Journal of Thermal Engineering
Yildiz Technical University Press, Istanbul, Turkey
Vol. 2, No. 5, pp. 907-912, October, 2016.

http://eds.yildiz.edu.tr/journal-of-thermal-engineering/Articles
Manuscript Received May 29, 2015; Accepted June 08, 2015

This paper was recommended for publication in revised form by Regional Editor Derya Burcu Özkan

A NUMERICAL INVESTIGATION OF THE FLOW IN WATER JET NOZULES

***Ahmet Çağrı Bilir**
Marsis External Fire Fighting Systems Co.
Research & Development Department
34944, Tuzla, Istanbul, Turkey

Ali Doğrul
Yildiz Technical University
Dept. of Naval Architecture and
Marine Engineering
34349, Besiktas, Istanbul, Turkey

Taner Coşgun
Yildiz Technical University
Dept. of Naval Architecture and
Marine Engineering
34349, Besiktas, Istanbul,
Turkey

Ahmet Yurtseven
Yildiz Technical University
Dept. of Naval Architecture and
Marine Engineering
34349, Besiktas, Istanbul,
Turkey

Nurten Vardar
Yildiz Technical University
Dept. of Naval Architecture and
Marine Engineering
34349, Besiktas, Istanbul,
Turkey

Keywords: Nozzle, water jet, k-w turbulence model, CFD
** Corresponding author: Ahmet Çağrı Bilir*
Phone: +90 506 4289720, Fax: +90 216 3949236
E-mail address: ahmetcagribilir@gmail.com

ABSTRACT

In this study, flow inside the nozzles are investigated by means of finite volume method. Firstly, some analyses are carried out in 2-D in order to compare and validate the results with the experimental ones. Later, 3-D models are created to have different nozzle geometries. 3-D analyses are made and outlet mass flow rates, velocities and reaction forces are calculated in the same inlet pressure level for different nozzle geometries. Two equation k-ω turbulence model is chosen as the turbulence model. At the end of this numerical study, the nozzle geometry with minimum reaction force and maximum mass flow rate is determined thanks to computational fluid dynamics (CFD) based on finite volume method (FVM).

INTRODUCTION

Nozzles have a wide range of usage such as maritime systems, aviation systems, automotive systems and almost all industrial systems. Nozzles can be both used as a single equipment or as a part of an equipment. Nozzle geometry is one of the most important parameter on the effect of water jet flow in nozzles. Nozzle geometry in a jet flow should maintain the flow with high mass flow while generating less force in the same pressure. Therefore forming a nozzle geometry with low pressure loss is a desirable feature for jet flow. The nozzles may have a conical structure. The nozzles can also have different geometrical shapes consisting of convex and concave angle.

Y. Yu and M. Shademant (2012) have studied about investigation of the flow in different nozzle types. In the study, they have made CFD analyses in order to investigate the effect of the different nozzle geometries with applying different turbulence models for comparison [1].

Guoliang Hu et al. (2012) have carried out some analyses about jet characteristics and structural optimization of a liquamatic fire water monitor with self-swinging mechanism. They investigated nozzle flow with CFD method using ANSYS Fluent. Two equation RNG k-ε turbulence model is chosen as the turbulence model for the study. In the study, inlet and outlet diameter of fire water monitor, section shape, and inlet pressure are changed and observed. After defining the best case for jet characteristics of fire water monitor, they have produced the prototype and made some experimental studies. They have seen the optimum throw length of water with the maximum inlet pressure value by changing the inlet pressure. Also they have compared CFD studies and experimental studies [2].

Z. L. Fang et al. (2013) have studied flow field characteristics of organ pipe nozzle numerically and experimentally. They have considered the flow is transient in CFD analyses and standard k-ε equation turbulence model is chosen. In the study, cavity effect is investigated for different inlet pressures. They have also gained some experimental results in order to compare with the numerical ones [3].

N. Baisheng at al. (2011) have studied about numerical investigations of the flow field inside and outside high-pressure abrasive waterjet nozzle. In the study, they have made CFD analyses in order to investigate the effect of the different nozzle geometries with applying different inlet pressures and shock pressures for comparison. Standard k-ε equation turbulence model is chosen for all analysis [4].

