SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage:
https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

The Blockage Ratio Effect to The Spray Performances

Mohamad Rasidi Pairan[1], Sharul Azmir Osman[2,*], Ahmad Nabil Md Nasir[1], Nur Hazirah Noh@Seth[1], Mohd Hizwan Mohd Hisham[1], Adjah Naqkiah Mazlan[1], Hanifah Jambari[1], Muhamad Afzamiman Aripin[1]

[1] Department of Technical and Engineering Education, Faculty of science Social and Humanities, UTM, Skudai, Malaysia
[2] Centre for Energy and Industrial Environment Studies (CEIES), Faculty of Mechanical and Manufacturing, UTHM, Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 3 December 2021
Received in revised form 5 April 2022
Accepted 7 April 2022
Available online 11 May 2022

Keywords:
Mist spray; cooling system; spray pattern

ABSTRACT

Nozzle sprays are used in wide range of application. The used of nozzle application is depend on the spray characteristics, by which to suit the particular application. This project studies the effect of the air blockage ratio to the spray characteristics. This research conducted into two part which are experimental and simulation section. The experimental was conducted by using particle image velocimetry (PIV) method, and ANSYS software was used as tools for simulation section. There are two nozzles were tested at 1 bar pressure of water and air. Nozzle A (with blockage ratio 0.316) and nozzle B (blockage ratio 1.000). Both of the sprays performances generated by the nozzles was examined at 9 cm vertical line from 8 cm of the nozzle orifice. The validation result provided in the detailed analysis shows that the trend of graph velocity versus distance gives the good agreement within simulation and experiment. From result, nozzle A generated a wider spray angle and higher water droplet velocity which are 31.41 degree and 37.317 m/s compared to nozzle B which has produced 27.13 degree of spray penetration angle and 16.49 m/s water droplet velocity. As a conclusion, blockage ratio has affected the spray system by increasing the velocity of air inside the spray system. This is happened at a condition of 1 bar air pressure.

1. Introduction

The term "water mist" is refers to very fine particle of water sprays in which 99% of the volume of the spray is in drops with diameters less than 1000 microns and that remains suspended in air for an extended period of time [1]. The first finding of the application on water mist is during 1950's and 1960'S about water mist fire protection system [2]. This water mist system is rather cheap and effective system compared to available system such as conventional sprinklers and halon gaseous agent [3]. The principle of this system is by applied the high pressure to the water to generating very fine droplet of water and delivering them to the fire zone in fact due to its high specific heat and heat of vaporization coupled with the increases surface area allowing faster heat absorption [4]. Technically the mist system raised concerns due to the high pressures required to produce a fine

* *Corresponding author.*
E-mail address: shahrula@uthm.edu.my

https://doi.org/10.37934/arfmts.95.1.99109

spray, the potential for blocking of the small orifice nozzles and doubts the long- term ability to maintain the equipment [5]. The performance of water-mist nozzles is greatly influenced by the behavior of sprays that produced. Hence, the characterization of sprays is essential to predict and evaluate the behavior of sprays produced by nozzles using the CFD [6]. The nozzle is designed based on the characteristic's method and the CFD method for various performance parameters analysis [7]. The dimensions used in the simulations are obtained from the theoretical relations developed and were verified experimentally to successfully produce mist spray [8]. Due to the nature of the complex atomization process, CFD analysis had become a very useful tool to solve the atomization process, However, there are not much application for small-scale low-pressure atomizer as indicated by Fung *et al.,* [9] Thus this research applied low pressure of water and air to produce water mist by using external mixing technique to suitable used in industry sector. This project studies the effect of the air blockage ratio to the spray characteristics. This research conducted into two part which are experimental and simulation section

2. Methodology

In this study, ANSYS software and Particles Image Velocimetry (PIV) were used to conduct the project in order to observing the velocity and penetration angle generated by the nozzle A. ANSYS 16.1 software was used to simulate the output result generated by the virtual nozzles model meanwhile PIV method was used to analysis the output result generated by the nozzle A. The same method was adopted in previous study, especially in determining the validity of the results [10]. Selected value of air pressure and water pressure was set at 1 bar. The Boundary condition setup for the simulation model is based on previous researcher [11].

Simulation was conducted by using ANSYS Fluent 16.1, the nozzle model was constructed using SOLIDWORKS. The nozzle designed and modeled for three-dimensional (3D) domain to get a better result [12]. Two types of fluids were used in this simulation and experiment which are water and air. Water and air temperature are set to 293K and 300K, the flow rate of the water for pressure 1bar was at 0.00128 kg/s, meanwhile for air pressure 1 bar was set as 0.00087 kg/s.

Figure 1 show the simulation model of nozzle system, this set of nozzles consist of three outlets orifice holes which are one for water orifice another two for air orifice. The diameter of each nozzle orifices was 0.5mm.

