Journal of Complex Flow, Vol. 4 No. 1 (2022) p. 12-18

FAZ Publishing

JCF

Journal of Complex Flow

Journal homepage : www.fazpublishing.com/jcf

e-ISSN : 2672-7374

Prediction on Transmission Flow Path of Human Coughs into Airborne

Goh Carmen[1], Ishkrizat Taib[1,*], Nor Azali Azmir[1], Ahmad Mubarak Tajul Ariffin[1], Nor Adrian Nor Salim[1], Suzairin Md Seri[1], NM Razif Noraini[2], Nurul Salihah Abdullah[2], Amir Abdullah Muhamad Damanhuri[3], Mohd Idain Fahmy Rosley[3]

[1] Faculty of Mechanical and Manufacturing Engineering
University Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor,MALAYSIA

[2] National Institute of Occupational Safety and Health (NIOSH)
Bandar Baru Bangi, Selangor, MALAYSIA

[3] Faculty of Mechanical and Manufacturing Engineering Technology
Universiti Teknikal Malaysia Melaka (UTeM), Durian Tunggal, Melaka, MALAYSIA

*Corresponding Author

Received 23 April 2022;
Accepted 4 May 2022;
Available online 15 May 2022

Abstract: Coronavirus disease 2019 has become a global pandemic that causes illnesses ranging from normal fever to more severe diseases to be hospitalized ordie. For understanding the mechanisms of the droplet's transmission are incomplete.This project adopts computational fluid dynamics (CFD) method to investigate the dispersion, and the transport velocities after people being cough into the air with andwithout the facemask. This project has employed 2 types of simplified three- dimensional models covering with and without a facemask. An ejection process of the saliva droplets from the mouth was to mimic the real human cough into the air. The size of the saliva droplets defines as 50-70 µm in diameter, infer the size range which may potentially be more infectious. Besides, consider the effect of the droplet's flow velocities which were 6 m/s, 14 m/s, and 22 m/s that cause the motion of the droplets in a confined room condition. The relationship between the saliva droplet flow characteristics and the dispersion of the droplet's particle velocities deposited on the areas was investigated. Hence, the results show that the higher the velocity, the higher the dispersion rate. However, for the people who wear a facemask, it provides great protection and decelerates the outspread rates. Further research is required to take account of other factors such as the environmental relative humidity, ventilation, indoor versus outdoor.

Keywords: Droplets, mask, flow characteristic, cough, dispersion, velocities

1. Introduction

In Wuhan, China, an outbreak of pandemic disease is highly contagious among the people at the end of 2019. Several weeks later, this pandemic spread out to other countries and the name of the pandemic is called "Coronavirus disease (COVID-19)". [1] Covid-19 is an illness caused by a virus that can spread primarily through droplets or aerosols of the human on a cough. Droplets or aerosols carrying the pathogen [2] are released from the respiratory tract like a mouth when the infected are coughing. The flow characteristic of cough is an important factor in the transmission of the Covid-19 to the next person [3]. Pathogens carried by droplets diffuse in the atmosphere and travel over a long distance [1].

Besides, pathogens can continue to live and remain in the atmosphere and are susceptible to others directly or indirectly. The transmission of Covid-19, droplet released with momentum and received by another individual's respiratory tract; indirectly physical contact with the droplet that remains on a surface; directly inhalation the droplets diffuse in the ambient air [4].

The velocities of the droplets are introduced into the environment that affect the transmission of the droplet [5]. Particle image velocimetry system (PIV) is being used by Zhu et al [6] to measure the velocity of cough flow. Figure 1 shows the result of the initial velocity of the cough's droplets between the range from 6 m/s to 22 m/s. The

**Corresponding author: iszat@uthm.edu.my*

strength of the cough flow behavior in the far-field can be characterized by the of peak cough velocity magnitude [5]. Therefore, the droplets released from the mouth will moving along with the initial velocity and travel up to a distance [7]. The geometry of the chamber was $3.5\ m \times 3.0\ m \times 2.5\ m$ set by Zhu et al [6]. Firstly, the droplet expelled from cough when the transmission over a distance ($> 30cm$) at high velocity, then the velocity of the droplet was reduced and gradually dispersed in the ambient air as shown in Figure 1.

