Computational Fluid Dynamics (CFD) Analysis of the Hyper-mist Formation Using Single-Orifice Nozzles Used to Extinguish the Fires in Containers or Closed Chambers

S. B. Mane Deshmukh, and J. S. Pal

Department of Mechanical Engineering

(Tolani Maritime Institute, Induri)

(Talegaon-Chakan Road, 410507, India)

{Corresponding author's email: sagarm@tmi.tolani.edu)

Abstract - The current research investigates into Computational Fluid Dynamics (CFD) analysis of hyper-mist formation using single-orifice nozzle. Several parameters play important roles in the proper functioning of the fire extinguishing systems. The mass flow rates determine the volume of water delivered per unit time, which directly impacts extinguishing efficiency of the system. The pressures at the base of nozzle influence the velocity and trajectory of the discharged water. Velocity at the exit of the nozzle dictates the momentum and dispersion of the hyper-mist. These parameters collectively govern the system's capability to combat fires efficiently. Looking at the importance of the study, the numerical simulations were performed for different mass flow rates (i.e. 0.025, 0.050, 0.075, 0.100, 0.125 kg/s) of water in the single orifice nozzle and different output parameters (which directly impacts on performance of the fire extinguishing system) were closely monitored spray travel distance, spray pattern formation, spray angle, turbulent kinetic energy and residence time of the water droplets in container. This study is mainly applicable to Class A fires on board ship.

Keywords: fire extinguishing system, single -orifice nozzles, spray patterns, spray distance, residence time, class A type of fire.

INTRODUCTION

The fire extinguishing on ships is one of the critical operation which aims to safeguard the lives and to protect on board systems. Following the guidelines provided by International Maritime Organization (IMO) is very important in designing the effective fire extinguishing systems. In order to improve the understanding of fire suppression mechanisms, various research endeavours have been pursued. A critical feature of fire suppression is the immediate and sufficient supply of water to the site of the fire to confirm prompt extinguishment. Depending on the nature of the heat generated, specific scheme or type of water supply should be provided to the fire sites/locations. Example: a) If the hot spot is just generated and flame is not started yet, jet of water should be used to avoid ignition, (b) If the small flame has started, water mist should be supplied, (c) If flames in bigger scale have generated, hyper-mist is the best option. This study is mainly focused on the systems capable of generating hyper-mist from specialized nozzle (single orifice). The numerical study conducted will help in improving the maritime safety and operational efficiency. Optimization of the design parameters and operation parameters of fire extinguishing systems the will reduce the unnecessary expenditure, which leads to resource conservation and cost savings. Moreover, enhanced understanding of hyper-mist formation facilitates better utilization of on-board water resources, crucial for fire fighting and emergency response scenarios.

Water mist refers to the formation of the water droplets with a diameters less than 1000 microns [1]. Following are the main advantages of using water mist for fire suppression (Class A type) as compared other systems in which gaseous agents or conventional sprinklers are used:(a) Water mist systems are non-toxic, making them safe for occupants and first responders during and after a fire incident, (b) Unlike some gaseous agents, water mist does not displace oxygen or pose risks of asphyxiation in enclosed spaces, (c) Water mist systems are environmentally friendly, as they do not release harmful chemicals or substances into the atmosphere, (c) The cost of installing and maintaining water mist systems is generally lower compared to other fire suppression systems, providing economical benefits over the system's lifetime, (d) Water mist systems use significantly less water compared to conventional sprinkler systems, reducing water damage and allowing for quicker clean-up post-fire, (e) Water mist systems are highly effective in extinguishing Class A fires by rapidly cooling the fire, suppressing flames, and preventing re-ignition.

The formed water mist does not behave like the gaseous agent used in the other systems. All the water mist formed is not used in the fire suppression process. It is distributed in following different partitions: [2]: Blown away droplets before reaching the fire (a) Water Droplets which penetrate into the fire plume. Sometimes, water droplets reach out to burning surfaces under the fire plume and it results into pyrolysis by cooling, and the resultant steam dilutes oxygen in nearby surrounding, (b) Water droplets which impacts on surfaces like vertical walls, ground/floor and ceiling of the compartments, which cools them or they will be wasted, (c) Water droplets which are vaporized into steam during its travel in the compartment and contributing to cool the fire plume/hot gases/compartment/other surfaces, (c) Water droplets which contributes in pre-wetting the adjacent combustibles to prevent fire spread.