In addition, usage of adjustable nozzles are greatly increased and flow velocity and mass flow can be adjusted in this kind of nozzles. Because of widely usage of the nozzles, several studies have been made [5]. Various numerical simulation and experimental study have been done on the selection of the best contraction angle in conical nozzle. For this reason, in this study, not only conical nozzles but also different nozzles with convex and concave angle are designed and investigated numerically. The flow structure within special nozzle type is very complicated. Therefore the mass flow-pressure, reaction force-pressure and velocity-pressure relationship are observed. With the help of a commercial code so called ANSYS Fluent, different combinations of the nozzles are simulated and the results are discussed.

MATHEMATICAL MODEL

The flow field is incompressible, steady, isothermal turbulent flow. In a Cartesian coordinate system continuity equation tensor form is as follows [2-4]:

$$\frac{\partial \rho u_i}{\partial x_i} = 0 \qquad (1)$$

Momentum equation of incompressible viscous fluid motion is as follows [2-4]:

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}(u_i u_j) = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu\frac{\partial u_i}{\partial x_j} - \overline{u_i'}\,\overline{u_j'}\right) \qquad (2)$$

Standard k-ε equation turbulence model in jet simulation [4]:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial t} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \qquad (3)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial t} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{C_{1\varepsilon}\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (4)$$

$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}, C_{1\varepsilon} = 1.44,\ C_{2\varepsilon} = 1.92\ , C_\mu = 0.09\ ,\ \sigma_k = 1.0\ ,$ σ_k=1.3

SST k-ω equation turbulence model in jet simulation [4]:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - Y_k \qquad (5)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial k}{\partial x_j}\right] + G_\omega - Y_\omega \qquad (6)$$

$\mu_t = a^* \frac{\rho k}{\omega}, a^* = 1, \sigma_k = 1.0, \sigma_k$=1.3, $G_k = \mu_t S^2$, $G_\omega = a\frac{\omega}{k}G_k$

VALIDATION FOR NUMERICAL METHOD

For validation of the current numerical approach, the case which Zhang et al. [5] have investigated is chosen. The geometry is consisting of a 13.5^0 conical nozzle with straight pipe. The results are compared with the ones from the literature. The nozzle has an axisymmetric structure while there is a straight pipe section at the nozzle exit. And 2-D models are shown below following the mesh structure as an example for validation case. Comparison of the results are given below via axial velocity component.

The inlet diameter of nozzle is 50 mm, outlet diameter is 20 mm, contraction angle of nozzle is $13,5^0$. The 2-D computational domain is 25d*8d, the straight pipe section at the exit of the nozzle is 1/4d while d is the outlet diameter of the nozzle.

Entrance boundary condition is velocity inlet, velocity of jet in the nozzle is 20m/s, velocity of the accompanying jet is 10m/s. The outlet boundary condition is outflow. The boundary condition on the axis is axisymmetric. The wall surface satisfies the no-slip condition, the standard wall function method is used near wall region.

The numerical analysis for the validation shows a good agreement with the results of Zhang et al. [5].

