CFD Letters 15, Issue 11 (2023) 110-117

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Simulation Analysis for the Distribution of Fire Water Sprinkler

Foong Meng Kit[1], Mohamad Hisyamuddin Ishak[1], Mohamad Faris Fauzi[1], Tengku Izzat Azraei Tengku Mohamad Muhibuddin[1], Ishkrizat Taib[1,*]

[1] Computational Fluid Dynamics BDE 40403, Faculty of Mechanical and Manufacturing Engineering, University Tun Hussein Onn Malaysia (UTHM), Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 12 June 2023
Received in revised form 15 July 2023
Accepted 10 August 2023
Available online 1 November 2023

Keywords:
Computer fluid dynamics (CFD); Fire Water Sprinkler; distribution of discharged water; simulation

ABSTRACT

Fire Water Sprinkler is one of reliable equipment to control fire in building. This equipment also known as a simple and cost-effective method to control fire growth in a building. The systems work by blowing the water droplet into the burning area. The objective of this project is to analyse the behaviour and distribution of discharged water from fire water sprinkler. The computer fluid dynamics (CFD) is used to study the dispersion of water into the room or case study area. However due to limitation of study, this simulation only carried out by using a room and single fire water sprinkler. This also important factor as the validation of the model is based on previous study. In addition, the project also to determine the effect of different operating pressure on water discharged from fire sprinkler. The project started with the data finding. It was important to identify the reference data to ensure the generate result was reasonable. The input water inlet pressure is in the range of 10 to 15 bar. The geometry of room was set to 4.0 m x 3.0 m x 4.0 m. The fire was assumed to start at the rectangular table which is the dimension is 1.0 m x 1.0 m and 0.5 m above the floor in the middle room. For boundary condition, the inlet parameter is velocity and the input magnitude set at 44.7 m/s under 10 bar. 1000 number of iterations was set in this project. The results showed the model of 15 bar operating pressure was managed to dispense the water jet down to rectangular table. The 15 bar operating pressure of the model has greatest velocity discharged up to 12.896 m/s. As conclusion, the operating pressure was affected the discharged of the water velocity. The increase of the operating pressure was led to greater water velocity discharged.

1. Introduction

The There are many types of the fire water sprinkler that applied in room in the building [1-10]. The fire water sprinkler was applied in room to prevent the fire spread to the whole building. The fire water sprinkler heat was detected any heat or high temperature, the valve will open and release the high pressure of water from pipe. The water that released were distributed for the whole building. The types of the fire water sprinkler have different types of the water distribution and different time rate to distribute for the whole building. The distribution and time rate of the water from fire water sprinkler was contributed to the efficiency of fire water sprinkler function.

* *Corresponding author.*
E-mail address: iszat@uthm.edu.my (Ishkrizat Taib)

https://doi.org/10.37934/cfdl.15.11.110117

However, there were many factors that can affect the dispersion of water into the room. So, computer fluid dynamic (CFD) software was chosen to conduct the simulation regarding the dispersion of fire water sprinkler. The purpose of the simulation was to understand and analysis the distribution of water dispersion for future improvement regarding the dispersion of water by using fire water sprinkler

1.1 Fire Water Sprinkler

The method that used in fire water sprinkler is water-based fire suppression. This method was widely used due to their effectiveness, simplicity and accessibility [1]. Furthermore, the fire water sprinkle also used to prevent fire growth by dispersing water over a wide area within the fire environment [2]. It also one of spray dispersion method that simple and cost- effective to reduce fire growth [3]. Figure 1 shows the example of fire water sprinkler.