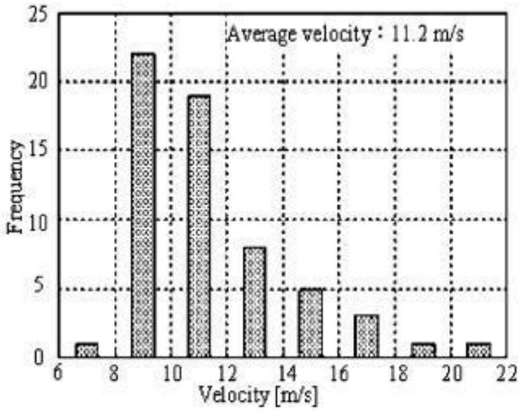


Fig. 1 – Velocity frequency distribution [6]

Wearing a standard mask is a predominate contribution to block the main path of infection of pandemics [8]. Although the mask is not fitting tightly close to the human face and the droplets still can pass through the leakage of the mask which was the top, side, and bottom but still can reduce the cough directly through the front of the mask with relative low speed [9]. It has minimal the forward momentum of the particles results in lower transmission energy into the surrounding air. The addition of a simple facemask, the respiratory droplet is mitigating hence reduce the infection risk. Therefore, the number of droplets pass through the mask had been efficiently reduced to lower the spread of the pathogen. For masks, material have a good capability to filter or stop the particle movement [8] especially large droplets will not be able to penetrate the mask and remain trapped in the areas of the surface of mask [10].

After the droplets being trapped, the particles were being absorbed by the filter of the mask. However, unmasked coughing may produce a turbulent jet with the content of the virus and spread directly to the surrounding. Furthermore, World Health Organization [11] provides evidence of suppressing the transmission of virus-laden droplets where the virus circulating in the environment, mask can be used as a barrier to minimize the infection rate. Figure 2 had shown a) cough without wearing a mask b) loose fit of mask c) cough with mask [9]. According to Zhang et al [12] have shown, by using the Lagrangian model, research the influence of the cough droplets transport in a conference room without ventilation. Brownian force, drag force and gravity of the coughed is being considered in these studies. CFD numerical simulation was the main research method.

(a) (b) (c)

Fig. 2 – (a) Cough without wearing mask (b) loose fit of mask (c) cough with mask [9]

The simulation uses unsteady state and $RNG\ k - \varepsilon$ turbulent model. Droplets to be considered incompressible when the particles with constant density. Result of coughed droplets affected by inertial force, when inertial force decreased gradually the main affected will shift to gravity drag force and Brownian force. Moreover, in Zhu et al [6] research work, the dispersion of droplet by human cough are tested in a calm indoor environment. The flow field inside the calm indoor was simulated with CFD analysis. The condition of the indoor environment was kept in steady state and standard $k - \varepsilon$ turbulent model. Aerosol with constant density assumed the type of fluid is incompressible. An indoor flow field was generated, average of 6.7 mg of the droplets was expelled with 22 m/s velocity in each cough. Result in obtaining the number of the droplets that adhered to the outlet.

The present study is to investigate the dispersion of the droplet's flow characteristics into the air with or without the mask and determine the velocities of the droplet particle deposited on the areas. By using the Computational Fluid Dynamics (CFD) to analyses the saliva's droplet flow behavior in the indoor environment. Computational modelling is used to simulate the numerous parameters by using mathematics, computer science to study the behavior of the saliva's droplet flow characteristics [13]. Through modelling the dispersion of saliva's droplets, researcher obtain a better understanding of the transfer of airborne particles [14].

2. Methodology

In this section, brief methodology will be present. The process for the simulation process starting is the data collection form previous literature, design the stirred tank reactor, meshing process, set up the boundary condition and run the simulation. The process of designing a tank agitator and impeller blades was done using the Solidwork software while the data analysis was carried out by using Ansys Fluent software.

2.1 Geometry of Case Study

A 3-D model of a room and 2 type of simplified head model covering with and without mask are generated with SolidWorks design modeler. The geometry of the room is $2\ m(length) \times 2\ m(width) \times 2\ m\ (height)$ and no ventilation are considered in the model room. In the first set up of the experiment, a simple head model without mask is assumed stay middle themodel room.

