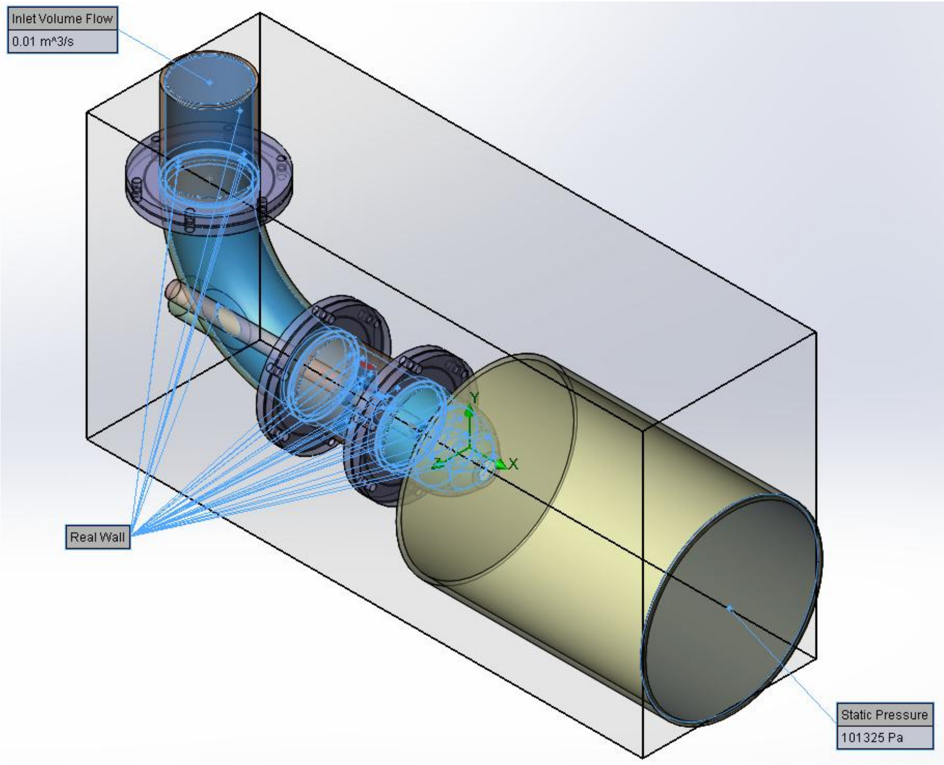


Fig. 5 – Inlet, outlet and wall conditions.

All components of the nozzle (flow adjusting needle, needle rod, constant and variable diameter parts of the nozzle, guide bush and their supports, elbow, adduction pipe) are defined as a real wall with the roughness of 60 µm.

3. Method and Results

The simulations were executed using a CFD program with taking into consideration of turbulence model k-ε.

For the CAD model of the nozzle the dimensions are similar with the Pelton turbine experimental model that exists at the Fluid Mechanics, Hydraulic Machines and Fluid Power Laboratory from Technical University “Gheorghe Asachi” from Iaşi.

The main purpose of the numerical simulation is to obtain the value of the pressure loss coefficient for the designed Pelton nozzle.

$$\zeta = \frac{2\left(p_{out} - p_{in}\right)}{\rho V^2} \quad (1)$$

The significance of terms from relation (1) is: ζ is the pressure loss coefficient, p_{out} and p_{in} are the average pressures from the outlet and the inlet sections of the nozzle, ρ is the density of the working fluid (water) and V is the average velocity from the inlet section.

In order to obtain the value of the pressure loss coefficient for the Pelton nozzle directly from the numerical simulation, a specific goal was created (Fig. 6).