Fig. 1. The fire water sprinkler

2. Research Methodology

In order to carry an experiment, some of parameter is set differently to achieve result expected. Based on [1] four nozzles were arranged in a square configuration typical of fire sprinkler system installations. Volume flux of water delivered to the floor was measured 1.5 m below the array at locations on a 50 mm grid. The total mean volume flux was 5.9 mm/min, with local measurements ranging from 25% to nearly 400% of the mean value. As in Malaysia market, the fire sprinkler head designed to operate under working water pressure of 12 bar. In this case, the input of water inlet pressure is in the range of 10 to 15 bars. The inlet would be set as velocity inlet after the water velocity values were obtained. Table 1 shows the conversion between bar, mm H20 and water velocity by using Engineering Toolbox Converter.

Table 1
Velocity-head conversion

Model	Water velocity inlet (m/s)	Operating Pressure (mm H20)	Operating Pressure (bar)
A	44.7	101971.62129779	10
B	49	122365.94555734801	12
C	51	132563.10768712702	13
D	52.9	142760.26981690602	14
E	54.8	152957.431946685	15

2.1 Governing Equations

Reynolds-Averaged Navier-Stokes equations (RANS). RANS consists of continuity equation and momentum equation. Newtonian and incompressible fluid flow simulation can be solved by using RANS. In this study, the flow is assumed in steady state, 3-dimensional and fully developed. Energy equation also will be reviewed.

Continuity equation:

$$\frac{\partial u_i}{\partial u_i} = 0 \tag{1}$$

Momentum equation

$$\rho\left(\frac{\partial u_i}{\partial t_i} + u\frac{\partial u_i}{\partial x_j}\right) = \frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_i \partial x_i} + f_i \tag{2}$$

2.2 Geometry of the Model

The analysed room equipped with fire sprinkler on top was a compartment with dimensions of 4.0m × 3.0m × 4.0m. The wall was considered as solid concrete walls. A rectangular table with area of 1.0m × 1.0m was placed 0.5m above the floor in the middle of room. Figure 2 shows the detail drawing of tested room equipped with fire sprinkler sketched by using SolidWorks 2017.

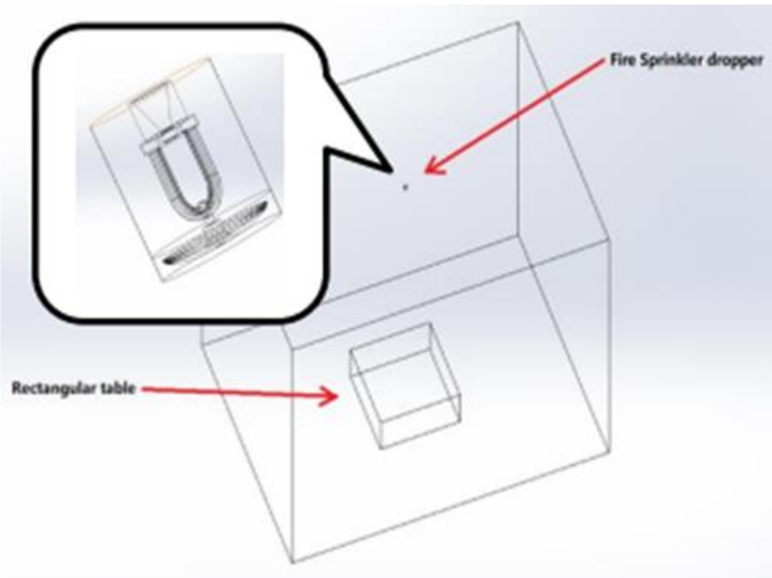


Fig. 2. The geometry of model

2.3 Mesh Setup for Fire Water Sprinkler

The mesh method is set to automatic generation. The proximity and curvature enabled successful refinement on critical regions such as curves and edges. The element size will be altered or refined in Grid Independent Test (GIT) step until there are no significant change towards the desired parameters. The details of fine mesh generated for fire sprinkler region and the first element size generated is 0.08 m.

2.4 Mesh Quality of Fire Water Sprinkler

Mesh metric plays a very important role to indicate the quality achieved during the meshing process. In this project, skewness ratio mesh metric is selected as reference to study the mesh quality produce in building suitable mesh. The maximum given skewness must less than 0.90 or below. Based on Table 2, the obtained skewness is 0.86879 and this showed that first set up element size of 0.08m has produce “Good” quality mesh with total elements of 561215. Table 2 shows the generated of the mesh quality.

