A STUDY OF THE CIRCULATION OF COVID-19 VIRUSES FLOW IN A PARTICULAR SPACE

MOHAMAD AKMAL BIN MOHD SHUKRI

UNIVERSITI SAINS MALAYSIA

2020

A STUDY OF THE CIRCULATION OF COVID-19 VIRUSES FLOW IN A PARTICULAR SPACE

By:

MOHAMAD AKMAL BIN MOHD SHUKRI

(Matrix No.: 137826)

Supervisor:

Dr. Nur Hidayah Binti Mansor

July 2021

This dissertation is submitted to

Universiti Sains Malaysia

As partial fulfilment of the requirement to graduate with honors degree in

BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering

Engineering Campus

Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed……………………………………………(Mohamad Akmal Bin Mohd Shukri)

Date………………………………………………………………… (11/7/2021)

STATEMENT 1

This thesis is the result of my own investigations, except where otherwise stated.

Other sources are acknowledged by giving explicit references.

Bibliography/references are appended.

Signed……………………………………………(Mohamad Akmal Bin Mohd Shukri)

Date………………………………………………………………… (11/7/2021)

STATEMENT 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed……………………………………………(Mohamad Akmal Bin Mohd Shukri)

Date………………………………………………………………… (11/7/2021)

ACKNOWLEDGEMENT

This thesis is finally completed as a result of continuous help and support from many people and party who were involved with this project, directly and indirectly. Therefore, I would like to express my sincere gratitude firstly to the School of Mechanical Engineering, Universiti Sains Malaysia for the opportunity given to me to enroll and complete my higher education in Mechanical Engineering through this four-year course offered here.

I would also like to state my utmost appreciation and acknowledge the assistance of my supervisor, Dr. Nur Hidayah Binti Mansor. The helpful guidance and insight given by her is very valuable towards the progress and completion of this research.

Besides that, a big thank to the staff of Computer Application Laboratory of Mechanical Engineering School, Mr. Jamari Sadli for allowing me and other students to use the available computers in the lab through remote access for 24 hours a day, seven days a week for the whole semester. The ability to use the lab at any convenient times greatly helps me to do required simulations for this research at my own comfortable pace.

Last but not least, I would very much like to thank my beloved parents and family members for their endless love, prayers and encouragement. Their continuous support helps me a lot through my ups and down during the conduction of this research.

Table Of Contents

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

3D	3-Dimensional
CFD	Computational Fluid Dynamic
COVID-19	Corona Virus Disease 2019
DPM	Discrete Phase Modelling
FYP	Final Year Project
HEPA	High-Efficiency Particulate Air
HVAC	Heating, ventilation and air conditioning
LAMMPS	Large scale atomic/molecular massively parallel simulator
MD	Molecular Dynamics
PIV	Particle Image Velocimetry
PTAC	Packaged Terminal Air Conditioner
PV-PE	Personalized Ventilation And Personalized Exhaust
RH	Relative Humidity
SARS	Severe Acute Respiratory Syndrome
WHO	World Health Organization

ABSTRAK

Situasi pandemic COVID-19 telah berlarutan selama hampir dua tahun tanpa menunjukkan sebarang tanda untuk berhenti dalam masa yang terdekat. Maka, amatlah penting untuk para saintis, jurutera dan juga para penyelidik untuk bersama-sama melakukan pengajian yang mapan dan berterusan berkaitan dengan perihal virus COVID-19, untuk mencari penawar dan juga langakah pencegahan yang sewajarnya untuk membendung jangkitan virus ini kepada orang ramai dalam kadar yang segera. Penyelidikan tentang peredaran virus COVID-19 di dalam sesuatu ruang adalah amat penting untuk meningkatkan pemahaman terhadap cara penyebaran virus tersebut. Maka, kajian ini dijalankan bertujuan untuk menggunakan program pengiraan dinamik cecair (CFD) seperti program Ansys Fluent untuk mencipta dan mensimulasikan satu model yang efektif untuk melambangkan mekanisma batuk oleh pesakit COVID-19 dan juga partikel-partikel yang tersebar di dalam sesuatu ruang untuk mengkaji dengan lebih mendalam tingkah laku virus-virus tersebut dan juga cara pergerakan mereka. Hasil kajian mendapati bahawa titisan-titsan daripada batuk tersebut boleh dibahagikan kepada dua jenis titisan. Jenis titisan pertama adalah titisan pernafasan dengan saiz antara 5µm hingga 10µm. Jenis titisan kedua adalah titisan nuclei ataupun aerosol dengan saiz yang lebih kecil daripada 5µm. Berdasarkan simulasi yang telah dijalankan, corak penyebaran titisan pernafasan yang besar mengikut rapat arah ke mana batuk dilepaskan oleh pesakit COVID-19. Aerosol pula mempunyai corak penyebaran yang amat terjejas dengan aliran udara di dalam sesuatu ruang dan juga pergolakannya. Langkah yang palik efektif untuk mencegah penyebaran virus COVID-19 yang disebabkan oleh titisan pernafasan yang besar adalah dengan megelakkan diri daripada berada di hadapan orang yang mungkin dijangkiti virus tersebut jikalau individu tersebut bernafas secara kuat, batuk ataupun bersin. Manakala untuk penyebaran virus yang disebabkan oleh titisan nukeli atau aerosol yang kecil pula, pihak berkaitan hendaklah memastikan sistem pengudaraan di dalam sesuatu ruang adalah efektif untuk mengelakkan virus tersebut daripada tersebar luas, di samping mengamalkan pemakain pelitup muka agar tidak menghirup udara yang mungkin membawa virus COVID-19 tersebut.

ABSTRACT

COVID-19 pandemic situation have been ongoing for almost two years now, and the infection cases is not showing any sign of stopping just yet. Thus, it is very important for the scientists, engineers, and researchers alike to continue the comprehensive study to find both the cures and the most effective prevention method to help mitigate the continuous infection cases of COVID-19 viruses among the people. Studying the circulation of COVID-19 viruses flow in a particular space is necessary to further understand the spreading mechanism of the viruses. Therefore, this research aims to utilize the computational fluid dynamic (CFD) simulation software which is Ansys Fluent, to developed and simulate an effective model of cough mechanism of individuals infected with COVID-19 and the particles releases alongside the cough jet inside a confine space to study the behaviour of the particles and track their movement. The final results of this research concluded that, the cough droplets from infected person consist of two types of droplets, namely respiratory droplets (5μm to 10μm size) and droplets nuclei or aerosols (smaller than 5μm). Based on the conducted simulation, the spreading pattern of coarse respiratory droplets is highly directional, that is it follows the direction of the infected person cough jet. The aerosols however, would be greatly affected by the airflow and it's turbulence in a space. The most effective way to avoid the transmission of COVID-19 viruses through respiratory droplets is through avoiding sitting or standing in front of infected person if the person are performing any vigorous respiratory activity such as heavy breathing, coughing or sneezing. To prevent the viruses spread through aerosols, careful planning of air ventilation and practicing wearing face mask would be the most effective method to prevent inhalation of the viruses possibly present inside the space.