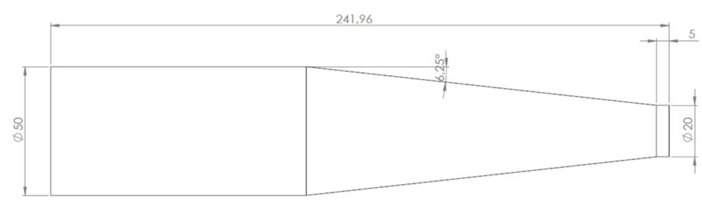

Figure 1 13.5^0 conical nozzle with straight pipe

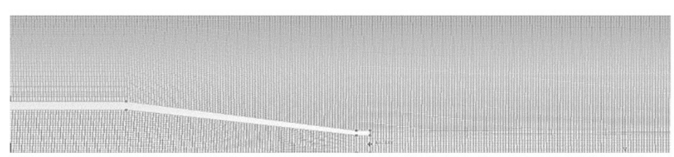

Figure 2 Mesh structure of 13.5^0 conical nozzle with straight pipe

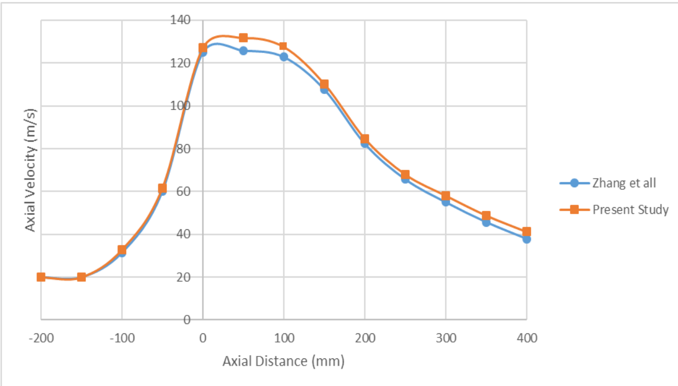

Figure 3 Comparison with Zhang et al. [5] via axial velocity component

NUMERICAL APPROACH

Mesh Dependency

Different mesh numbers are obtained by refining the mesh structure for the conical-1 nozzle geometry. Four different cases are investigated for three different pressure levels consisting 8, 10 and 12 bar, respectively.

Table 1 Mesh numbers used in mesh dependency study

Mesh structure	Coarse	Medium	Fine	Finest
Mesh number	178663	305087	651651	858354

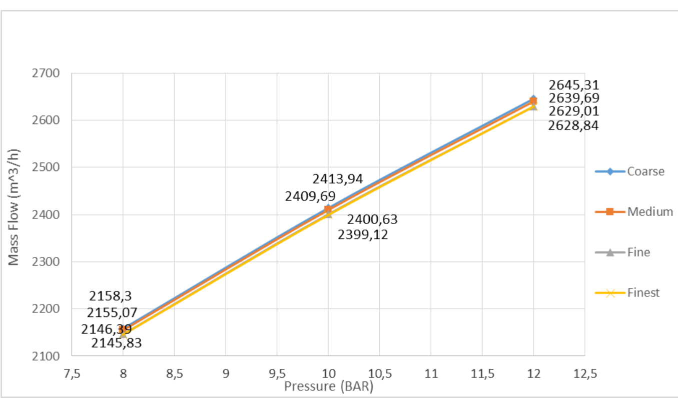

Figure 4 Comparison of the mass flow for different mesh structures

According to the results, fine and finest cases behave similar by means of mass flow rate. So, the fine case is chosen as the reference mesh number and all other nozzle geometries are investigated with fine mesh structure.

The figure given below shows that mass flow rate of different inlet pressures for coarse, medium, fine and finest mesh, respectively. Results of the higher mesh cases are almost same. In addition, the difference in mass flow rate is approaching zero while the mesh number gets lower.

NUMERICAL SIMULATIONS AND RESULTS

It is expected form the nozzle in the jet flow to pass more mass flow while creating less force. So, it is obligatory to determine the nozzle geometry with low pressure loss. Another important point in nozzle design is to minimize the force created by the nozzle geometry. In this study, five different nozzle geometries are designed and investigated in order to obtain the optimum geometry by means of flow performance. For this purpose, numerical analysis are carried out for all geometries which are called conical-1, conical-2, pelton-1, pelton-2 and reverse pelton, respectively.

In this study, five different nozzles are designed with the same inlet and outlet diameters and length to make a good comparison. The inlet diameter is 260 mm and the outlet diameter is 140 mm. while the length is considered as 365 mm. Conical-1 has an angle of 40^0, and conical-2 has an angle of 60^0. Other two nozzle geometries are called pelton-1 and pelton-2 because of the similarity with pelton structure. And the last one is called reverse pelton because of the reverse angle. All these five nozzle models are presented in the figures below.