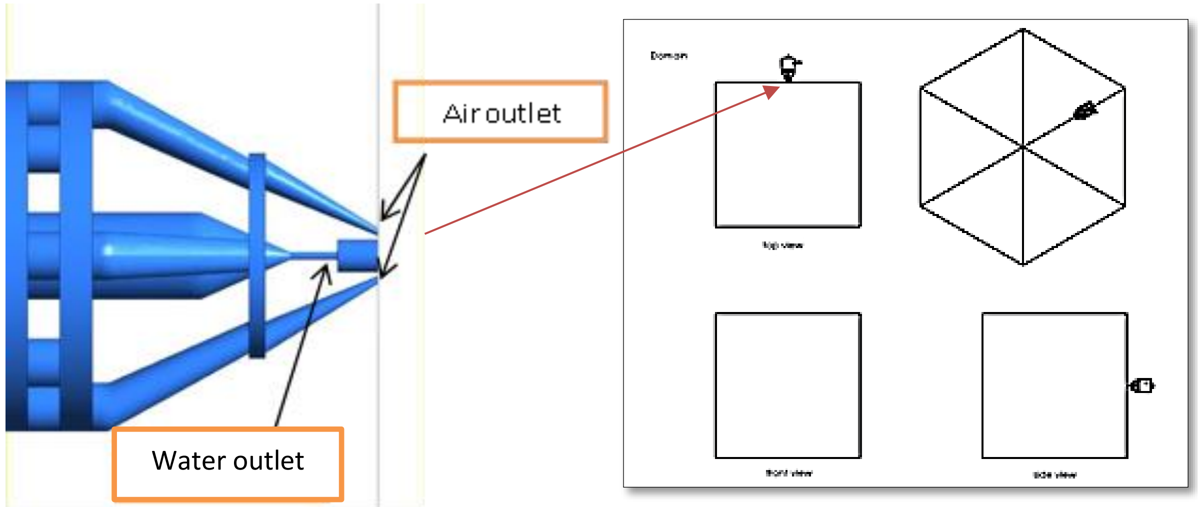


Fig. 1. Show the details of water and air outlet of the nozzle system

Meanwhile for Figure 2, it shows the computational meshing for the whole simulation domain, the domain was mesh by a combination of course and fine mesh. In details, at the middle part of the domain, the fine meshing was applied in order to get precession in the result. Based on the grid independence test on the model, it is clearly shows that, this model consists of two inlets namely as water inlet and water outlet as labeled in the Figure 2.

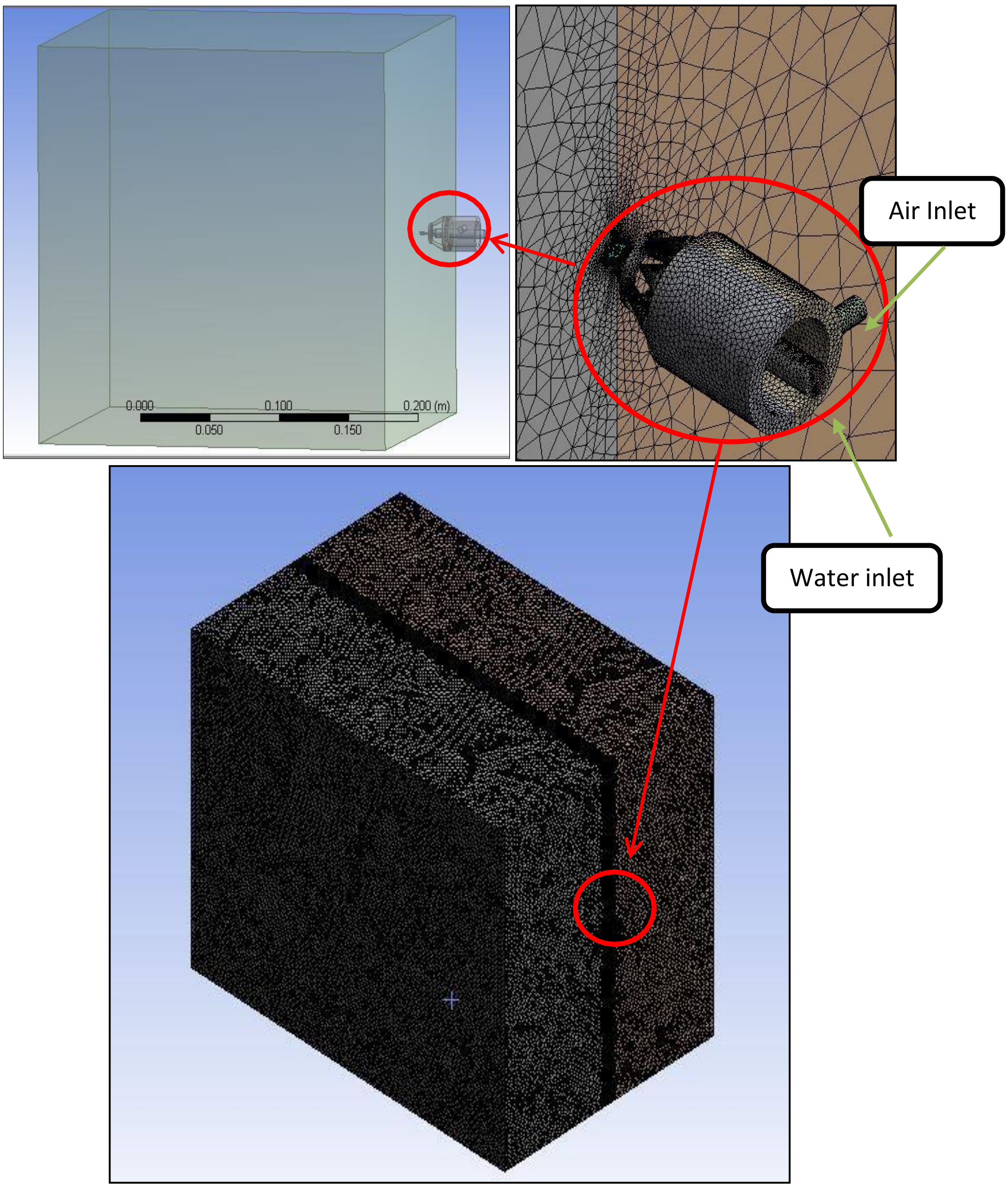


Fig. 2. Computational meshing domain

Grid independency test was a method to investigate by comparing the result produced using difference meshes level in mesh sequencing. The importance of mesh independency test was to ensure that the discretization errors in the particular simulation were relatively small. Generally, the grid independency test was the study of mesh quality prior to the result of the solution. In this test, the nozzle was tested under 3 different classes of grid. The grid was classified into three types which were coarse, medium and fine grid. Details of the number of elements and nodes for each type of grid were shown in Table 1. The velocity of air was measured along the x-axis from the centre nozzle tips.