CHAPTER 1 INTRODUCTION

1.1 Overview of COVID-19 Diseases

COVID-19 is a short term for the Corona Virus Disease 2019, which is a very infectious disease that when infected, the patient can experience mild to moderate respiratory illness and may recover without the need for special medical treatment. But for the older people and people who have underlying medical conditions such as diabetes, cancer or any chronic respiratory disease like asthma, occupational lung disease and others have a very high risk to develop serious illness from this virus that could eventually be fatal. According to the statistics available online, the virus COVID-19 has already infected about 51.5 million people and had killed 1.27 million of them as of now, and the graphs is unlikely to flatten out in the near future just yet. This proves how deadly the virus is and further support the needs for more academic research in order to facilitate the effort of preventing further deaths and infection of this virus to all the people around the world.

According to World Health Organization (WHO), the virus spreads primarily through small liquid particles that is saliva or discharge from the nose of already infected person that cough, sneezes, or breathe heavily. These liquid particles are in different sizes. The larger liquid particles are referred to as “respiratory droplets” while the smaller one are called “aerosols”. Infection through respiratory droplet usually happen when people are in direct or close contact (less than 1 metre distance) with the people who are infected by the virus. Infection through aerosol transmission however, can happen in other type of situations where the people are not even in direct contact with each other. The settings for aerosol type transmissions are such as in indoor locations, where it is crowded and there is no proper and sufficient air ventilation in the spaces. Aerosol type transmission is also known as airborne transmission.

In this Final Year Project (FYP), the circulation of COVID-19 viruses flows in a particular space, specifically when an infected person is coughing and sneezing is going to be studied and analyse. By studying and analysing the circulation of the viruses flow, hopefully a better prevention method of the virus infection can be made so that we can recover from this global pandemic. Related research paper on air-flow in a room by air-conditioner or wind is going to be studied. This project also requires the study of COVID-19 flow and particle involved from various reliable resources while also conducting the simulation of the flow, to find out how far the particle of the virus can spread.

1.2 Project Background

COVID-19 cases is constantly increasing right now with no vivid indication of stopping anytime soon. The virus is very infectious just like other previous case with severe acute respiratory syndrome (SARS) outbreak before. According to the WHO (WHO et al., 2020), there are many identified possible modes of transmission for the spread of COVID-19 viruses which include contact and droplet transmission, airborne transmission, fomite transmission and others. This project is going to be focus on the contact and droplet and also the airborne transmission mode of the virus particle by means of modelling and simulating the flow of the virus particle inside a properly defined ventilated space or room.

Just like other SARS disease, COVID-19 virus is mainly spread by the infected person through their various sizes of small liquid particles produce as a product of their respiratory activities, such as breathing, coughing and sneezing. To better understand how the virus can spread, the mechanism that makes the spread possible, which means the mechanism of breathing, coughing, and sneezing of a human must first be investigated. For example, according to research done by Gao and Niu (Gao & Niu, 2006), they estimated the normal breathing process of their human model in the simulation to have frequency of 17 times per minute under light physical work with an air rate of 0.141 litre/second. Having a sufficient information and parameters data about the respiration process is important in order to design an accurate model of the virus flow.

One of the biggest issues related to this project is the selection of suitable model and parameters to be considered for the components of particle and the ambient condition surrounding them that would heavily affect the final results of the particle flow simulation. There are so many factors that can be considered if one is going to model an accurate representation of particle flow as in the real world. Based on the research done by Gao and Niu (2006), Bourouiba (2020), and Zhang et al. (2019), factors that may be considered into the simulation are such as the ambient temperature, humidity, airflow, droplet size, droplet momentum, the water content inside the droplets, density of droplets, exhaled air temperature and the multiphase of the liquid particles generated through sneezing and coughing process. To include all the parameters into the simulation would required complex formula and would also be very time-consuming. Thus, only the most important parameters will be chosen with some suitable assumptions will be made in order to compensate for real life situation of the actual particle and fluid flow. Results from the simulation such as the distance of the virus particle spread, the lifetime of the respiratory droplets and the fraction of inhaled particle inside an exposed person can then be obtained.

1.3 Problem Statement

In the past, related works were concentrated on the respiratory droplet from previously established SARS viruses. Recent works considered the new viruses which is COVID-19 in their study (Bourouiba, 2020; Chen, 2020). Because of the COVID-19 is a new viruses, therefore the data and research done on them are still insufficient. Many research papers available on this topic now are not able to completely confirm the infections happen in their study are really due to airborne transmission or other modes of transmission when patient coughing and sneezing because of insufficient data and research done. The main goals of this paper are to facilitate and contribute to the findings of the current research done by all the paramedics and researchers alike, to further confirms or reject the possibility of airborne transmission of COVID-19 virus by studying and analysing the virus flow circulation in a particular space when patient coughing and sneezing to help prevent the infection.

1.4 Objectives

1. To develop and simulate an effective model of the cough mechanisms of humans which takes into account the presence of COVID-19 particles in the respiratory droplets.

2. To develop a simulation model of a ventilated room with proper fluid dynamic parameters inside a CFD software in order to find out the trends of the flow of COVID-19 particles presence in the respiratory droplets.

1.5 Scope of Work

The phenomenon of a person sneezing and coughing is going to be investigated based on the findings obtained by thoroughly studying the previous research papers done by multiple researchers before. By investigating the phenomenon, important data such as the duration of the sneezing and coughing process as well as the size and particle content of the respiratory droplet as a result of coughing and sneezing process can be obtained. All this information are important in order to develop an accurate model of the sneezing and coughing in upcoming simulation process. Then, by using a computational fluid dynamic (CFD) software such as the Ansys Fluent software, the circulation of COVID-19 viruses particle that flow alongside the sneezing and coughing respiratory droplet in the space of the event happening will be simulated and studied. The space for the event of the coughing and sneezing is chosen to be a properly ventilated room with displacement ventilation system as the air-condition property. Based on the results of the simulation, MATLAB software will be used to analyse and study the flow of the COVID-19 particles inside the space in order to find out how far the virus and related particle involved can spread.

