Characteristic study of flat spray nozzle by using particle image velocimetry (PIV) and ANSYS simulation method

M. Rasidi Pairan, Norzelawati Asmuin, Nurasikin Mat Isa, and Farid Sies

Articles you may be interested in

Study the velocity and pressure exerted in front of the filter surface in the kitchen hood system by using ANSYS
AIP Conference Proceedings **1831**, 020007 (2017); 10.1063/1.4981148

Characteristic Study of Flat Spray Nozzle by Using Particle Image Velocimetry (PIV) and ANSYS Simulation Method

M.Rasidi Pairan [1, a)], Norzelawati Asmuin [1, b)], Nurasikin Mat Isa [1] and Farid Sies[1]

[1]Department Energy & ThermoFluid Engineering,
FKMP, UTHM, Batu Pahat, Malaysia.

[a)]uncle_sid@yahoo.com.my
[b)]norzela@uthm.edu.my

Abstract. Water mist sprays are used in wide range of application. However it is depend to the spray characteristic to suit the particular application. This project studies the water droplet velocity and penetration angle generated by new development mist spray with a flat spray pattern. This research conducted into two part which are experimental and simulation section. The experimental was conducted by using particle image velocimetry (PIV) method, ANSYS software was used as tools for simulation section meanwhile image J software was used to measure the penetration angle. Three different of combination pressure of air and water were tested which are 1 bar (case A), 2 bar (case B) and 3 bar (case C). The flat spray generated by the new development nozzle was examined at 9cm vertical line from 8cm of the nozzle orifice. The result provided in the detailed analysis shows that the trend of graph velocity versus distance gives the good agreement within simulation and experiment for all the pressure combination. As the water and air pressure increased from 1 bar to 2 bar, the velocity and angle penetration also increased, however for case 3 which run under 3 bar condition, the water droplet velocity generated increased but the angle penetration is decreased. All the data then validated by calculate the error between experiment and simulation. By comparing the simulation data to the experiment data for all the cases, the standard deviation for this case A, case B and case C relatively small which are 5.444, 0.8242 and 6.4023.

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 vaporation 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 spray, the potential for blocking of the small orifice nozzles and doubts the long-term ability to maintain the equipment [5]. 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.

METHDHOLOGY

In this study, ANSYS software and Particles Image Velocimetry (PIV) were used to conduct the project in order to observing the velocity and angle penetration generated by the new development spray. ANSYS 16.1 software was used to simulate the output result generated by the virtual new development spray model meanwhile PIV method was used to analysis the output result generated for fabricated new development spray. Selected values of pressure of air and water are selected from 1 bar, 2 bar and 3 bar. "Air to liquid ratio" and "standard deviation" are defined as:

$$W^{-1} = \dot{m}_1 / \dot{m}_A \tag{1}$$

Where;

$\dot{m}_l$ = mass flow rate of liquid

$\dot{m}_A$ = mass flow rate of air

$$\sigma = \frac{\sqrt{\sum \left(U_{xi_{numerical}} - U_{xi_{experimental}} \right)^2}}{n} \tag{2}$$

Where;

$U_{xi_{numerical}}$ = local numerical velocity

$U_{xi_{experimental}}$ = local experimental velocity

n   = number of data

Simulation was conducted by using ANSYS fluent 16.1, the new development nozzle model was constructed using SOLIDWORKS. 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 2 bar and 3 bar were set as 0.00128 kg/s, 0.00223kg/s and 0.003217kg/s meanwhile for air pressure 1 bar, 2bar and 3 bar the flow rate were set as 0.00087kg/s, 0.00138kg/s and 0.00186kg/s.