Table 1
Grid independency test

Item	course	medium	fine
Nodes	192878	452668	878471
elements	452668	2519119	5055593

It was clearly seen that the trend of velocity versus distance of each set of meshing has almost similar trend in Figure 3. However, the average velocity of each mesh was 743.8565 m/s, 821.443 m/s and 825.3102 m/s respectively. Since the changes between medium to fine mesh was relatively small compared to the changes between medium to coarse mesh, which are 0.47% and 0.45%, thus medium mesh setup was selected to run all the nozzle.

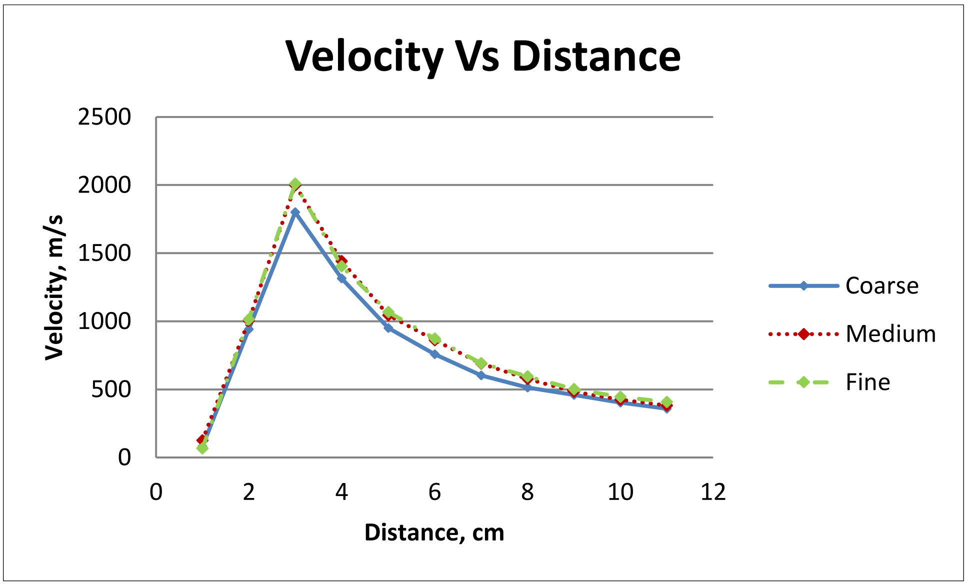


Fig. 3. Grid independency test

The simulation numerical setup and the boundary condition for this simulation was set as in the Table 2 and Table 3.

Table 2
Numerical setup

Item	Physical Properties
General	Steady State
Turbulence Model	k-epsilon realizable
Nozzle A	Blockage ratio:0.316
Nozzle B	Blockage ratio:1.000