1.6 Project Requirements

This project is a simulation based, thus no special experimentation instruments is needed. Instead, this project would require the use of a computer that are able to run analytic and engineering software, and the specific software that is needed to be used is a computational fluid dynamic (CFD) software such as the Ansys Fluent.

Ansys Fluent is a type of CFD software available that is very useful to be used for fluid simulation. The software could provide a fast solution for virtually any fluid or multiphysics application with high accuracy and robustness. Ansys Fluent is very useful for predicting fluid flow, solving heat and mass transfer problem, chemical reactions and other related phenomena.

CHAPTER 2 LITERATURE REVIEW

2.1 Current Situation of Pandemic

The COVID-19 virus outbreak starts in China. Thus, many early research papers were published by local researchers there. Three separate works by three different group of researchers in China, which are Chan et al. (2020), Zou et al. (2020), and Hu et al. (2020) reported many similar cases where the individuals tested positive for the COVID-19 virus shows no identifiable symptom at all like many more other infected individuals. According to online articles from WHO official website (*Coronavirus Disease (COVID-19)*, 2020), common symptoms shared by many people who got infected and tested positive for the virus include having a fever, dry cough and suffered from body tiredness. Other less common symptoms are symptoms such as body aches or pains, sore throat, diarrhoea, headache and many more. Infected individuals that shows no symptom at all are referred to as asymptomatic individuals. Rothe et al. (2020) research confirmed the virus transmissions from asymptomatic carriers have been identified.

As stated before, COVID-19 viruses is transmitted through the small liquid particles produce by infected person through their respiratory process which is breathing, sneezing and coughing. According to WHO (WHO et al., 2020) on their official website, the liquid particles with diameter larger than 5 to 10μm are called "respiratory droplets" while the smaller liquid particle with diameter of smaller than 5μm are called "droplet nuclei" or mostly known as "aerosols". Smaller aerosols particles is the one that is responsible for the airborne transmission in a particular space, as their smaller size cause them to more likely follow the airflow, which results in very small Stokes number (B. Zhang et al., 2019). The airborne aerosols that contain the COVID-19 virus would cause infection to a person if it is inhaled into the body. The larger respiratory droplets which contain the virus is also infectious to a person in contact with the droplets if they touch their mouth, nose or eyes afterwards without cleaning their hand first. Gao and Niu (2006) in their research paper also states that the initial size of the large respiratory droplets from humans mouth or nose can decrease in diameter after they are deposited to the ambient air due to evaporation process. Their diameter can continue to decrease until their diameter reach smaller than 5 to 10µm, which means that they are now becoming aerosols. Gao and Niu (2006) continue to state that when their size is small enough, they can be suspended in the air due to aerodynamic drag force (determined by Stoke's law) that can easily overcome the gravity force, which correspond to the research by Zhang et. al. (2019).

A research work done by Zhu et al. (2020) established the presence of COVID-19 viruses in the respiratory tract of investigated patients. This research thus categorized COVID-19 in similar group as other previous viral respiratory diseases that as the name implies, the illnesses were caused by dangerous viruses that share similar traits and affect humans upper respiratory tracts. Recent study on influenza, which is also a type of viral respiratory disease has been confirmed that the viable virus can be emitted by an already infected individual through normal respiratory activity such as breathing and talking, without the needs to cough or sneeze (J. Yan et al., 2018). As the COVID-19 is a very transmissible viruses compared to other viral respiratory diseases prior to this, it is very likely that the infection is possible without coughing or sneezing activity by an infected person which follows the case stated before with previously similar diseases.

Research made by Asadi et al. hear that, because there is no definitive measures on the outreach of the COVID-19 virus particles, due to various complicated unknowns that would affect the actual transmission route and longevity of the viruses aerosols, a more cautious approach to plainly inform the public that every individual can emits potentially infectious aerosols even without violent respiratory activity such as coughing and sneezing is deemed necessary (Asadi et al., 2020).

2.2 Research on sneezing and cough jet and their emitted particles

The particle flow simulation made by Gao and Niu (2006) more closely represent the fine droplets flow, for those with aerodynamics diameter less than 2.5µm as they utilise the tracer-gas diffusion analysis to do the simulation. Simulation by Zhang et. al.(2019) involves the study of the particle spread in cough only without the inclusion of sneezing process inside the ventilated space unlike Gao and Niu (2006) which includes both sneezing and coughing. But, Zhang et. al. (2019) were able to modelled a simulation that include both the larger respiratory droplets and smaller aerosols particle by using Lagrangian-Eulerian model. Both of the research papers shows some similar results that concluded that the cross-infection to the other person due to high velocity and long transport of pathogen containing particles from sneezing and coughing process is highly directional.

Figure 2.1: Gao and Niu (2006) smoke visualization of exhalation flow from mouth.

Research made by Chen (2020) involves the determination of the effects of changing ambient temperature and humidity on the water droplet lifetime in the ambient condition. Different droplet size effect on their lifetime will also be investigated. Even though this study didn't imply that the COVID-19 virus will be deactivated when droplet lifetime end, the effects of the relative humidity and temperature on the virus carrying drop will provide useful guidelines when considering the indoor air quality settings and HVAC operating condition settings for private or public spaces such as schools, malls and even hospitals. Generally, the study shows that the droplet lifetime will be longer when the relative humidity is high. Larger size of droplet also increase the lifetime, as larger size of droplet means there are more moisture content inside the droplet and would take longer time to evaporate the droplet. Bourouiba (2020) in his research stated that larger droplet would settle on the surfaces in a room faster than they would evaporate, whereas the small droplet would evaporate faster than they would settle on surfaces. His study also emphasize the importance of considering the warm and moist atmosphere within the turbulent gas cloud from human in the droplet flow studies. This is because the presence of turbulent gas cloud would allowed the contained droplets to evade evaporation for much longer than an isolated droplet. The pathogen-bearing droplets can also travel much further in the presence of the turbulent gas cloud that carry them forward as oppose to the situation for isolated droplet. Bourouiba research shows that the gas cloud of human sneeze and the pathogen-bearing droplets of various sizes contained inside it can travel as far as 7 to 8 meter in distance.

