NUMERICAL ANALYSIS OF NOZZLE FOR HYDRO-TURBINE

Prof. Sunil Shukla[1], Dr.Pushpendra kumar Sharma[2], Deepak Bisen[3]

[1] Professor & Guide, [2] HOD & Co-Guide, [3] M.Tech Scholar,

Mechanical Department, NIIST, Bhopal (India)

ABSTRACT

Due to periodic change of the relative position between the water jet and the bucket in Pelton turbines the flow is unsteady and has a free surface. The efficiency of the energy exchange depends on the flow structure of the water on the bucket and, therefore, the bucket design. Lately, more and more technical papers related to the numerical simulation of the flow in Pelton turbines have been published in the literature. And this review study investigated numerical analysis for hydro-turbine of nozzle. The equations governing this type of flow are the continuity equation and Navier-Stokes equation. Also, the velocity distribution inside the jet is investigated for different positions of needle tip.

Keywords: Hydro Classification, Pelton Turbine, Nozzle, Analysis of Nozzle,

I. INTRODUCTION

1.1. Hydro Power Plant

Hydro power plant is a system of mechanism which will initially store water to generate the potential head, then this potential head is converted to kinetic head (through nozzle), this kinetic head is used to create mechanical power by hydro turbines. Hydro turbine are connected to the generators for conversion of mechanical power to electrical energy.

1.2. Pelton Wheel Turbine

The Pelton wheel is a water impulse turbine. It was invented by Lester Allan Pelton in the 1870s. The Pelton wheel extracts energy from the impulse of moving water, as opposed to its weight like traditional overshot water wheel. Although many variations of impulse turbines existed prior to Pelton's design, they were less efficient than Pelton's design; the water leaving these wheels typically still had high speed, and carried away much of the energy. Pelton's paddle geometry was designed so that when the rim runs at ½ the speed of the water jet, the water leaves the wheel with very little speed, extracting almost all of its energy, and allowing for a very efficient turbine.

1.3. Nozzle

A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. A nozzle is often a pipe or tube of varying cross sectional area, and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. Nozzle with flow regulating device

Velocity increases rapidly in the passage through the nozzle, and surface are exposed to the risk of wear. They can be severely eroded by abrasive matter in the water, resulting in a disturbed jet, loss of efficiency, and wear on the bucket. The nozzle mouth-ring and the spear tip are made of bronze, stainless steel, or other erosion-resisting material, and can be easily replaced.

Discharge is regulated by the spear moving axially. It is stopped when the spear move forward to the discharge edge of the mouth-ring to close the orifice. Behind this the spear widens to a diameter larger than that of the mouth edge by about 20 or 30 per cent.

1.4. Water Turbine

A water turbine is a rotary engine that takes energy from moving water. ter turbines were developed in the 19th century and were widely used for industrial power prior to electrical grids. Now they are mostly used for electric power generation.

II. EXPERIMENTAL FACILITY AND THE INJECTOR GEOMETRY

The testing rig consists of the hydraulic circuit, control elements, measurement equipment and power consumers, as shown in figure 1.

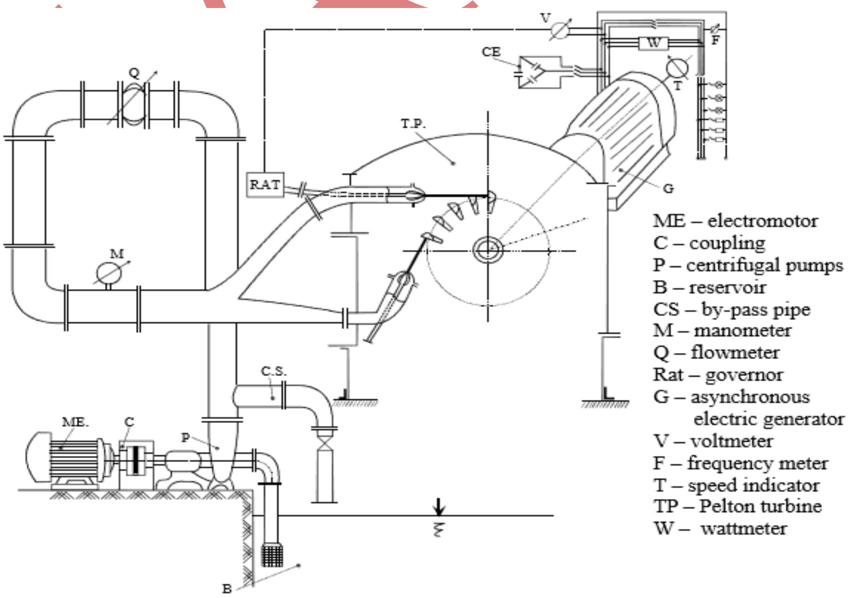


Figure 1. "Gemenele" testing rig from the Laboratory of Hydraulic Machinery, "Politehnica" University of Timişoara

Pelton turbine belonging to the testing rig has a rotor with 20 bucket shaped blades. The guide vane is axial – injector type. By opening or closing the needle tip from the regulating nozzle different operating points are achieved.