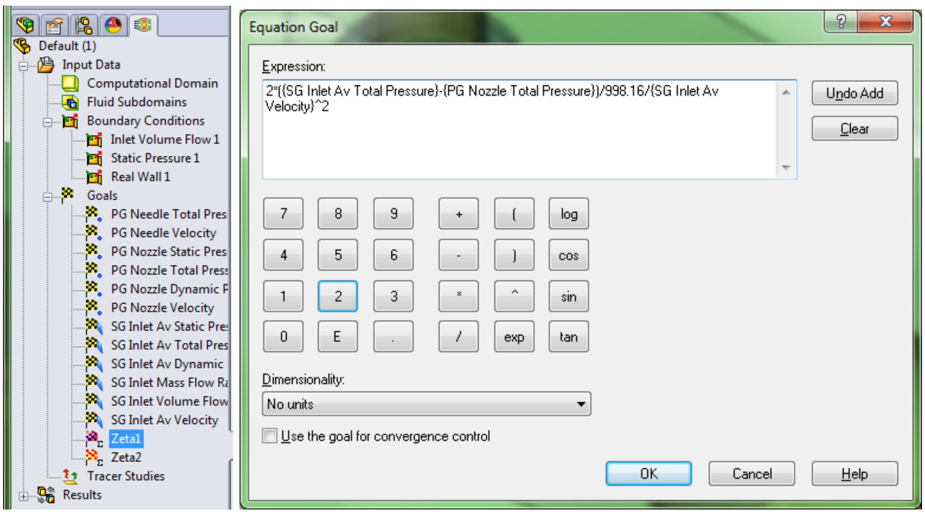


Fig. 6 – Pressure loss coefficient goal.

The values of the nozzle pressure loss coefficient goal and for all related goals are presented in the Table 1:

Table 1
Values of the Goals Obtained with the CFD Analysis

Goal Name	Unit	Averaged Value
SG Inlet Av Static Pressure	[Pa]	646674,6947
SG Inlet Av Dynamic Pressure	[Pa]	2443,875408
SG Inlet Av Total Pressure	[Pa]	649118,5701
SG Inlet Av Velocity	[m/s]	2,193291812
PG Nozzle Static Pressure	[Pa]	416795,2707
PG Nozzle Dynamic Pressure	[Pa]	214736,7935
PG Nozzle Total Pressure	[Pa]	631532,0642
PG Nozzle Velocity	[m/s]	20,67150809
SG Inlet Mass Flow Rate	[kg/s]	9,975617393
SG Inlet Volume Flow Rate	[m^3/s]	0,01
Zeta (pressure loss coefficient)	[ ]	7,325150639

The value of the pressure loss coefficient is similar to a control valve.

Figs. 7 and 8 present the velocity distribution inside the nozzle and Figs. 9 and 10 shows the streamlines of the particles through the nozzle and after that for the flow of 10 l/s.

A relatively uniform velocity field can be observed in the longitudinal section of the nozzle. This shows a hydrodynamically correct design of the nozzle.

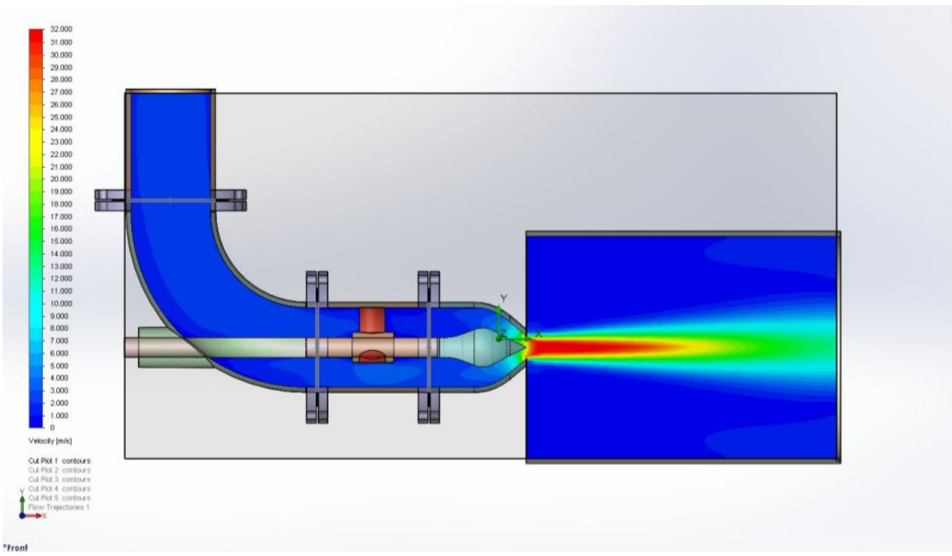


Fig. 7 – Velocity distributions.

