JAES | JOURNAL OF APPLIED ENGINEERING SCIENCE | www.engineeringscience.rs

ISTRAŽIVANJA I PROJEKTOVANJA ZA PRIVREDU

Indexed by

DETERMINING THE EFFECT OF NOZZLE GROOVE ON THE FLUID FLOW THROUGH VISCOUS 2D PLANAR FLUID

Reem Ahmed
Elgeraf sharg Technical College, Department of Mechanical Engineering, Sudan

Obai Younis
Prince Sattam Bin Abdulaziz University, College of Engineering at Wadi Addwaser, Department of Mechanical Engineering, Alkharj, Saudi Arabia;
University of Khartoum, Faculty of Engineering, Department of Mechanical Engineering, Khartoum, Sudan

Ali Mohammed Hamdan
University of Bahri, Department of Mechanical Engineering, Bahri, Sudan

Dania Ahmed
University of Khartoum, Faculty of Engineering, Department of Mechanical Engineering, Khartoum, Sudan

Ali Ahmed Al
Abdulatif Alhamad University of Technology, Faculty of Engineering, Sudan

Ibrahim Ahmed
Sudan University of Science and Technology, Department of Nuclear Engineering, Sudan

Key words: *fluid flow, groove, nozzle, CFD*

doi:10.5937/jaes0-31154

Cite article:

Ahmed R., Younis O., Hamdan Mohammed A., Ahmed D., Al Ahmed A., Ahmed I. (2021) DETERMINING THE EFFECT OF NOZZLE GROOVE ON THE FLUID FLOW THROUGH VISCOUS 2D PLANAR FLUID, *Journal of Applied Engineering Science*, 19(4), 954 - 961, DOI:10.5937/ jaes0-31154

Online aceess *of full paper is available at:* www.engineeringscience.rs/browse-issues

Istraživanja i projektovanja za privredu
Journal of Applied Engineering Science

Original Scientific Paper

doi:10.5937/jaes0-31154

Paper number: 19(2021)4, 875, 954-961

DETERMINING THE EFFECT OF NOZZLE GROOVE ON THE FLUID FLOW THROUGH VISCOUS 2D PLANAR FLUID

Reem Ahmed[1], Obai Younis[2,3], Ali Mohammed Hamdan[4], Dania Ahmed[3], Ali Ahmed Ali[5], Ibrahim Ahmed[6]*

[1]*Elgeraf sharg Technical College, Department of Mechanical Engineering, Sudan*
[2]*Prince Sattam Bin Abdulaziz University, College of Engineering at Wadi Addwaser, Department of Mechanical Engineering, Alkharj, Saudi Arabia*
[3]*University of Khartoum, Faculty of Engineering, Department of Mechanical Engineering, Khartoum, Sudan*
[4]*University of Bahri, Department of Mechanical Engineering, Bahri, Sudan*
[5]*Abdulatif Alhamad University of Technology, Faculty of Engineering, Sudan*
[6]*Sudan University of Science and Technology, Department of Nuclear Engineering, Sudan*

The study aims to determine the effect of the nozzle groove on fluid flow through the viscous 2D planar fluid. To fulfil the study's aim, a numerical method was adopted to introduce grooves of different dimensions from the nozzle exit. The study adopts SolidWorks software that was used to design nozzles and introduce groove shaped nozzles, each consisting of six different designs. The nozzle base model used in this study was similar to the one used in a previous study. The procedure was performed with different pressures (8, 10, and 12 bar) at a similar firefighting nozzle. The velocities contours were predicted based on the choice of nozzle section during the numerical stimulation. The present study results demonstrated a new approach that can be used for the increasing velocity at various types of modified nozzles through grooves at different pressures and locations. For grooves, dimensions 1×1 (mm) and location 15 mm at 8 bar, 10 bar and 12 bars showed no effect on velocity as it reduces velocity by increasing surface area. The velocity increases with increasing pressure in the proportion relationship. It clearly explains that the groove does not affect velocity as it rises due to increase in pressure. It is because the groove reduces the velocity by increasing surface area. The study concludes that the use of groove increases the velocity of water that further improves nozzles operation.

Key words: fluid flow, groove, nozzle, CFD

INTRODUCTION

Nozzle device is used for controlling the characteristics and direction of flowing fluid that increases its velocity as it enters or leaves an enclosed pipe or chamber. The nozzle is configured for a bi-directional flow, where pressure decrease in one direction increases the pressure of fluid in the other direction [1]. It is just like a tube or pipe of a definite cross-sectional area used to modify or direct the fluid flow. Therefore, nozzles play an essential role in controlling the flow rate, direction, speed, shape, mass, and pressure of stream emerging from them [2]. The velocity of the fluid within a nozzle increases at the cost of its pressure energy. The primary purpose of the nozzle is to restrict flow that builds up pressure, which is used to project the water stream. One restriction in the form of correct nozzle size is maintaining fluid flow and developing optimum pressure and velocity. For instance, the spray nozzle works at high pressure as the liquid is supplied to a swirl chamber through tangential holes. The liquid in the form of a thin conical sheet is discharged from the nozzle exit orifice and empties into the ligaments [3].

