Terbit *online* pada laman web jurnal : http://metal.ft.unand.ac.id

METAL: Jurnal Sistem Mekanik dan Termal

| ISSN (Print) 2598-1137 | ISSN (Online) 2597-4483 |

Research article

Effect of Convergent Angle on the Fluid Dynamic of Fire-Fighting Nozzle

Awinda Sari Riawan

Department of Mechanical Engineering, Universiti Tun Hussein Onn Malaysia, Batu Pahat, Johor, Malaysia

ARTICLE INFORMATION

Article History:
Received : 10 January 2025
Revised : 15 April 2025
Accepted : 30 April 2025

KEYWORDS

Simulation study
Internal Flow
Fire Fighting
Nozzle

CORRESPONDENCE

E-mail: awindasriawan@gmail.com

A B S T R A C T

The nozzle is extensively used in many industrial contexts with variable shapes to satisfy particular application requirements, influencing fluid properties. Firefighting sprinkler nozzles have a variety of shapes, leading to research on the possibility of flow treatment devices to improve nozzle performance using numerical analysis. This simulation study examined three different geometry angles of the nozzle (15, 20, and 25 degrees) with different parameters (inlet pressure) and found that the optimal geometry performed the best in terms of pressure retention and velocity output, which are critical for efficient fire suppression operations, with performance varying significantly depending on nozzle angle and applied pressure. A nozzle with a 20-degree angle geometry showed the best performance, with good pressure retention and velocity output, both of which are important for efficient fire suppression. Developing a highly efficient and high-performing nozzle is critical for increasing efficiency and effectiveness in firefighting operations.

1. INTRODUCTION

Nozzle geometry directly affects the performance of jets, which are further used for subsequent applications such as cleaning, cutting, and drilling [1]. Nozzles are essential components in a variety of technical and industrial applications since they transform pressure into kinetic energy to regulate fluid direction, flow rate, and velocity. The geometry of a nozzle, namely the convergent angle and throat length, has a substantial impact on fluid dynamics, affecting the efficiency of applications such as fire suppression systems, rocket engines, and hydraulic systems. Rotating multi-orifice nozzles can be used in oil and gas field drilling [2], [3]. Various types of non-circular nozzles have been investigated in recent years for abrasive jet cutting and spray irrigation [4], [5]. Among these, the cone-straight nozzle is most widely used in the oil field due to its relatively simple process.

Fire suppression is an important aspect of ensuring public safety, with the concomitant advancements in technology and engineering that have led to the development of enhanced fire suppression systems offering better flow and pressure performance [6]. Clutter in the form of structural supports, cables, pipes, machinery, and similar items can adversely affect suppressant flow, making it difficult to optimise fire suppression systems in many applications [7], [8], [9]. Since the mechanism of the water spray pattern created by a normal sprinkler system is crucial to developing and working against clutter fully, greater attention has been dedicated to investigating this pattern quantitatively. Many elements affect the water spray flow pattern, including water flow rate, spray nozzle type, droplet size, sprinkler head placement, ventilation conditions, and objects in the path [10], [11].

The complexity of the model used to model this problem is due to the fact that it includes several variables such as flow, flow pattern, and velocity, not forgetting the geometric features of the nozzle such as diameter, roughness, and angle of

https://doi.org/ 10.25077/metal.9.1.29-37.2025

inclination. This means that sources of uncertainty especially in determining the length of the hose used with the nozzle diameter at the flow velocity are not identified. One important parameter that significantly affects nozzle behaviour is the angle of convergence. The convergent angle determines the velocity at which the fluid converges towards the nozzle throat. Previous research has shown that certain convergent angles can cause boundary layer transitions and separation, which affect the flow characteristics and performance at the nozzle [1], [12], [13]. The effect of a given convergent angle and flow velocity on boundary layer transition and separation is significant with resistance to skin friction initially increasing and then decreasing with an increase in flow velocity when the angle is greater than 15° [14].

The convergence angle exerts an impact on the acceleration and velocity of fluid, potentially enhancing velocity with larger angles, yet leading to the occurrence of boundary layer separation and flow instability [12], [15]. Throat length, the narrowest section of the nozzle, stabilises flow and enhances jetting performance, with optimal lengths ensuring minimal pressure loss and efficient fluid acceleration [15], [16]. Many researchers have investigated the influences of various parameters on a jetting angle, jetting length, and impact force, especially in breaking rocks, such as coalbed, sandstone, and gas hydrate [17], [18], [19]. Many researchers have investigated the influences of various parameters on a jetting angle, jetting length, and impact force, especially in breaking rocks, such as coalbed, sandstone, and gas hydrate [17], [18], [19].

