



Water Spray System as Reducing Negative Effect of Welding Emissions

Muhammad Ilyas Ismail¹, Mohamad Farid Sies^{1,*}, Denny Wijanarto²

¹ Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

² Departments of Mechanical Engineering, Faculty of Engineering,
Universitas Muhammadiyah Surakarta, INDONESIA

*Corresponding Author

Received 10 May 2021;
Accepted 2 July 2021;
Available online 30 Oct. 2021

Abstract: The water spray system is a type of technology where it helps to keep the Local Exhaust Ventilation (LEV) system clean when the smoke is discarded into the environment and prevent the worker from inhaling welding fumes. The objective of this research is to conduct a simulation of a filter box with water spray as a filter welding fumes based on the effects of applying different liquid pressure by using Computational Fluid Dynamics. A geometric model of a spray nozzle is designed using SolidWorks 2019 and analysed via a CFD software, ANSYS Fluent. The inlet and outlet dimension parameter of the filter box used is 124.24mm, while the orifice of the V6 nozzle is 0.25mm. The working fluids used in this simulation are carbon-monoxide, nitrogen-oxide, and water-liquid. The CFD simulation was carried out for different pressure at the V6 nozzle ranging from 1 bar to 3 bar. The amount of volume fraction for carbon-monoxide and nitrogen-oxide showing drop in number according to the amount of water-liquid volume fraction. The simulation showed that this model could be used and securely used for an indoor workshop. The results from simulation shows the two (2) bar pressure offers the most compatible water spray for the dimension of V6 nozzle. This simulation is likely to succeed because it can reduce the welding emissions before discharged to environments.

Keywords: Computational Fluid Dynamics, Exhaust Ventilation, Welding Emissions, Volume Fraction

1. Introduction

Many small and medium enterprises (SMEs) have poor working conditions that cause safety and health problems for workers. Any type of respiratory disease or lung disease affects most welders working in the construction industry, factories, mines, refining, metallurgy, rail, petrochemical, ironwork, shipbuilding, or steel industries. Toxic fumes are produced by welding processes, such as nitric oxide, carbon monoxide, ozone, and nitrogen dioxide [1-3]. To overcome welding emissions from the health of welders, a prevalent example of engineering safety equipment used to control workers' exposure to chemicals that are harmful to health is the local exhaust ventilation system.

1.1 Local Exhaust Ventilation (LEV)

The local exhaust ventilation is commonly used in the laboratory, construction, construction industry, automobile industry, and workshop to reduce or remove exposure to contaminants resulting from toxic chemicals [4,5]. The toxic chemicals produced due to work in the industry need the LEV system to isolate the worker from inhaling the chemical fumes. The most cost-effective approach for managing air pollution in industrial buildings is local exhaust ventilation systems [6]. Fig. 1 shows the basic component of the local exhaust ventilation system [7]. The LEV system consists of four basic elements, such as the hood, duct system, air cleaner, and fan as additional discharge methods.

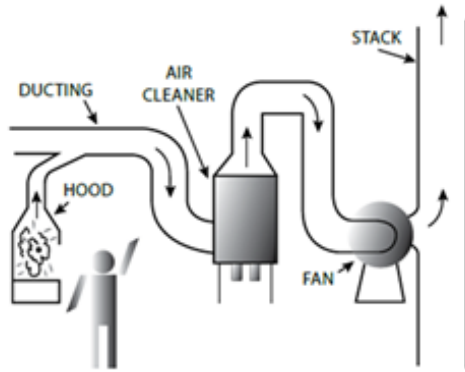


Fig. 1 – Component of LEV

ANSYS Fluent software contains the broad, physical modelling capabilities required to model flow, turbulence, heat transfer, and reactions for industrial applications. Fluent is covering an extensive range and unique models. These advantages are significant as the software can meet the criteria of a water spray system [8-10]. Therefore, objective of this research is to conduct a simulation of a filter box with water spray as a filter welding fumes based on the effects of applying different liquid pressure by using Computational Fluid Dynamics.

2. Designs and Methodology

A simulation on the filter box with a water spray nozzle requires a specific test to measure and identify the volume fraction of each fluid: air, water-liquid, carbon-monoxide, and nitrogen-oxide. The simulation will be conducted using Ansys Fluent and the model created by using SolidWorks to ensure that the necessary processes will run smoothly.