Published by FAZ Publishing
http://www.fazpublishing.com/jcf

Next, second set of the experiment, a simple head model with mask is stay middle the model room. Figure 3 shown the schematic of the room with a head model without mask while figure 4 shown the schematic of the room with a head model with mask.

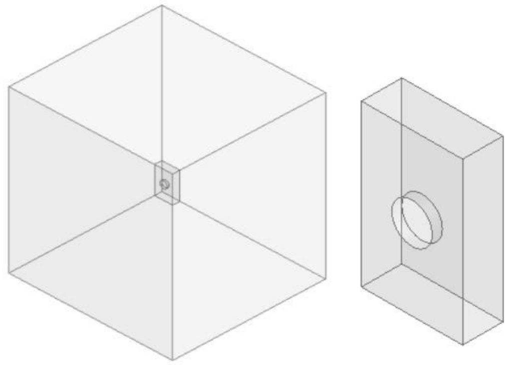


Fig. 3 – Schematic of the room with a head model without mask

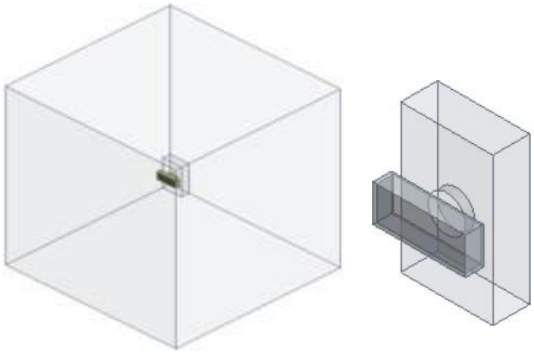


Fig. 4 – Schematic of the room with a head model with mask

2.2 Governing Equation

The dispersion of the droplets generated by coughing are modelling using a commercial software to study the percentages of the area of particles deposited on the predicted areas. Ansys Fluent 19.2, is used to simulate the dispersion of droplets. In this approach, the fundamental governing equations of the droplets flow are the Continuity, momentum, turbulent kinetic energy, k, energy and turbulent dissipation rate, $\in$ can be written as following equation [15].

$$\frac{\partial(\rho\phi)}{\partial t}+\nabla.\left(\rho\phi\vec{V}\right)=\nabla.(\Gamma\nabla\phi)+S \tag{1}$$

2.3 Discretization and Boundary Condition

The type of mesh generated was tetrahedrons. Number of grid nodes was 124970, while the number of the grid element was 504185. While the head model with facemask was meshed until the result had shown were independent of the grid. The type of mesh also generated was tetrahedrons. Number of grid nodes was 120614, while the number of the grid element was 498400. Figure 5 shows the physical domain of the room model is being meshed.

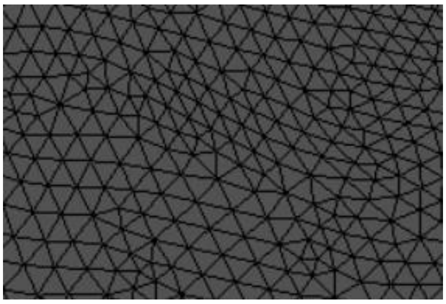
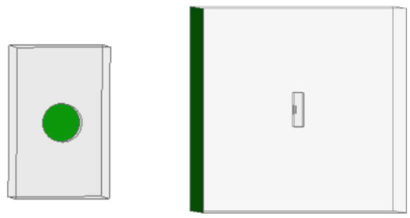


Fig. 5 – Physical domain of room model is being meshed

Figure 6 shows the inlet (mouth) of the head model and outlet of the domain. The table 1 shows the parameter assumption and boundary conditions. According to droplets expelled from human cough, the important factors that govern the flow dynamics are the exhaled initial velocities and pressure of the mouth. Droplets released by human cough are incompressible Newtonian fluids with density 1.225 kg/m^3 behavior associated with constant density. The unsteady state is used as an input simulation of droplet travel based on a discrete phase model. Droplets are assumed to be inviscid flow to simplify the simulation. Droplets have different velocities when expelled from the mouth.