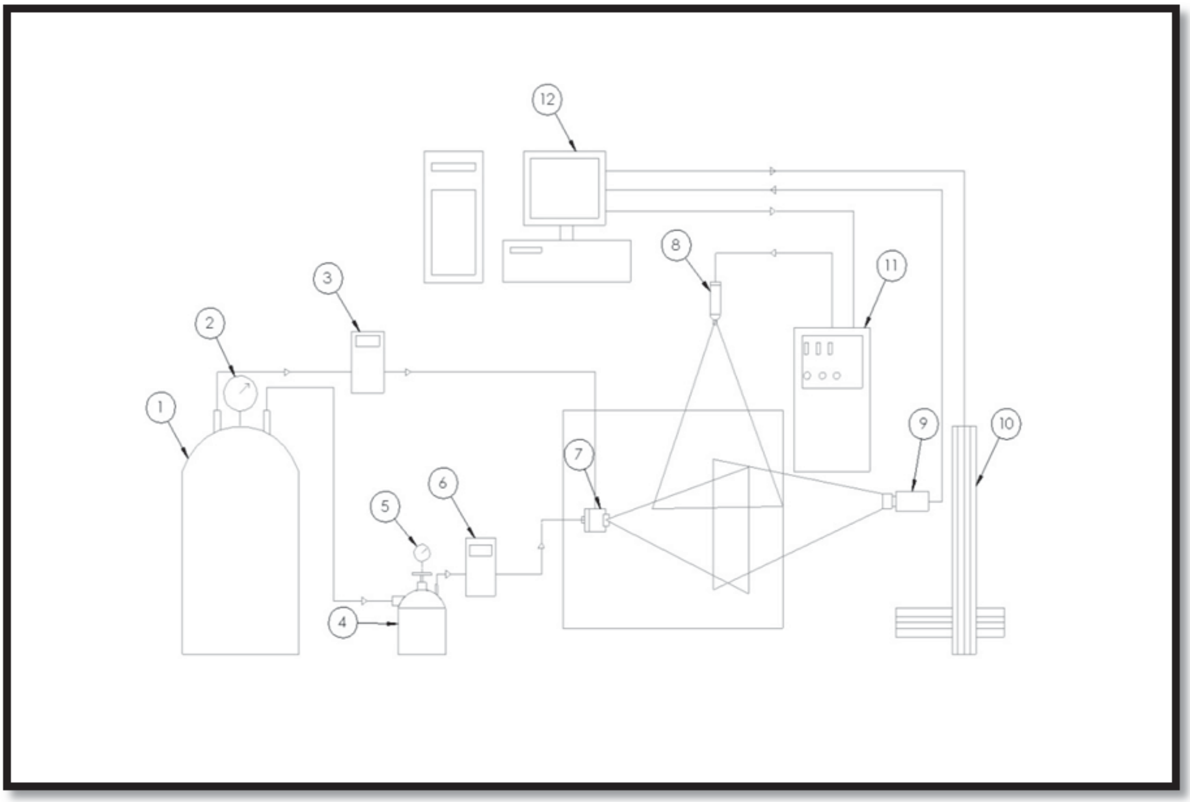
Table 3
Boundary Condition

Pressure	Mass flowrate (kgs-1)	
(bar)	Water	Air
1.0	0.00128	0.00087

3. Experiment Setup

This section will discuss the apparatus, set up and rig setting to carry out this experiment, the schematic diagram of the experiment facility is shown in Figure 4 and the real setup shown in Figure 5. The experiment setup as illustrated in the schematic diagram in Figure 3 was follow by the previous researcher [13]. There are consist of 5 important devices to run this project which are air compressor, pressure tank, pressure gauge, flow meter, 6 mm tube pipe, atomizer nozzle and lastly is spray catch basin. Clean water was placed in pressure tank; the air from the air compressor was supplied to give some pressure to the water in the pressure tank. Thus, the water in pressure tank then flows out from the pressure tank through a 6 mm tubing pipe. An inline pressure gauge was connected at the pressure tank and water flow rate was connected to the 6mm tube where the water flow rate and water pressure could be measure. The 6 mm tubing then connected to the inlet of the water nozzle.

i. The assist agent to atomize the water was supplied from the air compressor. The clean air from the air compressor then connected to the air flow rate through 6mm tube where the air flow rate could be read before supply to the inlet of nozzle spray. The water-air mixture was at the exit of the orifice nozzle. By controlling the pressure regulator of the water pressure and the air pressure, various spray pattern and droplet size could be produced.
ii. The pressure tank used was from Spray System Co. the water in pressure tank was pressurized using the air supplied from the air compressor. An inline pressure regulator was mounted at the top of pressure tank then 6mm tube connected to flow rate meter.



(1) Air compressor	(5) Pressure gauge (liquid)	(9) CCD camera
(2) Pressure gauge (air)	(6) Flow meter (liquid)	(10) Transverse
(3) Flow meter (air)	(7) Nozzle	(11) Power supply
(4) Pressure tank	(8) Laser	(12) Computer

Fig. 4. Schematic diagram of the apparatus to observe the velocity of droplet and flow pattern

Fig. 5. Experiment Setup

4. Results

Two nozzles were tested and the results were compared between experimental and simulations in order to observe the similarity of the water spray penetration angle, water droplet velocity and water droplet size. From Figure 6, it shows the graph of water particles velocity versus distance at 1bar of water and air pressure. The graph consists of two lines which plotted depending on the experiment and simulation data. Based on the graph in Figure 6, the result for experimental and simulation has almost similar trend. At this pressure condition the highest velocity generated by the experiment and simulation are 48.0224 ms-1 and 50.1340 ms-1. The highest water droplet velocity happened at the center of the orifice. However, the water droplet was decreased from point 0 to 4 and 0 to -4. Figure 7 shows the water droplet velocity contour, it is clearly shows that the water droplet velocity was decreased from the high velocity to low velocity as the particles flowing out from the nozzle orifice. The result validated by standard deviation as defined in Eq. (1) [14].

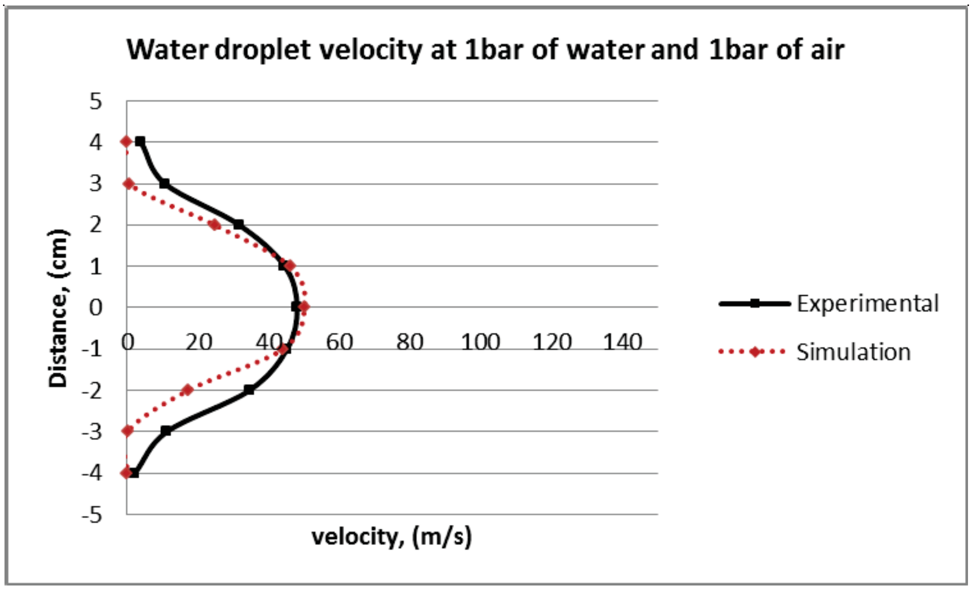


Fig. 6. Droplet velocity versus distance (1bar)