The supply of the water mist in the nearby zones of fire extinguishes fires because of the displacement of oxygen and extraction of the heat from these zones [3, 4]. Different ways proposed to supress the fire using the water mist are as follows: a) reducing the evaporation of the fuel along with cooling and dilution of the fire zone [5, 6]. The attenuation of the radiant heat, kinetic effect of mist on fire, combustible mixture dilution by entrained air [7, 8].

Looking at the significance of the water mist system and its usage on board ship, it is necessary to thoroughly understand how this system behaves for different pressures of water flow, spray pattern formation, spray angles, distance travelled by the spray etc.

Water Mist Characteristics:

The efficiency and effectiveness of water spray formation system depends on the type of spray formed by a particular nozzle. The important parameters (water spray related) are as follows [9]

(a) Rate of flow per unit area (fire region)

(b) Distribution of water spray in and about the fire zones/areas

(c) Direction of application

(d) Size of the droplet and its distribution

(e) Entrained velocity of air and

(f) Droplet velocity relative to entrained air, flame velocity, and fuel types.

Water mist formation in confined spaces has been extensively studied using Computational Fluid Dynamics (CFD) simulations. Researchers have employed various turbulence models, such as the Reynolds-Averaged Navier-Stokes (RANS) or Large Eddy Simulation (LES), to capture the complex flow phenomena involved in mist generation [10-11]. Pressure conditions play a crucial role in determining the behavior of water mist. Studies have investigated mist formation under different pressure environments, including atmospheric, elevated, and reduced pressures. These investigations have provided insights into how pressure affects droplet size distribution, mist concentration, and dispersion patterns [12-13]. The water spray angle and droplet sizes significantly influence mist formation characteristics. CFD simulations have been used to analyze the impact of varying spray angles and droplet sizes on mist distribution, coverage area, and effectiveness in suppressing fires [14-15].

METHODOLOGY IMPLEMENTED

In the current work, the space (where fire can occur) was considered as a 2D diagram. The quantity of the water plays important role in forming the water mist and hence extinguishing the fire in the required space. CFD simulations were performed using the ANSYS FLUENT- 16.

Geometry:

The simulations were performed in the 2D geometry of dimensions 100 mm X 60 mm to understand the different phenomenon like formation of the water spray, its spreading in the 2D space and distance travelled by the water droplets in the 2D space, velocity of the water droplets etc. the details of the geometry are mentioned in the Fig. 1 and Table 1.

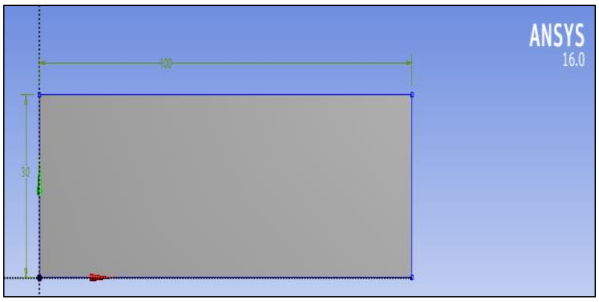


Fig 1. Geometry of confined/closed room (2D)

Table 1: Geometry dimensions

Sr. No.	Parts of the Geometry	Dimensions, mm
01	Inlet	60
02	Outlet	60
03	Length	100
04	Injector inner diameter (orifice)	0.1
05	Orifice Length	1

Mesh File:

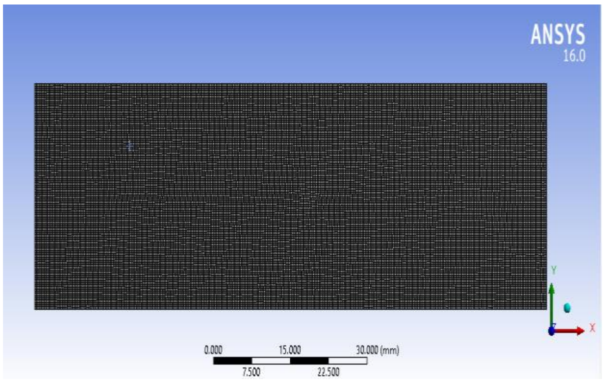


Fig. 2 Mesh Details

They align well with rectangular or straight-edged boundaries, simplifying mesh generation and ensuring accurate boundary condition application without excessive refinement. Quadrilateral meshes also enhance computational efficiency, benefiting large-scale simulations by optimizing solver performance and reducing solution times. **Mesh details used:** Type of mesh: quadrilateral, Nodes: 33724, Elements: 33291.

