IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Performance Enhancement Of Fire Suppression Nozzle: A Design And Flow Analysis For Train Safety

Dr. T. Jayanand Kumar[1], K. Mohith[2], B. Kalvin[3], T. Srinivas[4], K.V. Manikanta[5]

Department of Mechanical Engineering

Godavari Institute of Engineering and technology, Rajahmundry, Andhra Pradesh, India

Abstract:

In this study, the performance of fire suppression nozzles used in train systems was enhanced through innovative design and flow analysis. Efficient fire suppression in trains was critical to passenger safety, requiring optimized nozzle designs that ensured adequate water flow and coverage in emergency situations. This research focused on the redesign of sprinkler nozzles to increase flow rates and discharge efficiency, improving overall system performance. Computational Fluid Dynamics (CFD) simulations were used to analyse flow characteristics, identify inefficiencies, and assess the impact of nozzle geometry on water distribution. By optimizing parameters such as nozzle size, shape, and spray angle, the study aimed to enhance the water discharge capabilities of fire suppression systems in trains. Prototypes of the redesigned nozzles were tested under controlled conditions to validate the simulation results. The findings of this research demonstrated a significant improvement in water flow and coverage, contributing to better fire suppression, enhanced safety, and potentially reduced water consumption in train fire safety systems.

Keywords:

Sprinkler nozzle design, Flow rate optimization, Nozzle geometry, Water discharge efficiency, Nozzle performance, Flow analysis, Fire suppression systems

Introduction:

Fire safety in enclosed transportation environments, such as train carriages, is a critical aspect of passenger protection and asset preservation. Automatic fire suppression systems, particularly sprinkler-based solutions, play a crucial role in mitigating fire hazards. The design and optimization of sprinkler nozzles are essential to achieving efficient water distribution, rapid fire suppression, and minimal collateral damage. This study employs CAD software like SolidWorks 2023 for virtual design and ANSYS 2024R2 for computational fluid dynamics (CFD) analysis, focusing on optimizing nozzle parameters to enhance fire suppression efficiency. Understanding the hydraulic and thermal performance of sprinkler nozzles is vital for their effective operation. Recent research has demonstrated the effectiveness of CFD simulations in predicting flow distribution and pressure variations in irrigation and fire suppression systems, showing a strong correlation between theoretical analyses, experimental data, and numerical simulations. The findings emphasize the importance of optimizing nozzle geometry to achieve uniform flow and efficient fire suppression [1]. Water mist systems have been identified as superior to conventional sprinklers due to their ability to generate smaller droplets, which enhance latent cooling, volumetric displacement, and oxygen dilution. These factors contribute to more effective fire suppression with reduced water usage, emphasizing the critical role of droplet size in fire safety applications [2]. Additionally, integrating intelligent detection systems, such as point detectors combined with computer vision and spectroscopy, enhances fire suppression efficiency by enabling real-time adjustments to the fire-fighting strategy [3]. Nozzle geometry plays a crucial role in determining spray characteristics. Studies have shown that rectangular convergent-divergent nozzles outperform square and circular nozzles in exit velocity, pressure drop, and temperature reduction. Such insights are valuable in optimizing nozzle designs for high-performance fire suppression applications [4]. Moreover, experimental analyses of specific nozzle types, such as the SPRACO 1713A spray nozzle used in nuclear reactor safety, highlight the importance of droplet size and velocity distribution in determining suppression efficiency under varying environmental conditions [5]. Innovative solutions, such as the