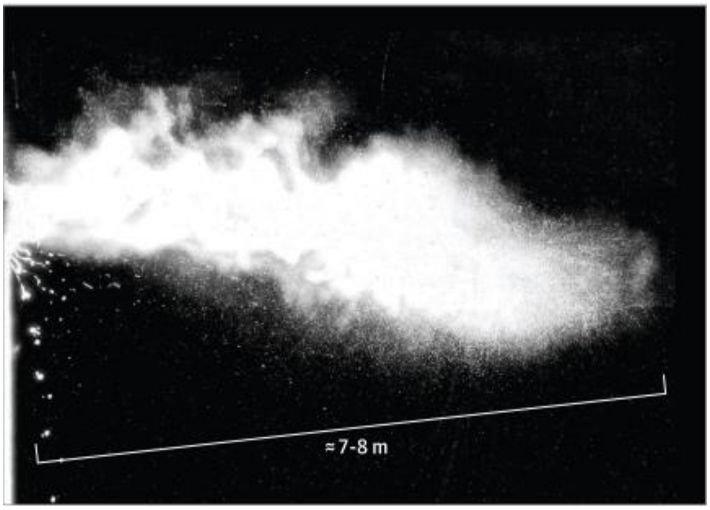

Figure 2.2: Multiphase turbulent gas cloud from a human sneeze by Bourouiba (2020)

Research papers from Wei and Li (2017) as well as Liu and Novoselac (2014) concern mainly about the topic of human cough jet and their role in the transport of airborne particles. Both of their research make use of artificial cough jet setup to visualize and analyse the parameters of the cough jet such as the velocity, penetration distance, and trajectories of the particles contained within the cough jet. The actual human cough jet might be different from the artificial cough jet developed in both of these research, thus the use of the parameters obtained from these studies must be taken with proper precautions and considerations.

Vansciver et al. (2011) in their research paper also contribute a lot of important data on average human cough parameters. In their research, cough generated infectious aerosols are of their interest while developing strategies for the mitigation of disease risks ranging from the common cold to SARS. In their work, the velocity field of human cough was measured using particle image velocimetry (PIV). A total of 29 subjects were involved in the project, and they individually coughed into an enclosure seeded with stage fog. Cough flow velocity profiles, average widths of the cough jet, and maximum cough velocities were successfully measured. The important data that can be extracted from their work is that the maximum cough velocities ranged from 1.5 m/s to 28.8 m/s, while the average width of all coughs ranged between 35 to 45 mm.

Shadloo-Jahromi et al.(2020) done a research that focus more on the vertical travelling of COVID-19 virus containing droplets rather than many other papers who study on the particles spread in term of both horizontal and vertical flow as a whole. They adopted molecular dynamics (MD) simulation method to investigate the behaviour of water droplets in a room environment. Main finding of the study is to predict vertical travelling and lifetime of aerosols at different manipulated thermodynamic conditions. Different sizes of water droplets were also considered, to find out the effect of air temperature on the behaviour of hovering droplets. Large scale atomic/molecular massively parallel simulator (LAMMPS) was employed to achieve the atomistic studies. OVITO programme which is an analysis and visualization tools for LAMMPS was also used to visualize the atomistic output data.

Extensive research was made by Bourouiba et al.(2014) presented the results of a combined experimental and theoretical investigation of the fluid dynamics of such violent expiratory events. The key findings of their research is that the turbulent multiphase cloud gives a great effect in extending the range of the majority of pathogen-bearing drops that accompany human coughs and sneezes. Smaller droplets with the diameter of less than 50 µm can remain suspended in the cloud long enough for the cough to reach heights of about 4 to 6 meter where ventilation systems can be contaminated. In their study, droplet of 10 µm diameter evaporates in 0.027 s, during which it would fall a distance of approximately 0.08 mm at a settling speed of approximately 3 mm/s. This implies that the droplets could remain suspended in a cough or sneeze cloud metres away from the infected person who coughed. The ambient conditions also influence the buoyancy of the cloud and so the range of contamination of its suspended droplets. Changes in season to the indoor conditions from summer to winter also give changes to buoyancy effect on the particles. This results in a variation of the range of deposition of the order of metres for the relatively large droplets (diameter $d > 50$ µm) to dozens of metres for the smallest droplets and droplet nuclei (diameter $d < 10$ µm). They also noted that droplet evaporation can enhance the cloud buoyancy, increasing its vertical momentum and thus the chances of the cloud reaching higher vertical distance and possibly reach the ceiling and contaminating the ventilation system.

Another research by Gupta et al. (2009) involves the study of flow dynamics and characterisation of cough for a number of 25 subjects, consist of both males and females volunteers. The research made by them concludes that many parameters involved during coughing phenomenon such as the flow rate of cough, number of cough (single or sequential, the temperature of the turbulent gas cloud, mouth opening area during cough and others are vastly different from each individuals. This suggests that any kind of simulations cannot perfectly describe the actual results of any type of research that involves coughing phenomenon as the parameter vary from person to person, and thus proper acknowledgement on the uncertainties from the results obtained from those simulations must be made.

2.3 Ventilation as a viable prevention method

There are a lot of recent papers that adopted CFD simulation in order to simulate, and investigate the flows of pathogenic particles in a properly modelled room with specified airflow condition or ventilation. All those research aim to achieve the same objective as the current research which is to utilize the CFD simulation involving COVID-19 virus carrying particles to find the most effective way to prevent the virus from infecting people that present in that confined space. Research done by Bhattacharyya et al. (2020) involves CFD simulation to analyse and minimize the spread of COVID-19 virus in a hospital isolation room. The concern of the research is to investigate the effectiveness of conditioned air released from the air-conditioning unit mix with aerosol type sanitizer to assess the possibility of using that method to deliver the aerosol to every point of space of the isolation room to kill the virus. If the method is possible, then it will be useful to help protect the lives of doctors, nurses and health care workers by reducing their risk of virus infection when they are working in the isolation rooms.

Zhang et al. research (2017) utilise numerical simulation method to study the flow of coughed droplet particles in an air conditioned conference room occupied by 12 individuals with one of them made to be the polluter. The study of the particles flow in the room involve four separate ventilation cases, namely the lateral-supply top-return, upper-supply top return, bottom-supply upper-return and finally a no ventilation cases for comparison and benchmarking purposes. The main objective of the research made by Zhang et al. is to identify which of the ventilation cases is the most effective in controlling the spread of the diseases in the indoor environment.

Both Yang et al. (2018) and Y. Yan et al. (2020) conduct a research to study the effects of cough-jet by an infected person on the airflow and contaminant transport in a common airliner cabin section with the usual ventilation system adopted inside an airplane cabin. The cabin of the airliner cabin section as well as the seat and a number of passengers consist of polluter and receiver individuals was modelled in the simulation.