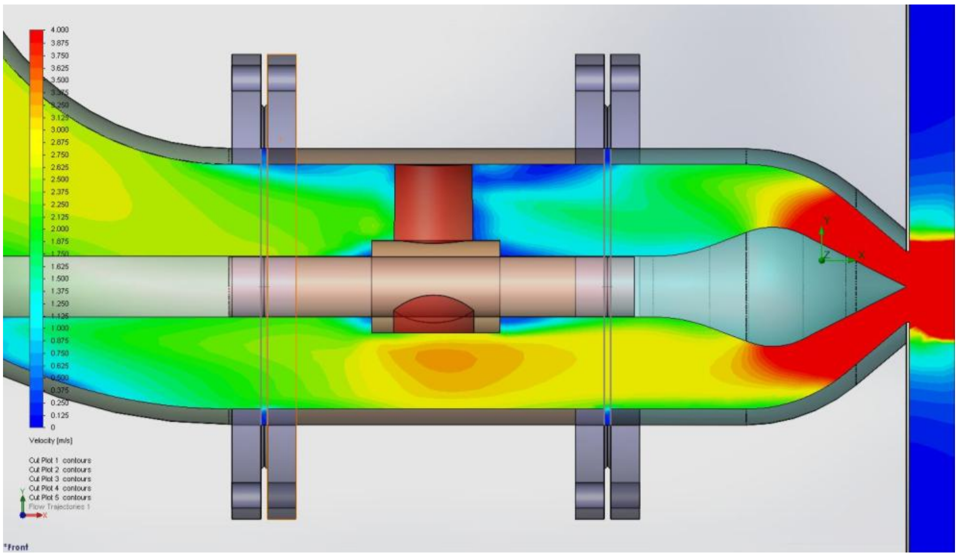


Fig. 8 – Velocity distributions (detail).

Variation range for speed and vorticity inside the nozzle is:

Velocity [m/s]	0	32.896
Vorticity [1/s]	0.031	27745.19

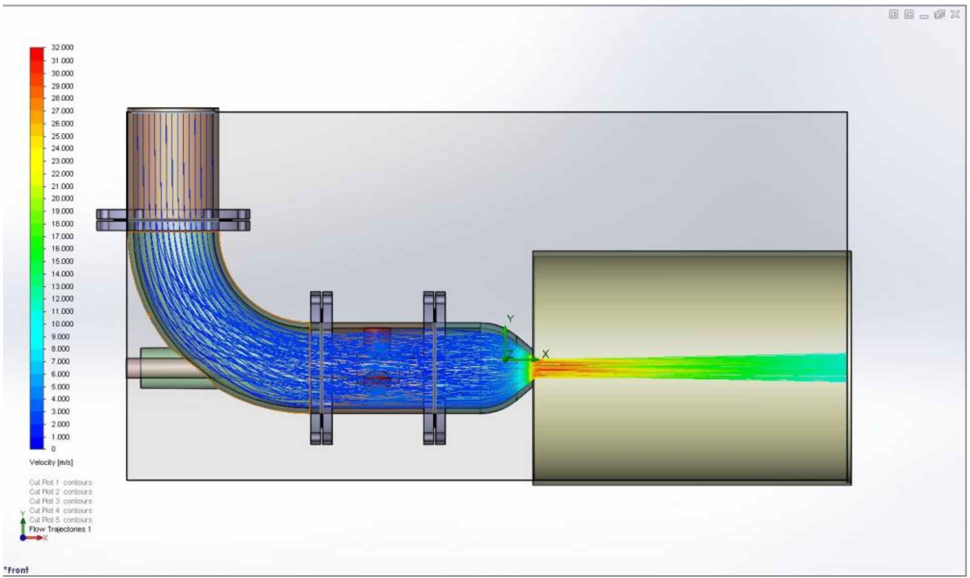


Fig. 9 – Streamlines.