Table 2
Mesh quality generated

Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.900000)
Smoothing	Medium
Mesh Metric	Skewness
Min	1.0623 e-004
Max	0.86879
Average	0.22556
Standard Deviation	0.11862
Statistics	
Nodes	103021
Elements	561215

2.5 Grid Independence Test

Grid independence test is being conducted in this project to determine the best element size for this geometry. In this project, the simulation is being conducted three times by using different element size and point 2 is being used as the validation point in the empty mosque case. Table 3 shows the test number, element size and element number produced.

Table 3
The number of Grid Independence Test

GIT	Element Size (m)	Amount of Element
1	0.08	561215
2	0.07	727223
3	0.06	978377
4	0.05	1410067
5	0.04	1925447

2.6 Setup of the Model

The model of water flow through fire sprinkler in room is taken as steady state problem. So, it was solved as multiphase (air & water) simulation using the volume of fluid multiphase model. For the phase interaction, the surface tension between air and water is set to 0.072 n/m constant value. Next, for boundary conditions, it required to define the inlet parameters. The surface that had chosen as water inlet can be referred in Figure 3 and it was set as velocity inlet. The velocity magnitude was input as 44.7 ms−1under 10 bar operating water pressured. For air phase in inlet, the volume fraction was selected as zero primarily.

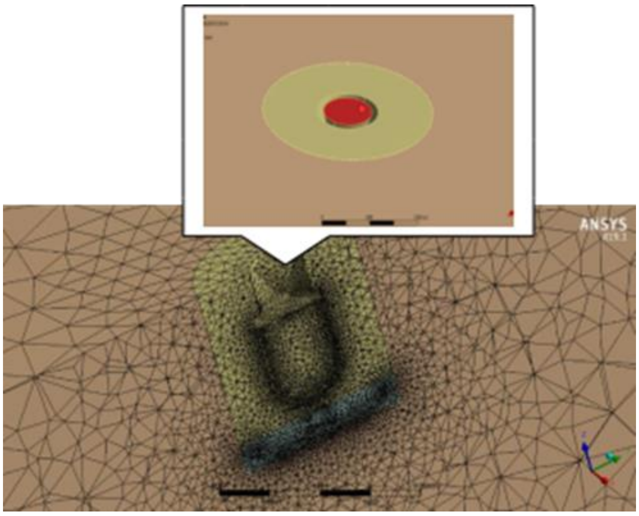


Fig. 3. Selected surface as water inlet

The solution settings can be started by selecting the methods used in simulation. Under pressure-Velocity Coupling, the PISO scheme had been used. For spatial discretization, the least square cell-based technique was implied in gradient and PRESTO! for pressure part. The momentum simulation was set to First Order Upwind in order to increase the stability result obtained. The initialization of display settings can be started with creating new window. First window is used for contour graphic to show the distribution of water in room. This can be done by first adding a new plane which is Plane 8. The scale residuals can be viewed in second window.

2.7 Initialization of Display

The initialization of display settings can be started with creating new window. First window is used for contour graphic to show the distribution of water in room. This can be done by first adding a new plane which is Plane 8. The scale residuals can be viewed in second window. The setting of first window is shown in Figure 4.