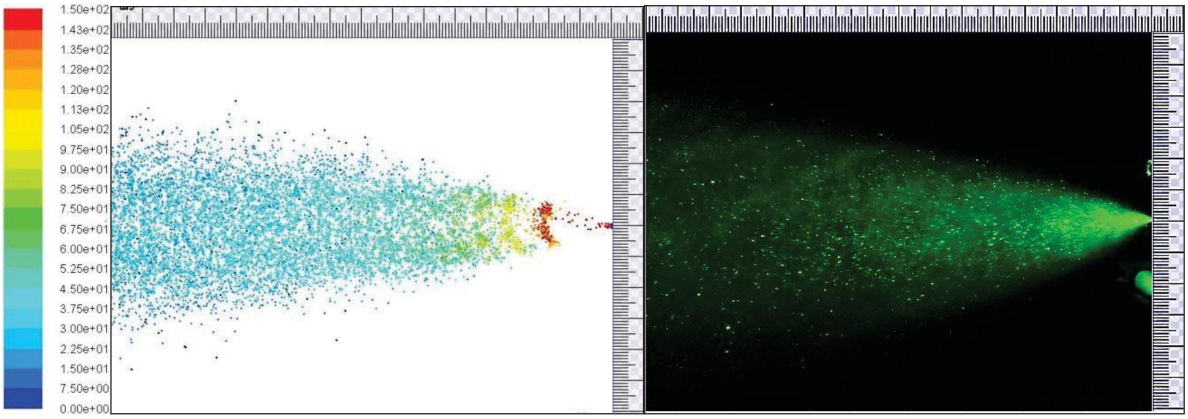


Fig. 7. Water droplet velocity contour and water droplet capture by camera (1bar)

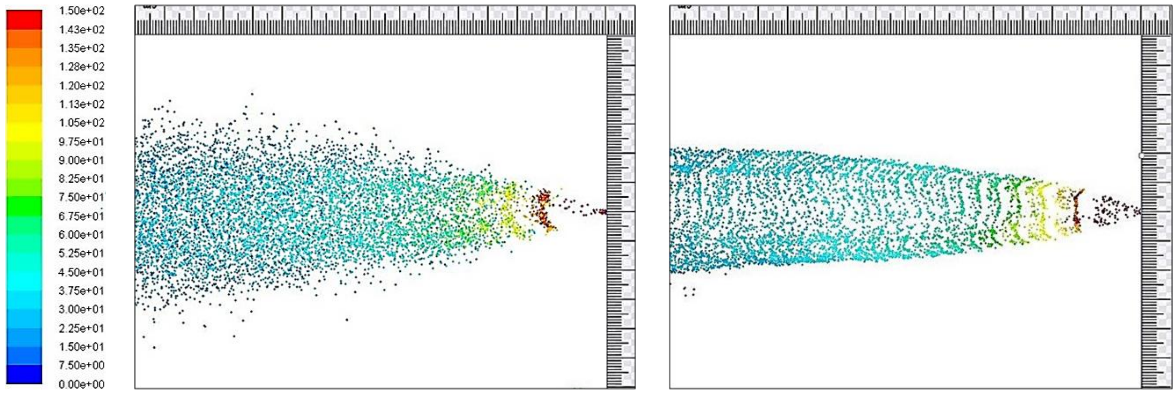
$$\sigma = \frac{\sqrt{\Sigma(U_{x_{inum}} - U_{x_{iexp}})^2}}{\mathrm{N}} \qquad (1)$$

where
$U_{x\,num}$ = *numerical droplet velocity*
$U_{x\,exp}$ = *experimental droplet velocity*
N = numbers of data

By comparing the velocity simulation values to the experiment values, the standard deviation for this validation is 0.451 where it relatively small less than 5 %, thus it shows the simulation giving high validity to the present simulation [15]. Hence, the current simulation approach was satisfactory and validated.

4.1 The Effect of Blockage Ratio

As the Nozzle A model was validated, then the domain model of nozzle A was modified in order to eliminated the blockage ratio to form a new nozzle which is known as nozzle B, two nozzles were studied and the results were compared in order to observe the effect of the blockage ratio to the water droplet velocity angle of penetration. Figure 8 and Figure 10 show the qualitative and quantitative results of water droplets velocity produced by nozzle A and nozzle B. There were two spray patterns were illustrated by as the outcome of each nozzle. Based on the Figure 8, it was obviously shown that the result generated by the nozzle A and nozzle B has a different spray pattern. Nozzle A has a random splash particle distribution compared to nozzle B which is more organized.



(a) (b)

Fig. 8. Spray patterns generated by nozzle A and nozzle B

The water droplets velocity continuously decreased as droplet move way randomly far from the exit of the nozzle. This phenomenon represented by the sequences changes of velocity contour from the red colour to the blue colour. The droplet water velocity was measured 9 cm from the nozzle orifice and 9 cm from vertical y-axis as shown in Figure 9. All the nozzles were tested under 1 bar pressure of water and 1 bar pressure of air.