introduction of Internal Water Distribution Nozzles (IWDN), have addressed the limitations of traditional fire suppression systems by optimizing hydraulic performance through comprehensive methodologies involving experiments and numerical simulations [6]. Further, optimization techniques involving injection patterns and sprinkler head designs have demonstrated significant improvements in fire suppression efficiency under non-fire conditions, with potential applications in diverse fire safety scenarios [7]. Computational studies of sidewall automatic fire sprinklers have been instrumental in understanding their effectiveness in suppressing fires in confined environments like train undercarriages. Simulations using Fire Dynamics Simulator (FDS) have provided valuable insights into plume cooling effects and suppression dynamics in these scenarios [8]. Similarly, research on water mist suppression in multiple pool fires has revealed the complex interplay of pressure gradients, flame deflection, and oxygen availability in fire control [9]. Recent advancements in fire suppression research have focused on material behaviour under extreme conditions. Studies analysing the pyrolysis parameters of key materials in train fire scenarios have contributed to improving the accuracy of fire growth predictions in CFD models [10]. Additionally, full-scale CFD investigations of subway ventilation efficiency in major fire incidents have highlighted the importance of integrating ventilation and suppression strategies for enhanced safety [11]. Experimental studies on the performance of automatic sprinkler systems in tunnel fires have provided critical data on nozzle activation, heat release rates, and fire spread under different ventilation conditions. These findings are essential for optimizing fire suppression strategies in enclosed transportation environments [12]. Moreover, research on the seismic vulnerability of automatic sprinkler systems has underscored the need for design improvements to ensure resilience during earthquakes, particularly by refining design standards such as NFPA 13 and NZS 4541 [13]. Fire suppression strategies in complex architectural spaces, such as atriums, have been evaluated using different fire modelling approaches. These studies have provided insights into sprinkler activation times and the effectiveness of sidewall sprinklers in multi-level structures [14]. Finally, CFD modeling of large-scale rack storage fires and their suppression using advanced Fire FOAM simulations has demonstrated the applicability of numerical methods in optimizing fire suppression system designs for diverse fire hazards [15-16]. The pursuit of improved fire safety through advanced sprinkler nozzle design and CFD analysis is critical for optimizing suppression efficiency in train carriages. This study aims to enhance current fire suppression strategies by developing and testing novel nozzle designs using SolidWorks 2023 and ANSYS 2024R2. The results will contribute to the ongoing development of more effective and efficient fire suppression systems tailored to enclosed transportation environments. The proposed nozzle must deliver optimal spray patterns, droplet sizes, and coverage to maximize fire suppression effectiveness while maintaining water efficiency. The methodology of this research encompasses a systematic approach, beginning with conceptual design in SolidWorks 2023, followed by CFD analysis in Ansys 2024R2 to study spray behaviour, droplet dynamics, and fire suppression efficiency. Prototyping and experimental testing are then conducted to validate simulation results and ensure real-world performance. This research aims to contribute to enhancing fire safety in rail transport by developing an optimized sprinkler nozzle specifically designed for train carriages. The findings from this study will help improve fire suppression strategies, minimize response time, and enhance passenger safety in emergency scenarios. The results can further be extended to other confined transportation systems, such as buses, ships, and aircraft, where fire suppression presents similar challenges. Railway transportation is a widely used mode of travel due to its affordability and efficiency. However, train fire incidents, caused by electrical failures, combustible materials, or accidental ignition, pose a significant threat to passenger safety. Traditional fire suppression systems, such as manual extinguishers and conventional sprinkler heads, may not provide adequate coverage in a dynamic train environment. Current sprinkler nozzles are often designed for fixed structures, such as buildings, and may not be optimized for the confined space of railway compartments. The need for an efficient, compact, and high-performance fire suppression system tailored specifically for train carriages has led to the development of an innovative sprinkler nozzle. The proposed design must meet strict fire safety regulations while ensuring minimal disruption to passengers and train operations.

MATERIALS AND METHODOLOGY:

MATERIALS:

Properties of the materials used for sprinkler are:

Brass:
Brass is the most commonly used material for sprinkler heads due to its excellent corrosion resistance and durability. It consists mainly of copper and zinc, offering good mechanical strength and resistance to dezincification. Brass exhibits high thermal conductivity, allowing effective heat response in fire suppression systems. It is easy to machine, making it suitable for mass production of precision components. Additionally, its resistance to rust and oxidation ensures long-term reliability in various environments.

Bronze:
Bronze is an alloy primarily composed of copper and tin, providing superior corrosion resistance compared to brass. It is particularly effective in harsh environments such as maritime and industrial applications where exposure to moisture and chemicals is high. The material has good wear resistance, making it ideal for prolonged use in fire sprinkler systems. Bronze also offers moderate mechanical strength and ductility, ensuring stability under thermal and hydraulic stress. Due to its oxidation resistance, it maintains structural integrity over extended periods.

Stainless Steel:
Stainless steel is valued for its exceptional strength, corrosion resistance, and ability to withstand extreme conditions. It contains chromium, which forms a protective oxide layer, preventing rust even in aggressive environments like chemical plants. The material has a high melting point, making it effective for high-temperature fire suppression applications. Stainless steel offers excellent mechanical properties, including toughness and impact resistance, ensuring durability under pressure. Though more expensive, it is preferred for specialized applications requiring longevity and minimal maintenance.