The inlet velocities boundary of mouth is range between 6-22 m/s [6] with a constant mass flow rate 2.4e-9 kg/s [6]. The cough jet velocity varies with time. The droplets are injected into the simulation domain using 3 different velocities which are 6m/s, 14m/s, and 22m/s. In this research, the domain defines those 5 walls as far-field region and ground as a non-slip wall. The walls of a domain are far-field, pressure is equal to 0 while the ground assumes as the non-slip wall boundary condition of the velocity of wall particles is 0m/s. The distance of particles travel depends on the velocity of particles.



(a) (b)

Fig. 6 – (a) Inlet (mouth) of the head model (b) Outlet of the domain

Table 1 – Parameter assumption and boundary condition

Parameters	Values
Size of the domain	2.0m x 2.0m x 2.0m
Type of fluid	Droplet's particles
Size of droplets	50-70μm
Density	1.225 kg/m3
Initial Velocity (Mouth)	6m/s, 14m/s and 22m/s
Flow Rate	2.4e-9kg/s
Temperature	36°C
Room	Far-field

Published by FAZ Publishing
http://www.fazpublishing.com/jcf

3. Results and Discussion

3.1 Grid Independency Test

To vary the mesh element size and checking the output result for each mesh can be found out from the Grid Independence test (GIT). When varying the mesh of the domains does not affect the result much then can select the minimum mesh size for the final solution output to get the accepted level of tolerance. The result of the grid resolution as shown in Figure 7 representation the data of velocity magnitude near horizontal line of the mouth with 90K, 100K, and 124K nodes. Mesh analysis facilities in the decision of how fine mesh should be used to obtain results with satisfactory accuracy. Green line representative 124970 of grid nodes, while the number of grid element is 504185 is being chosen for the simulation. The result of the grid resolution as shown in Figure 8 representation the data of velocity magnitude near the vertical line of the mouth covering by mask with 95K, 116K, and 122K nodes.

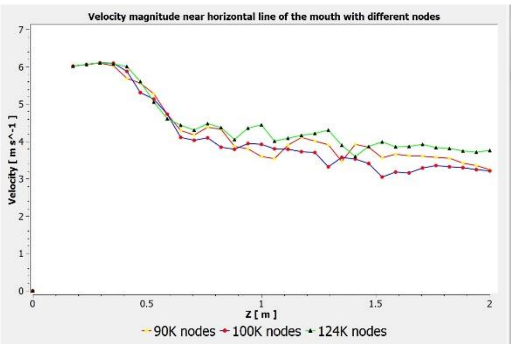


Fig. 7 – Velocity magnitude near horizontal line of the mouth with 90K, 100K and 124K nodes

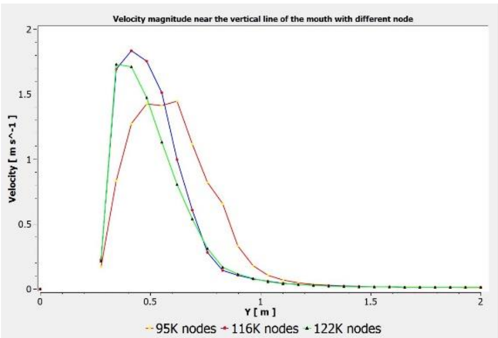


Fig. 8 – Velocity magnitude near vertical line of the mouth with 95K, 116K and 122K nodes

3.2 Validation of the Results

From the previous study [17], 3 sinusoidal function curves are plotted with different mesh. Figure 9 shows the respiratory air velocity magnitude near the vertical line of the mouth with fine mesh, medium mesh, and coarse mesh. From the figure, fine mesh, and medium show both curves are almost similar. While the course mesh is slightly difference from the medium mesh and fine mesh. Therefore, we can say that medium and fine mesh have more accurate results as compared to coarse mesh.

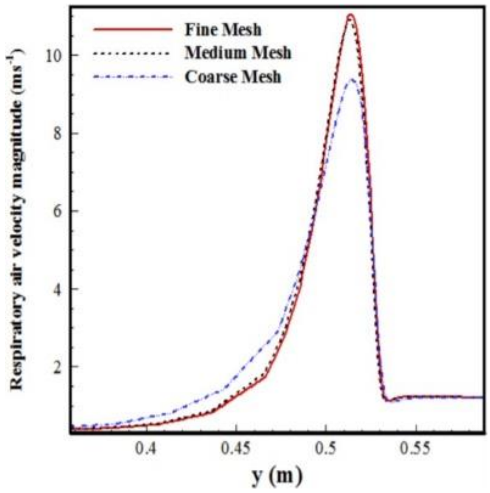


Fig. 9 – Respiratory air velocity magnitude [17]

The CFD simulation results with image of the dispersion of the droplet'sflow characteristics after people cough without and with mask at an initial velocity 6m/s, 14m/s, 22m/s shown in Figure 10 and 11 respectively.