On the other hand, research paper put out by Qian and Zheng (2018) serves the purpose to update all the important findings by previous researchers prior to their work, which focus heavily on the knowledge of airborne transmissions of pathogens and seek the improvement of ventilation efficiency to prevent infection. Qian and Zheng concluded their paper by stating that ventilation is useful to control airborne infection but is not efficient in preventing droplet-borne transmission. Droplet-borne transmission involves the exposure to larger droplets, smaller droplets and particles when a person is in close contact with an already infected person. Thus, the best prevention measure for droplet-borne transmission is to keep an adequate social distance from the infected person. Qian and Zheng proposed an improved downward ventilation system, and state that using PV-PE (personalized ventilation and personalized exhaust) system can reduce the risk of airborne infection effectively. Important points from their paper is also that the displacement ventilation system is not suggested for isolation rooms.

One of the question arises since the start of the pandemic regarding airborne transmissions of COVID-19 particles inside a room is whether or not air conditioning and ventilation system increases the risk of virus transmission. Global Heat Health Information Network in their online article (*Do Air Conditioning and Ventilation Systems Increase the Risk of Virus Transmission? If so, How Can This Be Managed?*) gives a detail explanation on this topic based on the findings made from existing research papers and journals related on the air conditioning ventilation systems. The short answer to that question is that any air conditioning or ventilation systems that are well-maintained and operated should not results in the increase of virus transmission risk. Fans are considered safe for personal room use but should be avoid if the room or space is occupied by several people. They also reminded the readers that every air conditioning and ventilation system for both residential and high occupancy building should have regular inspection, maintenance, and regularly cleaned to prevent virus transmissions. Despite all that, recommended physical distancing and social hygiene must still be practiced for the room occupants. The recommended room settings are of temperature between 24℃ and 27℃ for cooling during the warmer weather, and the relative humidity value RH% between 50% and 60% (CDC, 2015). They also suggest the action of increasing the outdoor air exchange, minimizing air blow from one person directly to another if the use of fans is unavoidable. This will greatly reduce the potential spread of any airborne or aerosolized virus.

Referring to the study made by Qian and Zheng (2018) , a ventilation system that is properly maintained should not cause a significant risk of virus transmission. The most efficient filters should be used they also must be changed according to the manufacturers recommendations. Not only that, the duct systems must also be cleaned periodically. If the air conditioning and air ventilation systems are not properly maintained and cleaned, the system itself could recirculate the contaminated air, and might also create indoor conditions for the room or space (temperature and humidity percentage) that supports the virus lifespan.

Building that make use of central ventilation system and/or climate control system should use the most efficient filters available. Higher efficiency filter (MERV 13 to 16, EPA 2019) should be considered. Any healthcare facilities that house COVID-19 patients should have negative-pressure and HEPA-filtered (high-efficiency particulate air) ventilation as it able to capture viruses more effectively and prevent the virus from spreading to other facilities (Perry et al., 2016).HEPA-filtered ventilation is also required for a busy non-hospitals building where the ventilation system function like a closed circuit to clean the recirculated air. Another method is to install a distributed HVAC (heating ventilation and air conditioning) units such as packaged terminal air conditioner (PTAC) which did not require the help of central mechanism to distribute and recirculate the air (de Groot et al., 2013; Dietz et al., 2020; Peeri et al., 2021). To reduce the lifespan of COVID-19 viruses in indoor spaces, climate settings of low or cold temperatures which falls below 21℃ must be avoided (Chin et al., 2020). On top of that, low humidity (dry) settings of below 40% must be avoided as these are the optimal conditions for the longevity of the viruses (K. H. Chan et al., 2011; van Doremalen et al., 2020).

Reducing virus circulation in public spaces is one of the important COVID-19 viruses infection prevention steps. Due to many of the recent infection cases shows a trend of pre-symptomatic transmission (Gandhi et al., 2020), people should continue practicing physical distancing in public spaces. The practice of wearing a face protection mask should not be underestimated as it can help prevent the virus droplets from speaking and coughing during social interactions from reaching surfaces or other people in the surrounding, which helps reducing potential spreading from pre-symptomatic individuals (Klrsten Koehler & Ana Rule, 2020).

Proper and frequent sanitisation of hands and surfaces in public or occupied spaces is important to prevent virus transmission. This is because, even though the virus does not reproduce outside of living cells, they can survive on surfaces and air (stay airborne) for a sufficient duration to facilitate transmission (Fears et al., 2020; van Doremalen et al., 2020).

Two groups of researchers (Cheng et al., 2020; Ong et al., 2020) has recently conducted an experiment on the air samples quality inside a hospital building where COVID-19 patients were allocated. In both of the research done, the patients were confirmed to have notable viral load in their respective respiratory secretions. However, the findings of the experiments reported that there was no SARS-CoV-2 virus discovered inside the taken air samples. Both researchers concluded that these situation happen because the ventilation system are well-maintained and were adjusted to the proper setting for a hospital as recommended, resulting in efficient ventilation and air circulation inside the building. A separate study was conducted by Schwartz et al (2020) regarding the lack of COVID-19 transmission on an international flight. They stated that the cause of virus infection between passengers of an airline cabin were more biased by the share of contacts between the infected passengers before boarding the flight instead of airborne infection that may happen inside the airline cabin when they are seated close to a COVID-19 carrier, as there are very few secondary cases of infection after a prolonged air travel found. This further supports the effect of efficient air filter and well ventilated airline cabin manage to prevent further virus spread from an infected person to other passengers who are seated in the close vicinity.

During January 26 to February 10 2020, a number of COVID-19 cases were reported at a local restaurant in Guangzhou, China, involving 3 family clusters. The restaurant was informed to be a 5-floor building with air-conditioned equipped, without any windows. The infection cases involved the customers who dine on the third floor dining area only. The particular dining floor occupies spaces of 145m^2, with the table arrangement to have only 1m distance from each other. In the dining space, a total of 83 customers at 15 tables were having a meal during that day when the virus outbreak happened. Among the 83 customers, 10 were tested positive while another 73 of them were classified as close contacts and were then quarantined for 14 days. The study of this virus outbreak cases were done by Lu et al. (2020), and they stated in their findings that the main reason of the virus spread was due to the direction of the airflow inside the dining area which was affected by the ventilation system configuration. They mentioned that the ventilation system was such that both the air outlet and the return air inlet for the central air conditioning system were located above one of the dining table there. This means that there were air recirculation happened which can possibly facilitate transmission of the virus among the customers. They recommended every operating restaurants to increase the distance between every dining table and improve the ventilation in order to prevent such cases from repeating.