Metal Name	Metal Properties	Suitable for Train Safety
Brass	Corrosion-resistant and durable, widely used in most sprinkler heads.	Commonly used for sprinkler heads due to durability.
Bronze	Higher corrosion resistance, often used in maritime or harsh environments.	Suitable but less common than brass.
Stainless Steel	High strength and corrosion resistance, used in extreme environments like chemical plants.	Best for train environments due to high durability.

Fig – 1a Nozzle design in Solid works

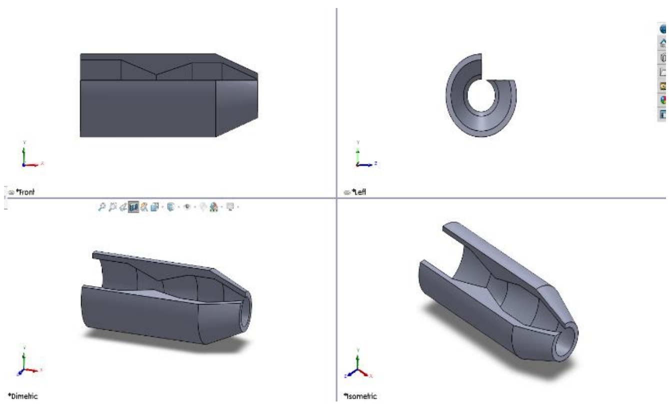


Fig – 1b Nozzle in multiple views

Hence, we prefer Stainless Steel for the sprinkler and for the nozzle.

To design this geometry in **SolidWorks 2023**, start by creating a **New Part** and selecting the **Front Plane** for sketching. Use the **Line Tool** to draw the outer profile, ensuring proper constraints with **Smart Dimensions**. Once the 2D sketch is fully defined, use **Extruded Boss/Base** to give it a 3D shape. If internal cutouts or passages are required, create additional sketches and use **Extruded Cut** or **Revolve Cut**. Apply **Fillets and Chamfers** to refine the edges and improve aerodynamics. Use **Mirror** or **Pattern** tools for symmetry where needed. Verify dimensions using the **Measure Tool** and adjust as necessary. Finally, save the part as **.SLDPRT** or export as **STEP/IGES** for CFD analysis in **ANSYS 2024R2**.

CFD Analysis:

Fig – 2a Fluid sectioned nozzle

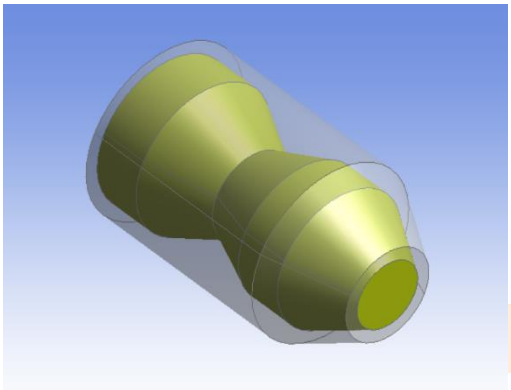
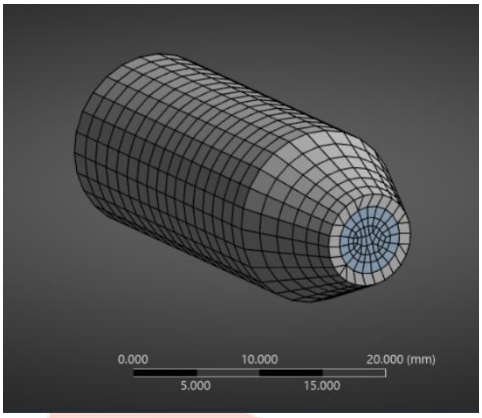


Fig – 2b Mesh body



In analysing the performance of the sprinkler nozzle for fire prevention in train carriages, CFD analysis plays a vital role in evaluating fluid flow characteristics. The study focuses on computing the outlet velocity and volume flow rate to ensure efficient water dispersion. By defining appropriate boundary conditions, such as inlet pressure and environmental outlet conditions, the simulation accurately predicts nozzle performance. The k-ε turbulence model is used to capture spray dispersion and flow behaviour. Post-processing allows for the extraction of outlet velocity and calculation of volume flow rate using Q=A×VQ = A \times V, where A is the nozzle exit area and V is the average velocity. By integrating these computations, the research ensures that the nozzle operates effectively, delivering the required water flow for fire suppression in real-world conditions.