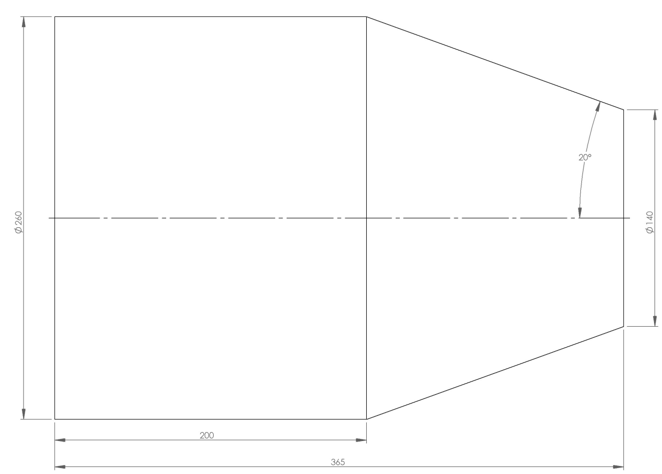

Figure 5 Conical-1 nozzle

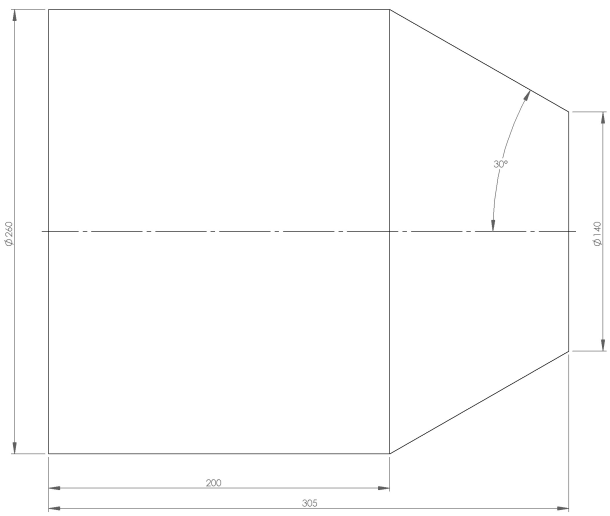

Figure 6 Conical-2 nozzle

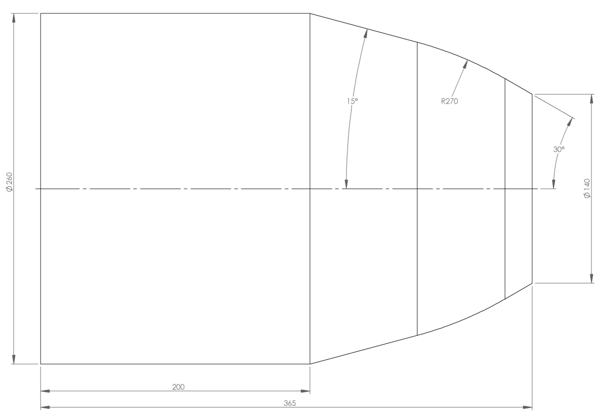

Figure 7 Pelton-1 nozzle

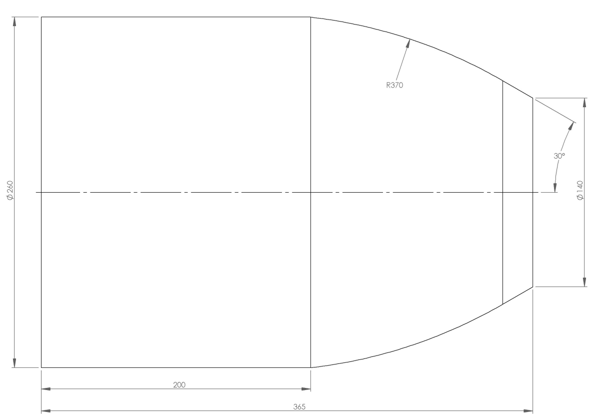

Figure 8 Pelton-2 nozzle

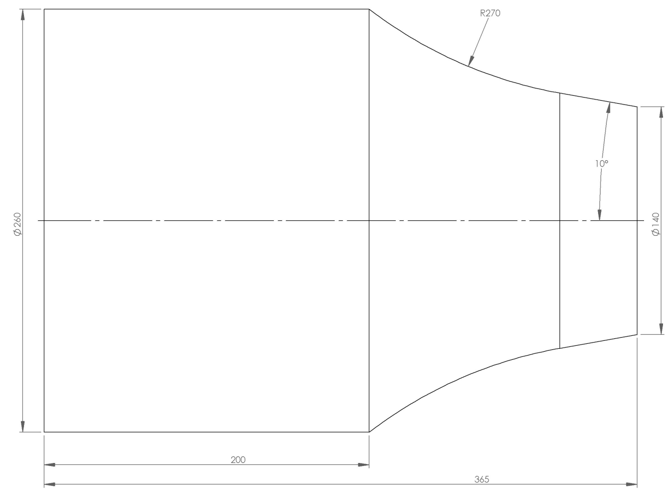

Figure 9 Reverse Pelton nozzle

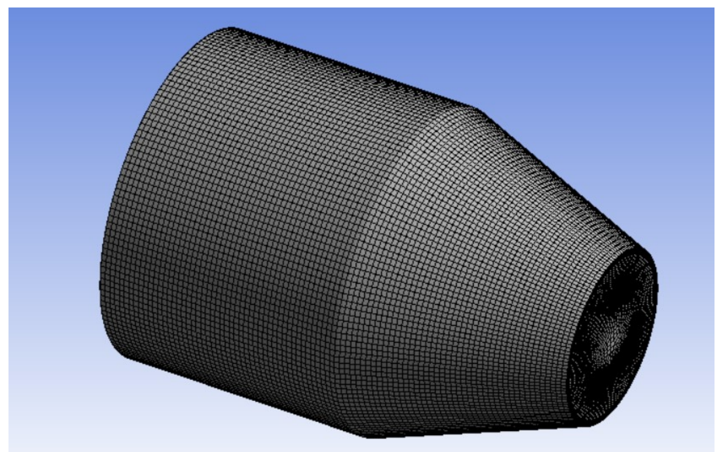

Figure 10 Mesh structure of conical-1 nozzle

During a jet flow, average outlet velocity in nozzle exit should be high in order to throw the water away. For this reason, relations for mass flow-pressure, force-pressure and velocity-pressure are examined in this study. Because of the impossibility of calculating the reaction force with the one dimensional force equation, ANSYS Fluent software package is preferred and the problem is solved in 3-D.