The main sizes of the injector belonging to the Pelton turbine testing rig are shown in figure 2.

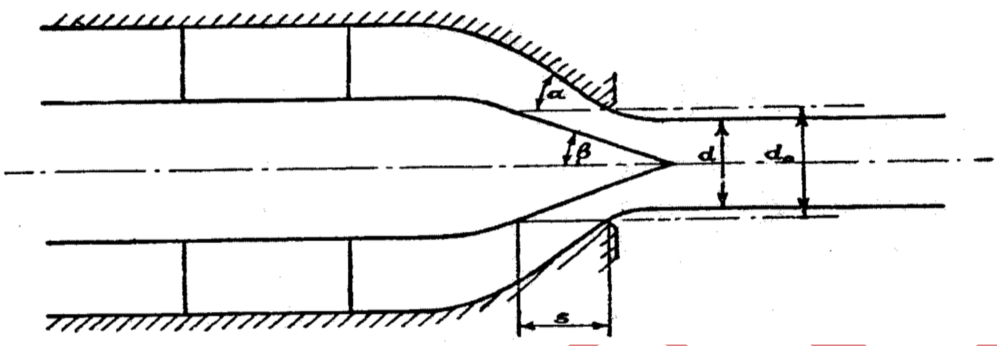


Fig. 2 – Diagram Of Nozzle And Spear

In the nozzle passage, represented in axial section in fig.2 by the outline of a nozzle and a spear head, the surface are shaped so as to reduce the annular area steadily toward the exit, where they are usually conical. The cone of the spear tip is drawn straight to a point at the apex, or it may be extended to a point farther out by a curved profile. In some designs there is a marked difference between the angles of the nozzle and spear cones and in other little or none. Short "onion-shaped" spears are to be found, where the half-angle α of the nozzle may be as large as 60^0 and β of the spear 45^0 , but usually the spear is elongated, with α from 30^0 to 45^0 and β from 20^0 to 30^0 .

III. BASIC EQUATIONS NUMERICAL ANALYSIS METHOD

3.1. Basic Equations

$$\frac{1}{r}\frac{\partial}{\partial r}(rv_r)+\frac{\partial v_z}{\partial z}=0$$

$$\frac{\partial v_z}{\partial t}+\frac{\partial v_z^2}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}(rv_z v_r)=$$

$$=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\nu\frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\frac{\partial v_z}{\partial r}-\frac{\partial v_r}{\partial z}\right)\right]$$

$$\frac{\partial v_r}{\partial t}+\frac{1}{r}\frac{\partial}{\partial r}\left(rv_r^2\right)+\frac{\partial}{\partial z}(v_z v_r)=$$

$$=-\frac{1}{\rho}\frac{\partial p}{\partial r}+\nu\frac{\partial}{\partial z}\left(\frac{\partial v_r}{\partial z}-\frac{\partial v_z}{\partial r}\right) \qquad (1)$$

The operation of tangential hydraulic turbines of Pelton type is based on fundamental equation of turbo-machinery, which derive from momentum of momentum equation. The flow is complicate due to periodic change of the relative position between the water jet and the bucket and also due to the jet interference.

In this chapter we consider the jet issuing from the Pelton turbine nozzle which is shown previous in fig. 2. The flow is assumed to be axisymmetric and without swirl flow. This type of flow is governed by the continuity equation and also the Navier-Stokes equations, presented above.

For solving these equations, the following boundary condition are introduced

-at the inlet of the injector: v_{z} = const.,

$v_r = 0$

-at the centerline of the jet:

$$\frac{\partial v_z}{\partial r} = 0, \quad v_r = 0$$

On the solid wall of the needle: $v_z = 0$, $v_r = 0$. Also. At exit of the nozzle, $\frac{vr}{vz}$ = tanβ, β is the incline angle of the nozzle exit.