Solver settings:

Pressure based (to avoid compressibility effect), Velocity formulation absolute (only water- liquid particles are moving and walls are stationary). Time – Transient (properties are changing with respect to time), 2D space – Axisymmetric (generally a case for the discrete phase modelling with plain-atomizer orifice).

Models:

Viscous model (turbulent): k - epsilon (2 equation), standard, near wall treatment - standard wall function.

Discrete Phase Model:

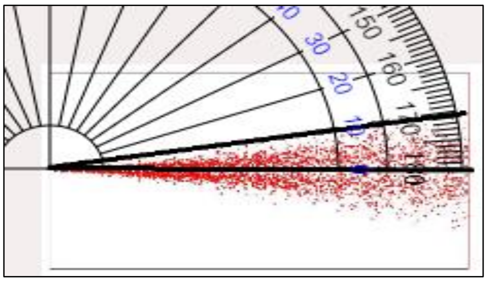
The discrete phase model are generally used for the simulating the dispersed particles (here it is water droplets in the spray) in continuous phase. It is necessary to find out the trajectory followed by the water droplets formed in the water spray. It also helps in understanding the interaction between the different water droplets. **Following important setting were done in CFD:** Without interaction with the continuous phase, Particle treatment: unsteady particle tracking, Tracking: tracking parameters: maximum number of steps=10000 (It specifies the maximum number of time steps over which FLUENT will track the trajectory of a particle in a simulation, If particles move slowly or the simulation runs for a long time, a higher maximum number of steps might be necessary to accurately capture particle behavior) ,step length factor=5 (It controls the step size used by the numerical integration scheme to calculate particle trajectories through the fluid domain. A factor of 5 typically indicates that the time step used in the particle trajectory calculations is 1/5th of the local characteristic time scale of the flow), Physical models=breakup (specifically refers to the activation of models designed to predict and simulate the breakup of droplets or particles based on selected breakup mechanisms and associated parameters). **Injections:** It is essential setting in FLUENT for simulating the movement of water droplets in fire extinguishing systems by defining the way in which particles are introduced in the domain and tracking their physical behavior. This helps to support the design and optimization of fire suppression strategies to enhance safety and efficiency in real-world applications. Following important setting were done in CFD: Injection name-water-injection, Injection type-plain-orifice atomizer (This is particularly useful for studying dispersed phase dynamics, such as spray formation and particle trajectories) , Number of particle streams=2 (this helps users to simulate scenarios involving multiple distinct groups or sources of particles, facilitating more accurate and detailed simulations of particle dispersion, interaction, and complex flow phenomena), Particle type – inert (water droplets were not doing any reactions with the available media).

