PAPER • OPEN ACCESS

The effect of air conditioning outlets on the spread of respiratory disease in Mosque's environment

To cite this article: H M Khan and S N Al-Saadi 2021 J. Phys.: Conf. Ser. 2069 012188

View the article online for updates and enhancements.

You may also like

- Identity of the architecture of mosques in Russia in the late XX-early XXI century (as exemplified by Tatarstan) Khanifa G. Nadyrova and Dilyara A. Nadyrova
- <u>The intensity of prayer room's natural</u> <u>lighting</u> K A Mannan
- <u>Design and implementation of the korean</u> <u>muallaf center application as a guide to</u> <u>praying and igro procedures for korean</u> <u>muallaf</u> W A A Praditasari, I Kholis and A A Budiman





DISCOVER how sustainability intersects with electrochemistry & solid state science research



This content was downloaded from IP address 3.145.178.157 on 04/05/2024 at 13:35

Journal of Physics: Conference Series

doi:10.1088/1742-6596/2069/1/012188

The effect of air conditioning outlets on the spread of respiratory disease in Mosque's environment

H M Khan¹ and S N Al-Saadi¹

¹Sultan Qaboos University, Department of Civil and Architectural Engineering, P.O. Box 33, Sultan Qaboos University, 123, Oman

Abstract. Mosques are places for daily worship for Muslims, where they attend prayers five times/day. As a common prayer practice, worshippers conduct prayers in standing groups side-byside in rows touching shoulders and ankles. Furthermore, in their praying practice, worshippers touch the floor with their forehead four to eight times in a single prayer, which is an important factor in picking or spreading infection disease. Mosques are usually air-cooled by mechanical means with a poor ventilation system. The prayer practices, coupled with the poor ventilation system increase the risk of spreading respiratory diseases like COVID-19. This study utilizes a Computational Fluid Dynamics (CFD) package to evaluate disease particles' movement around rows of worshipers. The research evaluates the impact of air outlet locations on the spread of the disease particles. The results indicated that the locations of air outlets relative to the infected person may significantly help to spread the particulates. In mosque environment, the ceiling diffusers are recommended, and sidewall outlets should be avoided. In addition, it was concluded that a minimum of 2 meters between occupants as suggested by WHO is not deemed enough to control the spread of disease in mosque environment and a minimum of 3 m is necessary. The study calls to review the guidelines by the World Health Organization (WHO) for mosques and similar environment.

1. Introduction

The mosque is a widespread building in the Islamic world, where the Muslims meet five times a day for worshiping. The number of occupants during the Friday noon prayer reaches the maximum. During this time, the mosque is packed with rows of hundreds of worshipers facing one direction toward the holy city of Mecca, The Muslim's praying includes many body movements with standing, kneeling, bow and prostrations, and this happens in cycles from two to four depending on the prayer time.

With the beginning of the SARS COVID-19 Pandemic of 2020, the World Health Organization (WHO) advised to avoid mass gathering, especially in closed spaces like Mosques. The infection may spread when inhaling respiratory droplets generated during speaking, breathing, coughing, or sneezing. The air droplets have a range of sizes from relatively large droplets (50 to 100's μ m) to smaller droplets (1 to 10's μ m). According to the Center for Disease Control and Prevention (CDC), both sizes may contribute to disease and the infection can transfer by both droplet and airborne transmission.

Many research papers were conducted on the droplets flow from the 20th century, such as Wells' work from 1934[1]. However, in the last decades, numerical methods like Computational Fluid Dynamics (CFD) are commonly used for modeling droplet flow. One example of CFD is the work of Gao and Niu[2], where

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI. Published under licence by IOP Publishing Ltd 1

Journal of Physics: Conference Series

they found the exposure to pollution is directional, and it is more in the case of two humans facing each other. They also showed that the type of ventilation can affect personal exposure to pollutions, which is less in case of displacement ventilation. Other researchers simulate droplets as particles in the Eulerian-Lagrangian model, and the air is modelled as a continuum phase and droplets as a discrete phase[3]. The particles' trajectory is simulated in the Lagrangian frame. The particles (droplets) are assumed to be a spherical mass of water with a density much higher than the air, which makes them under the effect of gravitation and drag force. Busco et. al [4] studied the biomechanics of a human sneeze and suggested a new simple model for simulating the spread of pollutions in closed spaces. The study highlighted the importance of evaporation. After the lapse of a few seconds in the air, the particles will either evaporate, deposit on surfaces, or remain in the air (airborne). The evaporation will depend on many factors such as the air humidity and the droplet's size[5], and it will gradually reduce the diameter of the droplets. The particles dimension will affect the movements of the particles in the space. The larger droplets over 100 μ m have a high momentum to penetrate the room faster than others but are affected by drag forces and will soon fall off due to gravity force. Smaller particles will stay nearly in the same vertical levels, and turbulent dispersion and evaporation will quickly reduce their size to small passive particles[3,5].