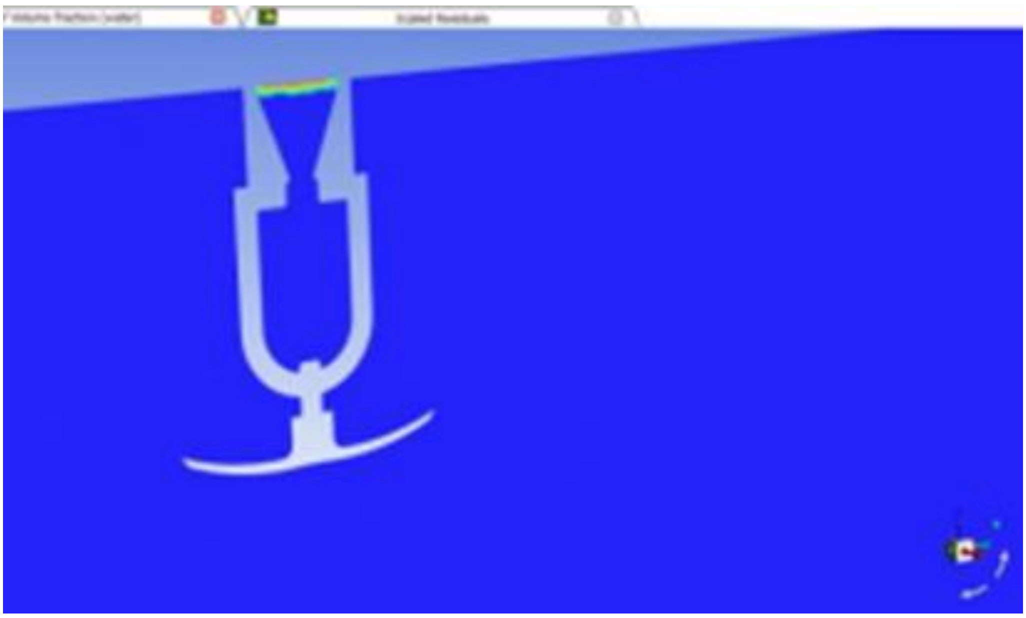


Fig. 4. Create of Plane 8 to display the water distribution in new window

3. Results and Discussion

The simulation was run for geometry of a room with the dimension 4.0m × 3.0m × 4.0m. There were 5 Grid Independence Test involved which are the element is 0.08 m, 0.07 m, 0.06 m, 0.05 m and 0.04 m. The difference percentage of GIT 4 and 5 is smallest as 2.98 %, hence the element size of 0.05 m is used for other model in simulation. The factor that choosing in empty geometry condition is because the simulation time consumed is shorter. Table 4 shows the Grid Independence Test with the element size, average velocity magnitude and difference percentage. The result for the average velocity was represented in the graph and comparison between each of element size. Figure 5 shows the graph of the average velocity for each element size.

Table 4
The Grid Independence Test Results

GIT	Element Size (m)	Average velocity magnitude (m/s)	Difference percentage (%)
1	0.08	1.532589	14.15
2	0.07	1.315678	8.66
3	0.06	1.201763	23.65
4	0.05	0.917526	2.98
5	0.04	0.890214	-