There are various sizes, styles, and shapes of nozzles with a large number of combinations that can be used to encounter the hydraulics problem on the fire ground [4]. The combination of nozzles with different engineering applications is used to generate sprays and jets [5]. They are also widely used as aviation systems, maritime systems, automotive systems, and industrial systems as single equipment or part of equipment [6]. The effect of nozzles on flow characteristics in the spiral nozzle was investigated by Matsuo et al. [7] using an annual slit. The study used velocity distribution downstream of the nozzle exit that was compared with the computational results.

Similarly, the flow characteristics of jet expansion pipe nozzles were investigated using the flow visualizations technique [8]. A review was carried out for a free jet from the round pipe installed with a collar at the jet exit. The results demonstrated no difference in jet spreading rate of pipe nozzle with collar and the conventional jet i.e., pipe nozzle without the collar. However, the conventional jet was much smaller than the jet from pipe nozzle with collar.

Another study conducted by Alam et al. [9] investigated the impact of flow parameters at nozzle exit by stimulating turbulent flow using Reynolds-averaged Navier-Stokes (RANS) equations via nozzles of different dimensions. The results showed that the dimension of nozzle significantly affected the discharge coefficient. Similarly, Mohamed et al. [10] also studied pressure influence on the flow characteristics using RANS and two transport equations for modelling turbulence. The results showed

*o.elamin@psau.edu.sa

that there is a numerical simulation of the compressible flow in a convergent-divergent nozzle. The mechanism of inflow nozzle using direct numerical simulation (DNS) on spatially evolving round jets for the laminar and turbulent jet was investigated by numerous studies [11, 12]. The results showed that it is essential to allow inflow entrapment in the simulation of turbulent jets. Various studies [13, 14] illustrated a shock wave position for various pressure ratios by investigating different shapes of nozzles using the numerical investigation method. The results showed a significant impact of nozzle geometry on the flow structures, including the shock wave location.

The structure of spray and distribution of particle size at nozzle exit was determined by Hespel et al. [15]. The results demonstrated that the atomization process and exit of fluid from the nozzle depend on cavitation. Moreover, the significance of nozzle geometry on subsequent atomization and fluid flow was validated by Agarwal and Trujillo [16], Zhang et al. [17], and Kumar and Sahu [18]. These results may correspond to the study conducted by Satyanarayana et al. [19], showing that fluid properties significantly depend on the cross-section of the nozzle that further affects the flow within the nozzle.

Experiments on two-dimensional nozzles were also performed by Mashida and Sou [20] and Sou et al. [21] for analyzing cavitation in liquid jets under different Reynolds number and cavitation condition. Another study in 1999 was conducted by Badock et al. [22], who focused on the impact of nozzle geometry and internal flow on the velocity of fuel droplets and spray characteristics. However, these studied failed to conclude some promising results. Considering these failures, Hespel et al. [15] aimed to determine particle size distribution and spray structure close to nozzle exit. The results revealed the significant role of cavitation in the process of atomization and exit of liquid and ambient gas from the nozzle. Only a few studies have mainly focused on the numerical simulation of flow within the nozzles.

A recent study by Liao and Deng [23] conducted a numerical simulation of nozzle water jet to analyze exit nozzle diameter, length of the cylindrical section, and convergence angle of the nozzle water jet. The results demonstrated that there is a significant effect of hydraulic cutting on the optimal nozzle parameters. Recently, it is observed that alterations are required in building sizes and elevations in case of fire for maintaining water velocity and flow. Fire can cause loss of lives and monetary loss; therefore, it is essential to manage it with new techniques to prevent hazardous effects. It is believed that more water with increased velocity helps in eliminating obstacles on the way during firefighting. These changes are only possible by generating some new technique. Therefore, there is a need to focus on the effects of nozzle shapes on water flow characteristics in firefighting nozzles with grooves of different dimensions. The present study evaluates the impact of nozzle groove on fluid flow through the viscous 2D planar fluid.

METHODS

The main stages of study analysis were; pre-processing, simulation, and post-processing. The initial phase of modelling, i.e., the pre-processing step, mainly include; defining geometry, mesh generation, defining boundary conditions, and solver parameters. Constructed models are used at the stimulation stage; whereas, analysis and visualization of the obtained results are conducted at the post-processing stage in the form of contours.

Defining nozzle dimensions

The designing of nozzles is done using SolidWorks software that was modified to introduce groove shaped nozzles, each consisting of six different designs (Figure 1). These figures show multiple shapes of grooves with different dimensions and locations. The dimensions and distance of grooves from the nozzle exit are illustrated in Table 1.