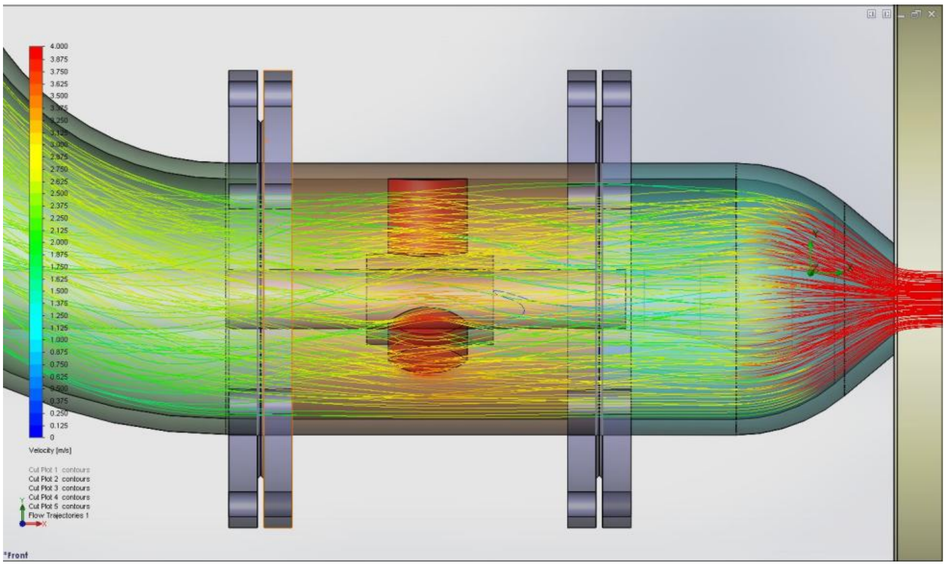


Fig. 10 – Streamlines (detail).

Fig. 11 presents the velocity distribution outside the nozzle and the position of four reference cross-sections and Fig. 12*a* and 12*b* shows the velocity distributions in these sections for the flow of 10 l/s.