The design of the filter box with a water spray nozzle will be modelled using SolidWorks 2019 software before export into ANSYS Fluent 2020 R1 software. The filter box that was created with the same size of inlet and outlet pipes and the design of the V6 nozzle. The references of the filter box parameters are determined based on previous research for this filter. The analysis parameters consist of liquid, gas, and different ranges of inlet pressure in the V6 nozzle between 1 bar and 3 bar. The inlet velocity is the same, which is 2.7 m/s. The filter box with V6 nozzle is shown in Fig. 2 while the parameters are summarized as in Table 1 and 2.

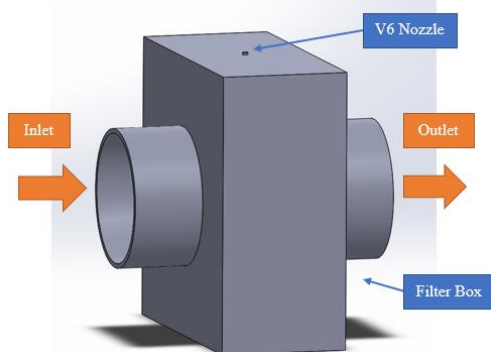


Fig. 2 – Filter box with V6 nozzle

Table 1 – Parameters of the filter box

Parameter	Measurement
Inlet pipe	76.20 mm x 142.24 mm
Outlet pipe	76.20 mm x 142.24 mm
Diameter of inlet nozzle	2.7 mm
Diameter of outlet nozzle	0.25 mm
Filter box sizes (L x W x H)	139.90 mm x 304.80 mm x 304.80 mm

In the ANSYS Fluent, to get the better meshing, the face sizing be controlled at one location: the nozzle to produce the best skewness that can affect the calculation in solution [11].

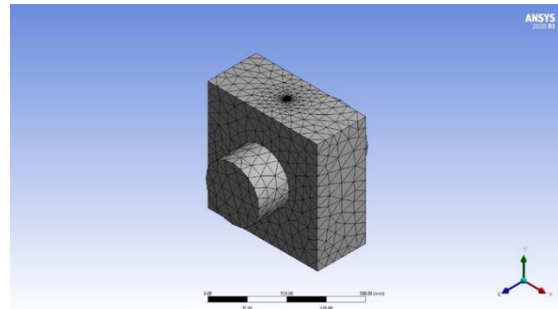


Fig. 3 – Meshing of the filter box model

The simple scheme was selected under pressure velocity coupling. For spatial discretization, least-square cell-based was choose for gradient and body force weighted has been set for the pressure. Volume fraction has been set as compressive in solution. For momentum, turbulent kinetic energy, turbulent dissipation rate, and energy in spatial discretization have been set as second-order upwind. Under relaxation factors in solution controls, the flow quantities have been set as pressure, $p = 0.3$, density, $\rho = 1$, body faces = 1, and volume fraction = 0.5 [12]. Turbulent kinetic energy and turbulent dissipation are set as 0.8, while turbulent viscosity is set as 1. The initialization method for this solution has been selected as standard initialization, which computes from Inlet 2 boundaries. In the initialization method, the patch was added in the fluid domain. The pressure has been set as 100000 Pa, 200000 Pa, and 300000 Pa for the mixture phase. The water-liquid volume fraction has been set as one, and the remainder, which is carbon-monoxide and nitrogen-oxide, has been set as zero. Underrun calculation, the number of time steps has been set as 1000, and the time step size is 2×10^{-3} to obtain two (2) seconds of time flow. The number of iterations was set as one.

3. Results and Discussion

Result and findings obtained through this research being analyzed. The result obtained is shown in the form of a graph, table, and figure. The result is explained in detail about the volume fraction throughout the filter box with the V6 nozzle. The result of the findings is compared to obtain the most efficient pressure for the V6 nozzle design.

3.1 Volume Fraction at Selected Line

The value of the volume fraction inside the filter box with the V6 nozzle must be located to take readings of the volume fraction. Several points must be discovered because the contour of the volume fraction for each end inside the filter box with the V6 nozzle is not uniform, so more points must be taken to achieve an accurate reading of the volume fraction. In this research study, all points are in the middle of the outlet pipe. A line was plotted to measure the contour of volume fraction on the plane. From there, ten samples were generated, and the data was successfully obtained. The line's location was plotted at the centre of a 76.2 mm length outlet pipe. The line also can be used to obtain others required data such as velocity. The volume fraction generated by ANSYS Fluent from the selected line inside the filter box's output pipe and two-point technique was utilised to identify the chosen line position for collect the ten data samples. Table 2, 3 and 4 shows the volume fraction at the filter box with different inlet pressure at the V6 nozzle. All the velocity and working fluid at the inlet filter box is the same.