Table 1: Dimensions and distance from exit for groove

Type of groove	Dimension(mm)	Distance from exit(mm)
A	1×1	10
B	1×1	15
C	2×2	10
D	2×2	15
E	1×1	7 and 15
F	2×2	7 and 15

The nozzle base model used in this study is similar to the one used by Zhang et al. [24]. All the nozzles presented in this study had these dimensions; 141.96 mm in length, 50 mm in diameter at the inlet, and 20 mm diameter at the outlet. The flow field considered in this case is steady, incompressible, and possess turbulent isothermal flow, with tensor form as follows in equation (1);

$$\frac{\partial \rho \mu_i}{\partial x_i}=0 \tag{1}$$

Equation 2 shows the formula to calculate the equation of incompressible viscous fluid motion;

$$\frac{\partial \mu_i}{\partial t}+\frac{\partial}{\partial x_j}\left(u_i u_j\right)=-\frac{1}{p}\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(v\frac{\partial u_i}{\partial x_j}-u_i u_j\right) \tag{2}$$

The different geometries of nozzles set in ANSYS FLUENT– Design Modeler platform specify the physical boundaries of the fluid. The present study selected type 2D, and each dimension for every nozzle were defined according to the model. The nozzle models for grooves are shown in Figure 2; whereas, the geometries are illustrated in Table 2.

Meshing and Boundary

The nozzle structure is axisymmetric, which comprises

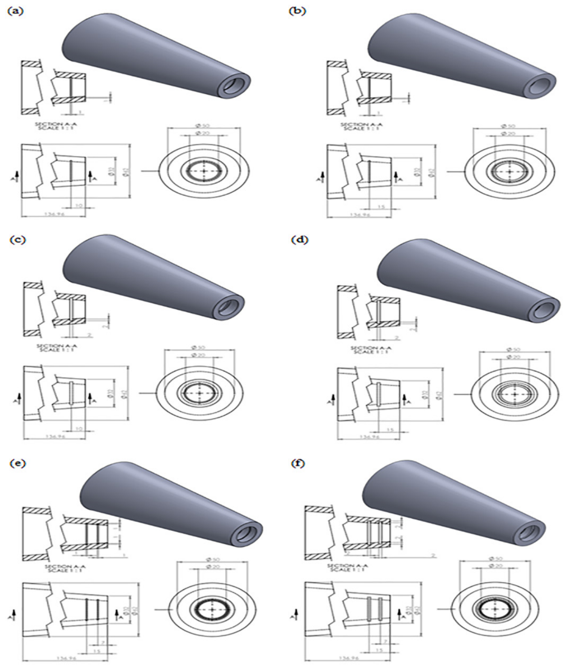


Figure 1: Groove designs; (a) dimension 1x1 at 10; (b) dimension 1x1 at 15; (c) dimension 2x2 at 10; (d) dimension 1x1 at 15; (e) dimension 1x1 at 7 and 15; (f) dimension 2x2 at 7 and 15 (all unit in mm)

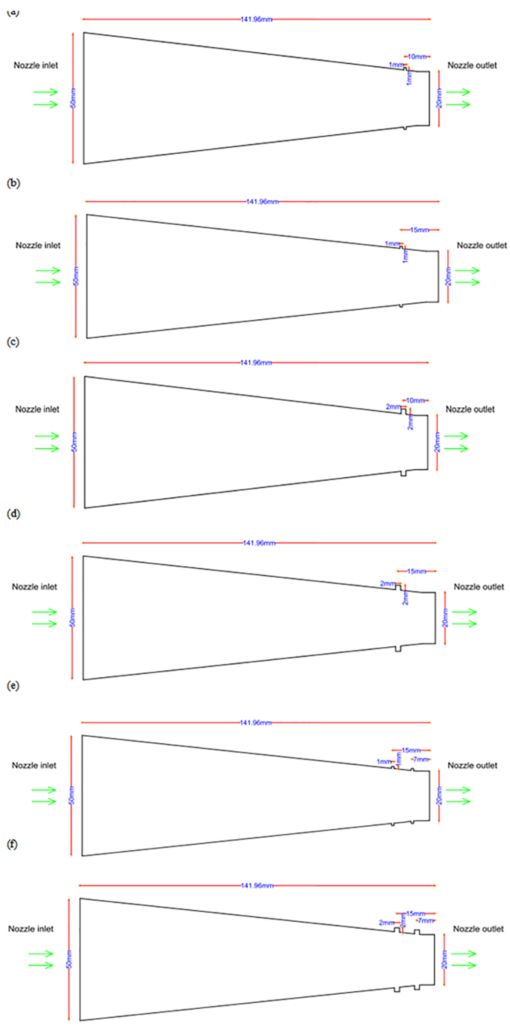


Figure 2: Groove models; (a) dimension 1x1 at 10; (b) dimension 1x1 at 15; (c) dimension 2x2 at 10; (d) dimension 1x1 at 15; (e) dimension 1x1 at 7 and 15; (f) dimension 2x2 at 7 and 15 (all unit in mm)

Table 2: Geometry of groove models

Nozzle Model type (Grooves)	Geometries parameters
Model 2a	One groove with dimensions of 1×1 mm and distance from nozzle exit 10 mm.
Model 2b	One groove with dimensions of 1×1 mm and distance from nozzle exit 15 mm.
Model 1c	One groove with dimensions of 2×2 mm and distance from nozzle exit 10 mm.
Model 2d	One groove with dimensions of 2×2mm and distance from nozzle exit 15 mm.
Model 2e	Double grooves with dimensions of 1×1 mm, 7mm distance for first ring and 15mm for second ring from nozzle exit.
Model 2f	Double grooves with dimensions of 2×2 mm, 7mm distance for first ring and 15 mm for second ring from nozzle exit.