Final conclusion that can be made is that other than to create a thermally comfortable indoor conditions for occupants inside a particular rooms or buildings, ventilation and air conditioning system serve an important purpose to improve the air quality that is circulating inside those spaces. Otter et al. (2016)conducted a comprehensive that examined the relationship between the lifespan and transmission of coronaviruses and influenza with respect to the various indoor temperature and relative humidity settings. The research concluded that at lower temperature and relative humidity level (below 21℃ and below 40%RH), there will be significant increase in survival time or lifespan of those coronaviruses and influenza on dry surfaces. This finding was supported by the finding from another research by K.H. Chan et al. (2011). They found that the SARS-CoV-1 can survive for at least two weeks in an indoor space where the temperature was set to22-28℃ and relative humidity of about 30-60%. However, previous research by WHO (2003) have already declared the same type of virus will be inactivated after 15 minutes in 56℃ temperature in a laboratory setting. van Doremalen et al. (2020) confirmed the SARS-CoV-2 and SARS-CoV-1 are both comparable in their behaviour of transmission characteristics. Thus, quoting from Sun et al. (2020), they stated that their studies successfully "evaluated the common epidemiological patterns of both SARS epidemics in China and identified cold, dry winter as a common environmental condition conducive for SARS virus infection to human beings" (p.8).

Even though low temperature and humidity conditions for indoor environment does contribute to the longer lifespan and transmission of the viruses, WHO (2021) advised the publics to avoid themselves from needlessly exposing their body to scorching sun or temperatures higher than 25℃ as it does not help in preventing COVID-19 infection. Instead, they strongly recommended the publics to protect themselves by practicing social distancing and washing hands before touching their eyes, mouth and nose.

CHAPTER 3: METHODOLOGY

3.1 Modelling the ventilated room and human dummies

The ventilated room and the human dummies as well as the table and chairs were modelled by using Solidworks program. The ventilated room was design to have a displacement ventilation system, where the air inlet is located at the lower side of the room wall, while the return outlet is located on the ceiling of the top opposite wall. The dimensions of the ventilated room is of the volume of $48.6m^3$, with 6m length, 3m wide and 2.7m tall. Inside the room are a long table of 3.1m length, 1.4m wide and 0.5m tall. Around the table, 6 chairs with the dimensions of 1m tall and 0.54m width were placed. The arrangement of the chairs around the table were such that two chairs were located on the farthest left and right side of the table, whereas the remaining 4 chairs were arranged to be equidistant from each other, with 2 chairs on each remaining side of the table. On the 4 chairs located at the center of the table, a human dummies were placed with the distance 1m from each other, with 1m measured from shoulder to shoulder for side by side seats, and measured 1m from mouth to mouth for the individual who sit opposite to each other. This 1m distance between every human dummies configuration was made in order to mimic the real situation at a meeting or dining table where people were advised to seat 1m away from each other during this COVID-19 pandemic. The human dummies were designed according to the mean measurement of human upper bodies, with 300mm shoulder span, and 575mm for the upper body length. Only the upper body and the head was designed for each dummies, without including lower limbs and hands, to reduce simulation time and complexity. Among 4 of the individuals, only one of them was made to be the COVID-19 infected person. The Figure 3.1 and Figure 3.2 shows the overall setup of the 3D model used in the simulation.

3.2 Meshing in Ansys Fluent

From Solidworks, the model of the room, furniture and human dummies was saved with ".igs" format before it was imported into Ansys Fluent CFD program. Boolean operation with subtract option was performed on the room and furniture model inside the DesignModeler window in Ansys, in order to modify the imported

Solidworks model created into a 3D model that is more suitable for Ansys Fluent simulation. After the Boolean operation, the room and the furniture inside it are now combined into one body, while the 4 human dummies inside it were made to be a separate solid bodies. Meshing was done by using the built in meshing software inside Ansys Fluent program. The number of nodes present in the meshing done are 12966 and the number of elements are 59868. From the mesh metric data, the average value of skewness of the mesh is 0.26487 while the average value of orthogonal quality is 0.73366. Both the average value for the skewness and orthogonal quality of the mesh is satisfactory. This is because both the value of skewness and orthogonal quality of the mesh created fall inside the "Very good" range of values as can be observed from the Figure 3.1 below (Muñoz, 2015).

Skewness mesh metrics spectrum:

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

Orthogonal Quality mesh metrics spectrum:

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

Figure 3.1: Skewness and orthogonal quality mesh metrics spectrum.

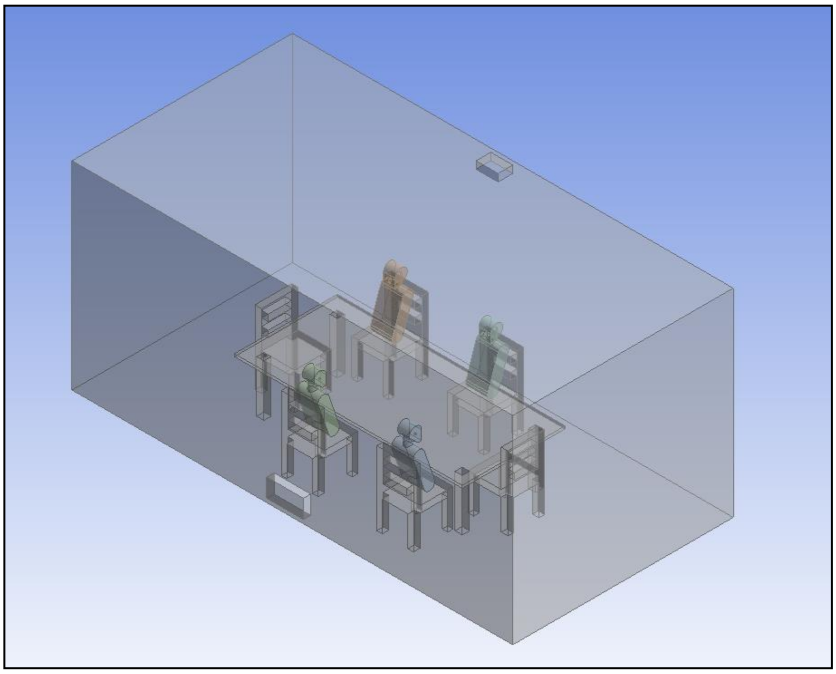

Figure 3.2: Isometric view of the ventilated room, furniture and human dummies setup