Previous research has shown that certain convergent angles can cause boundary layer transitions and separation, which affect the flow characteristics and performance at the nozzle [1], [12], [13]. The effect of a given convergent angle and flow velocity on boundary layer transition and separation is significant with resistance to skin friction initially increasing and then decreasing with an increase in flow velocity when the angle is greater than 15° [14]. They studied the internal flow of the nozzle by studying the effects of flow numerically and conducted a study on cone strength nozzles comparing the k-epsilon RNG with the Les model to accurately capture the details of the flow near the nozzle wall [20].

This study attempts to enhance the reliability and efficacy of firefighting equipment and techniques by comparing theoretical predictions to real measurements and firefighting standards. This study helps to the refinement of firefighting procedures for safer and more effective fire management by analysing the elements that impact them statistically. By designing the nozzle convergent angle, it is more efficient to design a nozzle that maximises efficiency and performance in fire suppression operations for maximum efficiency and performance in its application use.

2. METHODOLOGY

2.1 Geometry of Nozzle

The nozzles are designed on Ansys modern, the design in where D_1 is 15 mm, D_2 is 4 mm, L is 20 mm, Throat length is 5 mm and the angle is set at 15, 20, and 25 degrees shown in Figure 1 The internal of the nozzle and other dimensions are kept constant to isolate the effect of low characteristics.

Table 1. Geometrical parameters of 3-D nozzle

Parameter	Value	Unit
Length	20	mm
Diameter inlet	15	mm
Diameter outlet	4	mm
Throat length	5	mm
Angle	15 - 25	degree

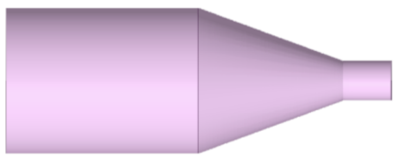

Figure 1. Nozzle geometry 20-degree, 3-D nozzle models

2.2 Meshing

Meshing is a crucial process for accurately representing the flow field inside and outside the cone-straight nozzle during simulations. The meshing process involves dividing the computational domain into small elements to discretise the geometry for numerical simulations. Mesh refinement techniques are used to ensure that the grid near the wall is fine enough to capture turbulent structures and boundary layer transitions effectively. Proper meshing helps in accurately capturing flow characteristics, such as velocity profiles and pressure distributions, which aids in the validation of computational models against

 https://doi.org/ 10.25077/metal.9.1.29-37.2025

experimental data. The meshing process plays a significant role in obtaining reliable results by providing a framework.

2.3 Boundary Condition

At the inlet, the constraints are velocity-inlet with pressures of 1 bar, 3 bar, and 5 bar, a diameter of 15 mm, a velocity of 5 m/s, and a turbulence intensity of 1.06%. At the outlet, the constraints are a pressure outlet with a pressure of 0 gauge pressure and a diameter of 4 mm. The nozzle body has smooth walls with no-slip shear conditions and cone angles of 15°, 20°, and 25°. The working fluid used is water at normal temperature with a density of 997 kg/m³ and a dynamic viscosity of 0.79722.

2.4 Simulation

The double-precision pressure-solving method works by solving each equation independently, using the standby calculation method, turbulent viscosity model, standard pressure, pressure-velocity coupling using SIMPLE, and second-order momentum. To achieve stable flow conditions, the RNG k-epsilon model is used, and its calculation will be set as the initial condition for the LES simulation.

2.5 Governing Equation

The Reynolds-Averaged Navier-Stokes equation (RANS) is used to solve the analysis for Newtonian and incompressible fluid flow which is shown as follows:

Continuity equation:

$$\frac{\partial \mathrm{v}}{\partial x}+\frac{\partial \mathrm{u}}{\partial y}+\frac{\partial \omega}{\partial z}=0 \quad (1)$$

X-momentum equation:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+\omega\frac{\partial u}{\partial z} \quad (2)$$

$$=-\frac{1}{\rho}\frac{\partial P}{\partial x}+v\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial \omega^2}\right]+\frac{1}{\rho}\left[\frac{2(-\rho\overline{u'v'})}{\partial x}+\frac{2(-\rho\overline{u'^2})}{\partial y}+\frac{2(-\rho\overline{u'v'\omega})}{\partial z}\right] \quad (2)$$