Point properties- Choosing the different mass flow rate of water droplets in fire extinguishing systems is critical to achieve effective fire suppression, to ensure the safety of occupants and systems available on board. It involves balancing factors such as spray coverage, droplet size distribution, system design considerations, and compliance with relevant standards and regulations. Following important setting were done in CFD: Start time - 0 sec, Stop sec – 100 sec, Vapour pressure – 1000 (Pa), Injector inner diameter = 0.1 mm, Orifice Length – 1mm, Corner radius of the curvature=0.01mm, Constant A =4.9. **Physical models:** Drag law - dynamic drag (This is crucial for accurately modeling the interaction between particles (or droplets) and the surrounding fluid. Dynamic drag considers the particle's size, shape, and Reynolds number to compute drag forces dynamically, accounting for varying flow conditions). Enabled break up (Breakup is important in applications such as spray painting, combustion, and industrial processes where the size distribution of droplets significantly affects process efficiency and outcome). Break up model – KHRT (The KHRT model considers instabilities in the interface between the particle and the surrounding fluid, which can lead to breakup. It's particularly suitable for scenarios where turbulence or shear forces play a significant role in breakup dynamics). Turbulent dispersion-discrete random walk model (The discrete random walk model simulates the random movement of particles influenced by turbulent eddies, providing a more realistic representation of particle dispersion compared to simpler models. This is essential for accurately predicting the spatial distribution of particles in turbulent flows). **Boundary Conditions:** In FLUENT, the specified boundary conditions such as 0 gauge pressure at both inlet and outlet, along with turbulent intensity settings and hydraulic diameters, ensure accurate simulation of fire extinguishing scenarios by modeling realistic flow dynamics and particle dispersion. The "escape" type boundary conditions for discrete phase particles allow for realistic particle movement through the domain. Wall conditions enforce no slip and stationary behavior, crucial for accurate representation of particle-wall interactions. Time step settings and solution methods like coupled pressure-velocity coupling and second-order spatial discretization ensure numerical stability and efficient computation, facilitating precise predictions of water droplet or mist behavior in fire suppression applications, thereby optimizing system design and operational effectiveness. Following important setting were done in CFD: **Inlet:** Pressure = 0 gauge (pa), Turbulent Intensity specification method: Turbulent Intensity = 5%, Hydraulic diameter=60mm, Discrete phase boundary condition type – escape. **Outlet:** Pressure = 0 gauge (pa), Turbulent Intensity specification method: Backflow turbulent Intensity = 5%, Backflow hydraulic diameter = 60mm, Discrete phase boundary condition type – escape, **Wall:** No slip and stationary, Discrete phase boundary condition type – reflect, Time step =3e^{-6} sec, Maximum iterations per time step=10, Total time of the simulation = 3 e^{-3} sec. **Solution Methods:** Pressure velocity coupling: Coupled, Spatial discretization: Gradient: green gauss cell based, Pressure: standard, Momentum: second order, Turbulent kinetic energy: second order upwind, Turbulent dissipation rate: second order upwind Transient formulation: first order Implicit.

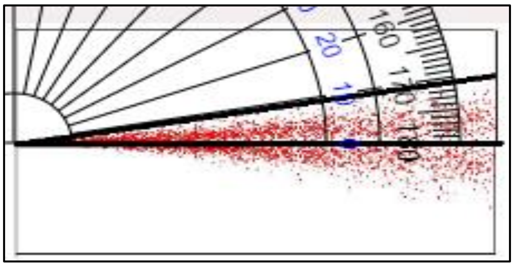
RESULTS AND DISCUSSION:

Looking at the importance of the water mist formation and its usage to extinguish the fire on ship or in marine filed, simulations were performed in the 2D space of size 100 mm X 60 mm. The main target was to understand the variation in the different parameters like static pressure, velocity magnitude, turbulent kinetic energy, density and particle residence time at different location in the computational domain for the different flow rates of the water through the plain-orifice-atomizer. This data further can be used to develop real time system. The changes in angle of spray, size of the water droplet formed and distance travelled by the water spray was observed for different mass flow rates of the water. Mainly water spray pattern formation for the different values of the mass flow rate were monitored. Table 2 presents the data related to water (droplet) particle residence time varying with respect to mass flow rate.

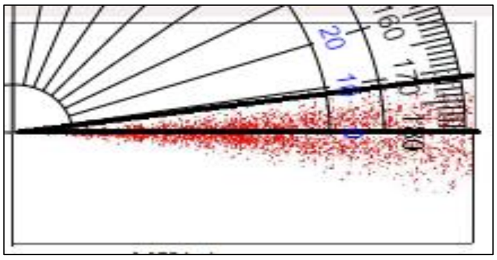
The results showed that (a) The water droplets residence time near to the orifice was less than the location away from the orifice. The shorter residence time close to the orifice gives clear indication about the quick dispersion and carried away from the nozzle due to the initial momentum imparted by the injection. This is important for effectively covering a wider (major portion) area with the extinguishing agent (eg. Water), which helps to ensure that the fire is suppressed evenly across the affected zone, (b) The location of water droplets released from the orifice were found to be shifting away from the orifice exit/tip for increased mass flow rates of water. This indicates enhanced momentum and reduced rebounding effects. This phenomenon suggests that higher flow rates propel droplets further from the nozzle, potentially improving coverage and effectiveness in fire suppression. Minimizing rebounding helps to optimize water resource utilization by ensuring more droplets reach the targeted area, enhancing efficiency and containment of fires or other hazards. (c) The spray angle for the stream of water droplets exiting from the orifice was found to be decreasing with increasing mass flow rates of the water [Refer Fig. 3]. The range of the water spray angle was varied between 14 deg. (at mass flow rate = 0.125 kg/s) to 16 deg. (at mass flow rate = 0.025 kg/s). This signifies more concentrated and targeted