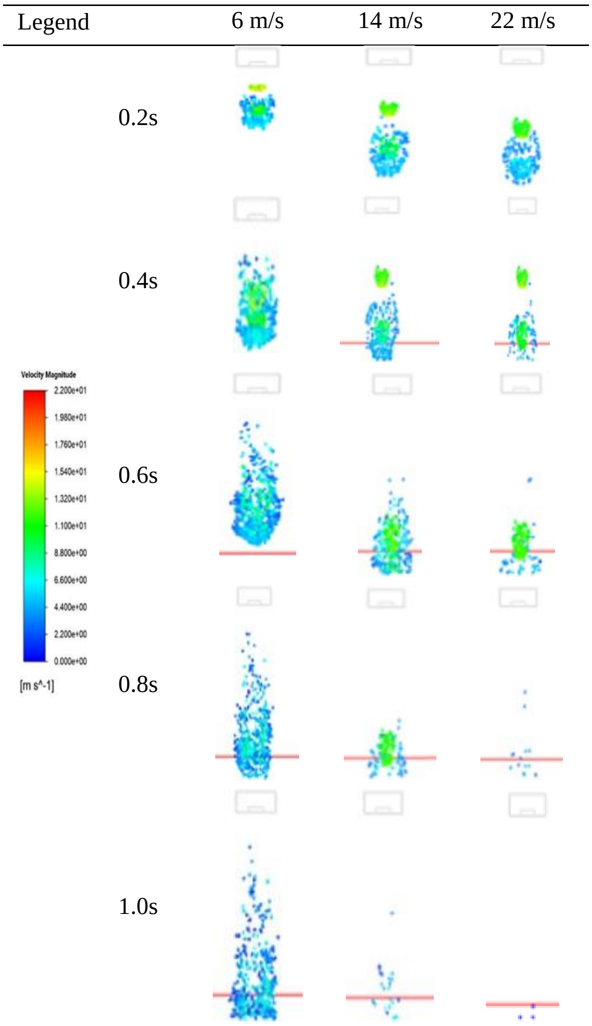


Fig. 10 – Dispersion of the droplet's flow characteristics after people cough without mask

Published by FAZ Publishing
http://www.fazpublishing.com/jcf

Regarding the time from 0.2 s to 1.0 s refer to thetime when the people cough ends. The cough flow direction was visualized through the slowest to the highest speed in the present simulation. Figure 10 illustrates clearly the distances reached horizontally. It is noticeable when t = 1.0 s at an initial velocity of 22 m/s, almost all the droplets are dispersed toward the outlet of the domain. While Figure 11 also shows that when t = 0.2 s at an initial velocity of 6 m/s, droplet particles are trapped in the mask and start to deviate far from face seal leakage as the time across.