Y-momentum equation:

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+\omega\frac{\partial v}{\partial z} \quad (4)$$

$$=-\frac{1}{\rho}\frac{\partial P}{\partial y}+v\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right]+\frac{1}{\rho}\left[\frac{2(-\rho\overline{u'v'})}{\partial x}+\frac{2(-\rho\overline{u'^2})}{\partial y}+\frac{2(-\rho\overline{u'v'\omega})}{\partial z}\right] \quad (5)$$

Z-momentum equation:

$$u\frac{\partial \omega}{\partial x}+v\frac{\partial \omega}{\partial y}+\omega\frac{\partial \omega}{\partial z} \quad (6)$$

$$=-\frac{1}{\rho}\frac{\partial P}{\partial x}+v\left[\frac{\partial^2 \omega}{\partial x^2}+\frac{\partial^2 \omega}{\partial y^2}+\frac{\partial^2 \omega}{\partial \omega^2}\right]+\frac{1}{\rho}\left[\frac{2(-\rho\overline{\omega'v'})}{\partial x}+\frac{2(-\rho\overline{\omega'^2})}{\partial y}+\frac{2(-\rho\overline{u'v'\omega})}{\partial z}\right] \quad (7)$$

3. RESULT AND DISCUSSION

3.1. Grid Independence Test

Grid independence test is a study of determining the appropriate mesh setting for the models created by running the simulation for each increase in the number of nodes and elements. Mesh setting is refined as the number of nodes and elements increases until the simulation result shows an acceptable deviation percentage from the two recent mesh settings. After consideration for accuracy and CPU solution time, the mesh with the lower number of nodes and elements between the two mesh settings was chosen for the entire study. For this study, the mesh of the curved diffuser was generated in a Hexahedron cell structure.

As depicted in Figure 2, presents the relationship between mesh nodes and velocity for nozzles with convergent angles of 15°, 20°, and 25°, demonstrating the outcomes of the grid independence test. It is observed that fewer meshes result in converging velocity values, such as stabilising at 0.04 m/s with 2,000,000 cells at a 15° angle. Upon analysing different situations, it became apparent that employing 3,000,000 cells for a 20° angle can lead to a consistent velocity of 0.05 m/s. On the other hand, approximately 4,000,000 cells are needed for the 25° angle in order to attain grid independence, as a result of the varying flow dynamics.

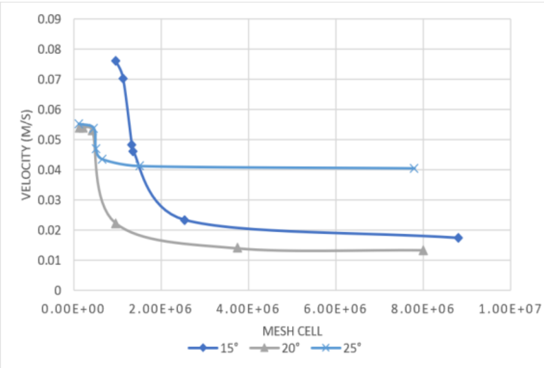

Figure 2. Mesh node vs velocity out of the different geometries of the nozzle

The accuracy and validity of the simulation model and the results were compared with data from a study conducted by T. Jiang. Simulation results at different angles in Figure 3 and 4 show that the comparison indicates notable differences in the velocity and outlet pressure predictions between the present and previous works. The 15° and 20° angles show relatively minor discrepancies, suggesting consistency in the modelling approach for these angles. However, the 25° angle exhibits significant differences, especially in velocity at higher inlet pressures and in the outlet pressure trend. The variations we observed can be explained by differences in the experimental conditions, computational methods, and assumptions used in the simulations. The greater discrepancies at higher pressures, especially for the 25° angle, may indicate complexities in flow dynamics that are not consistently accounted for in the studies. This underscores the need to improve and validate the simulation models to guarantee their accuracy and dependability in predicting nozzle performance under different operating conditions.