Figure 1 show the computational domain of new development water mist model, this set of water mist spray consist of three orifice holes which are one for water orifice another two for air orifice. The diameter of each nozzle orifices was 0.5mm.
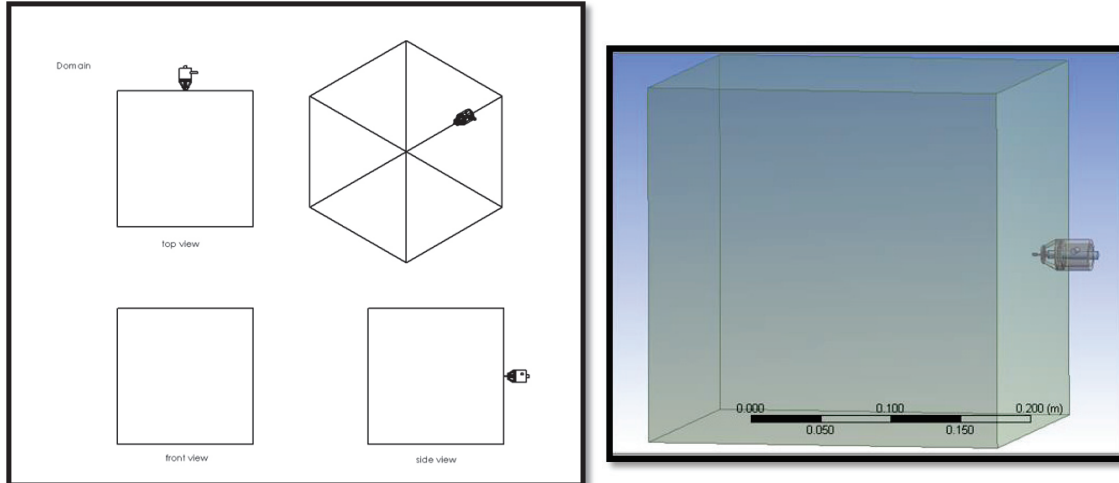

FIGURE 1. Computational domain new development nozzle

Item	Physical Properties	
General	Steady State	
Turbulence Model	k-epsilon realizable	

Pressure (bar)	Mass flowrate (kgs^{-1})	
	Water	Air
1.0	0.00128	0.00087
2.0	0.00223	0.00138
3.0	0.00321	0.00186

Experiment setup

This section will discuss the apparatus, set up and rig setting to carry out this experiment, A schematic diagram of the experiment facility is shown in figure 2 and the real setup shown in figure 3. The experiment setup as illustrated in the schematic diagram in figure 2 was follow by the previous researcher [6]. There are consist of 5 important devices to run this project which are air compressor, pressure tank, pressure gauge, flow meter, 6mm 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 6mm 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 6mm tubing then connected to the inlet of the water nozzle.

1. 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 produce.

2. 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.
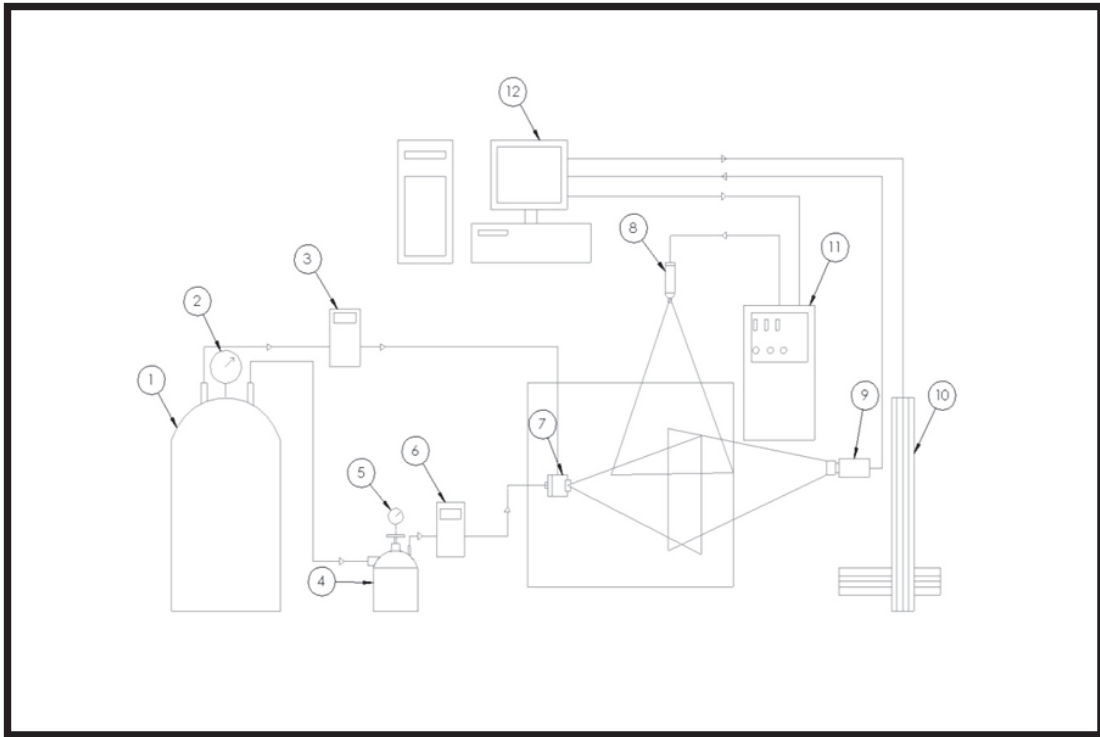
FIGURE 2. Schematic diagram of the apparatus to observe the velocity of droplet n flow pattern (6).