Flow analysis for different types of nozzle geometries are carried out. The flow is considered as steady and single phase. The fluid density is considered as 1025kg/m^3 while the dynamic viscosity is taken as 0,001003 kg/m-s which refers to the sea water. As turbulence model, k-w SST is chosen after several analyses with different turbulence models. Some analyses are made for conical-1 case with different mesh numbers in order to obtain mesh independent results. The numerical analyses are made for 8, 10 and 12 bar pressure levels for each nozzle geometry.

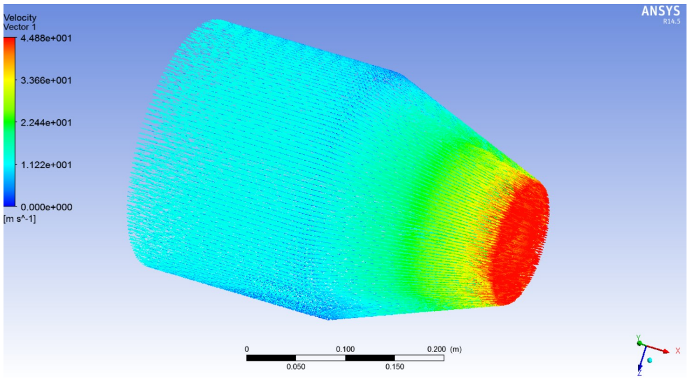

Figure 11 Velocity vectors for Conical-1 nozzle at 10 bar inlet pressure

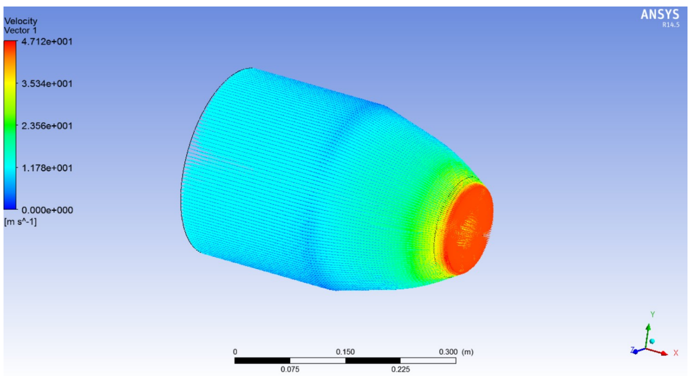

Figure 12 Velocity vectors for Pelton-1 nozzle at 10 bar inlet pressure

It can be seen from figures 4.8 and 4.9 that the velocity is approaching zero at contraction area of the nozzle because of the sharp edge on nozzle geometry. Streamlines make a sudden change in flow direction which causes stagnation points where the velocity is zero.