-on the free surface:

$$\frac{\partial v_z}{\partial r} = 0, \ \frac{\partial v_r}{\partial r} = 0, \ p = p_{at}.$$

3.2. Numerical Analysis Method

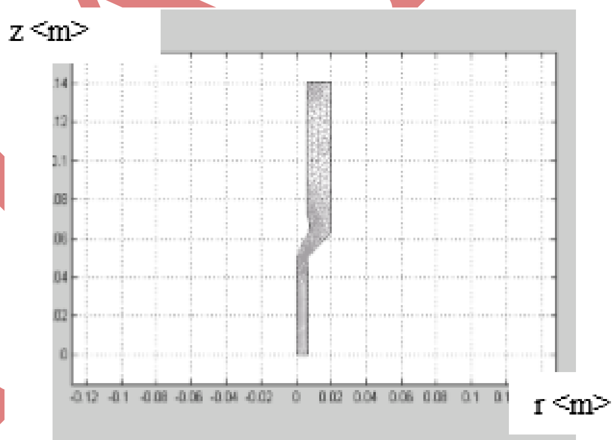


Figure 3. The Finite Element Grid for Simulation

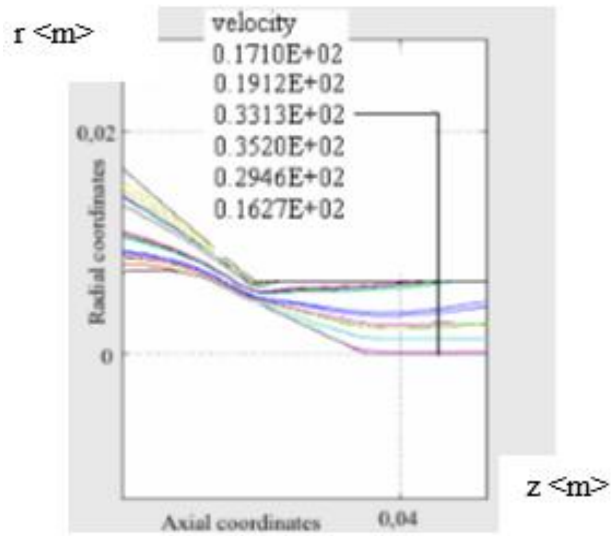
The three equations from (1) are solved by finite difference method using triangular mesh system for a good handling of the free surface. The finite element grid is represented in figure 3. The number of the cells from effective grid, which varies by the position of the free surface, is approximately 50000.

The numerical simulation results are obtained for five different positions of the needle tip. The needle stroke is about 15 mm. The complete opening of the nozzle, namely the nozzle is fully open, corresponds to the fully retracted position of the needle.

In figure 4 and figure 5 the numerical simulation results are represented for two positions of the needle from regulating nozzle.

In figure 6, for turbine head H =50 m and the position of the needle indicated by its relative value s/smax = 0,2, we determined the axial velocity distribution in the jet, for two axial positions along the free jet, downstream the nozzle. These two positions are at 1D0 and 2D0 from the nozzle, where D0 is the jet diameter.

Therefore, by changing the position of the needle from regulating nozzle obtain the flow rate variation and that caused the modification of the velocity distribution. This fact is presented in figure 4 and figure 5. Also, looking at figure 6, we observe that there is a low velocity region in jet near the axis and it spreads with the decrease of needle opening.



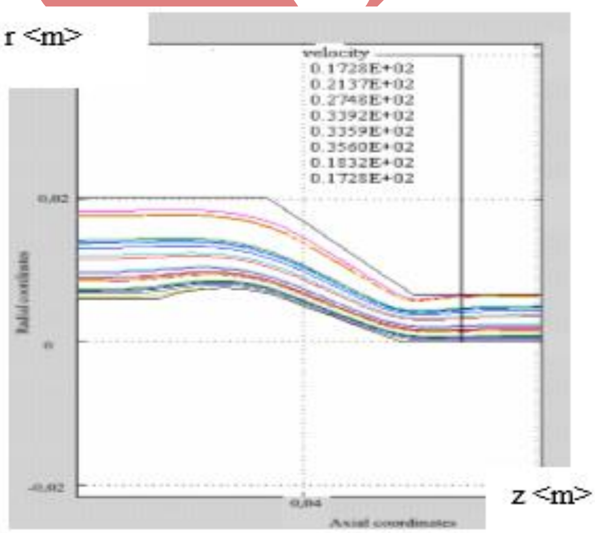


Figure 4. Velocity Distribution For S/S_{max}= 0,2 Figure 5. Velocity distribution for s/smax = 0,8