Table 3: Mesh type and number of nodes

Mesh	Coarse	Medium	Fine
Number of nodes	8,00,000	1,200,000	1,400,000

a straight pipe ending at the nozzle exit. The nozzle is divided through grid division with an inlet diameter of 50 mm, outlet diameter of 20 mm, and needle diameter of 16 mm. The contraction angles of nozzles were adjusted at 30°, 40°, 50°, 60°, 70°, and 80°. The shapes and area of nozzles were calculated through ICEM software. Velocity inlet describes the boundary condition at the entrance. The velocity of the fluid at nozzle entrance is maintained at 20m/s; whereas, the boundary condition is axisymmetric on the axis and outflow is measured accordingly.

Appropriate boundary conditions that describe the physics of the fluid flow need to be defined in the CFD solver. It is considered that the nozzle has an inlet and an outlet. The inlet pressure is eight par for the first case, 10 bar for the second case and 12 bar for the third case. The nozzle outlet pressure is 1.10235 bar. For walls condition, the nozzle, rings, and grooves walls are considered hydraulically smooth, so the boundary condition at the wall is described by the 'no-slip condition', which is the relative velocity between the wall and the fluid is set to zero. Flow analysis was conducted in the steady-state mode for all models.

Mesh dependence study and validation

For the mesh dependence study, three different number of mesh were tested, coarse, medium and fine as shown in Table 3.

Figure 3 shows the volume flow rate computed at three different pressure values (8, 10 and 12 bar) for using the three different types of mesh mentioned in Table 1. It can be noticed that the results achieved by medium and fine mesh are almost identical. Therefore, the medium mesh is selected as a reference mesh number for all cases in this study.

Validation is the primary mean for evaluating the accuracy and reliability in computational simulations. A well-documented benchmark experimental or numerical data must be used to validate the process that assesses modelling uncertainty. To validate the numerical approach utilized in this study, the obtained CFD results were compared with the numerical data of Zhang and Zhu [24]. Figure 4 presents the comparison between the current study and Zhang and Zhu [24] for axial velocity component. It is clear that a very good agreement between the results of current study and Zhang and Zhu [24] is obtained.

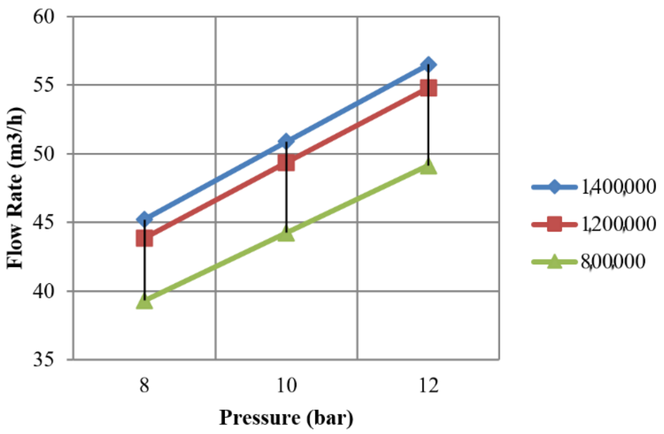


Figure 3: Flow rate for different mesh number

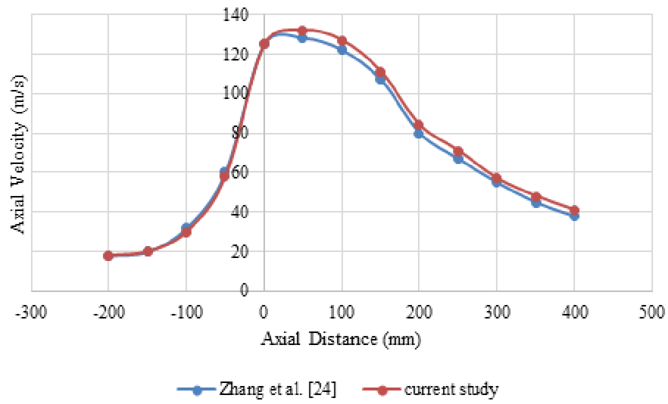


Figure 4: Axial velocity component (current study Vs. Zhang et al. [24])

Analysis

The nozzles were designed using SolidWorks software; while, ANSYS FLUENT 15.0 that is a computational fluid dynamics software package was used for stimulating the incompressible and viscous planar fluid across the nozzles. ANSYS ICEM was used for mesh generation and ANSYS FLUENT for flow analysis.

RESULTS AND DISCUSSION

The effect of groove dimensions 1×1 (mm) and location 10 mm at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle is depicted in Figure 5. The figure shows that velocity increased with increasing in pressures proportion relationship, which clearly explains that the groove has no effect on velocity as it increases due to increase in pressure. This is because the groove reduces the velocity by increasing surface area.