(1) Air compressor
(2) Pressure gauge (air)
(3) Flow meter (air)
(4) Pressure tank

(5) Pressure gauge (liquid)
(6) Flow meter (liquid)
(7) Nozzle
(8) Laser

(9) CCD camera
(10) Transverse
(11) Power supply
(12) Computer

FIGURE 3. Experiment setup

RESULT

A new development mist nozzle was tested and the results were compared between experimental and simulations in order to observe the similarity of the water spray penetration angle and water droplet velocity. From figure 4, 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 4, 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 0 to 4 and 0 to -4. Figure 5 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 equation 2. By comparing the simulation data to the experiment data, the standard deviation for this case was 5.444.
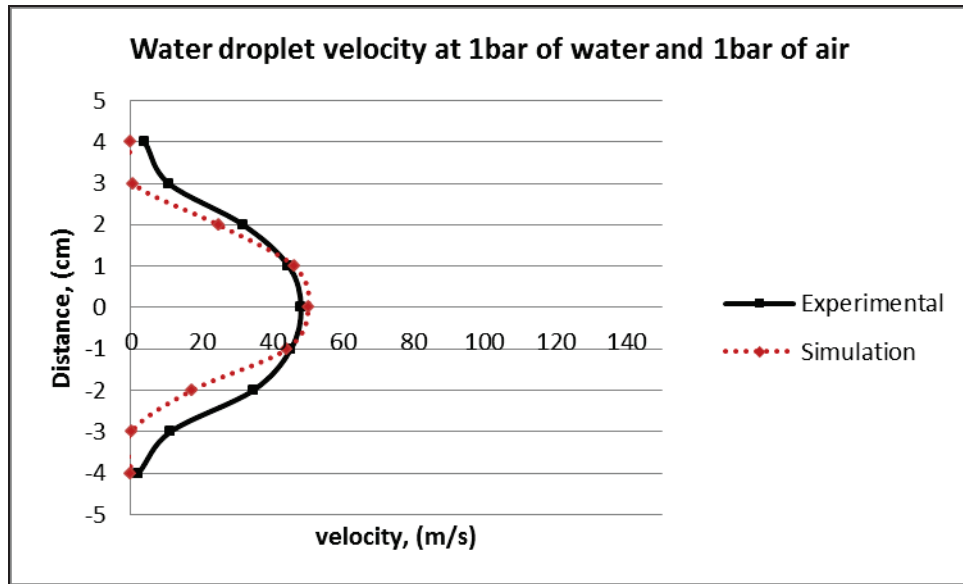

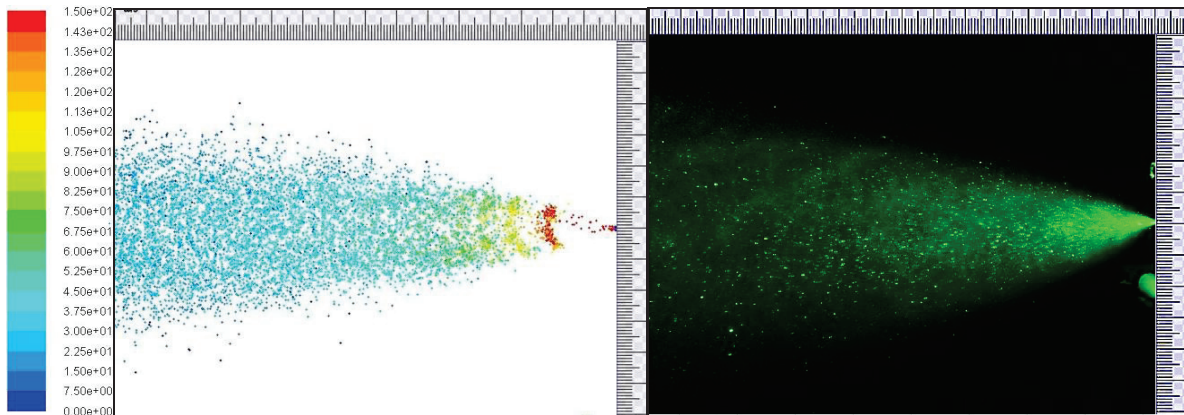
FIGURE 4. Droplet velocity versus distance (1bar)