Legend	6 m/s	14 m/s	22 m/s

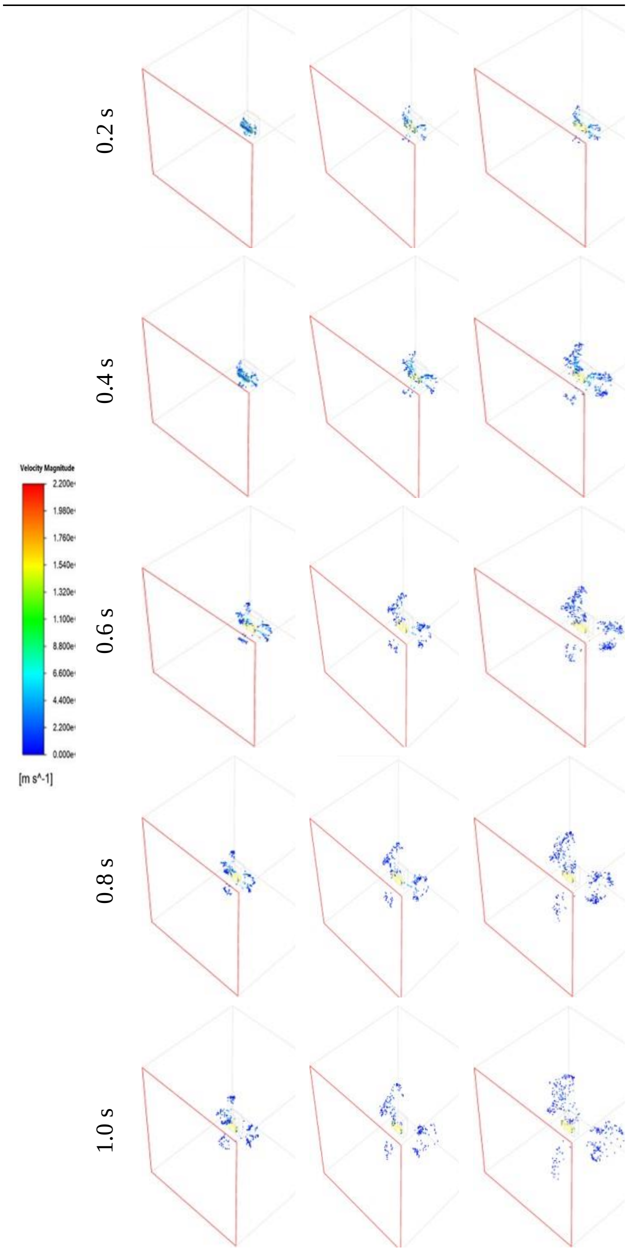
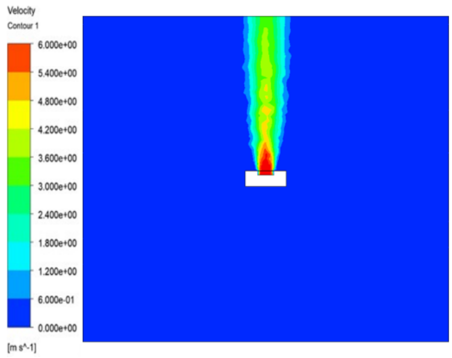


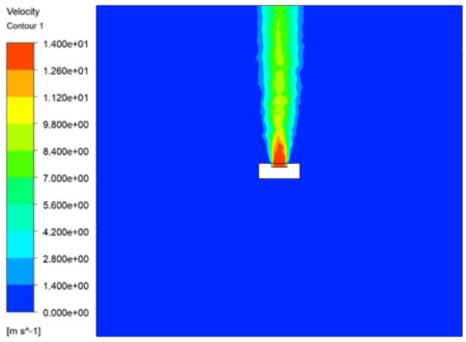
Fig. 11 – Dispersion of the droplet's flow characteristics after people cough with mask

3.3 Velocity Contour with and without Mask

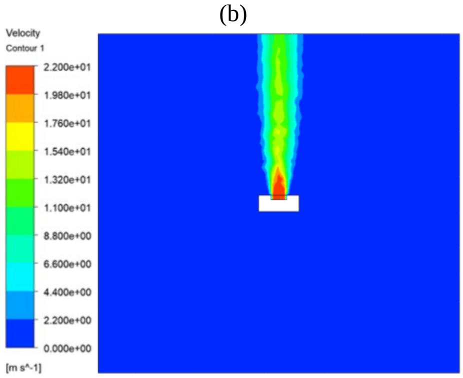
The velocity of the cough flow is substantially reduced as going far from the inlet of the mouth. Figures 12 and 13 show the velocity contour with 6 m/s, 14 m/s, and 22 m/s test cases during the room simulation in ZX and YX planes respectively. Velocity field notes that at higher velocities, the higher the turbulence as well. Droplets expelled from the mouth andformed the vortical structure in an indoor space. As time increases the vortex being created when the main flow exhausted from the mouth. Droplets trapped in the flow circulation inside the domain. The airflow coming from the mouth generates a vortex that initially drive droplets perpendicularly to the main flow. The position of the vortex heading the main flow of the droplets depends on the velocity. Therefore, the highest the velocity of the airflow coming from the mouth, the drastically affects the transport of the droplets exhausted from the mouth.