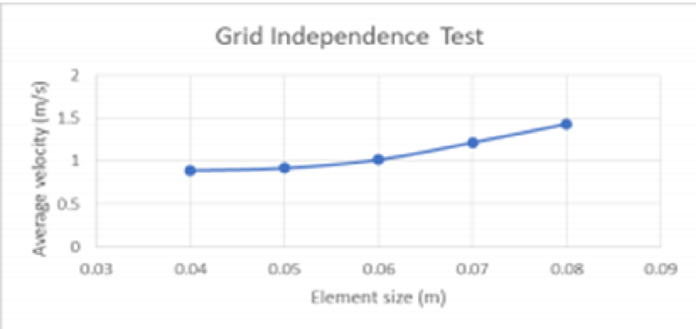
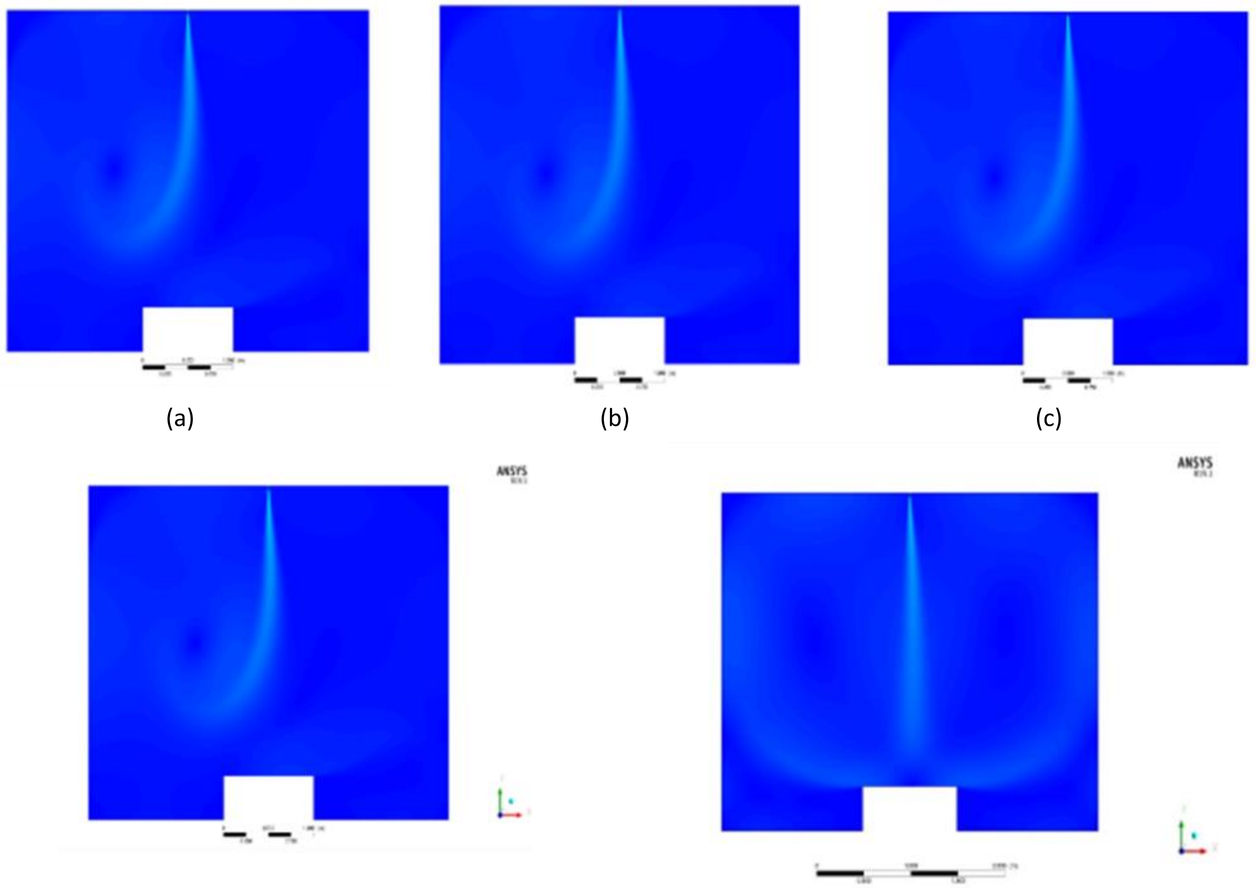


Fig. 5. Average velocity

3.1 Variation of Velocity at Different Operating Pressure

In post-processing, the contours were performed to study the water discharged behaviour of fire sprinkler. It can be noticed that the model E (15 bar) manage to dispense the water jet down to rectangular table satisfactory. For other models such as A,B,C and D, the pattern is about the same and the dispensation of water jet is not complete at bottom region near to rectangular table. The result of water discharged patterns at different operating pressure is shown in velocity contours of Figure 6.