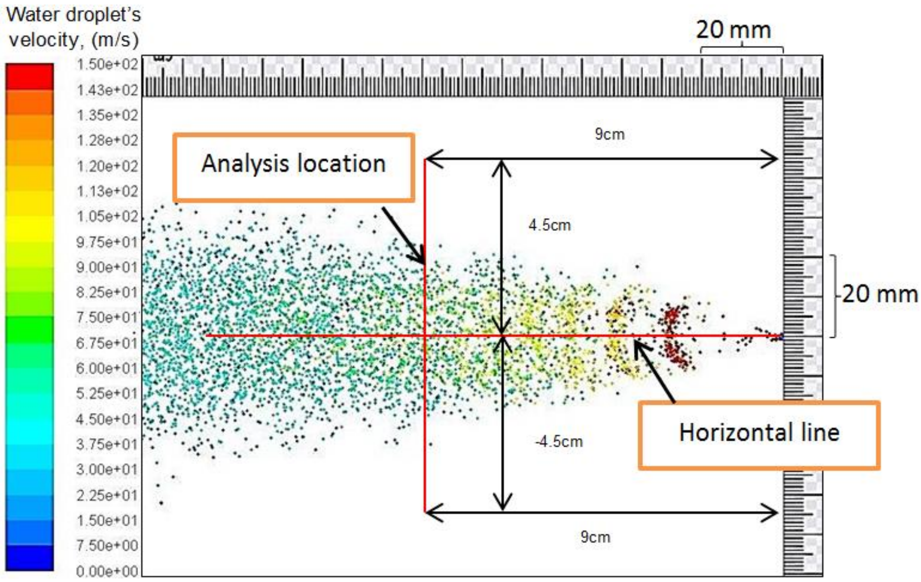


Fig. 9. Water droplets analysis location as it exiting the nozzle

Based on the graph in Figure 10, it is clearly shows that nozzle A has higher water droplet velocity at all the point, thus this give nozzle A a higher average droplet velocity compares to nozzle B which are 37.31657 m/s and nozzle B 16.49 m/s respectively. The highest velocity was achieved at center of the nozzle. The maximum droplet velocity for nozzle A and nozzle B are 50.154 m/s and 30.96 m/s.

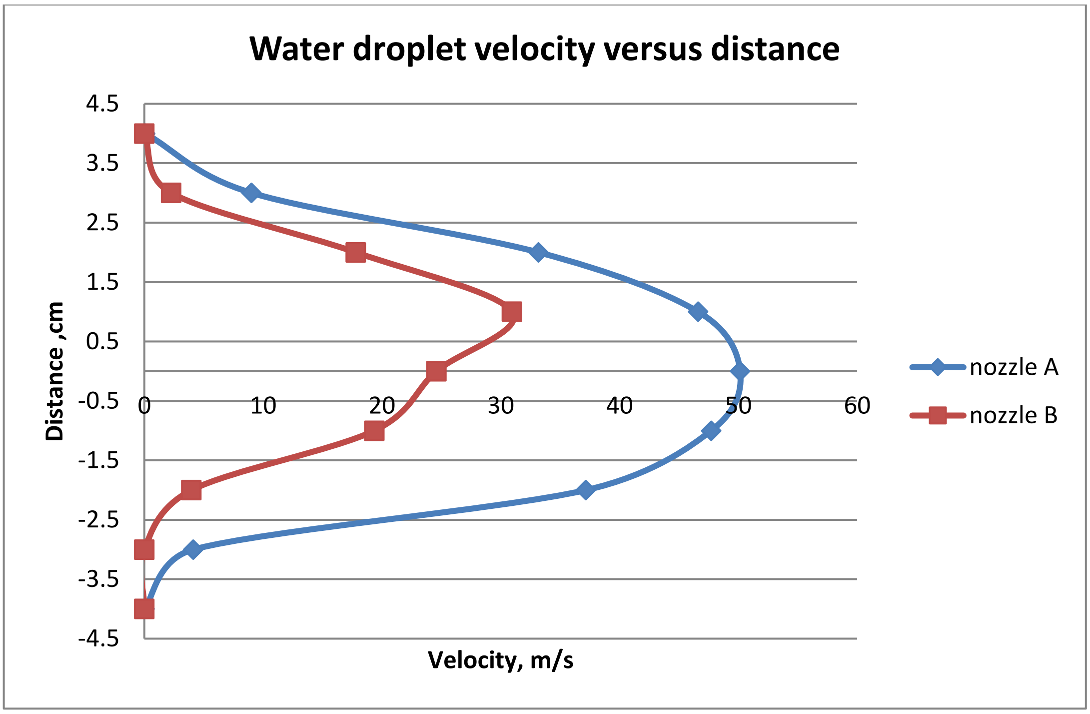


Fig. 10. Water droplet velocity of nozzle A vs water droplet velocity Nozzle B

Figure 11 illustrated the water droplet particle diameter generated by each nozzle. The characteristic of the water droplet diameter was different compared to the characteristic of the water droplet velocity phenomena. As for the water droplet velocity, the velocity decreased while the water droplet moves way far from the nozzle orifice. Meanwhile, for droplet diameter, the water droplet diameter significantly unchanged even it moves away far from the nozzle exit. However, both nozzles generated difference water droplet diameter values.