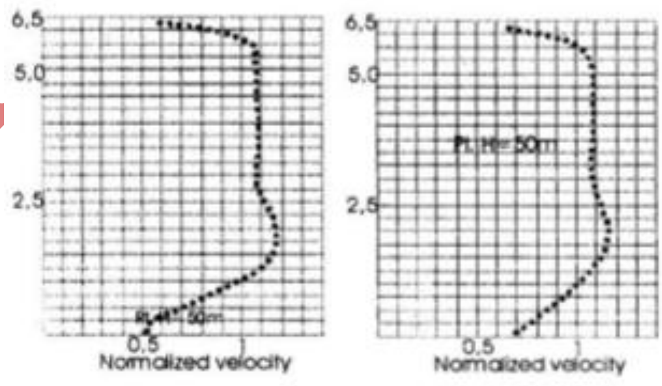


Figure 6. Axial Velocity Distribution at D_o and $2D_o$ from the Nozzle

We also proposed an expression for velocity which taking into account the velocity distribution with the jet axis z and also with the radius r of the jet. This proposed expression is a polynomial function of third degree, like:

$$v(z,r) = A\cdot z + B\cdot r + C\cdot z\cdot r + D\cdot z^2 + E\cdot r^2 + \\ + F\cdot r^2\cdot z + G\cdot r\cdot z^2 + H\cdot z^3 + I\cdot r^3 \qquad (2)$$

Analyzing the velocity distributions represented in figure 6, for three values of z and different values of r, we determine all the nine constants from relation (2).

These constants are: A = -6,495·10-3; B = 1,183; C = 0,065; D = 5,462·10-3; E = -0,154;

F = 5,209·10-3; G=7,084·10-4; H = -1,241·10-4; I = 2,495·10-3.

All these values are for z and r in millimeters. So, the velocity function in z and r is:

$$v(z,r) = -6{,}495\cdot 10^{-3}\, z + 1{,}183\, r - 0{,}065\, z\, r + \\ + 5{,}462\cdot 10^{-3}\, z^2 - 0{,}154\, r^2 + \\ + 5{,}209\cdot 10^{-3}\, r^2\, z + 7{,}084\cdot 10^{-4}\, r\, z^2 - \\ - 1{,}241\cdot 10^{-4}\, z^3 + 2{,}495\cdot 10^{-3}\, r^3 \qquad (3)$$

In figure 7 we represented the calculated velocity distribution with the relation (3), at $1D_o$ from the relation (3), at $1D_o$ from the nozzle.

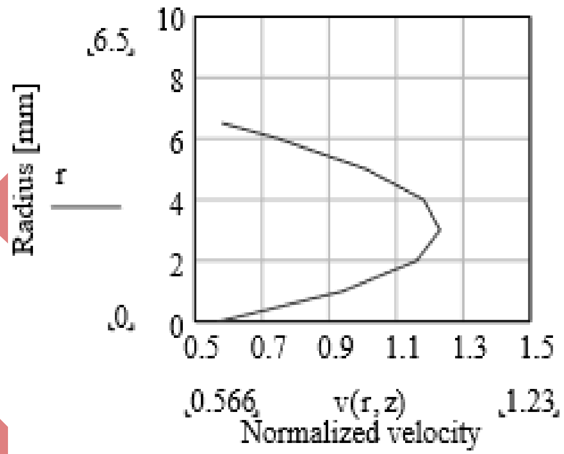


Figure 7. Calculated Velocity Distribution at $1D_o$ from the Nozzle Exit

Comparing the results represented in figure 6 and figure 7 we observed that the velocity function from relation (3) approximate with good accuracy the velocity distribution near the axis and near the free surface.