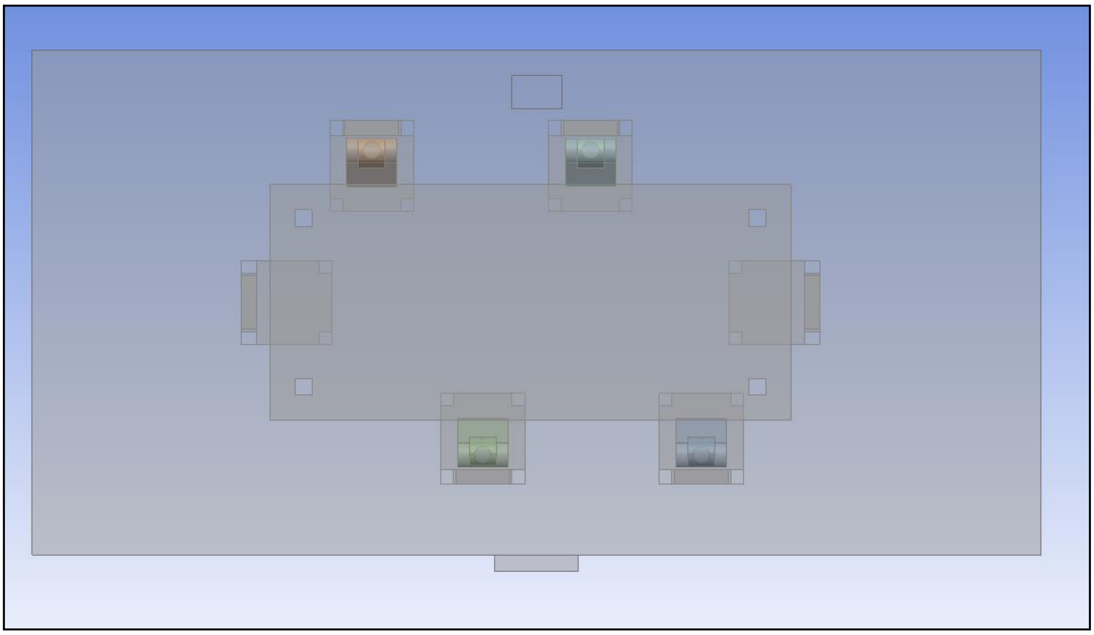

Figure 3.3: Top view of the ventilated room, furniture and human dummies setup. (lower bottom human dummy was chosen as the infected person)

3.3 Simulation settings and boundary conditions

The next step was the setup of the required parameters and condition before the calculation or simulation can be made. In the general tab, the solver chosen was Pressure-Based. The velocity formulation was chosen to be Absolute and the calculation will be made according to Transient time setting. Gravity value was set to be -9.81m/s in the Y-axis.

Energy equation settings was turned on as the parameters such as temperatures and velocity were needed for the simulation. Viscous model was set into Realizable k-ε as it provides more accuracy for cough with round jet flow. Species transport settings was enabled as well as the diffusion energy source settings. Discrete phase modelling (DPM) setting was enabled and the particle injection was set up to have surfaces type injection, with the mouth of infected person chosen to be the surface for injection. The particle type was set to be droplet and the material of the particles were water-liquid, with the water as the evaporating species. The particles sizes was chosen to follow Rosin-Rammlar diameter distribution setting in the Ansys Fluent program, resulting the distribution of variable sizes of particles is with 1.98μm min, 737μm max and 170μm mean sizes. The particles injection velocity was 10m/s and have temperature of 305K, with spherical drag law and discrete random walk model settings selected (B. Zhang et al., 2019).

The room inlet air velocity is set to be 3.6m/s with 5% turbulence intensity and turbulence viscosity ratio of 10. The air inlet temperature was set to 300K and have 0.012 H_2O species mass fraction with "escape" as the discrete phase boundary condition in DPM setting, which means that droplet particles will only get away from the boundary. The return oulet was set to have 0Pa gauge pressure, that makes the air entering the domain to leaves the domain through the outlet boundary. Same as the room inlet boundary condition, the oulet discrete phase boundary condition in DPM setting was set to "escape". The mouth of the infected person was defined as a wall with "escape" discrete phase boundary condition, in order to avoid the cough particles to flow back into the infected person mouth surface after the particles injection took place. And finally, the human dummies outer surfaces, furniture surfaces as well as the floor and walls of the ventilated room were defined as stationary walls with "trap" discrete phase boundary condition. This settings was chosen in order to make sure the particles will be trapped or stick to the surfaces of the furniture, room or human dummies so that the amount of particles or droplets deposited onto those specific spots can be observed. After the hybrid solution initialization, the simulation was performed to compute the movement of cough particles droplets after real time 20s of the initial particles injection. (B. Zhang et al., 2019)

The summary of the important simulation settings and the boundary conditions was listed accordingly in the table below.

CFD Model Selection:	
Steady/Unsteady	Transient
Dimension	3-D
Turbulence model	Realizable k-ε
Discrete phase model	Water-liquid droplets, Velocity=10m/s Temperature=305K Diameter distribution=Rosin-Rammlar Logartihmic
Boundary Conditions:	
Room inlet	Inlet velocity=3.6m/s Turbulence intensity=5% Turbulence viscosity ratio=10 Escape (DPM)
Room outlet	Gauge pressure=0Pa Escape (DPM)
Mouth of infected human dummy	Escape (DPM) Heat Flux= 0W/m^2 (Adiabatic wall)
Room walls/floor	Reflect (DPM) Heat Flux= 0W/m^2 (Adiabatic wall)
Human dummy outer surfaces	Reflect (DPM) Heat Flux= 0W/m^2 (Adiabatic wall)
Furniture surfaces	Reflect (DPM) Heat Flux= 0W/m^2 (Adiabatic wall)

Table 3.1: Summary of simulation settings and boundary conditions

CHAPTER 4: RESULTS AND DISCSUSSIONS

4.1 Steady state airflow results

In order to carry out a transient simulation of coughing in the ventilated room, the initial condition of the room must be already air conditioned before the cough particles was injected. Thus, a steady state simulation was first done without including the injection of coughing from the COVID-19 infected patient as the required initial condition. Figure 4.1 below shows the steady state pattern of the airflow inside the air conditioned room before the cough injection, simulated inside the Ansys Fluent CFD program. From the figure, it can be observed from the pathlines that the airflow of the room is more focused at the center of the room, whereas the left and side of the room is not that affected by the airflow. This means that the air turbulence is much higher in the center part of the room compared to anywhere else. This is due to the room outlet that was located exactly at the middle of the top ceiling of the room. From the contour plot on the left side of Figure 4.1, the highest velocity magnitude is at the point close to the room oulet with value of 6.32m/s, and it can be observed that for the whole room, the average velocity magnitude is about 0.5m/s.