FIGURE 5. water droplet velocity contour and water droplet capture by camera (1bar)

For case B, figure 6 represent the result gathered from the experimental and simulation, as the pressure of water and air was increased from 1bar to 2bar, the velocity of water particles was increased, it is shown in the graph in the figure 6, the highest water droplet velocity is 86.1780 ms^{-1} for the experiment meanwhile for simulation 82.5479ms^{-1} which happened at point 0. However, even though the velocity was increased, both of the simulation and experimental shows the similar trend of water droplet velocity with significant different of value. Referring to the water droplet velocity contour in figure 7, case B has more water droplet with high velocity which indicated by yellow to oren velocity range. By comparing the simulation data to the experiment data, the standard deviation for this case was 0.8242.
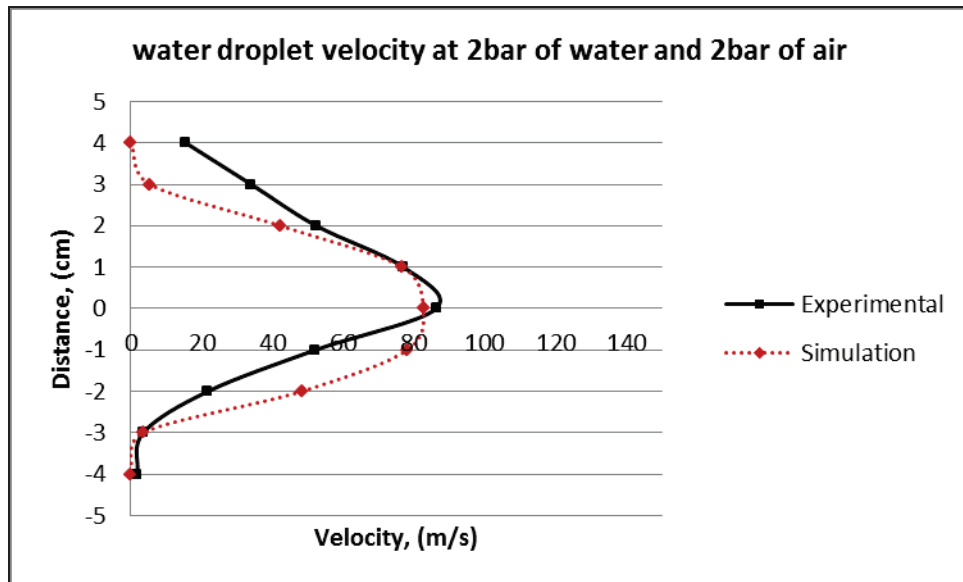
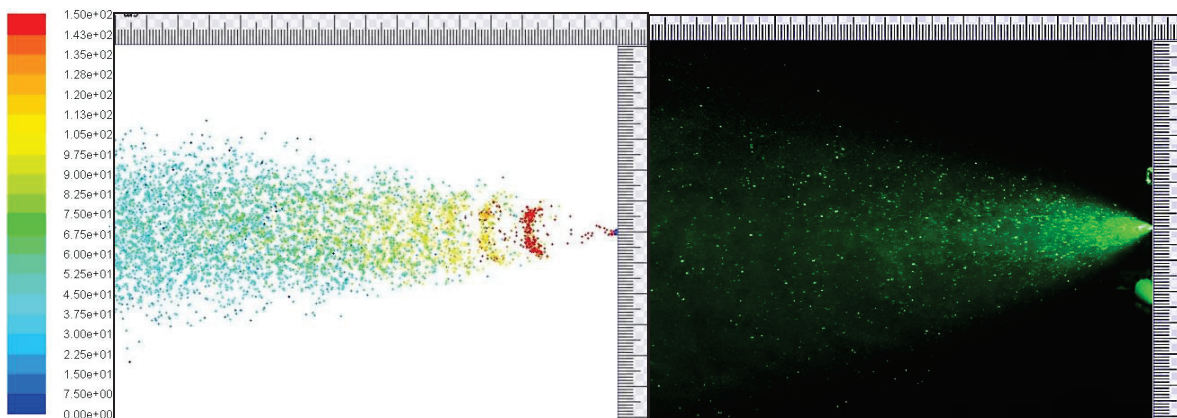**FIGURE 6.** Droplet velocity versus distance (2bar)