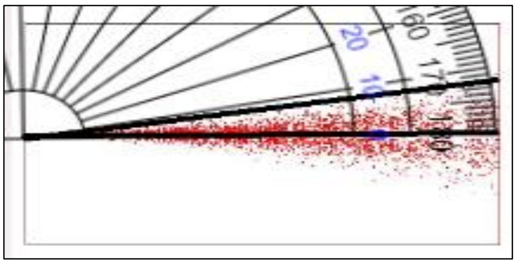
(a) Mass flow rate = 0.025 kg/s, angle of spray = 20 deg



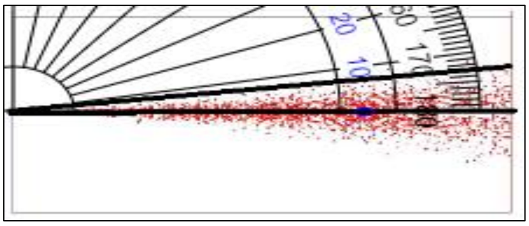
(b) Mass flow rate = 0.050 kg/s, angle of spray = 19.6 deg



(c) Mass flow rate = 0.075 kg/s, angle of spray = 17.8 deg



(d) Mass flow rate = 0.100 kg/s, angle of spray = 16 deg



(e) Mass flow rate = 0.125 kg/s, angle of spray = 14 deg

Fig. 3 Variation of the angle of spray with respect to mass flow rate dispersion of water droplets, which will help in improving the efficiency and effectiveness of fire suppression (by enhancing coverage and reducing water wastage), (d) The water droplets (mist) reached the last end of the room for all the mass flow rates of water, (e) the minimum residence time equal to $3e^{-6}$ sec and maximum residence time equal to $3.59e^{-6}$ sec were found for all the mass flow rates of water, (f) The static pressures at the location in the water droplets were observed to be negative and the range was observed between $-3.52\ e^{2}$ to 0 Pa for all the mass flow rates of water. The observation of negative static pressures ranging from $-3.52e^{2}$ to 0 Pa at the location of water droplets indicates a suction effect, influencing their trajectory and dispersion characteristics in the fire extinguishing system, (g) The turbulent kinetic energy was observed to be varying between 2.09 m^2/s^2 (was near to the orifice) to 4.63 m^2/s^2 (was away from the orifice). Higher Turbulent Kinetic Energy values away from the orifice indicate increased turbulence and fluid agitation. This turbulence plays a crucial role in dispersing water droplets more effectively throughout the fire-affected area, enhancing coverage and ensuring that the extinguishing agent reaches all necessary points efficiently.

CONCLUSION:

The study was mainly focused on understanding the behaviour of the water spray formation through single orifice nozzle in the 2D space. The different parameters like formation of the spray patterns, angle of spray, water droplet formation, distance travelled by the water spray, turbulent kinetic energy, static pressure, water droplet residence time was monitored for different mass flow rates of the water spray. Following were the key findings from the analysis: (a) Size of water droplet depends on the orifice size and mass flow rate, (b) The shorter residence time of water droplets near the orifice indicates rapid dispersion, facilitated by initial momentum, ensuring broader coverage of the fire-affected area. (c) This quick dispersion minimizes fire spread and ensures even suppression.