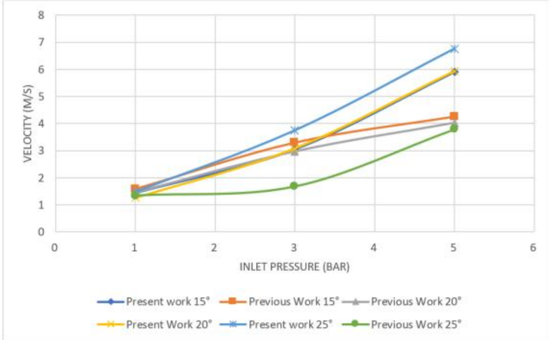

Figure 3. Comparison velocity of present work and previous work in convergent angle

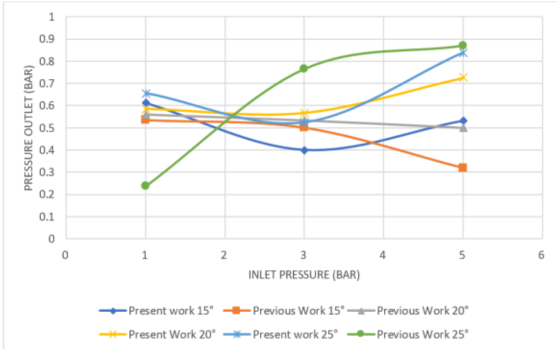

Figure 4. Comparison pressure of present work and previous work in convergent angle

Figure 5 shows the connection between input pressure (from 1 bar to 5 bar) and output velocity for nozzles with convergent angles of 15°, 20°, and 25°. For the 15° angle, the speed starts at around 0.6 m/s at 1 bar, drops to about 0.4 m/s at 3 bar, and then increases back up to around 0.5 m/s at 5 bar, demonstrating a non-linear connection. At a 20° inclination, a consistent velocity trend of 0.6 m/s is recorded from 1 bar to 2 bar, which then slightly reduces to roughly 0.55 m/s at 3 bar and progressively increases to approximately 0.7 m/s at 5 bar, suggesting a more uniform pattern. The 25° angle highlights the highest velocities, starting at approximately 0.7 m/s at 1 bar, slightly dropping to about 0.5 m/s at 3 bar, and then rapidly rising to around 0.8 m/s at 5 bar. This indicates that the 25° angle offers the most effective acceleration of a fluid through the nozzle at elevated pressures. In contrast, the 20° angle demonstrates steady performance, and the 15° angle displays intricate flow characteristics. These findings emphasise the significance of choosing the appropriate convergent angle to attain the desired velocity attributes in nozzle design, with the 25° angle proving the most

 https://doi.org/ 10.25077/metal.9.1.29-37.2025

efficient for high-velocity applications at increased pressures.

Figure 6 shows the association between inlet pressure (ranging from 1 bar to 5 bar) and outlet pressure for nozzles with convergent angles of 15°, 20°, and 25°. The nozzle with a 15° angle demonstrates a continuous linear rise in outlet pressure, starting at approximately 1.0 bar with 1 bar inlet pressure and reaching around 4.0 bar with 5 bar inlet pressure, highlighting a consistent association between inlet and outlet pressures. In contrast, the 20° angle commences with an outlet pressure of approximately 1.0 bar at 1 bar inlet pressure, swiftly escalating to around 4.7 bar at 3 bar inlet pressure, and then gradually increasing to roughly 5.0 bar at 5 bar inlet pressure. This indicates a heightened responsiveness to inlet pressure at lower levels, stabilising as pressures increase. Additionally, the 25° angle displays a constant rise in outlet pressure from about 1.0 bar at 1 bar inlet pressure to approximately 4.0 bar at 5 bar inlet pressure, albeit with slightly lower magnitudes than the 15° angle at higher pressure levels. These findings suggest that the 20° angle is particularly adept at producing higher outlet pressures, which is advantageous for applications necessitating elevated pressure outputs, while the 15° and 25° angles provide more consistent and foreseeable performance across varying inlet pressures. A comprehensive comprehension of these dynamics is essential for enhancing nozzle design for specific purposes, striking a balance between pressure requirements and flow efficiency.