FIGURE 7. water droplet velocity contour and water droplet capture by camera (2bar)

Figure 8 show the result gathered for case C, based on the figure 8. Case C has the highest droplet velocity of water compare to other cases, it has the highest droplet water velocity for all the point from 0 to 4 and 0 to -4 compared to the case A and case B. The maximum velocity is at the mid line of the nozzle which are 115.5413 ms^{-1} for experimental meanwhile 115.6980 ms^{-1} for simulation.. Both of the plotted experimental and simulation data show almost the similar trend. Based on the figure 9, the water droplet velocity contour indicates that there are more number of droplet water with the high velocity for the particular position compared to the case A and case B. By comparing the simulation data to the experiment data, the standard deviation for this case was 6.4023..
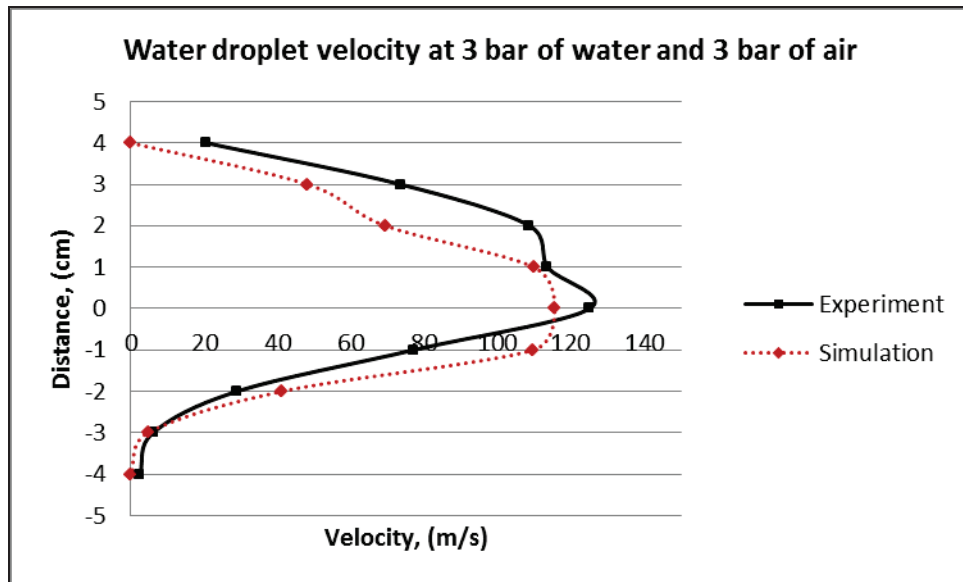
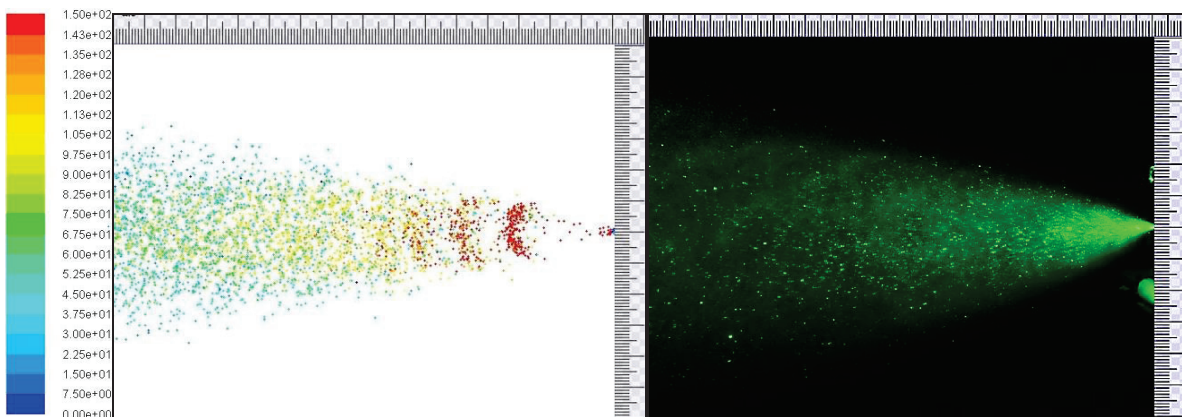**FIGURE 8.** Droplet velocity versus distance (3bar)