Table 2: variation of the particle residence time with mass flow rate

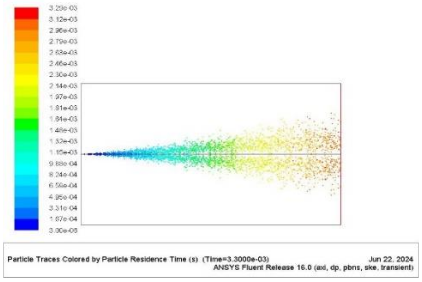
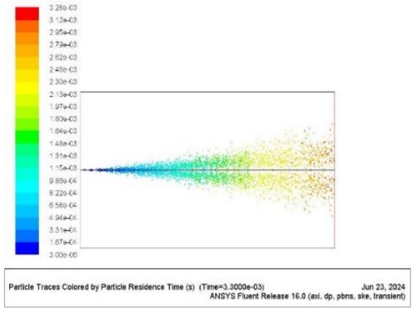
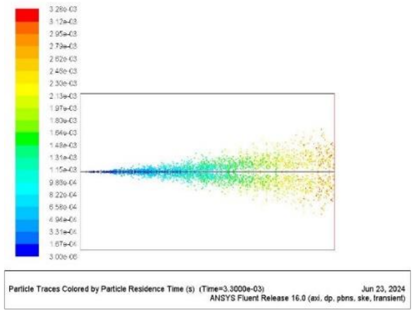
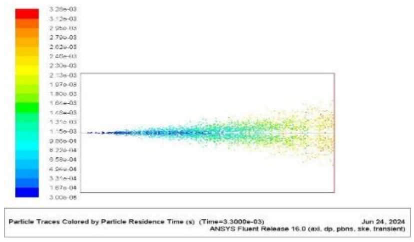
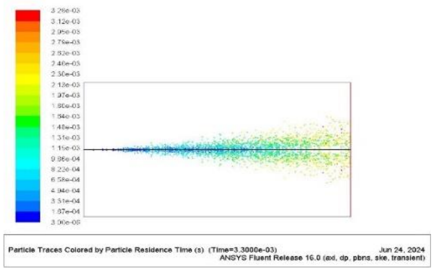
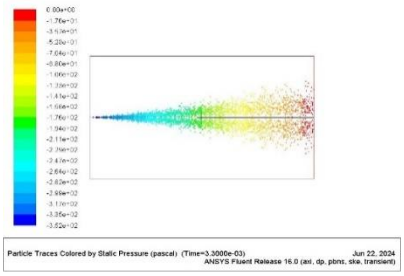
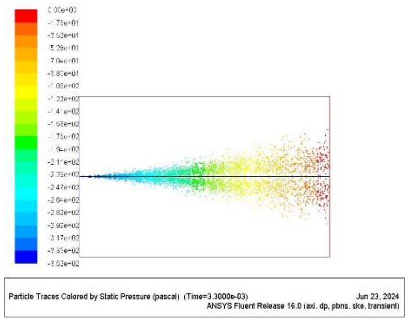
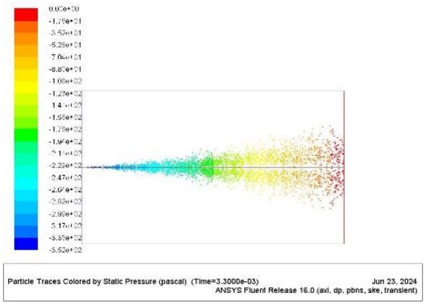
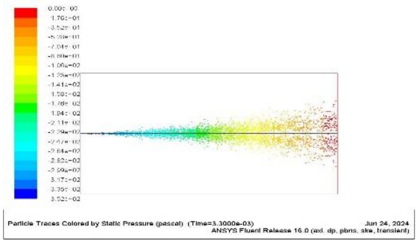
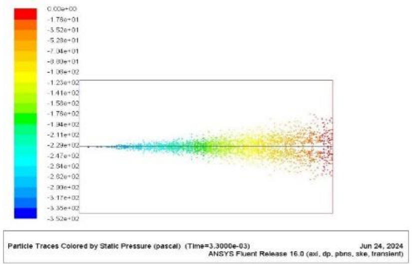
Mass flow rate, kg/s	**Particle residence time, Sec**
0.025	
0.05	
0.075	
0.100	
0.125	