Table 2 – Volume fraction with 1 Bar inlet pressure

Node Number	Carbon-Monoxide	Nitrogen-Oxide	Water-Liquid
1	0.422	0.423	0.024
2	0.418	0.433	0.019
3	0.410	0.447	0.010
4	0.404	0.460	0.001
5	0.406	0.442	0.005
6	0.416	0.400	0.028
7	0.432	0.353	0.069
8	0.452	0.305	0.128
9	0.470	0.272	0.180
10	0.476	0.263	0.197

Table 3 – Volume fraction with 2 Bar inlet pressure

Node Number	Carbon-Monoxide	Nitrogen-Oxide	Water-Liquid
1	0.461	0.442	0.003
2	0.424	0.419	0.019
3	0.353	0.380	0.077
4	0.283	0.342	0.158
5	0.194	0.300	0.336
6	0.161	0.286	0.443
7	0.139	0.276	0.524
8	0.166	0.290	0.473
9	0.217	0.317	0.363
10	0.247	0.332	0.294

Table 4 – Volume fraction with 3 Bar inlet pressure

Node Number	Carbon-Monoxide	Nitrogen-Oxide	Water-Liquid
1	0.421	0.372	0.048
2	0.419	0.393	0.037
3	0.415	0.427	0.023
4	0.414	0.447	0.014
5	0.417	0.429	0.018
6	0.432	0.378	0.056
7	0.463	0.298	0.146
8	0.480	0.252	0.210
9	0.489	0.232	0.253
10	0.491	0.228	0.262

Fig. 4 to 6 shows the volume fraction of working fluid inside the filter box with different inlet pressure at the V6 nozzle against the number of nodes.

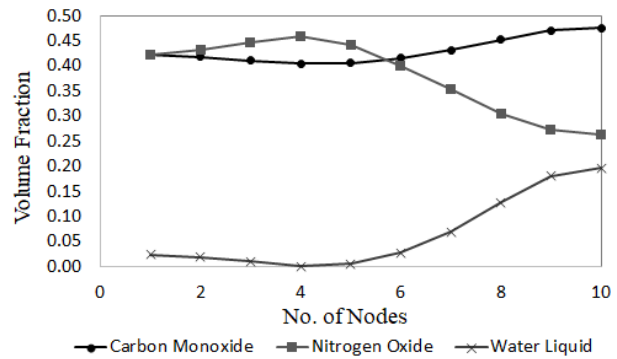


Fig. 4 - Volume fraction of working fluid Inside the filter box with 1 Bar of inlet pressure

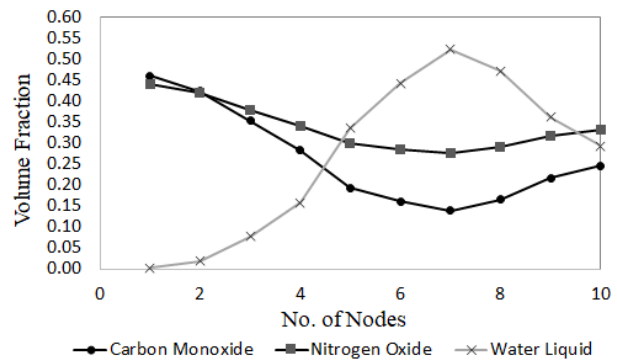


Fig. 5 - Volume fraction of working fluid Inside the filter box with 2 Bar of inlet pressure

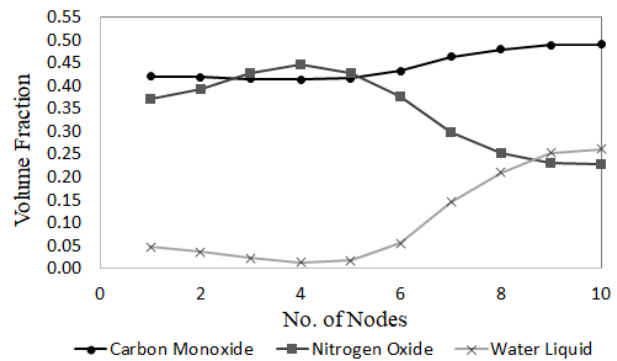


Fig. 6 - Volume fraction of working fluid Inside the filter box with 3 Bar of inlet pressure