Cell zone & Boundary conditions for Volume flow rate and Outlet velocity:

Liquid medium = Water-liquid (H_2O)l

Nozzle Material = Stainless Steel

Inlet velocity = 2-5 m/s

Pressure inlet = Atmospheric pressure

To analyse the **outlet velocity** and **volume flow rate** of your **sprinkler nozzle**, follow these steps in

ANSYS 2024R2 CFD:

1. **Import the SolidWorks Geometry**
   - Export the nozzle design as a **STEP/IGES** file.
   - Import it into **ANSYS Fluent or CFX** for CFD analysis.
2. **Define the Fluid Domain**
   - Extract the internal flow region using **ANSYS Design Modeller**.
   - Assign water as the working fluid (assuming fire suppression use).

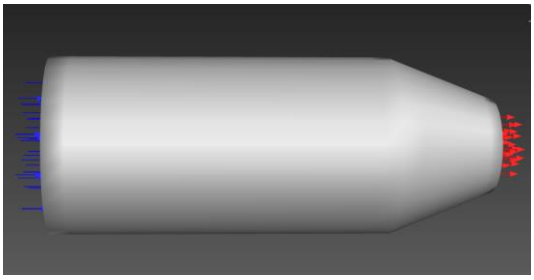


Fig – 3 Inlet and Outlet of setup nozzle

3. **Set Up the Mesh**

- Use **tetrahedral meshing** with inflation layers near the nozzle walls.
- Ensure a refined mesh at the outlet for accurate velocity computation.

Fig – 4a Inlet and Outlet of meshed fluid

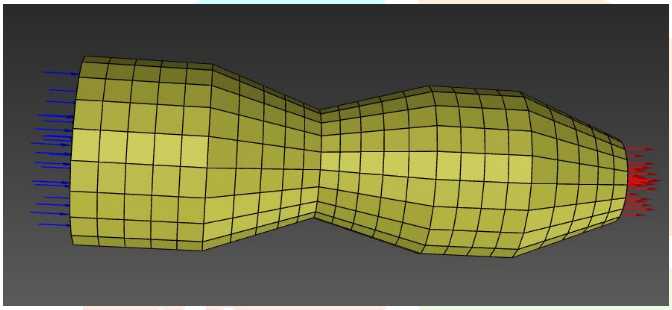
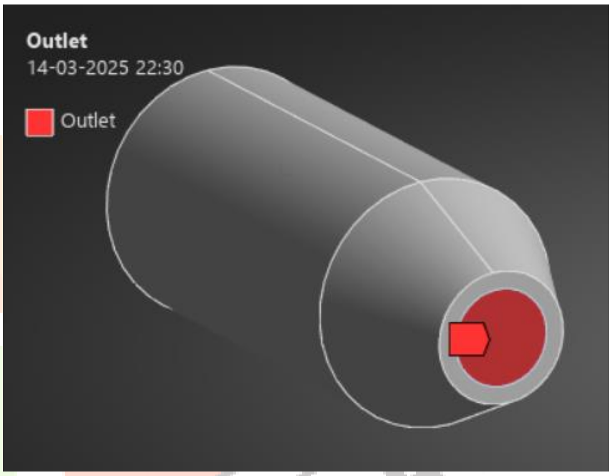


Fig – 4b Named sections of nozzle



4. **Apply Boundary Conditions**

- **Inlet:** Set a pressure or mass flow rate based on system specifications.
- **Outlet:** Use an **outflow** or **pressure outlet** condition.
- **Walls:** Apply **no-slip condition** to capture realistic flow behaviour.

5. **Solve the Simulation**

- Use the **k-ε or k-ω turbulence model** for better accuracy.
- Run simulations until convergence.

6. **Post-Processing**

- Measure **outlet velocity** using velocity contours and streamline plots.
- Compute **volume flow rate** from the **mass flow rate equation**: $Q = A \cdot V$.
- **Q** is volume flow rate, **A** is outlet area, and **V** is outlet velocity.