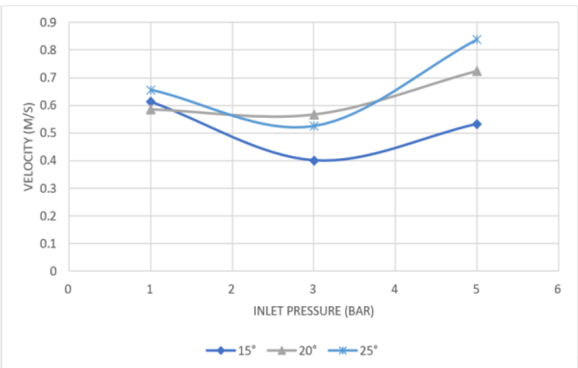

Figure 5 Inlet Pressure vs Velocity at Different Convergent Angles

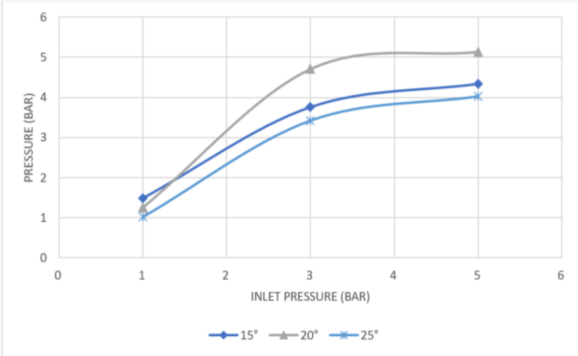

Figure 6. Inlet Pressure vs Pressure at Different Convergent Angles

3.2. Effect of Throat Length on Velocity and Pressure

Figure 7 illustrates the correlation between inlet pressure (ranging from 1 bar to 5 bar) and the velocity at the throat of the nozzle for convergent angles of 15°, 20°, and 25°. At a 15° angle, there is a notable increase in speed from around 0.1 m/s at 1 bar to about 0.6 m/s at 3 bar, followed by a slight decrease to approximately 0.5 m/s at 5 bar, indicating a non-linear trend. The 20° angle exhibits a more linear rise in velocity from approximately 0.3 m/s at 1 bar to about 0.7 m/s at 5 bar, demonstrating consistent growth with rising inlet pressure. The 25° angle starts with a lower initial velocity at 1 bar but shows a steady increase, reaching roughly 0.7 m/s at 5 bar, achieving the highest velocity among the three angles at higher pressures. These results suggest that the 25° angle provides the most efficient fluid acceleration at elevated pressures. In contrast, the 20° angle delivers consistent performance, while the 15° angle may not maintain velocity as effectively at higher pressures. This evaluation emphasizes the importance of selecting the appropriate convergent angle to optimize nozzle performance for desired velocity characteristics under varying pressure conditions.

Figure 8 portrays the relationship between inlet pressure (ranging from 1 bar to 5 bar) and outlet pressure for nozzles with convergent angles of 15°, 20°, and 25°. The 15° angle reveals a sudden increase in outlet pressure, starting at around 0.5 bar at 1 bar inlet pressure and rising to roughly 3.0 bar at 5 bar inlet pressure, with a more gradual incline above 3 bar. The 20° angle demonstrates a linear progression in outlet pressure from about 1.5 bar at 1 bar inlet pressure to roughly 3.5 bar at 5 bar inlet pressure, indicating a consistent trend. The 25° angle exhibits a steady enhancement in outlet pressure, beginning at approximately 0.5 bar at 1 bar inlet pressure and reaching about 2.5 bar at 5 bar inlet pressure, resembling the behavior of the 15° angle, albeit with slightly lower pressures at higher inlet pressures. These findings suggest that the 20° angle is the most effective for applications requiring higher outlet pressures, while the 15° and 25° angles offer more consistent and predictable performance across varying inlet pressures. Understanding these dynamics is crucial for optimizing nozzle design to achieve a balance between pressure and flow efficiency.