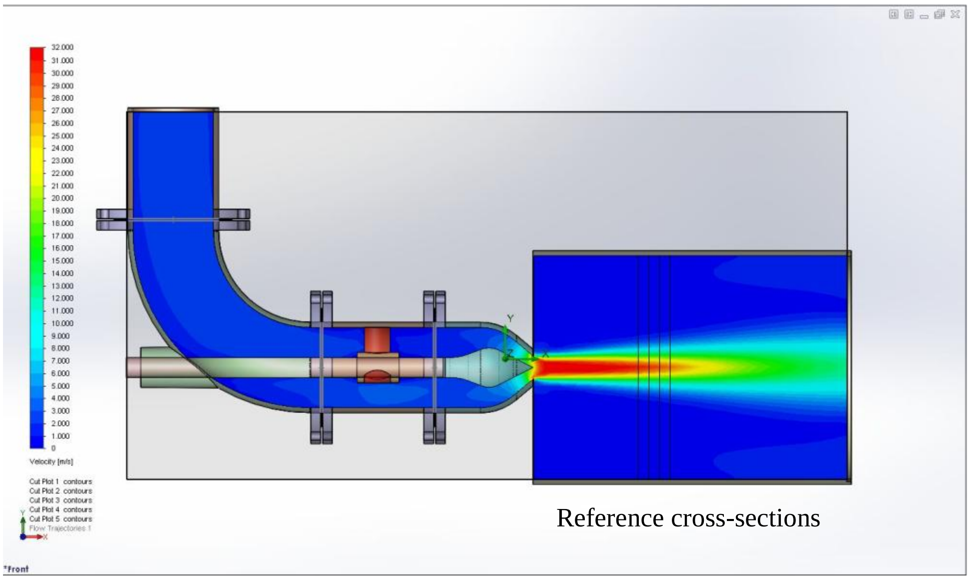
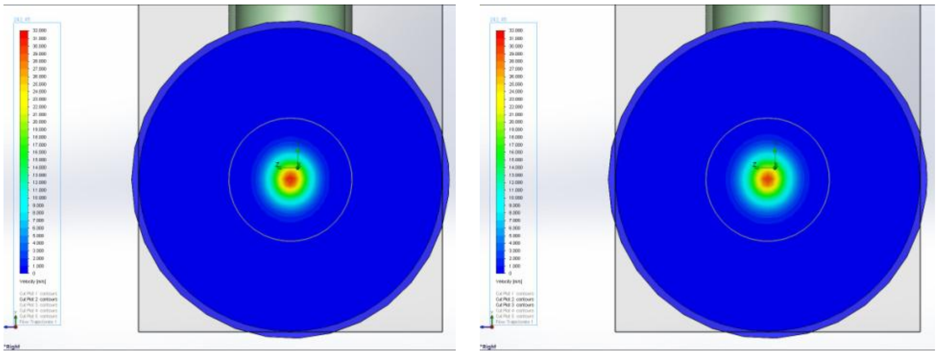
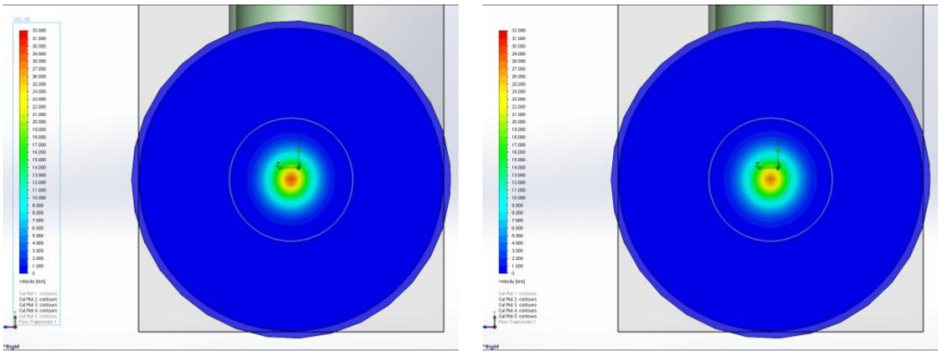


Fig. 11 – Position of reference cross-sections for velocity distributions at blade impact.



a



b

Fig. 12 – (*a*) – Velocity distributions for reference section 1 and 2; (*b*) – Velocity distributions for reference section 3 and 4.

In order to establish the decrease of velocity in the area of impact between the jet and the turbine bucket, we analyzed the velocity distribution in four cross-sections. As Fig. 12 shows, there is a decrease of velocity but the axial velocity remains at its maximum value.

4. Conclusions

The results of the analysis performed in this paper show a relatively uniform distribution of velocities in the designed Pelton nozzle. This shows that the nozzle design was well designed.

The fast increase in fluid velocity occurs only in the outlet area of the nozzle and the resulting jet maintains an approximately constant value of velocity at the center axis. This demonstrating the nozzle is well made in terms of impact between the jet and the bucket.

The nozzle pressure loss coefficient was determined using the pressure and velocity parameters obtained from the CFD analysis. The time and costs required to determine this coefficient can be substantially reduced by using the method proposed in the paper.

REFERENCES

Catanase A., Bărglăzan M., Hora C., *Numerical Simulation of a Free Jet in Pelton Turbine,* Scientific Bulletin of the Politehnica University of Timisoara, Transactions on Mechanics, Special issue HMH2004, 79-84 (2004).

Chukwuneke J.L., Achebe C.H., Nwosu M.C., Sinebe J.E., *Analysis and Simulation on Effect of Head and Bucket Splitter Angle on the Power Output of Apelton Turbine*, International journal of Engineering and Applied Sciences, **5**, *3* (2014).

Han L., Zhang G., Gong R., Wei H., Li W., *Physics of Bad-Behaved Flow in 6-Nozzle Pelton Turbine Through Dynamic Simulation*, 29th IAHR Symposium on Hydraulic Machinery and Systems, IOP Conf. Series: Earth and Environmental Science **240**, 9 pages, (2019).