2069 (2021) 012188

The work of Busco shows that particles larger than $100 \ \mu m$ can stay for a few seconds in the air before they evaporate or fall (deposit) on the ground. Smaller particles can remain in the air if the humidity is higher than 95%. Eventually, all the particles over 50 μm will fill down 2 meters away from the source (i.e., the mouth). Smaller particles will continue floating in the air similar to dust and other passive particles[5,6]. However, these particles will be affected by turbulent diffusion, reducing the movement, and increasing the side penetration.

Sun and Ji [3] showed the influence of ventilation types in a room environment. They concluded that displacement ventilation has the highest potential in removing the particles from the room. Verma and others [7] examined the role of air outlet positions in particles resident time and how it depends on the air flow rate and the position. Zhu et. al [6] evaluated the impact of the air conditioner position and found it helps in transferring the infection from one person to another.

The mosque is a particular case of aerosol simulation where the worshipers are not facing each other. The worshipers organize themselves in rows where there is more than one meter spacing between a row and another. Additionally, continuous movement during praying may change the distribution of the particles around the prayer. Furthermore, according to Islamic instruction, the prayer's forehead should touch the ground; this will get the mouth area close to the ground where the particles are settled after falling from the mouth level. After reviewing the literature, no research was found to deal with particulates distribution in the mosque's environment. This paper will focus on evaluating the particles distribution in the mosque environment. In particular, the work will evaluate the relationship between the air outlets distribution and spreading of particles through the zone. The simulation will be divided into two groups. The first group focuses on the spreading of fine particles in the air. The second group will deal with different particles' sizes and how far they go before depositing over the ground.

2. Numerical simulation

Virus transmitted in the air in droplet form (for sizes higher than 5 micros) and by airborne (i.e., for sizes lower than 5 microns) resulting from droplets emitted from the mouth and nose when coughing, sneezing, etc[8]. The first group of cases simulate airborne transmission between hundreds of prayers in a large hall by using trace gas. This methodology approach was used by many researchers and validated by Bivolarova et. al[9]. In the second group, the simulation is conducted to model the flow particles using the Lagrangian model around the worshipers in the mosque. The second group was needed to study a small and confined space with a limited number of worshipers. This simulation covers two phases: continuous and discrete. The indoor air is treated as a continuous phase, solved using the Eulerian model. On the other hand, the particle

8th International Building Physics Conference (IBPC 2021)		IOP Publishing
Journal of Physics: Conference Series	2069 (2021) 012188	doi:10.1088/1742-6596/2069/1/012188

as a discrete phase is tracked using the Lagrangian model. There is an interaction between the two phases where the exchange of mass, momentum and energy is simultaneous.

One symptom of COVID-19 is coughing, which will release millions of particles through the mouth cavity at high speed and cause the flow to be more turbulent. In this case, the software needs to solve conservation equations and global continuity equation to predict the flow field, energy equation, together with an appropriate turbulence model. Researchers in this subject used various versions of K-epsilon (k- ε) turbulence models such as RNG[2,5], Standard [3]and Realizable[4]. In this simulation work, the Realizable k- ε algorithm for turbulence simulation in STAR-CCM+ Ver 2020 was adopted. The airflow was modelled as a continuous phase of an ideal gas which consists of a mixture of dry air and water vapour. This mixture was used to determine the relative humidity which will subsequently be used to estimate the particles evaporation. The evaporation can reduce the particles' diameter and according to Stokes' law, it will reduce the gravity force and keep the particles longer in the air. The software will calculate the evaporation by using the quasi-steady evaporation rate, and this will simulate the mass reduction in particles due to the evaporation using the following equation:

$$\dot{m}_{evap} = -g^* \varepsilon A_p \ln \left(1 + B\right) \tag{1}$$

where, g^* is the mass transfer conductance which depends on the velocity and diameter of the particle, ε is representing the fractional evaporation rate, A_p is the particle surface area and *B* is Spalding transfer number. The evaporation depends on two material properties; the saturation pressure and the critical temperature. These two properties are related to the air temperature, and the relative humidity. The simulation parameters for the CFD model are assumed as per table 1. Sherwood number correlation and the Spalding mass transfer number were assumed in the simulation to model the convective mass transfer during the evaporation process. Ranz Marshall was used as a method for a closure of Sherwood number. The software injects streams of various sizes droplets from the mouth area according to the lognormal distribution from [10] with a mean diameter of 360.1 μm and geometric standard deviation equal to 1.5 and minimum size of 125 μm and maximum diameter of around 1 cm which is a fraction of the total area of the mouth.

The experiments from [4] show particles flow are mainly affected by evaporation and breakup. Therefore, the breakup dynamics and distribution by using Taylor Analogy Breakup (TAB) were considered[11]. The model uses mass-spring-damper system to simulate an oscillating and distorting of the droplet. In Stock's law, any falling droplet is affected by the drag force, so the drag coefficient is calculated using Liu model[12]. In addition, to show the effect of the transient in the carrier phase, the turbulent dispersion is modelled by using turbulent particle dispersion[13]. The model is necessary to simulate the particle dispersion due to local variations (i.e., eddies) in a flow field. In this case, the software calculates a random velocity component out from normal distribution with zero mean value and the eddy velocity scale during the eddy's time scale to calculate the standard division.

The particle energy balance in the software is found by using a generic form of the equation of conservation of energy:

$$m_p c_p \frac{dT_p}{dt} = Q_t + Q_{rad} + Q_s \tag{2}$$

the particles do not include the heat source (Q_s) and the radiation between particles and other surfaces are usually neglected[5], so equation two can be simplified as:

$$m_p c_p \frac{dT_p}{dt} = fhA_s \left(T - T_p\right) \tag{3}$$

Journal of Physics: Conference Series

here $m_p c_p$ are the mass and specific heat of the particles, A_s is the surface area of the particles, T is air temperature, T_p the particle temperature, f is a mass transfer correction, and h is the heat transfer coefficient calculated by Ranz-Marshall correlation.

The simulations for some cases include a rigid body motion performed by using the overset grid method. In this method, several regions may occupy the same space. Furthermore, the air will be background mesh and one or more overset grid attached to bodies will overlap with each other. The information can transfer between the grids in different regions by using linear interpolation. The simulation uses non-uniform density polyhedral meshes with a refined region in the injection area at the mouth. The mesh size is a few centimetres in the volumes around the manikin to tens of centimetres in the rest of the zone.

Case	Parameter	Reference
Human height	1.75 m	[3]
Human thermal condition	Heat flux=25 W/m^2	[3]
Mouth area	4 cm^2	[3]
Exhaled air velocity	2 m/s	
Air temperature out of the mouth	37°C	
Relative humidity for air out of the mouth	100%	[3]
Inlet air temperature and initial temperature	12 °C	[14]
Air outlet boundary conditions	Pressure outlet, velocity inlet	[2,5]
Zone wall boundary type	Adiabatic wall	[2]
Air velocity from the air outlet	3 m/s	[14]
Relative humidity for air out of the air outlet	55%	[3]
The duration of cough or sneeze	0.5 s	[3]
Droplets shape and contains	As spherical drop of water	[3]
Droplet realize direction (from the mouth)	Horizontal	[2]
Droplet expelled velocity	20 m/s	[3]
Droplet volume flow rate	250 l/min	[2]
Wall boundary condition for droplet	Splash (no rebound)	[7]
Mouth and air outlet boundary condition	Escape	
Time step	(non -uniform step) 0.01 to 0.1 s	[5]
Simulation time elapsed	45s	
Turbulent model	Realizable K-epsilon (k-ε)	

Table 1. Summary of the values and parameters used in the simulation.

The geometry model in the first group is shown in figure 1. The mosque is for more than 400 worshipers, representing a medium-size mosque as per[14]. Several air outlet cases can be included in the simulation. As shown in table 2, the ceiling cases include a square diffuser with side dimensions 60 cm whereas the wall cases are considered with rectangular grill with side dimension 120 cm by 50 cm. The worshiper model is shown in the figure, and the people inside the zone are divided into four groups. Each group is divided into two subgroups in the breathing stage; one inhaled and the second is exhaled. In the inhaled and exhaled stages, the mouth area's boundary condition is changed from inlet to outlet, and each stage will take around 4 seconds. Carbon dioxide was used as a trace gas and emitted from 5 people distributed on the group's edges and the centre.