FIGURE 9. water droplet velocity contour and water droplet capture by camera (3bar)

The water penetration angle were measured by using IMEJ J software, it is clearly shows that the water droplet penetration generated by the simulation and experimental are almost identical. The water droplet penetration angle produced by the experimental is 48 degree for case A. As the pressure of air and water was increased to 2 bar, the angle of water penetration also increased to 56 degree. However when the pressure of air and water were increased to 3 bar, the angle of water penetration is decreased to 50 degree. All the result of water penetration angle shows in the figure 10, figure 11 and figure 12.
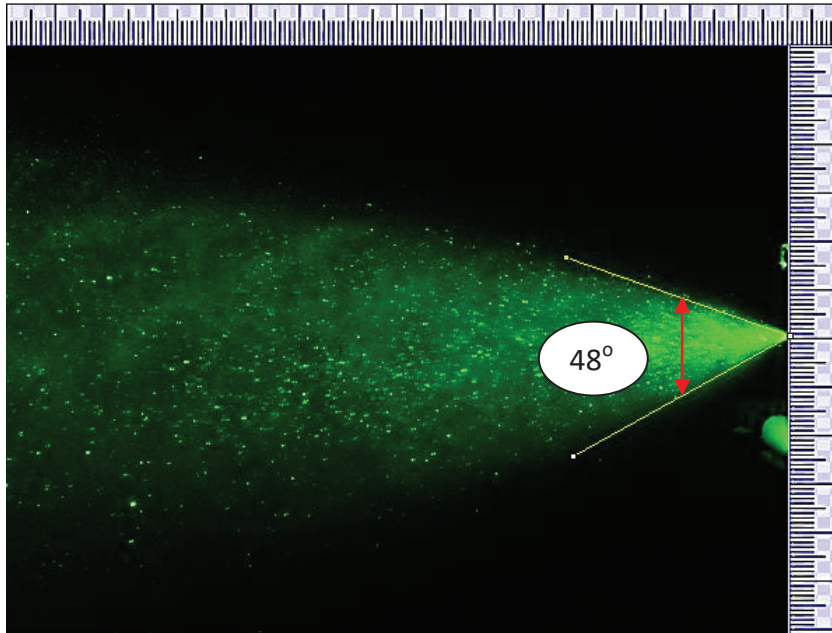
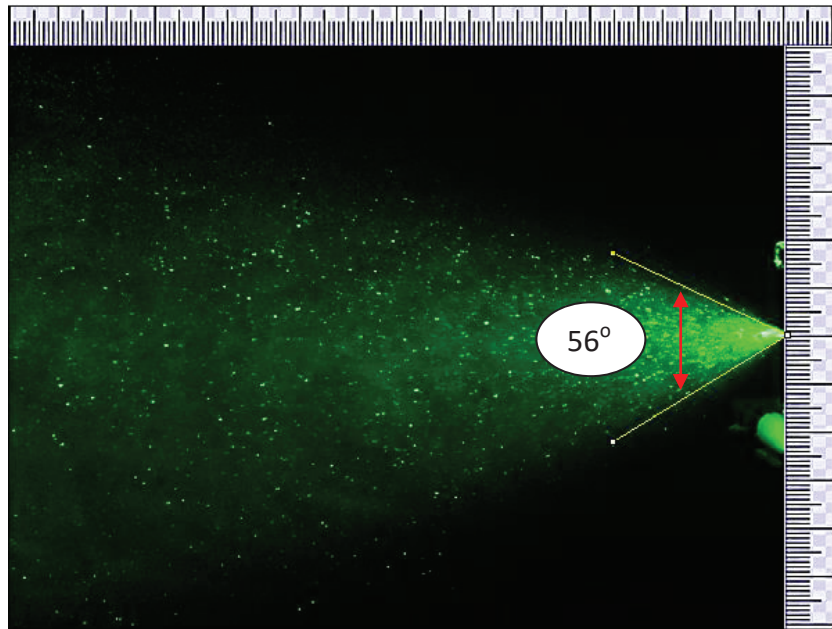**FIGURE 10.** Penetration angle between experiment and simulation (1bar)