(a)

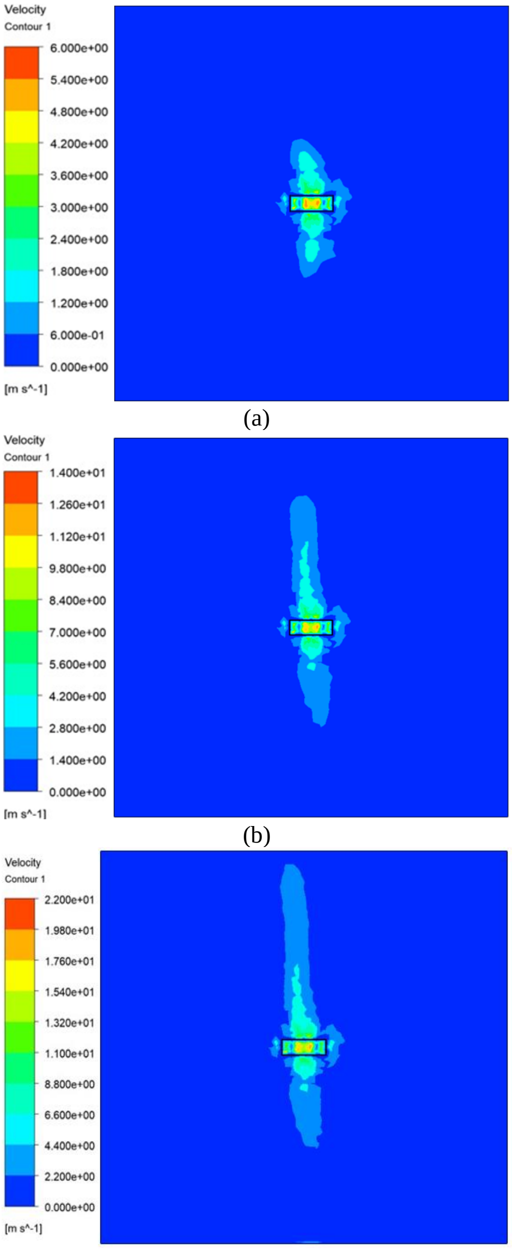


(b)



(c)

Fig. 12 – Velocity contour of a) 6 m/s, b) 14 m/s, and c) 22 m/s test case of the simulation without mask



(c)

Fig. 13 – Velocity contour of a) 6 m/s, b) 14 m/s, and c) 22 m/s test case of the simulation without mask

3.4 Comparison with and without Mask

The line graph depicts the velocity of the droplet particles deposited on the area with and without mask shown in Figure 14. From the figure, the case of wearing a mask demonstrates the velocity of thecough flow in the z-direction is substantially reduced by the presence of a mask. Covering with mask able to resist high velocity flow through the media. Whereas the case of without wearing mask shows the decrease of the velocity of the droplet as the droplets go further. Besides, the fluctuation of the blue line representative the unsteady flow of droplets dispersion in the present simulation. In the current study, the distance reached by the particles, with a focus on the various velocities and the dispersion of the droplet's flow characteristics after the people cough into the air with and without a mask, was investigated.

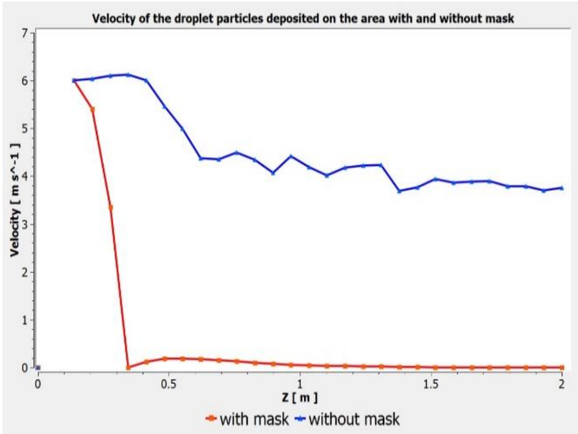


Fig. 14 – Velocity of the droplet particles deposited on the area with and without mask

4. Conclusion

In this present study, CFD simulation based on Lagrangian models was presented. There are 2 types of simplified head models covering with and without a mask. The behavior of the droplets due to cough was analyzed. The proposed simplified models can predict the droplet's flow characteristics after the people cough into the air with mask and without mask reasonably well. In this regard, it has been shown that the model cover with a mask was very efficient to reduce the velocity of the droplet dispersion. Also, it is significant to note that face seal leakage will decrease the performance. The purpose of simulated with different velocity magnitude by covering with and without mask deposited show higher velocity will speed the rate of transmission to the area. Wearing a mask, will provide greater protection to the wearer as the mask blocks the bulk of droplet released from another person and further decelerates the outspread of the particles.