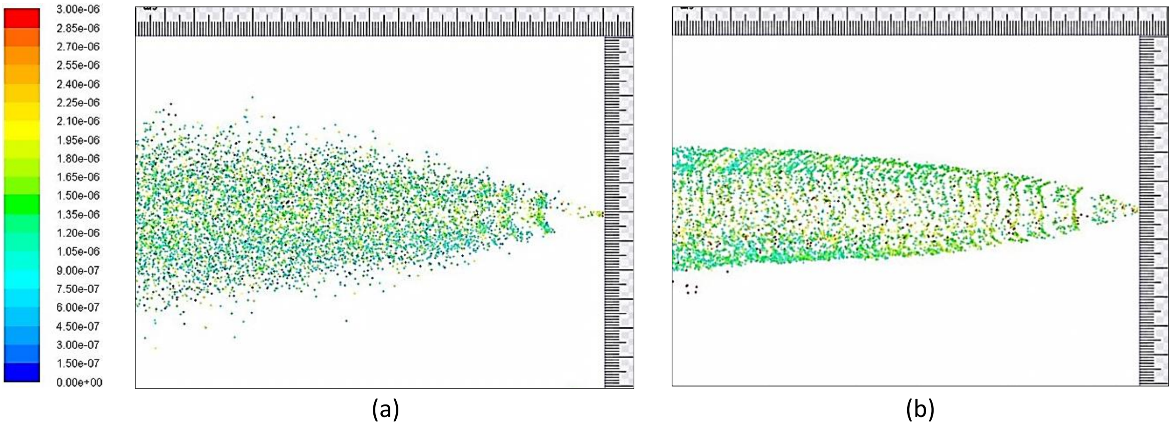


Fig. 11. Water droplets particle diameter of nozzle A vs water droplet velocity Nozzle B

Figure 12 represent the water droplets diameter for nozzle A and nozzle B, it is clearly shows that nozzle A has bigger water droplet diameter at all the point, thus this give nozzle A smaller average water droplet diameter compare to nozzle B which are 1.24199E-06 mm and nozzle B 6.79216E-07 mm respectively.

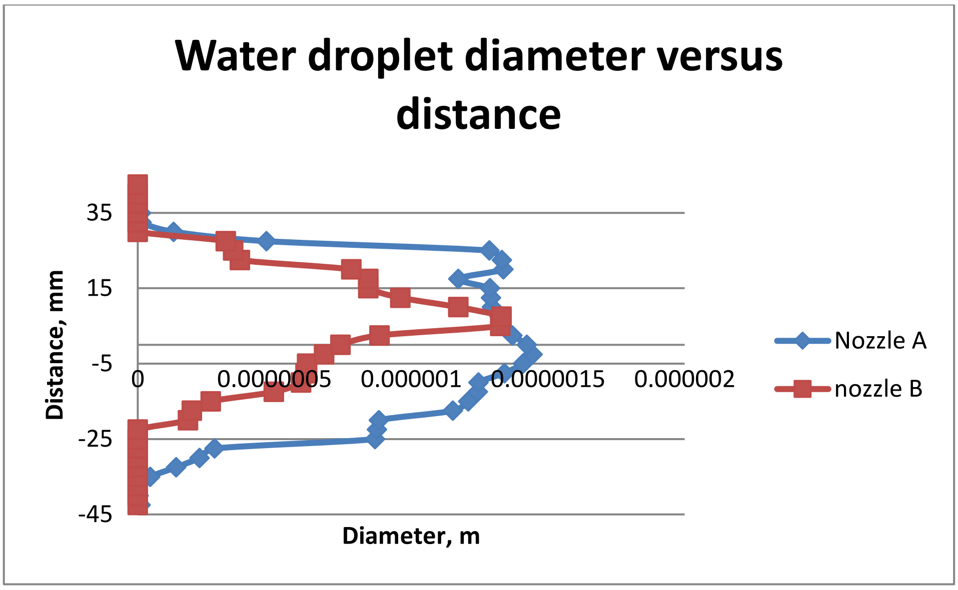
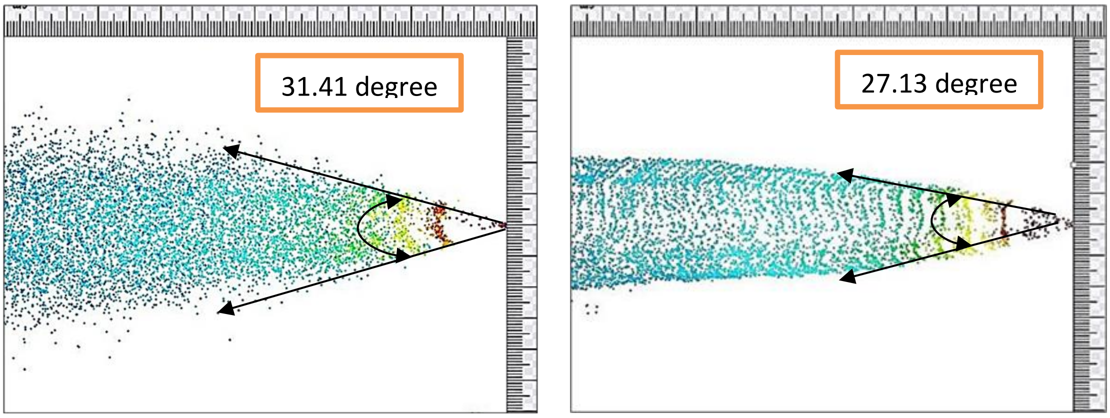


Fig. 12. Water droplet diameter vs distance of nozzle A and nozzle B

Other than water droplet size and water droplet velocity, Penetration angle also one of the important aspects to be included in this analysis, ImageJ Software were used to measure the spray penetration angle [16]. The penetration angle was measured 4 cm from the nozzle orifice. As shown in Figure 13(a) and Figure 13(b), the picture illustrated the differences angle of penetration of both nozzles, as can be seen nozzle A has wider penetration angle compare nozzle B, angle of penetration of nozzle A and nozzle B is 31.41 degree meanwhile for nozzle b is 27.13 degree.