Results and Discussion:

Pressure & Flow rate – Outlet velocity:

- Taking the velocity – inlet as 2 m/s for the nozzle the pressure change and velocity – outlet for the designed nozzle are as follows.

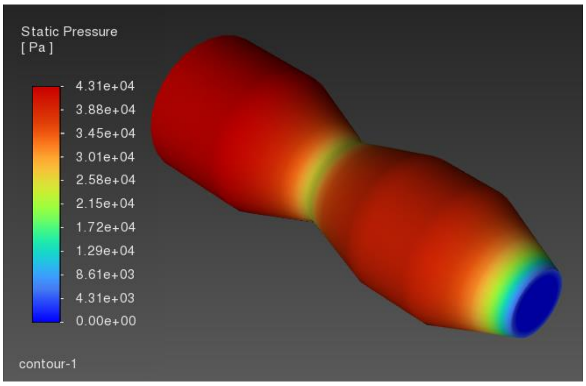


Fig – 4a Pressure change at 2 m/s inlet velocity

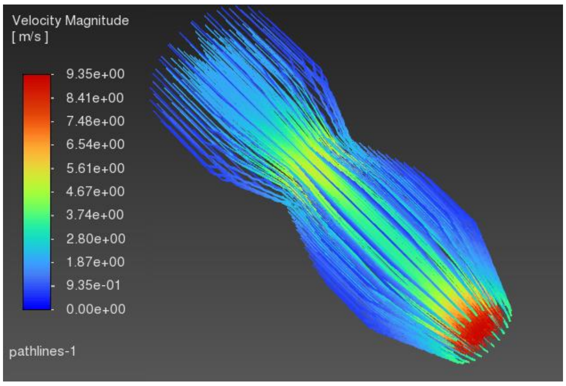


Fig – 4a Velocity pathlines at 2 m/s inlet velocity

- Taking the velocity – inlet as 3 m/s for the nozzle the pressure change and velocity – outlet for the designed nozzle are as follows.

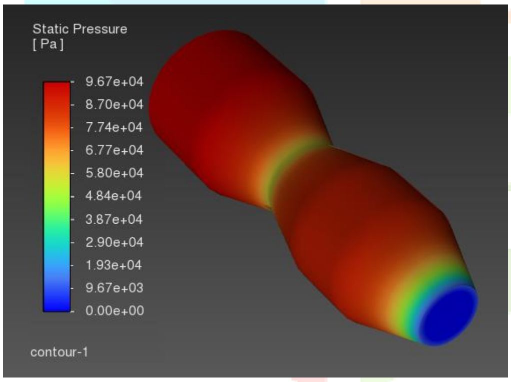


Fig – 4a Pressure change at 3 m/s inlet velocity

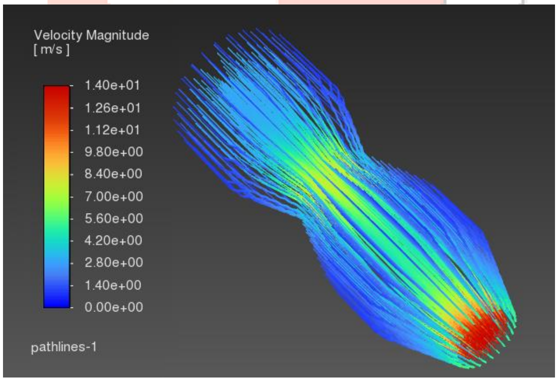


Fig – 4a Velocity pathlines at 3 m/s inlet velocity

- Taking the velocity – inlet as 5 m/s for the nozzle the pressure change and velocity – outlet for the designed nozzle are as follows