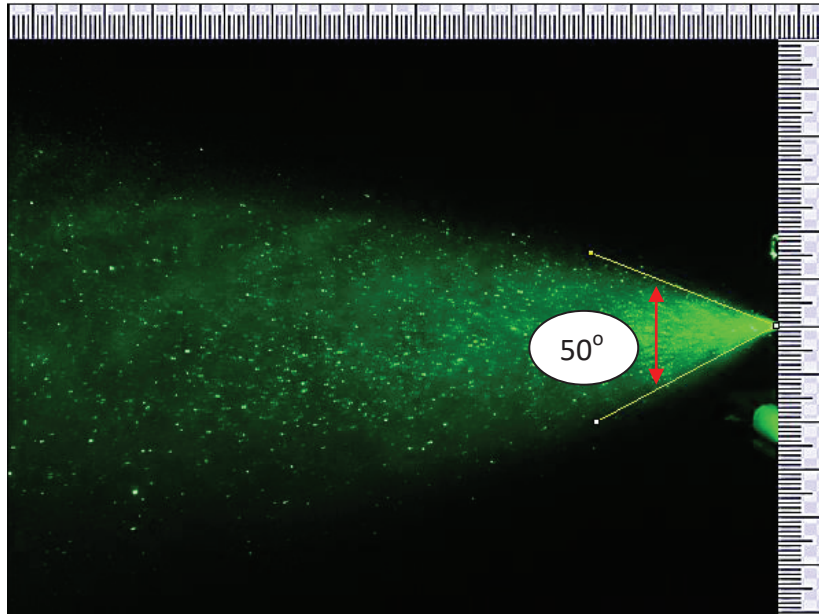**FIGURE 11.** Penetration angle between experiment and simulation (2bar)

FIGURE 12. Penetration angle between experiment and simulation (3bar)

CONCLUSIONS

Three cases were study to observe the velocity and the penetration angle for new development nozzle. Based on the result, it is simply shows by increased the pressure of water and air, the velocity of the water droplet will increased. However, for the penetration angle, the angle penetration was increased when the pressure is increased from 1 bar to 2bar, while, when the pressure is increased to 3 bar, the water droplet velocity still increasing but decreased for the water penetration angle. Hence, it can be concluded that the water droplet velocity and water droplet angle penetration performance can be significantly improved by controlling the ratio of water and air flow rate. It is noted that water and air pressure has a very strong effect on the water droplet velocity.

FUTURE SCOPE

This study is for the future scope to implement the new development mist spray to the commercial kitchen hood system as a tool to traps the small particles. It is very important to know the physical characteristic of the new development nozzle.

ACKNOWLEDGEMENTS

REFERENCES

1. Mawhinney, J.R. and Solomon R. (1997). "Water Mist Fire Suppression Systems, Fire Protection Handbook", 18th ed. National Fire Protection Association, Quincy, MA.
2. Milke, J. A. (1996). "Comparison of the Performance of Water Mist System Designs For Library Stack Areas", University of Maryland, College Park, Baywood Publishing Co., Inc.
3. Benjamin Piers Hume (Feb 2003). "Water Mist Suppression in Conjunction with Displacement Ventilation". School of Engineering University of Canterbury Christchurch, New Zealand,
4. Jones, A. and Nolan,P.F. (1955). "Discussion on the Use Of Fine Water Sprays or Mist for Fire Suppression". Journal Loss Prev. Process ind., Volume 8,Buterworth-Hainemann Ltd, Pp17-21

5. Mawhinney, J.R. and Eng P. "Characteristics of Water Mist for Fire Suppression in Enclosures". National Research Council, Institute for Research in Construction, National Fire Laboratory, Canada.

6. Norzelawati Asmuin, (Jan 2011). "Investigation Into Novel Matched Valve-Actuator Atomiser (Insert) Design For Compressed Gas Aerosols". Thesis Phd, School of Computing, Science and Engineering University of Salford,UK.