Figure 6 shows the effect of groove dimensions 1×1 (mm) and location 15 mm at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle. The representation shows that velocity increased with increase in pressures proportion relationship. Therefore, it can be stated that the groove has no effect and velocity mainly increases due to increasing pressure as the groove reduces the velocity by increasing surface area.

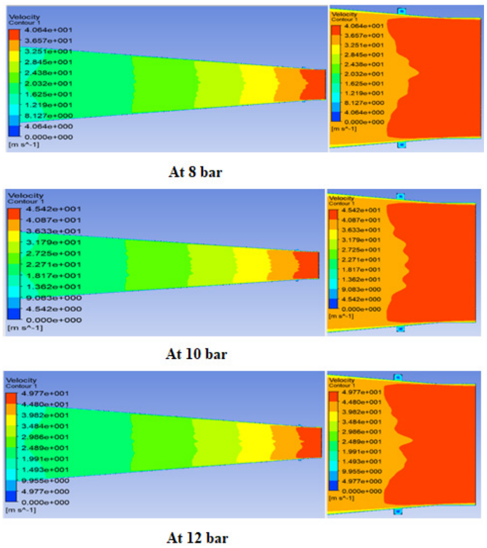


Figure 5: Velocity contours of first groove model with dimensions of 1×1 mm and distance from nozzle exit 10 mm

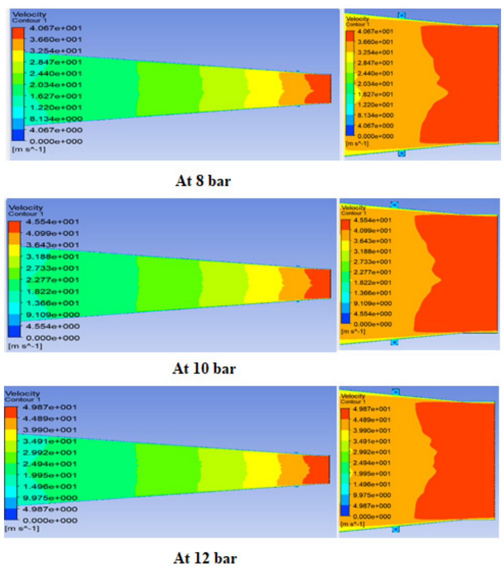


Figure 6: Velocity contours of second groove model with dimensions of 1×1 mm and distance from nozzle exit 15 mm

Figure 7 illustrates the effect of groove dimensions 2×2 (mm) and location 10 mm at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle. The velocity and pressure increase in a proportional relationship. The figure clearly shows that groove has no effect on the velocity because the groove reduces the velocity by increasing surface area.

Figure 8 shows the effect of groove dimensions 2×2 (mm) and location 15 mm at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle. This figure shows no effect on velocity because the groove reduces the velocity by increasing surface area.

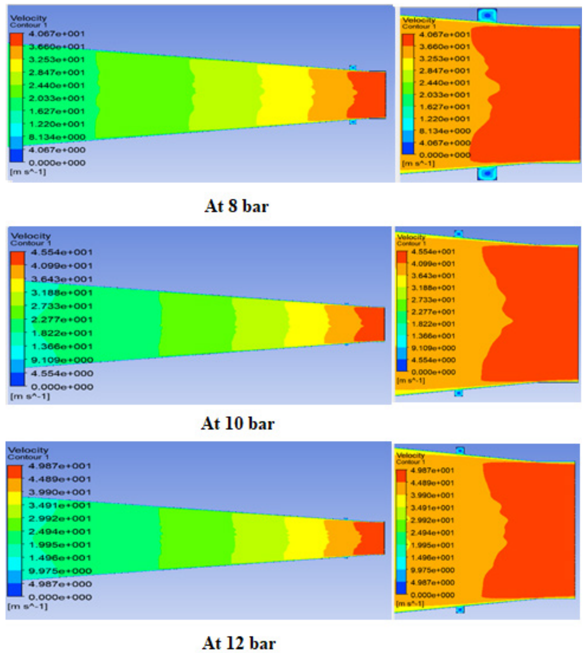


Figure 7: Velocity contours of third groove model with dimensions of 2×2 mm and distance from nozzle exit 10 mm

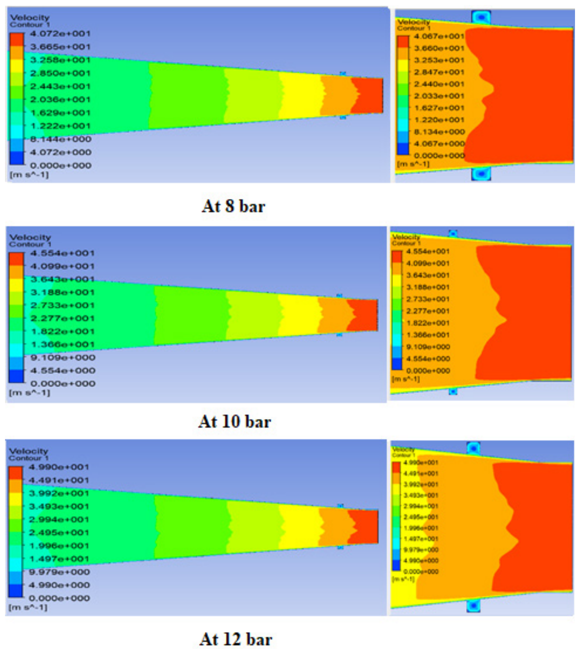
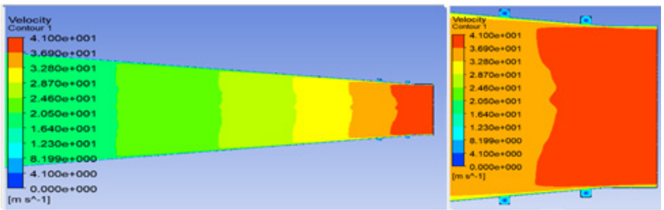