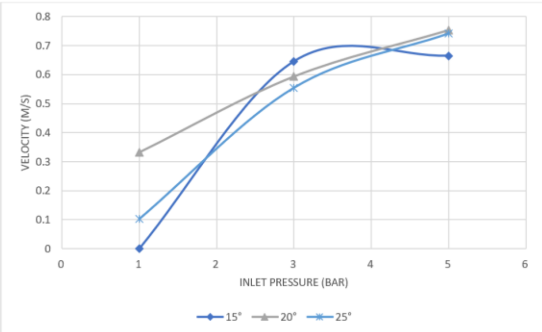

Figure 7. Relation between inlet velocity vs velocity at the throat nozzle

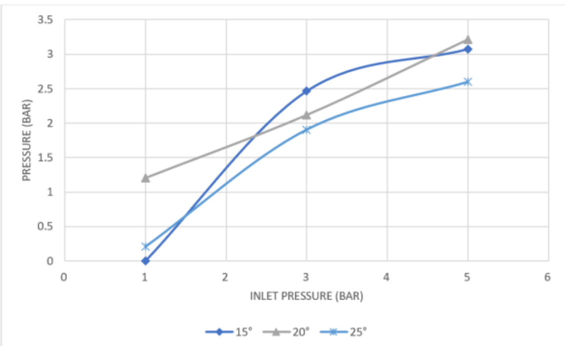

Figure 8. Relation between inlet velocity vs velocity at the throat nozzle

3.3. Velocity Vector and Pressure Contour at Different Convergent Angles

The velocity vector plots presented depict the fluid flow characteristics in the nozzle for three configurations, showing variations in flow stability and turbulence. Differences in flow behaviour influenced by nozzle geometries and operating conditions underscore the significance of optimising nozzle design for stable and efficient fluid dynamics. Figure 9 shows the nozzle performance statistics for convergent angles of 15°, 20°, and 25° for input pressures of 5 bar. The 20° angle provides considerable velocity improvements and stability via effective fluid acceleration, making it the best option for all pressure circumstances. The pressure contour plots in Figure 10 show that the 20° angle is the best choice for firefighting nozzles because of its consistent pressure distribution, while the 15° angle is slightly less effective, and the 25° angle has significant instability and pressure fluctuations, making it the least effective option.

3.4 Optimal Convergent Angle and Throat Length for Fire Fighting Application

The best variation in each parameter is defined by the pressure that is standard in fire fighting systems in general, with a minimum pressure of 0.5 bar at the outlet and a velocity of 6 m/sec. The results of velocity and pressure calculations show that a nozzle with a 20° convergent angle and a 5 mm throat length provides improved performance, high-velocity output, and constant fluid flow. This design ensures efficient water delivery and fire suppression due to its steady pressure distribution and minimum turbulence, making it the best choice for firefighting applications.

