BERKELEY PUBLICATION
AND RESEARCH INTERNATIONAL

03.31.2020

Journal of Pure and Applied Science Vol. 12 No. 9

A NUMERICAL INVESTIGATION OF THE FLOW IN WATER JET NOZZLES

SIBEUDU CHIWETALU EMENIKE AND OCHUBA NNAMDI
Department of Mechanical Engineering, Federal Polytechnic Oko

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 reactive 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 reactive force and maximum mass flow rate is determined thanks to computational fluid dynamics (CFD) based on finite volume method (FVM).

Keywords: *Investigation, Flow, Water, Jet Nozzles.*

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. 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 [1]. 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.

VALIDATION FOR NUMERICAL METHOD

For validation of the current numerical approach, the case which Zhang et al. [1] have investigated is chosen. The geometry is consisting of a 13.50 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 13,50. 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. [1].

NUMERICAL SIMULATIONS

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 400, and conical-2 has an angle of 60. 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 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/m3 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. As can be seen that as the velocity approaches zero in the section that starts nozzle contraction rate, velocity reaches the maximum value at the outlet section.

Because the outlet section is the narrowest section for all nozzles in this study. 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. 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.

CONCLUSION

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. 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

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

Irving H. Shames (1989) 'Mechanics of Fluids' Faculty of Engineering and Applied Science State University of New York at Buffalo

Zhou W H (2008) 'Simulation of flow field of high-pressure water-jet from nozzle with FLUENT (Lanzhou: Lanzhou Univ. of Tech) Ansys Fluent 14.5 Theory Guide

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