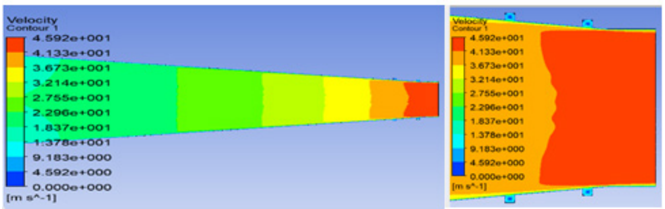
Figure 8: Velocity contours of fourth groove model with dimensions of 2×2mm and distance from nozzle exit 15 mm

Figure 9 illustrates the effect of double rings dimensions 1×1 (mm) and location 7 mm for the first and 15mm for second ring from the exit at 8 bar, 10 bar and 12 bars, in velocity of water during flow in the nozzle. The velocity increased with increasing pressures in a proportional relationship. There was no effect of increasing pressure on the velocity because the groove reduces the velocity as the surface area increased.

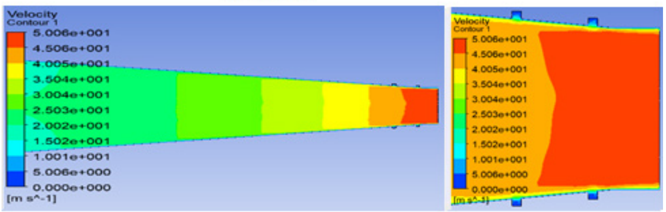
Similar to the previous settings, Figure 10 also shows proportional relationship between increasing velocity and pressure. The results clearly demonstrate that there is negative effect of all dimensions and distance of groove location on the velocity of water during flow, at different pressure 8, 10, and 12 bar. This might be due to increase in surface area that decreases the velocity. There is no significant effect of groove on the velocities and turbulence intensities, as observed by comparing the corresponding plots because of experiencing much stress across the nozzle all the way up to the ringed nozzle exit. These results are in agreement with Yu et al. [5] as the study reported little effect of groove extending or exit the nozzle on the characteristics of fluid flow. It is of significance, as understanding it will help in considering its effect in firefighting, so that necessary changes in the model can be suggested.



At 8 bar



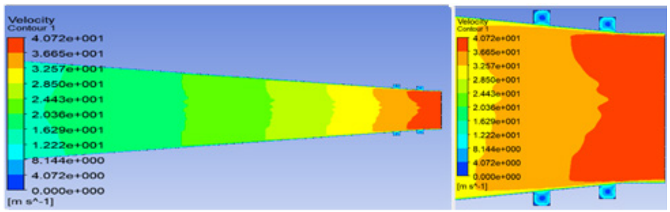
At 10 bar



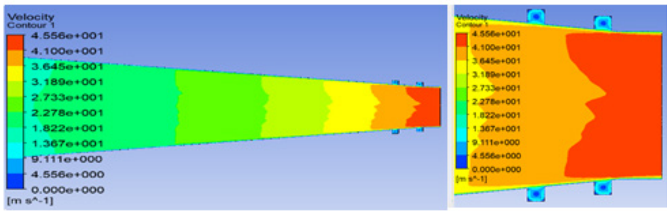
At 12 bar

Figure 9: Velocity contours of fifth groove model with dimensions of 1×1 mm, 7mm distance for first ring and 15mm for second ring from nozzle exit

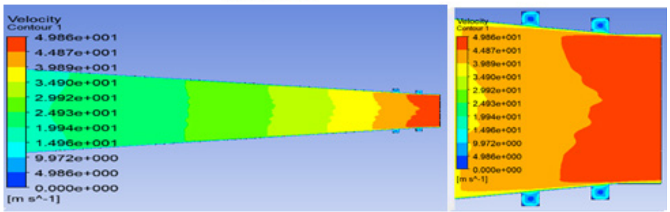
The aerodynamic performance of micro lobed nozzles ejectors of different dimensions was compared in an experimental and numerical study by Shan et al. [25]. The study used the k-ε standard turbulence model based on Reynolds-Averaged Navier-Stokes (RANS) simulations