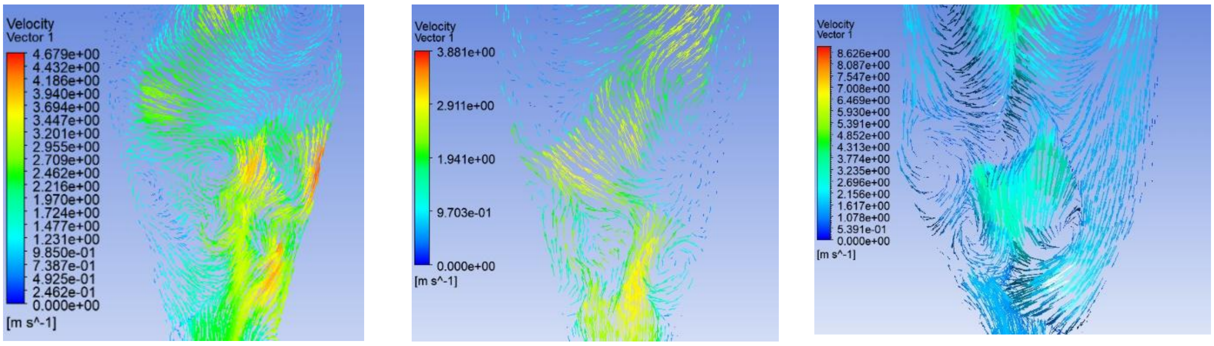

Figure 9. Velocity vector of convergent angle (15, 20, 25 degree) in 5 bar inlet

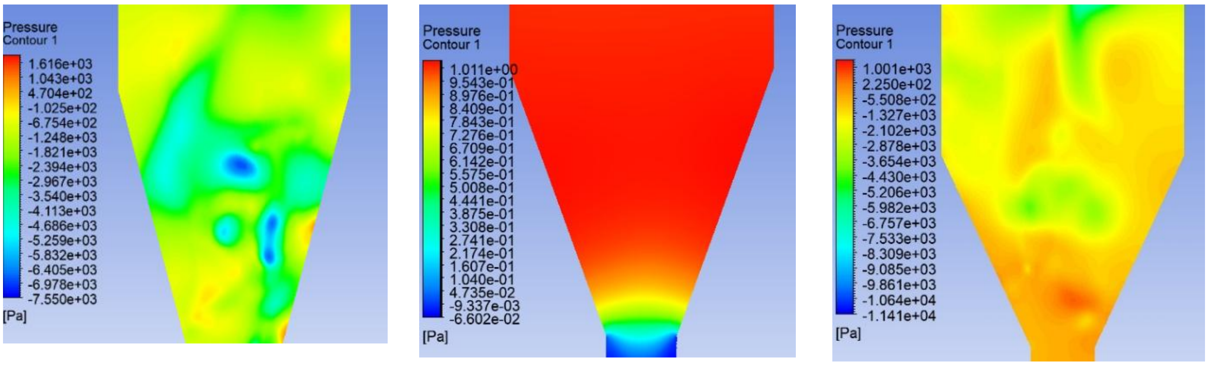

Figure 10. Pressure contour of convergent angle (15, 20, 25) in 5 bar inlet

4. DISCUSSIONS

The findings of this study provide valuable insights into the relationship between nozzle geometry and fluid dynamics in firefighting applications. The numerical simulations clearly demonstrated that a nozzle with a 20° convergent angle and 5 mm throat length offered the optimal balance between flow stability and performance. This configuration maintained consistent pressure distribution while achieving sufficient velocity output, making it particularly suitable for fire suppression operations. The 20° nozzle's superior performance - balancing velocity (0.6-0.7 m/s) and stability - confirms theoretical predictions about moderate angles minimising flow separation while maintaining efficiency.

Aligns well with established fluid dynamics principles regarding boundary layer behaviour in convergent. Especially noteworthy is the non-linear performance degradation observed at 25°, where despite achieving the highest velocities (0.8 m/s at 5 bar), the nozzle exhibited problematic flow instabilities. This suggests the existence of an optimal design window rather than a simple "more angle equals more performance" relationship. The methodology's rigour, particularly the grid independence verification requiring up to 4 million cells, provides a valuable case study in computational resource allocation for similar CFD projects.

However, several limitations merit consideration in interpreting these results. The study's focus on ideal conditions (steady-state, room-temperature water) necessarily simplifies the complex reality of firefighting scenarios where thermal effects, multi-phase flows, and dynamic operating conditions would influence performance. Additionally, the fixed throat length of 5 mm, while shown to be effective, invites further investigation into how this parameter interacts with the convergent angle across different scale applications.

5. CONCLUSION

This study examines the effect of the convergent angle of nozzles on fluid dynamics in firefighting applications through numerical simulations. Results show that nozzles with a convergent angle of 20° offer the best performance with stable pressure retention and optimal flow velocity, while a 25° angle causes flow instability, and a 15° angle is less efficient at high pressure. A throat length of 5 mm also proved effective in maintaining flow stability. These findings highlight the importance of nozzle geometry optimisation to improve the efficiency of firefighting operations. Further research could involve experimental validation and exploration of additional geometry parameters for nozzle design refinement. In conclusion, this numerical investigation highlights the significant impact of nozzle geometry on flow and pressure performance in fire-fighting applications. Among the tested designs, the 20° nozzle offers an optimal balance, making it a suitable choice for enhanced fire-fighting effectiveness. Future work may involve experimental validation and exploring additional geometric parameters to further refine nozzle design.

REFERENCES

[1] T. Jiang, Z. Huang, J. Li, Y. Zhou, and C. Xiong, "Effect of Nozzle Geometry on the Flow Dynamics and Resistance Inside and Outside the Cone-Straight Nozzle," *ACS Omega*, vol. 7, no. 11, 2022, doi: 10.1021/acsomega.1c07050.

[2] T. Jiang, Z. Huang, J. Li, and H. Li, "Effect of parameters on threshold pressure of sandstone tested by water jet," *International Journal of Rock Mechanics and Mining Sciences*, vol. 139, p. 104640, Mar. 2021, doi: 10.1016/j.ijrmms.2021.104640.

[3] L. Jingbin, H. Zhongwei, Z. Guangqing, L. Xin, and L. Huan, "Rock breaking characteristics of the self-rotating multi-orifices nozzle applied to coalbed methane radial jet drilling," *International Journal of Rock Mechanics and Mining Sciences*, vol. 136, p. 104483, Dec. 2020, doi: 10.1016/j.ijrmms.2020.104483.

[4] X. Zhou, H. Li, and Y. Jiang, "Study on water distribution uniformity of non-circular nozzles at low pressure," *Paiguan Jixie Gongcheng Xuebao/Journal of Drainage and Irrigation Machinery Engineering*, vol. 35, no. 5, 2017, doi: 10.3969/j.issn.1674-8530.16.0158.