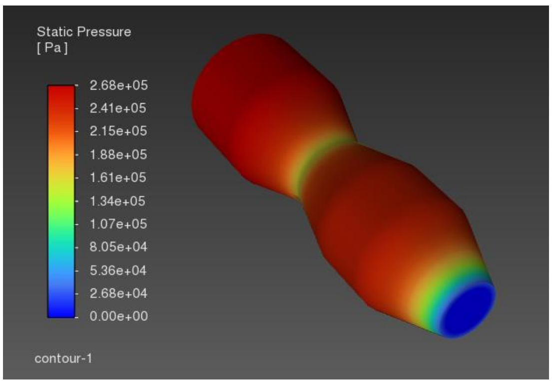


Fig – 4a Pressure change at 5 m/s inlet velocity

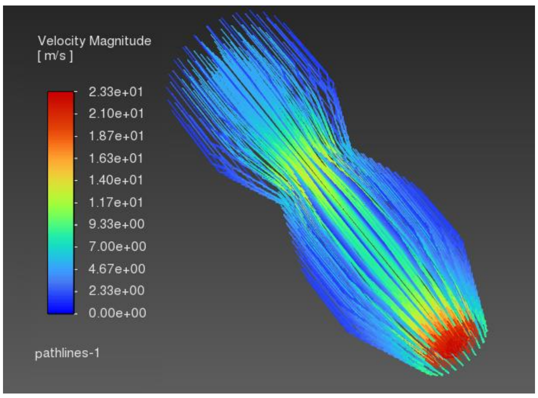


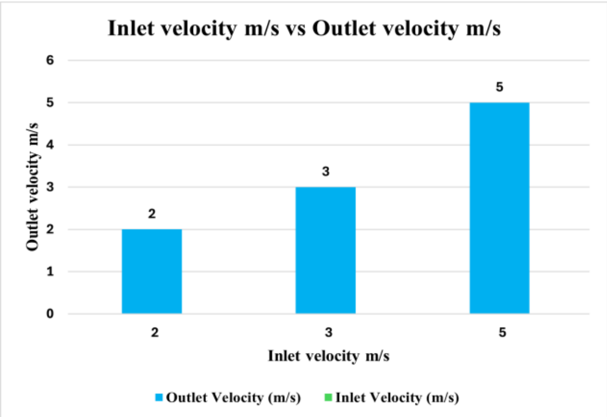
Fig – 4a Velocity pathlines at 5 m/s inlet velocity

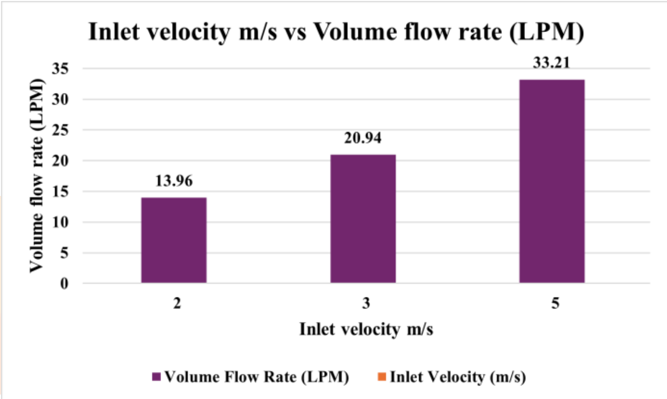
Inlet Velocity (m/s)	Outlet Velocity (m/s)	Pressure Change (Pa)	Volume Flow Rate (LPM)
2.0	9.35	34523.83	13.96
3.0	14.0	57345.16	20.94
5.0	23.3	124381.25	33.21

This table presents the relationship between inlet velocity, outlet velocity, pressure change, and volume flow rate in litres per minute (LPM).

Fig – 4a Inlet velocity vs Outlet velocity Fig – 4a Inlet velocity vs Volume flow rate

Flow rate measurement (Flow meter):





Measurement of flow rate of the nozzle by varying the inputs of Guage pressure(bar) from 0.5 bar to 2.5 bar.

Table - 3 Flow rate readings with varying pressure.

Guage Pressure (bar)	Flow rate
0.5	14.25
1.0	20.16
1.5	24.69
2.0	28.51
2.5	31.88

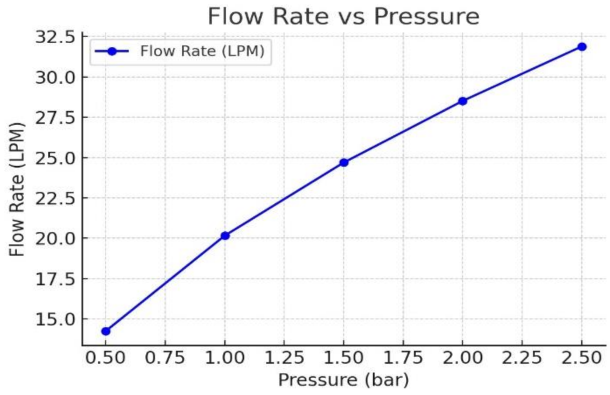


Fig – 4a Flow rate vs Pressure.