At 8 bar



At 10 bar



At 12 bar

Figure 10: Velocity contours of sixth groove model dimensions of 2×2 mm, 7mm distance for first ring and 15 mm for second ring from nozzle exit

and showed significant impact of nozzle shape on the fluid downstream. Another similar study by Babu and Mahesh [26] studied the impact of inflow condition on the ambient fluid exiting the nozzle in a jet. The study failed to emphasize on the significance of initial conditions as the fluid exits the nozzle. On the contrary, the present study has drawn numerous findings and conclusions regarding certain physical and methodological aspects of fluid flow through a nozzle. The study also showed significant impact of nozzle shape on the exit conditions in the post-contraction region. A similar study by Yu et al. [5] compared the experimental data to investigate the impact of nozzle geometry on the turbulence characteristics of fluid flow through the nozzles. It is also likely that the analytical model of the pressure attributes is authentic and beneficial to obtain an insight of the physical mechanism [27]. Moreover, the passive control device can control the base pressure, which result in an improvement in the base pressure and mitigating the base drag. In addition, passive control is very effective and efficient whenever there is preferable pressure gradient at the nozzle exit [28, 29]. The results demonstrated that presence of ring required increased inlet pressure for moving fluid through the nozzle as it increases the mean velocity and turbulence at the exit. These results are somewhat consistent with the present study that showed rings modifying the nozzle shape have significant impact on velocity.

CONCLUSION

The present study has investigated the effect of different nozzle shapes on fluid flow characteristics in firefighting nozzles. There was no effect of groove design with dimensions 1×1 (mm) and location 15 mm at 8 bar, 10 bar and 12 bar because increase in surface area reduces the fluid velocity. These results demonstrate that there is no effect of the nozzle modified with grooves on the fluid velocity. These results exhibit good expectations about the use of rings to increase velocity of water that further improve nozzles operation in the same dimension.

The study has demonstrated that flow quantities in any region were not changed by extending the nozzle by one nozzle exit diameter. Slight change in the wall pressure is likely to be observed by cutting groove in the proximity of exit. This is clearly explained by the present study as addition of ring changes the pressure field in the post-contraction zone. The baseline nozzle with same flow rate imposed on nozzle was not effective in increasing the mean velocity of fluid in the jet. These findings suggest that inserting grooves without rings do not present promising results in achieving higher mixing efficiencies in the baseline nozzle, as it creates exit flow that is conducive to mixing. However, the future studies need to use the model developed in this study to assess the effect of water separations on velocity with inclusion of more grooves along with rings.

ACKNOWLEDGEMENT

This publication was supported by the Deanship of Scientific Research at Prince Sattam bin Abdulaziz University, Alkharj, Saudi Arabia.

REFERENCES

1. Fisher B.A., Snitkoff J.R. (2018). U.S. Patent No. 10,138,716. Washington, DC: U.S. Patent and Trademark Office.
2. Amini G. (2016). Liquid flow in a simplex swirl nozzle. International Journal of Multiphase Flow, Vol. 79, pp. 225-235. https://doi.org/10.1016/j.ijmultiphaseflow.2015.09.004
3. Liu X., Xue R., Ruan Y., Chen L., Zhang X., Hou Y. (2017). Flow characteristics of liquid nitrogen through solid-cone pressure swirl nozzles. Applied Thermal Engineering, Vol. 110, pp. 290-297. https://doi.org/10.1016/j.applthermaleng.2016.08.150
4. Zhou, K., Wang, Y., Zhang, L., Wu, Y., Nie, X., Jiang, J. (2020). Effect of nozzle exit shape on the geometrical features of horizontal turbulent jet flame. Fuel, Vol. 260, p. 116356.
5. Mat, M.N.H., Asmuin, N.Z., Basir, M.F.M., Goodarzi, M., Abd Rahman, M.F., Khairulfuaad, R., Jabbar, B.A., Kasihmuddin, M.S.M. (2020). Influence of divergent length on the gas-particle flow in dual hose dry ice blasting nozzle geometry. Powder Technology, Vol. 364, pp. 152-158.
6. Bilir A.Ç., Doğrul A., Coşgun T., Yurtseven A., Vardar N. (2016). A numerical Investigation of the Flow in Water Jet Nozules. Journal of Thermal Engineering, Vol. 2, No. 5, pp. 907-912. https://doi.org/10.18186/jte.55087
7. Matsuo S., Kim T.H., Setoguchi T., Kim H.D., Lee, Y.W. (2007). Effect of nozzle geometry on the flow characteristics of spiral flow generated through an annular slit. Journal of Thermal Science, Vol. 16, No. 2, pp. 149-154. https://doi.org/10.1007/s11630-007-0149-4
8. Banat, R.A.A., Adam, A.M.H., Younis, O., Elsir, D. (2018). The Effects of Nozzle Shape on the Flow Characteristics-A Review. European Academic Research, Vol. 5, No. 8, pp. 4874-4886.
9. Alam M.M.A., Setoguchi T., Matsuo S., Kim H.D. (2016). Nozzle geometry variations on the discharge coefficient. Propulsion and Power Research, Vol. 5, No. 1, pp. 22-33. https://doi.org/10.1016/j.jppr.2016.01.002
10. Mohamed S., Mokhtar A., Chatti T.B. (2017). Numerical simulation of the compressible flow in convergent-divergent nozzle. International Journal of Heat and Technology, Vol. 35, No. 1, pp. 673-677. https://doi.org/10.18280/ijht.350328
11. Babu P.C., Mahesh K. (2004). Upstream entrainment in numerical simulations of spatially evolving round jets. Physics of Fluids, Vol. 16, No. 10, pp. 3699-3705. https://doi.org/10.1063/1.1780548
12. Anghan, C., Dave, S., Saincher, S., Banerjee, J. (2019). Direct numerical simulation of transitional and turbulent round jets: Evolution of vortical structures and turbulence budget. Physics of Fluids, Vol. 31, p. 065105.
13. Jassim E.I., Awad, M.M. (2013). Numerical investigation of nozzle shape effect on shock wave in natural gas processing. In Proceedings of World Academy of Science, Engineering and Technology (Vol. 78, p. 326). World Academy of Science, Engineering and Technology (WASET).
14. Jassim, E. I. (2019). Geometrical impaction of supersonic nozzle on the dehumidification performance during gas purification process: an experimental study. Arabian Journal for Science and Engineering, Vol. 44, pp. 1057-1067.
15. Hespel, C., Blaisot, J. B., Margot, X., Patouna, S., Cessou, A., Lecordier, B. (2010). Influence of nozzle geometry on spray shape, particle size, spray velocity and air entrainment of high pressure diesel spray. In THIESEL 2010-Conference on Thermo-and Fluid Dynamic Processes in Diesel Engines, pp. 383-394.
16. Agarwal, A., Trujillo, M. F. (2020). The effect of nozzle internal flow on spray atomization. International Journal of Engine Research, Vol. 21, pp. 55-72.