Nigussie T., Engeda A., Dribssa E., *Design, Modeling and CFD Analysis of a Micro Hydro Pelton Turbine Runner*, Hindawi International Journal of Rotating Machinery Volume 2017, Article ID 3030217, 17 pages, (2017).

Panthee A., Thapa B., Neopane H.P., *CFD Analysis of Pelton Runner*, International Journal of Scientific and Research Publications (IJSRP), **4**, *8* (2014).

Perrig A., Avellan F., Kueny J.-L., Farhat M., Parkinson E., *Flow in a Pelton Turbine Bucket: Numerical and Experimental Investigations*, Journal of Fluids Engineering, **128**, *2*, 350-358 (2006).

Rossetti A., Pavesi G., Cavazzini G., Santolin A., Ardizzon G., *Influence of the Bucket Geometry on the Pelton Performance*, Proc. IMechE Part A: J. Power and Energy 0(0), 1-13 (2013).

Santolin A., Cavazzini G., Ardizzon G., Pavesi G., *Numerical Investigation of the Interaction Between Jet and Bucket in a Pelton Turbine*, Proc. IMechE Vol. **223,** Part A: J. Power and Energy, 721-728 (2009).

Sharma A., Sharma P., Khotari A., *Numerical Simulation for Pressure Distribution in Pelton Turbine Nozzle for the Different Shapes of Spear*, International Journal of Innovations in Engineering and Technology (IJIET), **1**, *4*, December 2012, 95-107 (2012).

Sharma A., Sharma P., Khotari A., *Numerical Simulation and CFD Analysis for Energy Loss Computation in Fully Open Geometry of Pelton Turbine Nozzle*, International Journal of Latest Trends in Engineering and Technology (IJLTET), **2**, *2*, March 2013, 318-326 (2013).

Zhang J., Xiao Y.X., Wang J.Q., Zhou X.J., Xia M., Zeng C.J., Wang S.H., Wang Z.W., *Optimal Design of a Pelton Turbine Nozzle via 3D Numerical Simulation*, Asian Working Group- IAHR's Symposium on Hydraulic Machinery and Systems, IOP Conf. Series: Earth and Environmental Science **163**, 9 pages, (2018).

Zidonis A., Panagiotopoulos A., Aggidis G.A., Anagnostopoulos J.S., Papantonis D.E., *Parametric Optimisation of Two Pelton Turbine Runner Designs Using CFD*, Journal of Hydrodynamics, **27**, *3*, 840-847 (2015).

Zidonis A., Aggidis G.A., *Pelton Turbine: Identifying the Optimum Number of Buckets Using CFD*, Journal of Hydrodynamics, **28**, *1*, 75-83 (2016).

Zoppe B., Pellone C., Maitre T., Leroy P., *Flow Analysis Inside a Pelton Turbine Bucket*, Transactions of ASME, Journal of Turbomachinery, **128**, 3, 500-511 (2006).

DETERMINAREA COEFICIENTULUI DE REZISTENȚĂ LOCALĂ PENTRU UN INJECTOR PELTON, UTILIZÂND ANALIZA CFD

(Rezumat)

Determinarea coeficientului de rezistență locală pentru elementele de instalații cu geometrie complexă se face, de regulă, pe cale experimentală. În cazul injectorului turbinelor Pelton acest lucru este dificil deoarece, prin construcția sa, injectorul nu este o rezistență hidraulică de traseu ci una de capăt. Lucrarea propune o metodă de determinare a coeficientului de rezistență locală pentru un injector de turbină Pelton, utilizând analiza numerică de tip CFD. Metoda constă în determinarea distribuției de presiuni în secțiunile de intrare, respectiv ieșire din injectorul Pelton și calculul coeficientului de rezistență locală raportat la viteza medie a curentului de fluid din secțiunea de intrare în injector. Atât presiunile cât și vitezele în diversele secțiuni de lucru ale injectorului sunt determinate prin analiză numerică tip CFD, realizată cu pachetul FluidFlow integrat în Solidworks. analiza a fost efectuată pentru un injector de microturbină Pelton.