Velocity reaches the maximum value at the outlet section because of the outlet section is the narrowest in all nozzle types.

As it is known that the flow rate equation (Q=V*A) shows already the same result which means the maximum velocity should be in the narrowest section in order to obtain the same mass flow.

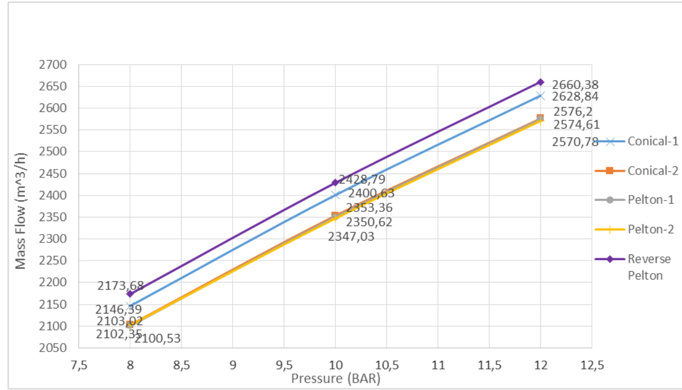

Figure 13 Comparison of the mass flow for different nozzle geometries

Anyone can see from the numerical results that the reverse pelton geometry maintains maximum mass flow rate with the same inlet pressure. Minimum mass flow rate is acquired with the pelton-2 geometry while conical-1 geometry gives best results following the reverse pelton geometry.

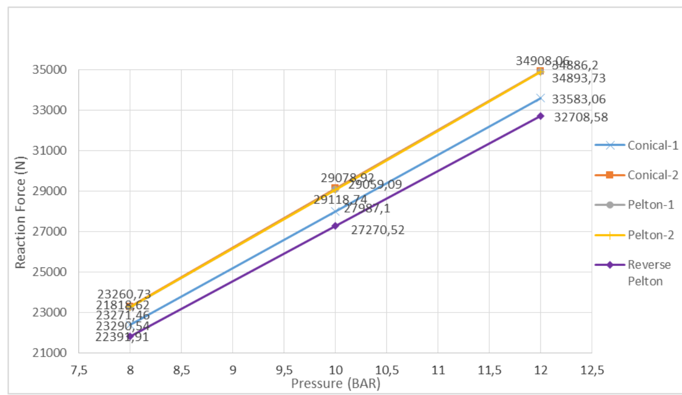

Figure 14 Comparison of the reaction force for different nozzle geometries

As can be seen from the numerical results that the reverse pelton geometry creates minimum reaction force with the same inlet pressure. Maximum reaction force is created with the pelton-2 geometry.

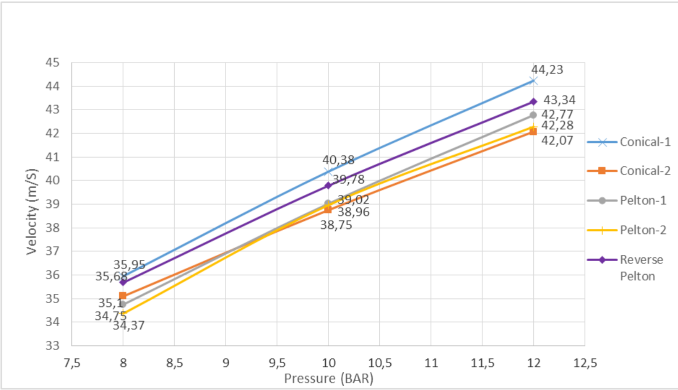

Figure 15 Comparison of the axial component of outlet velocity for different nozzle geometries

It can be seen from the numerical results that conical-1 geometry generates the maximum velocity in nozzle direction with the same inlet pressure.