Table 3: Variation of static pressure with variation of mass flow rate of water

Mass flow rate, kg/s	Static pressure, Pa
0.025	
0.05	
0.075	
0.100	
0.125	

(d) The shifting of water droplets away from the orifice with increased mass flow rates signifies enhanced momentum and reduced rebounding, optimizing water utilization and improving fire containment efficiency. (e) The decreasing spray angle with higher mass flow rates enhances droplet concentration and targeted dispersion, effectively minimizing water wastage and maximizing suppression efficacy. (f) Findings of consistent droplet reach across all flow rates and varying but controlled residence times underscore the reliability and predictability of the system in different scenarios. (g) The observation of negative static pressures and varying turbulent kinetic energy further supports efficient droplet trajectory and dispersion, which is crucial for comprehensive fire suppression and safety. (h) Data collected will help to design fire extinguishing system which will effectively manage fires by optimizing resource use and enhancing coverage, thereby contributing to improved fire safety outcomes.

ACKNOWLEDGEMENT

Authors wish to thank Dr. Sanjeet Kanungo (Principal), Capt. Manoj Hirkane, Sr. Vice Principal, Capt. Indranath Banerji, Vice Principal (Nautical science), Mr. Shailendra Kumar, Vice Principal (Marine Engineering), Dr. Ajaj Attar from Tolani Maritime Institute, Pune for their valuable inputs in preparing this article.

REFERENCES

[1] Mawhinney, J.R., "Fire Protection Water Mist Suppression Systems," NFPA Handbook - 18th Edition, 1997

[2] Pietrzak, L.M. and Ball, J.A., "A Physically Based Fire Suppression Computer Simulation - Definition, Feasibility Assessment, Development Plan, and Applications," Mission Research Corporation, MRC-R-732, April 1983.

[3] Braidech, M.M., Neale, J.A., Matson, A.F. and Dufour, R.E., "The Mechanisms of Extinguishment of Fire by Finely Divided Water," Underwriters Laboratories Inc. for the National Board of Fire Underwriters, NY, p.73, 1955.

[4] Rasbash, D.J. and Rogowski, Z.W., "Extinction of Fires in Liquids by Cooling with Water Sprays," Combustion and Flame, Vol. 1, 1957.

[5] Wighus, R., Aune, P., Drangsholt, G. and Stensaas, J.P., "Full Scale Water Mist Experiments," International Conference on Water Mist Fire Suppression Systems, Sweden, 1993.

[6] Wighus, R., "Engineering Relations for Water Mist Fire Suppression Systems," Proceedings: Halon Alternatives Technical

[7] B. Y. Lattimer and D. G. Bogard, "Radiant Heat Attenuation by Water Mist Sprays," *Fire Technology*, vol. 50, no. 2, pp. 397-417, Mar. 2014.

[8] H. Zhang and H. Zhao, "Experimental Study of the Effect of Water Mist on Flammable Gas Explosion," *Journal of Loss Prevention in the Process Industries*, vol. 41, pp. 403-411, Jan. 2016.

[9] T. G. Cleary, J. P. Yetter, and R. A. Gotts, “Water Mist Fire Suppression Systems: A Review of the State of the Art”, Fire Technology, vol. 54, no. 1, pp. 19-37, January 2018.

[10] Sarkar, S., Kumar, S., & Bose, N. (2018). Large Eddy Simulation of Water Mist Interaction in a Confined Space. Combustion Science and Technology, 190(7), 1171-1192.

[11] Choi, J., Kim, S., Lee, H., & Park, J. (2020). Computational Fluid Dynamics Analysis of Water Mist Characteristics in a Confined Space. Fire Safety Journal, 115, 103166.

[12] Smith, D., Johnson, J., & Brown, B. (2016). CFD Modeling of Water Mist Behavior in Enclosures at Different Pressures. Journal of Fire Sciences, 34(6), 485-504.

[13] Li, W., Li, S., & Zhang, Q. (2019). Numerical Simulation of Water Mist Fire Suppression in Enclosed Spaces under Different Pressure Conditions. Fire Technology, 55(5), 1807-1830.

[14] Wang, Z., Huang, X., & Shen, L. (2021). Influence of Water Spray Angle on Fire Suppression Effectiveness: CFD Simulation and Experimental Validation. Journal of Thermal Science and Engineering Applications, 13(6), 061009.

[15] Zhang, Y., Wang, F., & Liu, J. (2017). CFD Simulation of Water Mist Distribution in Confined Spaces under Different Spray Angles. Fire Safety Science, 12, 1067-1078.