(a) (b) (c)

(d) (e)

Fig. 6. The water discharged patterns at different operating pressure

Figure 6 (a) shows the dispersion of water by using lowest pressure setting. The results show the velocity contour for model A. The velocity produced at model A is the lowest velocity compare to others model. So that, the dispersion of water from fire sprinkler cannot reach the rectangular table at the floor of the room. Figure 6 (b) shows some improvement when the pressure of water was improved. The velocity of the water dispersion also increase and the dispersion of water almost reach the rectangular table. Figure 6 (e) show the most regular dispersion of water from the sprinkler. The dispersion of water reaches the rectangular table at the floor of the room, then it was distributing into the surrounding. Model E shows the uniform distribution water from sprinkler compared to the others model. So that, the most suitable pressure setting for fire sprinkler is as used in the model E which is 15bar.

4. Conclusions

In conclusion, based on the results obtained from the data collection, its show that the objective is achieved. From the result, the distribution of water from fire water sprinkler were determined by using the ANSYS simulation. The pattern of water distribution in room discharged from fire water sprinkler was represented in velocity contour, the figure shows that the pattern is almost the same for all models except model E. In details of study, the effect of operating pressure on water velocity discharged in room from fire sprinkler is evaluated by plotting the velocity profile. It shows that the increase of operating pressure has led to greater water velocity discharged in room. The significant raise of discharged velocity is occurred around the fire sprinkler head. In overall, the trend of velocity profile is going down with height but it shows a slight climb in striking region.

References

[1] Link, E. D., S. J. Jordan, T. M. Myers, P. B. Sunderland, and A. W. Marshall. "Spray dispersion measurements of a sprinkler array." *Proceedings of the Combustion Institute* 36, no. 2 (2017): 3305-3311. https://doi.org/10.1016/j.proci.2016.06.056

[2] Ren, Ning, Howard R. Baum, and André W. Marshall. "A comprehensive methodology for characterizing sprinkler sprays." *Proceedings of the Combustion Institute* 33, no. 2 (2011): 2547-2554. https://doi.org/10.1016/j.proci.2010.06.107

[3] Myers, Taylor, Arnaud Trouvé, and Andre Marshall. "Predicting sprinkler spray dispersion in FireFOAM." *Fire safety journal* 100 (2018): 93-102. https://doi.org/10.1016/j.firesaf.2018.07.008

[4] Beji, Tarek, Setareh Ebrahim Zadeh, Georgios Maragkos, and Bart Merci. "Influence of the particle injection rate, droplet size distribution and volume flux angular distribution on the results and computational time of water spray CFD simulations." *Fire Safety Journal* 91 (2017): 586-595. https://doi.org/10.1016/j.firesaf.2017.03.040

[5] Wang, Zhen, Wenhe Wang, and Qingsheng Wang. "Optimization of water mist droplet size by using CFD modeling for fire suppressions." *Journal of Loss Prevention in the Process Industries* 44 (2016): 626-632. https://doi.org/10.1016/j.jlp.2016.04.010

[6] Yuan, Liming, and Alex C. Smith. "Numerical modeling of water spray suppression of conveyor belt fires in a large-scale tunnel." *Process safety and environmental protection* 95 (2015): 93-101. https://doi.org/10.1016/j.psep.2015.02.018

[7] Trapp, Andrew C., and Ali S. Rangwala. "Analyzing the impact of in-rack sprinklers in a warehouse fire: A demonstration of the role optimization has in mitigating damage." *Fire Safety Journal* 73 (2015): 55-62. https://doi.org/10.1016/j.firesaf.2015.03.002

[8] Marshall, A. W., and M. Di Marzo. "Modelling aspects of sprinkler spray dynamics in fires." *Process Safety and Environmental Protection* 82, no. 2 (2004): 97-104. https://doi.org/10.1205/095758204322972744

[9] Jahn, Wolfram. "Using suppression and detection devices to steer CFD fire forecast simulations." *Fire Safety Journal* 91 (2017): 284-290. https://doi.org/10.1016/j.firesaf.2017.03.062

[10] Aghajani, Hamed, Siaka Dembele, and Jennifer X. Wen. "Analysis of a semi-empirical sprinkler spray model." *Fire safety journal* 64 (2014): 1-11. https://doi.org/10.1016/j.firesaf.2014.01.004