17. Zhang, X., He, Z., Wang, Q., Tao, X., Zhou, Z., Xia, X., Zhang, W. (2018). Effect of fuel temperature on cavitation flow inside vertical multi-hole nozzles and spray characteristics with different nozzle geometries. Experimental Thermal and Fluid Science, Vol. 91, pp. 374-387.
18. Kumar, A., Sahu, S. (2020). Influence of nozzle geometry on primary and large-scale instabilities in coaxial injectors. Chemical Engineering Science, Vol. 221, p. 115694.
19. Satyanarayana G., Varun C., Naidu S.S. (2013). CFD analysis of convergent-divergent nozzle. Acta Technica Corviniensis-Bulletin of Engineering, Vol. 6, No. 3, pp. 139.
20. Mashida, M., Sou, A. (2018). Effects of inlet edge roundness on cavitation in injector nozzles and liquid jet. International Journal of Automotive Engineering, Vol. 9, pp. 9-15.
21. Sou A., Maulana M.I., Isozaki K., Hosokawa S., Tomiyama A. (2008). Effects of nozzle geometry on cavitation in nozzles of pressure atomizers. Journal of Fluid Science and Technology, Vol. 3, No. 5, pp. 622-632.
22. Badock C., Wirth R., Fath A., Leipertz A. (1999). Investigation of cavitation in real size diesel injection nozzles. International journal of heat and fluid flow, Vol. 20, No. 5, pp. 538-544.
23. Liao W.T., Deng X.Y. (2017). Study on Flow Field Characteristics of Nozzle Water Jet in Hydraulic cutting. In IOP Conference Series: Earth and Environmental Science (Vol. 81, No. 1, p. 012167). IOP Publishing. https://doi.org/10.1088/1755-1315/81/1/012167
24. Zhang S.B., Zhu J.M. (2013). Numerical simulation of adjustable nozzles. In IOP Conference Series: Materials Science and Engineering (Vol. 52, No. 7, p. 072014). IOP Publishing. https://doi.org/10.1088/1757-899x/52/7/072014
25. Shan Y., Zhang J.Z., Huang G.P. (2011). Experimental and numerical studies on lobed ejector exhaust system for micro turbojet engine. Engineering Applications of Computational Fluid Mechanics, Vol. 5, No. 1, pp. 141-148.
26. Babu P.C., Mahesh K. (2004). Upstream entrainment in numerical simulations of spatially evolving round jets. Physics of Fluids, Vol. 16, No. 10, pp. 3699-3705.
27. Saha B.K., Songjing L.I., Xinbei L.V. (2020). Analysis of pressure characteristics under laminar and turbulent flow states inside the pilot stage of a deflection flapper servo-valve: Mathematical modeling with CFD study and experimental validation. Chinese Journal of Aeronautics, Vol. 33, No. 3, pp.1107-1118.
28. Joseph J, Rehman D, Delanaye M, Morini GL, Nacereddine R, Korvink JG, Brandner JJ. (2020). Numerical and experimental study of microchannel performance on flow maldistribution. Micromachines. Vol.11, No. 3, pp. 323.
29. Khan A., Rajendran P., Sidhu J.S.S. (2021). Passive Control of Base Pressure: A Review. Applied Sciences, Vol. 11, No. 3, pp.1334.

Paper submitted: 07.03.2021.
Paper accepted: 06.05.2021.