The volume fraction of working fluid in Fig. 4 shows the lowest volume fraction of water-liquid compared to the others two graphs. The water-liquid volume fraction affects the carbon-monoxide, and the water-liquid begin with a small-volume fraction and start increasing at node 5 until 10. The volume fraction of carbon-monoxide and nitrogen-oxide decreases as node 5 is increases. Compared to the volume fraction of working fluid in Fig. 4, Fig. 5 present the highest volume fraction of water-liquid. The water-liquid volume fraction starts to increase from node 1 until node 7, then decreasing to node 10.

The figures shows that the lowest volume fraction for carbon-monoxide and nitrogen-oxide. The water-liquid volume fraction starts increasing at node 1 until node 7, the volume fraction for the carbon-monoxide and nitrogen-oxide decreases at the same node. Fig. 6 shows a similar pattern as in Fig. 4. The increasing water-liquid volume fraction may be seen from node 5 to node 10. Both patterns indicate decreased carbon-monoxide volume percentage from node 4 to node 10. Both figures show the same nitrous oxide pattern, increasing at node 5 until node 10.

Comparing the three results of volume fraction working fluid with the different pressure inlet of the V6 nozzle, Fig. 5 shows the decreasing volume fraction for the carbon-monoxide and nitrogen-oxide. This figure also gives the lowest value of the volume fraction and the highest volume fraction of the water-liquid. From all the results for volume fractions with different pressure inlets of the V6 nozzle, the 2 Bar is the most suitable for the V6 nozzle. It can be seen through the water-liquid volume fraction, which is the 2 Bar that gives the most value of volume fraction. It means that the 2 Bar has better water spray particles than the other inlet pressure of the nozzle. It also has the most negligible value of volume fraction for carbon-monoxide and nitrogen-oxide. It proved that the more water-liquid volume fraction, the less value of volume fraction for carbon-monoxide and nitrogen-oxide.

3.2 Validation

Validation for this research, the simulation has computed average velocity at the outlet of the selected line to compare with the previous experimental result. At point 8, the experimental velocity is the same location as the velocity at the selected line in this simulation. The experimental velocity is 2.71 m/s being compared by using the percentage difference formula. The percentage difference is the calculation between two different results for determining the precision of the calculation. In this research, the percentage differences that need to be calculated are between the simulation of average velocity (experimental) with the experimental velocity (theoretical) from the previous study. The equation is given by,

$$\% \text{ error} = \left| \frac{\#_{\text{experimental}} - \#_{\text{theoretical}}}{\#_{\text{theoretical}}} \right| \times 100 \quad (1)$$

Based on the equation above, it is imperative to determine the simulation result is close enough to the experimental result. From this formula, it can assume that the simulation results have an error or not. Table 5 shows the average velocity generated from the ANSYS Fluent simulation. Due to no water spray in this research and the pressure at this nozzle was not taken. It is then necessary to compare the average velocity to the 2.71 m/s experimental velocities from the prior study using the percentage difference calculation to determine whether the average velocity is higher. Table 6 shows the formula for calculating the percentage difference between two values.

Table 5 – Average Velocity for 0 Bar from simulation

Pressure (Bar)	0
	3.08
	4.76
	4.48
	3.98
Velocity (m/s)	2.78
	1.86
	1.65
	1.41
	1.09
	0.54
Average Velocity (m/s)	2.56

Table 6 – Calculations of percentage difference for average velocity with experimental velocity

Pressure (Bar)	Calculation	% different
0	$\% \text{ error} = \left \frac{2.56 - 2.71}{2.71} \right \times 100$	5.53 %

By referring the Table 6, the percentage difference for 0 bar is less than 10%. It is acceptable because the previous study for the LEV does not has a filter box on it which is the geometry model only shaped in the cylinder pipe. After putting the filter box, it will be increasing the dimension for the geometry model. Then, the velocity flow will decrease because of the area inside the filter box. Consequently, the dimensions from the filter box affect the velocity flow inside the filter box.