CONCLUSIONS

During the jet flow, nozzles are used to adjust the outlet velocity to reach the desired distance in fire monitors. The most important parameter in nozzle design is to minimize the reaction forces while obtaining maximum mass flow rate from the nozzle.

In this study, jet flow in nozzles is investigated in a numerical manner. Firstly, a validation case is chosen from the literature and the commercial code is verified with the given results by considering the flow is 2-D. After that, 3-D models for different nozzle geometries are created. The first model called conical-1 is chosen for mesh dependency study. An optimum mesh number is gathered with respect to the mass flow rate. Optimum mesh number is used for all remaining nozzle geometries. All numerical analyses are carried out for 8, 10 and 12 bar pressure levels.

Sharp edge on nozzle geometry causes a change in the streamlines. This may decrease the velocity to zero at contraction area in all nozzle types.

The outlet section which is the narrowest in all nozzle types makes the velocity maximum.

While the inlet pressure increases the outlet velocity, reaction force and mass flow also increase.

For all pressure levels, reverse pelton nozzle gives maximum mass flow and minimum reaction force. But average outlet velocity reaches its maximum with conical-1 nozzle. So conical-1 is the best nozzle geometry in regard to the water throwing distance.

REFERENCES

[1] Y. Yu, M. Shademan, R. M. Barron, R. Balachandar (2012). “CFD study of effects of geometry variations on flow in a nozzle”, Engineering Applications of Computational Fluid Mechanics Vol.6, No.3 pp. 412-425 (2012), Department of Mechanical, Automotive & Materials Engineering, University of Widsor, ON, N9B 3P4, Canada.

[2] Hu G., Long M., Liang J., Li W. (2012). “Analysis of jet characteristics and structural optimization of a liquamatic fire water monitor with self-swinging mechanism”, Int J Adv Technol (2012) 59:805-813.

[3] Z. L. Fang, Y. Kang, X. C. Wang, D. Li, Y. Hu, M. Huang, X Y Wang (2013). “Numerical and experimental investigation on flow field characteristics of organ pipe nozzle”, 27th IAHR Symposium on Hydraulic Machinery and Systems (IAHR 2014), School of Power and Mechanical Engineering, Wuhan University, No.8 Donghu South Road, Wuhan, 430072, China.

[4] N. Baisheng, W. Hui, Li Lei, Z. Jufeng, Y. Hua, L. Zhen, W. Longkang, Li Hailong (2011) ‘Numerical investigation of the flow field inside and outside high-pressure abrasive waterjet nozzle’ China University of Mining & Technology (Beijing) 100083, China.

[5] S B Zhang and J M Zhu (2012) ‘Numerical simulation of adjustable nozzles’ School of Power and Mechanical Engineering, Wugan University, Wuhan, P R China.

[6] Irving H. Shames (1989) ‘Mechanics of Fluids’ Faculty of Engineering and Applied Science State University of New York at Buffalo.

[7] Zhou W H (2008) ‘Simulation of flow field of high-pressure water-jet from nozzle with FLUENT (Lanzhou: Lanzhou Univ. of Tech).

[8] T. Mabrouki, K. Raissi, A. Cornier, (1999), “Numerical simulation and experimental study of the interaction between a pure high-velocity waterjet and targets: contribution to investigate the decoating process”, Elsevier Wear 239 (2000) 260-273, Laboratoire des Procedes Conventionnels et Non Conventionnels, BD de l’Hopital 75013 Paris, France.

[9] Adriana C., Mircea B., (2004), “Numerical simulation of a free jet in pelton turbine”, The 6th International Conference on Hydraulic Machinery and Hydrodynamics Timisoara, Romania, October 21-22 2004, Armata Romana Street 5, Oradea, Romania.

[10] Yuan B et all (2012) ‘Numerical investigation on high-pressure convergent nozzle by comparative and statistical analysis method’ School of Power Mechanical Engineering, Wuhan University, No:8 Donghu South Road, Wuhan, 430072, China.

[11] ANSYS Fluent 14.5 Theory Guide

[12] www.cfd-online.com