[5] Z. He, G. Guo, X. Tao, W. Zhong, X. Leng, and Q. Wang, "Study of the effect of nozzle hole shape on internal flow and spray characteristics," *International Communications in Heat and Mass Transfer*, vol. 71, pp. 1–8, Feb. 2016, doi: 10.1016/j.icheatmasstransfer.2015.12.002.

[6] C. Li, J. Mao, Z. Kang, S. Zhao, and H. Ren, "Influence of Firefighting Intervention on Fire Spread Characteristics in Ship Engine Room," *J Mar Sci Eng*, vol. 11, no. 4, 2023, doi: 10.3390/jmse11040877.

[7] G. Grant, J. Brenton, and D. Drysdale, "Fire suppression by water sprays," 2000. doi: 10.1016/S0360-1285(99)00012-X.

[8] R. Bellas, M. A. Gómez, A. González-Gil, J. Porteiro, and J. L. Míguez, "Assessment of the Fire Dynamics Simulator for Modeling Fire Suppression in Engine Rooms of Ships with Low-Pressure Water Mist," *Fire Technol*, vol. 56, no. 3, pp. 1315–1352, May 2020, doi: 10.1007/s10694-019-00931-8.

[9] K. Farrell *et al.*, "Water Mist Fire Suppression Systems for Building and Industrial Applications: Issues and Challenges," *Fire*, vol. 6, no. 2, p. 40, Jan. 2023, doi: 10.3390/fire6020040.

[10] K. Prasad, G. Patnaik, and K. Kailasanath, "A numerical study of water-mist suppression of large scale compartment fires," *Fire Saf J*, vol. 37, no. 6, 2002, doi: 10.1016/S0379-7112(02)00004-8.

[11] S. A. Abdulrahman, K. Chetehouna, A. Cablé, Ø. Skreiberg, and M. Kadoche, "A review on fire suppression by fire sprinklers," 2021. doi: 10.1177/07349041211013698.

[12] R. Dehane *et al.*, "Impact of the convergent geometric profile on boundary layer separation in the supersonic over-expanded nozzle," *Open*

Physics, vol. 20, no. 1, pp. 1080–1095, Nov. 2022, doi: 10.1515/phys-2022-0185.

[13] Y. Yu, Y. Mao, T. Yu, and S. Xu, "Numerical Investigation on Mechanism of External Flow Field–Induced Separation Pattern Transition," *J Aerosp Eng*, vol. 36, no. 2, Mar. 2023, doi: 10.1061/JAEEEZ.ASENG-4555.

[14] S. Zhang, X. Tao, J. Lu, X. Wang, and Z. Zeng, "Design, optimization and CFD simulation of a nozzle for industrial cleaning processes based on high-pressure water jets," *Fluid Dynamics and Materials Processing*, vol. 11, no. 2, 2015, doi: 10.3970/fdmp.2015.011.143.

[15] J. Bahamon and M. Martinez, "Study of fluid-dynamic behavior in a convergent–divergent nozzle by shape optimization using evolutionary strategies algorithms," *Proc Inst Mech Eng G J Aerosp Eng*, vol. 237, no. 12, 2023, doi: 10.1177/09544100231163372.

[16] D. V. S. Siddhartha, K. Das Sadiq, and R. S. Sarath, "Comparative Study of Shock Formation in Bell and Conical Nozzle," in *Lecture Notes in Mechanical Engineering*, 2023. doi: 10.1007/978-981-19-6945-4_13.

[17] Wenjie; L.; & Xiaoyang; C, "Study on optimization simulation of high pressure water jet nozzle based on Fluent," *Energy Environ*, 2020.

[18] C. Xinying, "Hydraulic fracturing sand annulus jet nozzle structure analysis and optimization," *Oil Field Equip*, vol. 48, pp. 41–45, 2019.

[19] Tingbin C.; Wenling L.; Xiaoxiao L.; Zhengbiao J.; Yilong G., "Design and optimization of jet nozzle structure based on Fluent," *Chengdu Univ*, vol. 17, 2019.

[20] T. W. Jiang, Z. W. Huang, J. Bin Li, and Y. S. Zhou, "Internal flow mechanism of cone-straight nozzle," *Pet Sci*, vol. 18, no. 5, 2021, doi: 10.1016/j.petsci.2021.08.008.