Acknowledgement

The support of the Ministry of Education under the FRGS/1/2018/TK03/UTHM/03/ and the Universiti Tun Hussein Onn Malaysia (UTHM) grant number K058. This paper also acknowledged group research under National Institute of Occupational Safety and Health (NIOSH) lead by Mr. Razif.

References

[1] Peng, S., Chen, Q., and Liu, E., "The Role of Computational Fluid Dnamics Tools on Investigation of Pathogen Transmission: Prevention and Control," *Science of The Total Environment*. (2020): 23

[2] Diwan, S.S., Ravichandran, S., Govindarajan, R., and Narasimha, R., "Understanding Transmission Dynamics of COVID-19-Type Infections by Direct Numerical Simulations of Cough/Sneeze Flows," *Transactions of the Indian National Academy of Engineering*. (2020): 7

[3] Andrio, A., "Computational fluid dynamic (CFD), air flow-droplet dispersion, and indoor CO2 analysis for healthy public space configuration to comply with COVID 19 protocol" *Indoor Air* (2020): 9.

[4] Rajat, M., Rui, N., and Jung-Hee, S., "The flow physics of COVID-19," *Journal of Fluid Mechanics* (2020): 14.

[5] Sarah, G., "See how a sneeze can launch germs much farther than 6 feet," 17 April 2020. [Online]. Available: https://www.nationalgeographic.com/science/2020/04/coronavirus-covid- sneeze-fluid-dynamics-in-photos/. Scienc

[6] Shengwei, Z., Shinsuke, K., and Jeong-Hoon, Y., “Study on transport characteristics of saliva droplets.” *Building and Environment* (2005): 12.

[7] Mohammad, R.P., and Jose, C.P., "Numerical modeling of the distribution of virus carrying saliva droplets during sneeze and cough" *Physical of Fluids* p. 19, 2020.

[8] Christian, J.K., and Rainer, H., "Fundamental protective mechanisms of face masks against droplet infections" *Journal of Aerosol Science* (2020).

[9] Julian, W.T., Thomas, J.L., and Brent, C., "A schlieren optical study of the human cough with and without wearing masks for aerosol infection control," *Journal of The Royal Society Interface* 2020.

[10] Wayne, S., Scott, L., and Jessica, S., "Reducing aerosol dispersion by high flow therapy in COVID-19: High resolution computational fluid dynamics simulations of particle behavior during high velocity nasal insufflation with a simple surgical mask," *Journal of the American College of Emergency Physicians Open* 2020.

[11] "Coronavirus disease (COVID-19): Masks," 1 December 2020. [Online]. Available: https://www.who.int/emergencies/diseases/novel-coronavirus-2019/question-and-answers-hub/q-a-detail/coronavirus-disease-covid-19-masks, *Journal of Pediatrics & Neonatal* Biology 2020

[12] Yixian, Z., Guohui, F., Zhiqiang, K., and Yilin, C., "Numerical Simulation of Coughed Droplets in Conference Room," *Procedia Engineering*. (2017): 302-308.

[13] "Computational Modeling," 20 May 2020. [Online]. Available: https://www.nibib.nih.gov/science-education/science-topics/computational modeling. Epilepsy.

[14] Talib, D., and Dimitris, D., "On coughing and airborne droplet," p. 11, *Physics of Fluids* 2020.

[15] Jinliang, W., and Tin-Tai, C., "Numerical investigation of influence of human walking on dispersion and deposition of expiratory droplets in airborne infection isolation room," *Building and Environment*. 46(10) (2011): 1993-2002, 2011.

[16] Guo, H.F., Xin Ying, L., and Yi, Z., "Numerical Study of the Respiratory Aerosols Transportation in Ventilated Classroom," *Applied Mechanics and Materials* (2012): 204–208

Published by FAZ Publishing
http://www.fazpublishing.com/jcf