(a) (b)

Fig. 13. Water droplet diameter penetration angle of (a) nozzle A and (b) nozzle B

5. Conclusions

As a conclusion, the effect of the blockage ratios was investigated through the simulation. The simulation study concluded that the blockage ratio gives a significant effect to the spray penetration angle for each type of nozzles, by reducing the air blockage ratio from 1.000 to 0.316, it was affecting the spray physical characteristic. Figure 10 clearly shows that reducing the air blockage ratio increases the water droplet velocity, as compared to the blockage ratio 1.000, consequently, affected the penetration angle and water droplet diameter as in Figure 12 and Figure 13. It was found that Nozzle A has the best spray performance compared to Nozzle B. Thus, it was concluded that Nozzle A has the optimum performance characteristic of the spray. The wider the spray angle means a smaller droplet size is produced and more space to distribute the droplets.

Acknowledgement
The authors would like to thank the Universiti Tun Hussein Onn Malaysia (UTHM) for partly supporting this research. Communication of this research is made possible through monetary assisstance by Universiti Tun Hussein Onn Malaysia and the UTHM Publisher's Office via Publication Fund E15216

References
[1] Mawhinney, J. R., and R. Solomon. "Water Mist Fire Suppression Systems." *Fire Protection Handbook* (1997).
[2] Milke, James A. "Comparison of the performance of water mist system designs for library stack areas." *Journal of Applied Fire Science* 5 (1996): 185-202. https://doi.org/10.2190/M1GB-T8BK-CGGH-ND8Q
[3] Hume, Benjamin Piers. "Water Mist Suppression in Conjunction with Displacement Ventilation." *School of Engineering University of Canterbury Christchurch*, New Zealand (2003).
[4] Jones, A., and P. F. Nolan. "Discussions on the use of fine water sprays or mists for fire suppression." *Journal of Loss Prevention in the Process Industries* 8, no. 1 (1995): 17-22. https://doi.org/10.1016/0950-4230(95)90057-V
[5] Mawhinney, J. R. "Characteristics of water mists for fire suppression in enclosures." In *Proceedings of Halon Alternatives Technical Working Conference*, pp. 10-11. 1993.
[6] Asmuin, Norzelawati, Mohd Farid Sies, Mohamad Khairi Hassan, M. R. Pairan, N. A. Nor Salim, and H. Zakaria. "Simulation of water mist spray for commercial kitchen ventilation (CKV) system using Ansys-CFD." In *Applied Mechanics and Materials*, vol. 660, pp. 704-708. Trans Tech Publications Ltd, 2014. https://doi.org/10.4028/www.scientific.net/AMM.660.704
[7] Mohan Kumar, G., Dominic Xavier Fernando, and R. Muthu Kumar. "Design and optimization of de Lavel nozzle to prevent shock induced flow separation." *Advances in Aerospace science and Applications* 3 (2013): 119-124.
[8] Dikshit, S. B., D. B. Kulshreshtha, and S. A. Channiwala. "Numerical simulation of pressure swirl atomizer for small scale gas turbine combustion chamber." In *13th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*. Portoroz, Slovenia. 2017.
[9] Fung, Man Chiu, Kiao Inthavong, William Yang, and Jiyuan Tu. "CFD modeling of spray atomization for a nasal spray device." *Aerosol Science and Technology* 46, no. 11 (2012): 1219-1226. https://doi.org/10.1080/02786826.2012.704098
[10] Seth, NurHazirahBinti Noh, Norasikin Binti Mat Isa, Safiah Binti Othman, and Vijay R. Raghavan. "The effects of angle of attack on 3-dimensional turning diffuser on baffle performances." *ARPN Journal of Engineering and Applied Sciences* 11, no. 3 (2006): 1536-1541.
[11] Asmuin, N., and M. R. Pairan. "New approach to reducing water consumption in commercial kitchen hood." In *IOP Conference Series: Materials Science and Engineering*, vol. 243, no. 1, p. 012043. IOP Publishing, 2017. https://doi.org/10.1088/1757-899X/243/1/012043
[12] Shariatzadeh, Omid Joneydi, Afshin Abrishamkar, and Aliakbar Joneidi Jafari. "Computational Modeling of a Typical Supersonic Converging-Diverging Nozzle and Validation by Real Measured Data." *Journal of Clean Energy Technologies* 3, no. 3 (2015): 220-225. https://doi.org/10.7763/JOCET.2015.V3.198
[13] Pairan, M. Rasidi, Norzelawati Asmuin, Nurasikin Mat Isa, and Farid Sies. "Characteristic study of flat spray nozzle by using particle image velocimetry (PIV) and ANSYS simulation method." In *AIP Conference Proceedings*, vol. 1831, no. 1, p. 020009. AIP Publishing LLC, 2017. https://doi.org/10.1063/1.4981150
[14] Pairan, Mohamad Rasidi, Norzelawati Asmuin, and Hamidon Salleh. "Simulation of Film Cooling in the Leading Edge Region of a Turbine Blade Using ANSYS CFX." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 1, no. 1 (2014): 1-10.
[15] Grich, Nihel, Walid Foudhil, Souad Harmand, and Sadok Ben Jabrallah. "Numerical simulation of water spray transport along a plate of a heat exchanger." *Journal of Thermal Analysis and Calorimetry* 143, no. 5 (2021): 3887-3895. https://doi.org/10.1007/s10973-020-09356-w
[16] Isa, Kamariah Md, Kahar Osman, Azli Yahya, Zulkifli Abdul Ghaffar, Ahmad Hussein Abdul Hamid, and Salmiah Kasolang. "Studies on the spray characteristics of pressure-swirl atomizers for automatic hand sanitizer application." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 55, no. 1 (2019): 51-64.