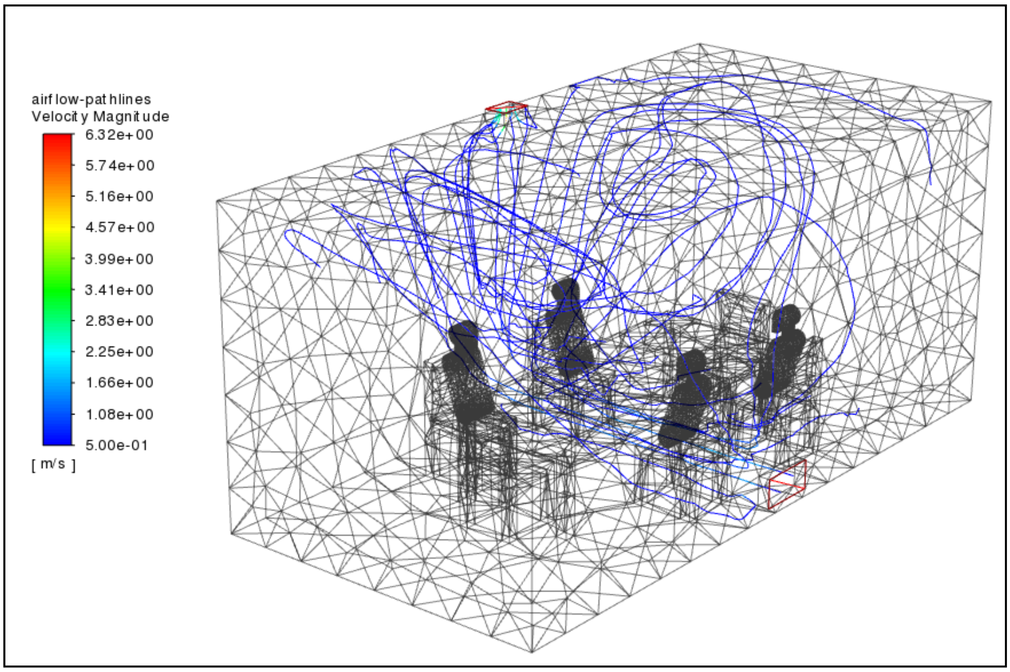

Figure 4.1: Airflow pathlines colored by velocity magnitude.

4.2 Transient simulation results and particles tracking

The results of transient simulation are as shown in the following Figure 4.2 to Figure 4.5. The figures represent the simulation results of the COVID-19 particles tracking inside the air conditioned room after 0.5s, 2s, 10s and 20s for Figure 4.2, Figure 4.3, Figure 4.4 and Figure 4.5 respectively. The wall of the air conditioned room are made hidden, in order to increase the clarity of particles tracking. From the contour plot on the left side of every figure below, it can be observed that the particles sizes ranges from 1.98μm to 7.37×10^{-4} or 737μm. These variable sizes of particles is because the particles sizes was chosen to follow Rosin-Rammlar distribution setting in the Ansys Fluent program, with 1.98μm min, 737μm max and 170μm mean.

Based on Figure 4.2 (0.5s after cough injected), majority of the particles of the injected cough follows the trajectory of the high speed cough jet with high inertia, before majority of the coarse droplets dropped due to gravity and deposited on the table surface. There are more coarse droplets deposited on the table top surface compared to finer size droplets, as the finer size droplets were suspended in the air as they are lighter in mass. It can be concluded that the deposition of coarse COVID-19 particles is highly directional, following the direction of the cough jet of the infected person. This means that the practice of sitting in the table arrangement as shown below (no sitting directly in front of each other) is very effective in preventing a person from contact with coarse size cough droplets, which most likely to carry COVID-19 virus.

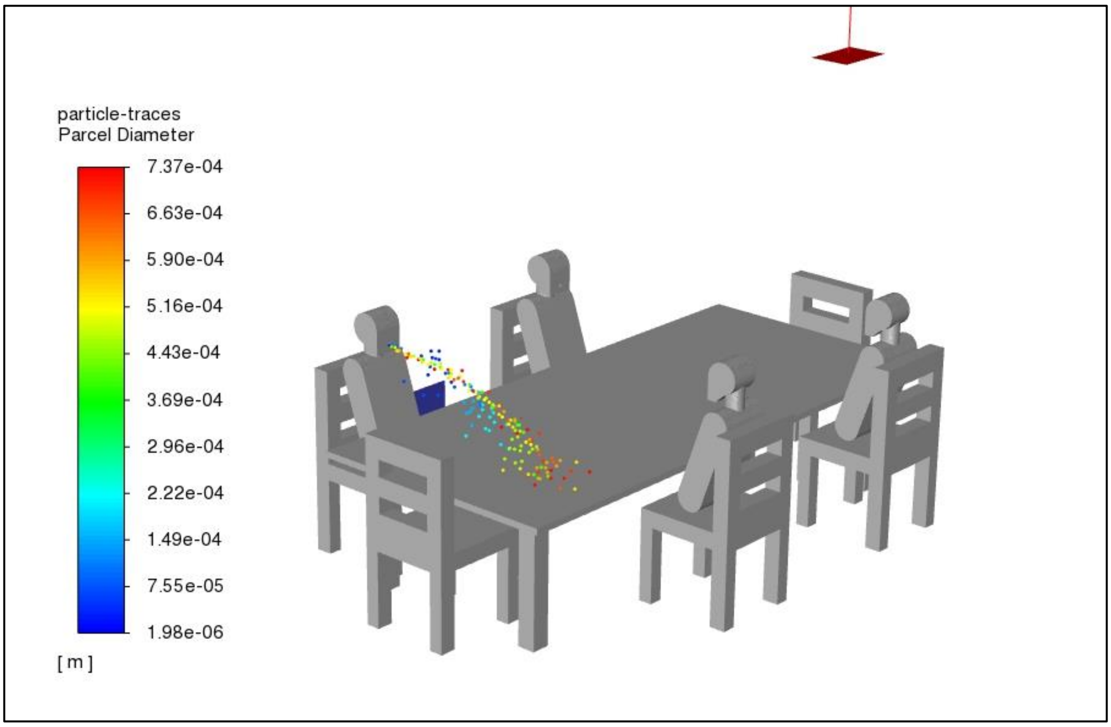

Figure 4.2: Particles tracking of cough particles after 0.5s of injection.

Referring to the particles tracking in Figure 4.3 which taken place 2s after the cough injection, it can be observed that most of the finer droplets with size of about 443µm and below were still suspended in the air, and were also spread in the vicinity area, mostly in front of the infected person. This spreading of particles is due to the air flow of the air condition. The coarse droplets were missing in the Figure 4.3 as they were already deposited on the table surface in the previous figure and also evaporated due to the ambient temperature.