4. Conclusion

As for conclusion, different pressure at V6 nozzle can affect the volume fraction for carbon-monoxide, nitrogen-oxide, and water-liquid. The results show that if the water-liquid volume fraction is high, it can be reducing the volume fraction for welding fumes which is carbon-monoxide and nitrogen-oxide. 2 bar inlet pressure for the inlet nozzle can be considered as the best as it gives good results compared to the others pressure. 2 bar offers the most volume fraction for water-liquid inside the filter box and it more compatible with the dimension for the V6 nozzle. Consequently, it can be concluded that a higher volume fraction of water-liquid is needed to reduce the welding fumes. Hence, the objective stated is achieved. The percentage difference for average velocity was achieved and indirectly makes the simulation valid and carried into further simulation study. The result of percentage difference shows below 10% of average velocity compared with the previous research. It is still acceptable because the different dimensions of model geometry will affect the velocity flow inside the filter box. As for recommendations, this filter box design is highly efficient to the pressure that supplies from the nozzle at the top of the body filter box. The nozzle design is successful in its capability to transmit the different pressure towards the filter box. scale model geometry that needs to mesh in small element sizes.

Acknowledgement

The author would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Industrial and Indoor Environment Research Group (IIERG) for its support and Departments of Mechanical Engineering, Universitas Muhammadiyah Surakarta for research collaboration.

References

- [1] Kumar, P., and Mistry, J., "Impact of Welding Processes on Environment and Health," *International Journal of Advanced Research in Mechanical Engineering & Technology*. 1(1) (2015): 17–20
- [2] Yin, X., Liu, J., Pei, J., and Hou, Y., "A New Calculating Model for the Suitable Area of Air Cleaners Purifying Indoor Gaseous Chemical Contaminants." *Procedia Engineering*. 121 (2015): 1467–1474.
- [3] Zhao, J., Feng, Y., Bezerra, M., Wang, J., and Sperry, T., "Numerical Simulation of Welding Fume Lung Dosimetry." *Journal of Aerosol Science*. 135(5) (2019): 113–129.
- [4] Hasan, N.H., Said, M.R., and Leman, A.M., "Local Exhaust Ventilation and Application: A Review," *Int. J. Eng. Technology*. 12(4) (2012): 39–43.
- [5] Liu, H., Liang, Y., Bowes, S., Xu, H., Zhou, Y., Armstrong, T. W., "Benzene Exposure in Industries Using or Manufacturing paint in China – A Literature Review." 1956-2005. *Journal of Occupational and Environmental Hygiene*. 6 (2009): 659-670.
- [6] Carlsten, C., Salvi, S., Gary W.K. Wong, Chung, K.F., " Personal Strategies to Minimise Effects of Air Pollution on Respiratory Health: Advice for Providers, Patients and the Public." *European Respiratory Journal*. 55 (6) (2020): 1-21
- [7] Guideline on Occupational Safety and Health for Design, Inspection, Testing and Examination of Local Exhaust Ventilation System 2008, Dept. of Occupational Safety and Health, Malaysia.
- [8] Aiman S. H. Al-Ammari, Yazan S. M. Altarazi, Abd. Rahim Abu Talib and M. I. Nadiir Bheekhun, "CFD Simulation of Local Exhaust Ventilation for Aviation Fire-Test Lab." *International Journal of Engineering & Technology*, 7 (4.13) (2018): 188-194
- [9] Hurnik, M., Blaszcok, M., and Popiolek, Z., "Air Distribution Measurement in a Room with a Sidewall Jet: A 3D Benchmark Test for CFD Validation." *Building and Environment*. 93(2) (2015): 319-330
- [10] von Hooff, T., Blocken, B., Aanen, L., and Bronsema, B., "A Venturi-Shaped Roof for Wind-Induced Natural Ventilation of Buildings: Wind Tunnel and CFD Evaluation of Different Design Configurations." *Building and Environment*. 46(9) (2011): 1797-1807
- [11] Sies, M.F., Ismail, N.T.J., and Ramli, M.N., "Airflow Analysis of Air Conditioning System for Lecture Hall: Study Case – Block G3, UTHM." *Journal of Complex Flow* 2(2) (2020): 1-5
- [12] Guo, C., Nie, W., Xu, C., Peng, H., Zhang, C., Li, S., Yue, N., Liu, Z., Yang, S., Ma, Q., & Li, M., "A Study of the Spray Atomization and Suppression of Tunnel Dust Pollution Based on a CFD-Based Simulation. *Journal of Cleaner Production*, 276 (2020): 123632