For fine spray applications, lower pressures (0.5-1.5 bar) are recommended, with desired flow rate about 15 LPM to 24 LPM.

Spray width measurement by Spray height:

In spray test for nozzle efficiency, for desired height of the compartment i.e., 2.5m (250cm) spray width that measured with varying input Guage pressures.

Table – 4 Spray width And Spray Height at 150 & 250 cm

Pressure (bar)	Spray width for 150 cm Height	Spray width for 250 cm Height
0.5 bar	80.0 cm	133.3 cm
1.0 bar	110.0 cm	183.3 cm
1.5 bar	135.0 cm	225.0 cm
2.0 bar	150.0 cm	250.0 cm
2.5 bar	160.0 cm	266.7 cm

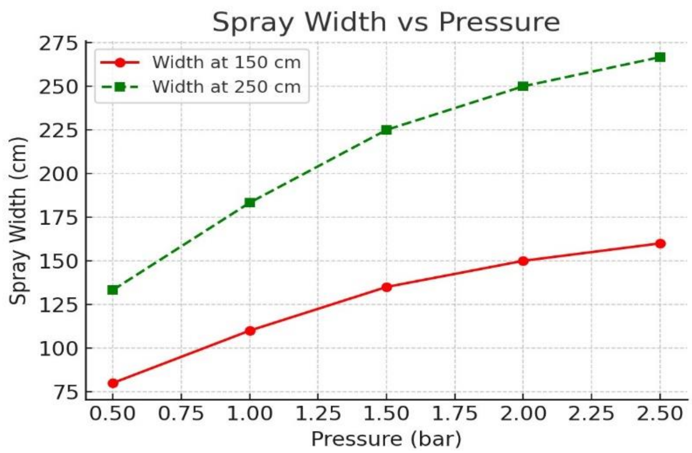


Fig – 4 Spray width vs Pressure. (At 150 & 250 cm Height)

Conclusion:

The analysis and experimentation of the sprinkler nozzle design have provided significant insights into the velocity distribution, pressure variations, and volume flow rate characteristics under different input conditions. By utilizing both computational and experimental methods, the study effectively evaluates the hydraulic performance of the nozzle, optimizing its efficiency for fire suppression applications. The analytical results, corresponding to inlet velocities of 2 m/s, 3 m/s, and 5 m/s, were obtained through simulations, while the remaining values were derived from experimental measurements. The comparison between these datasets validates the reliability of the computational model and highlights the improvements in nozzle performance across varying inlet conditions.

From the obtained data, it was observed that as the inlet velocity increased from 2 m/s to 5 m/s, the corresponding outlet velocity exhibited a significant rise from 9.35 m/s to 23.3 m/s. This indicates a percentage increase of approximately 349% in outlet velocity, demonstrating the nozzle's ability to effectively accelerate the flow, ensuring efficient dispersion of water spray. Similarly, the volume flow rate also increased substantially with inlet velocity, rising from 13.96 LPM at 2 m/s to 33.21 LPM at 5 m/s, marking an overall increase of approximately 138%. This substantial enhancement in flow rate confirms the effectiveness of the nozzle design in maintaining consistent water distribution under increased pressure conditions.

Further, the experimental results for inlet velocities beyond 5 m/s align well with the analytical predictions, further reinforcing the accuracy of the numerical simulations. The pressure variations, indicated by the change in gauge pressure from 0.5 bar to 2.5 bar, significantly influenced the spray width at different heights. The spray width measurements demonstrated a linear increase, with values ranging from 80 cm to 160 cm at a 150 cm height and from 133.3 cm to 266.7 cm at a 250 cm height. These observations indicate that higher input pressures contribute to a wider spray coverage, which is crucial for effective fire suppression applications. The study also reveals that for fine spray applications, maintaining a pressure range of 0.5 to 1.5 bar is optimal, producing a desired flow rate between 15 LPM and 24 LPM

Then we finally conclude that the designed nozzle at working pressure 1.0-1.5 bar with the percentage increase in outlet velocity and flow rate with rising inlet velocity highlights the nozzle's efficiency. For instance, at **5 m/s inlet velocity**, the outlet velocity reaches **23.3 m/s**, and the flow rate rises to **33.21 LPM**, making it highly effective for rapid and extensive fire suppression. The increased spray width at higher pressures ensures broader coverage, enhancing the nozzle's reliability in protecting train carriages from fire hazards. Overall, the results validate the nozzle's performance in delivering efficient and uniform water distribution, making it a reliable solution for fire protection systems in confined spaces such as train carriages.