IV. LITERATURE REVIEW

Abid A. Khan et al[1]Small-scale hydro or micro-hydro power has been increasingly used as an alternative energy source, especially in remote areas where other power sources are not viable. In the larger hydro projects potential energy of the water is converted to electrical power, whereas such small-scale hydro power systems utilize kinetic energy of the flow. These systems can be installed in small rivers or streams with little or no discernible environmental effects. Usually, these systems do not require a huge infrastructure like dam or major water diversions, rather use water wheels with least environmental impacts. This paper presents a report on how flow can be accelerated through convergent nozzles for run-of-river turbines in open flow channels. An existing canal in Pakistan has been used for the analysis based on its easily accessible data. The analytical and computational work presented here converts kinetic energy of water flow to electric power. **F. Montomoli et al [2]** Experimental and numerical investigation of the beneficial effect of higher conductivity materials in HP turbine nozzles. Most of the literature studies focus on the maximum temperature that a nozzle can withstand, whereas the effect of thermal gradients is often neglected. However thermal gradients have higher influence on the life of the components and they have to be given careful consideration. In this work it is shown that thermal gradients are reduced by using high conductivity materials and, as a consequence, the nozzles life is appreciably increased.A representative film cooled leading edge with an internal impingement plate was studied experimentally at Texas AM University. Two materials were used, namely polycarbonate and stainless steel, in order to highlight the impact of conduction on coolant effectiveness. Numerically conjugate heat transfer simulations have been carried out with an in house solver to analyse in detail the impact of conduction and internal convection. Both experimental and numerical results show that by increasing the conductivity in the solid region, the thermal gradients are strongly reduced. Numerically it is shown that using inserts of nickel-aluminide alloys in nozzles may reduce the thermal gradients from 3 to 4 times if compared to nowadays design. **SławomirDykas, et al [3]** Presents the computational results of the wet steam flow through the Laval nozzles for low and high inlet pressures. The results of the numerical modelling are compared with experimental data. The comparisons constitute validation tests of the condensation model implemented into an in-house CFD code solving the RANS equations for the real gas equation of state. The steam condensing flow is modelled by means of the single-fluid model, which means that the conservation equations are formulated for the vapour/liquid mixture. The water vapour properties are described by means of the local equation of state. An effective method of determination of water vapour properties are presented in the cases of expansion in the nozzle at high pressures. The presented results are compared with published experimental values. The validation of the in-house CFD code proves its usefulness for modelling the steam condensing flow for both low and high inlet pressures. **Abhijit Date et al [4]**The performance characteristics of a split reaction water turbine. The governing equations are derived by using the principles of conservation of mass, momentum and energy for a practical case, which includes consideration of frictional losses. The optimum diameter for a simple reaction turbine is defined and an equation for the optimum diameter is derived. Design and building procedures for a split reaction turbine are described. Using the equation for optimum diameter and assuming a loss factor (k-factor) of 0.05, optimum rotor diameters for different operating heads and rotational speeds are plotted and discussed. Measured performance of a 122 mm diameter split reaction water turbine rotor is presented. The relationship between k-factor and relative velocity for a split reaction turbine model is discussed with reference to experimental data.

V. CONCLUSION

Simulation for Pelton turbines is more difficult than for other hydraulic turbines because of the free surface flow downstream the nozzle and the unsteady flow in the bucket.The jet from a Pelton turbine nozzle has been numerically investigated, under the assumption of axisymmetric flow. The calculation shows that the needle considerably influences the velocity distribution in the jet. Also, the enlargement of the jet, the contraction diameter and its position are determined.

REFERENCE

[1] [1]Abid A. Khan, Abdul M. Khan, M. Zahid, R. Rizwan "Flow acceleration by converging nozzles for power generation in existing canal system"Renewable Energy, Volume 60, December 2013, Pages 548-552

[2] [2]F. Montomoli, M. Massini, H. Yang, J.C. Han "The benefit of high-conductivity materials in film cooled turbine nozzles"International Journal of Heat and Fluid Flow, Volume 34, April 2012, Pages 107-116

[3] [3]SławomirDykas, WłodzimierzWróblewski "Numerical modelling of steam condensing flow in low and high-pressure nozzles" International Journal of Heat and Mass Transfer, Volume 55, Issues 21–22, October 2012, Pages 6191-6199

[4] [4]Abhijit Date, AliakbarAkbarzadeh "Design and analysis of a split reaction water turbine"Renewable Energy, Volume 35, Issue 9, September 2010, Pages 1947-1955

[5] [5]E. Amiri Rad, M.R. Mahpeykar, A.R. Teymourtash "Evaluation of simultaneous effects of inlet stagnation pressure and heat transfer on condensing water-vapor flow in a supersonic Laval nozzle" ScientiaIranica, Volume 20, Issue 1, February 2013, Pages 141-151

[6] [6]Kai Shimokawa, Akinori Furukawa, Kusuo Okuma, Daisuke Matsushita, Satoshi Watanabe "Experimental study on simplification of Darrieus-type hydro turbine with inlet nozzle for extra-low head hydropower utilization"Renewable Energy, Volume 41, May 2012, Pages 376-382