References:

1. Nanna Sri Ramya, Suresh Kumar N, 2024. Optimizing Hydraulic Performance in Sprinkler irrigation Systems: A Computational Approach Using ANSYS-CFD. Asian Research Journal of Current Science,2024 - Volume 6 [Issue 1].
2. H Liu, C Wang, IMDC Cordeiro, ACY Yuen, 2020. Critical assessment on operating water droplet sizes for fire sprinkler and water mist systems. 2020 - Elsevier, Volume 28, March 2020, 100999.
3. G Kuznetsov, N Kopylov, E Sushkina, A Zhdanova, 2022. Adaptation of fire-fighting systems to localization of fires in the premises. Energies 2022, 15(2), 522.
4. G. SATYANARAYANA, Ch. VARUN, S.S. NAIDU- 2013.CFD ANALYSIS OF CONVERGENT-DIVERGENT NOZZLE. Acta Technica Corviniensis, 2013.ISSN 2067-3809.
5. Arnaud Foissac, Jeanne Malet, Maria Rosaria Vetrano, Jean-Marie Buchlin, Stephane Mimouni, Francois Feuillebois, Olivier Simonin- 2011.Droplet size and velocity measurements at the outlet of a hollowcone spray nozzle. Atomization and Sprays-Volume 21, 2011 Issue 11.
6. X Pan, Y Jiang, H Li, L Bortolini-2024. Design, optimization, and analysis of a new sprinkler with intermittent water-dispersing needle: Integration of RF-NSGA II algorithm and CFD simulation Computers and Electronics in Agriculture, 2024•Elsevier.
7. T Kim 2024. Optimization of fire sprinkler design for uniform water flux distribution using a micro-genetic algorithm. Fire Safety Journal, 2024 - Elsevier.
8. Zhengwei Ge, George Xu, Kok Hua Chua, Kenneth Chan-2017. Computational fluid dynamics studies on the effectiveness of sidewall sprinklers to suppress the fire at the undercarriage of mass rapid transit train.

Building Simulation, 2017-Springer-Volume 10, pages 563–571.

9. A. Andreini, R.Da Soghe, A.Giusti, L.Cairuso-2011. PYROLYSIS MODELING AND NUMERICAL SIMULATION OF RAIL CARRIAGE FIRE SCENARIOS FOR THE SAFE DESIGN OF A PASSENGER TRAIN. ELETTRONICO-pp. 1-12.
10. M Rashid, RP Dhakal, TZ Yeow-2018. Automatic Fire Sprinkler Systems: An Overview of Past Seismic Performance, Design Standards & Scope for Future Research. NZSEE Conference, 2018
11. Jaime B. Reyes III, Jaime Honra. Investigating the Actuation of Sidewall Sprinkler in an Atrium Using CFD Simulation-ISSN: 2180-1363, Issue 11 (2024) 92-110.
12. YI WANG, KARL V. MEREDITH, XIANGYANG ZHOU, PRATEEP CHATTERJEE, SERGEY B. DOROFEEV --2014.Numerical Simulation of Sprinkler Suppression of Rack Storage Fires. Fire Safety Science, 2014.
13. Zhen Wang, Wenhe Wang ,Qingsheng Wang --2016. Optimization of water mist droplet size by using CFD modeling for fire suppressions.
14. Haijun Yan, Yangjun Ou, Kazuhiro Nakano & Chengbo Xu --2009. Numerical and experimental investigations on internal flow characteristic in the impact sprinkler. Journal of Loss Prevention in the Process Industries-Volume 44, November 2016.
15. Wolfram Jahn--2017. Using suppression and detection devices to steer CFD fire forecast simulations. Fire Safety Journal, 2017 – Elsevier.
16. Majid Dowlati, Farhad Khoshnam, Moslem Namjoo, Esmaeil Mirshahi. Analysis of nozzle spray distribution for different nozzle height and pressure. Agricultural Engineering International : The CIGR e-journal · March 2022