The second group's geometry model is shown in figure 2, and it represents a zone of the full mosque (refer to figure 1) with only a few worshipers. Here the simulation includes the movement of the human

body where particles are injected at three times; one at the beginning of the praying and two are before dropping down to the floor as shown in the figure.



Figure 1. The model for the first group with shown the positions of four groups.

Name	Description
Roof	Air Flow from and to outlets on the ceiling level
Long side up-down	Airflow from (up outlets) on the left side to outlets to (ground outlets) on the right side (both long side)
Long side up-up	Airflow from (up outlets) on the left side to outlets to (up outlets) on the right side (both long side)



Figure 2. The model for group two cases and shows the stages of movements.

3. Results

The cases covered in the two groups are shown in table 2. The result is shown in figure 3 (a), where the term (ratio) represents the total amount of mass of trace gas inhaled by manikins divided by the total amount of gas release in the zone during the simulation. The four groups' distribution is shown in figure 3 (b) and the highest ratio is observed in regions where the gas source is located near the air outlets. As shown in figure 4, the highest ratio is in case (Long side up-down) where the near-ground air outlet forces the gas to move down in parallel to the rows. In the Ceiling case (i.e., Roof case), the air is returned at the ceiling level which will limit the particles distribution in the breathing zone. The results show that it is efficient to locate the air outlet at the ceiling, rather than the sidewalls to limit the distribution of fine particles.



Figure 3. Percentage of inhaled mass relative to the total released mass, (a) for the whole the mosque (b) in the four zone groups (refer to Figure 2 for mosques subzones).



Figure 4. Top view for the same zone, but various cases and shows the gas distribution (blue).

With the second group of cases, the side penetration of the particles was investigated. This is taken as an important scenario because the worshipers are not facing each other during the prayer and the most important direction of the spreading is the lateral direction between the two persons. The particles' vertical elevation is not crucial because the worshiper has many positions during the praying practice which includes standing, bowing, prostration, and sitting, consequently changing the body position. The simulation is divided into three stages (refer to figure 2). The first stage represents the standing position and continues for 20 seconds. The second stage involves rotating movement to simulate the bowing stage of praying then returning back to standing and this stage will take 4 seconds. The third stage is moving from standing position to the ground (4 seconds) and the prostration stage (13 seconds) and returns to the standing position



(4 seconds)—there is one time of cough at the beginning of every stage. Figure 5 shows the side penetration for the three cases; the figure compares the three stages among different particles' sizes.

Figure 5. Side penetration.

The results show that the penetration will always increase with the body movement where the particles are further pushed away. So, in case of still standing position (stage 1), the particles will rarely reach two meters. However, and when the person is at stage two and stage three, the particles may penetrate more than two meters, which shows the need for more research on the safe distance between occupants especially for special environment like a mosque. The average penetration of particulates is predominant for the long side

8th International Building Physics Conference (IBPC 2021)

Journal of Physics: Conference Series

2069 (2021) 012188 doi:10.1088/1742-6596/2069/1/012188

cases when compared to the Roof case. This finding highlights the need to avoid sidewall air outlets and recommends the ceiling air outlet in the mosque environment. It also indicates that 3 meters is deemed necessary when sidewall supply outlets are available in mosques. The infected person (particles injector) location relative to air inlet or outlet influences the particles' direction. The spreading is more when the injector is near the supply air outlet as in 1st injector.

4. Conclusion

This research focuses on how respiratory infection particles will spread in unique indoor environment like mosques and how air outlets may influence the particles propagation. The research simulates the spreading of both fine and larger size particles by using CFD. The results show that air outlets on the ceiling level will have less spreading of infection particles than air outlets located on sidewalls. The location of the contagious person can increase the spreading, especially in places near air inlets. Moreover, the human body movement can affect the spreading where the rotation of the human body can increase the particles' momentum, which further enhances the particles movement. Due to this special attribute, the minimum distance between occupants should not be less than 3 meters for mosques, compared to the current practice of 2 meters as suggested by WHO. The outcome of this research encourages for further study in this direction.

References

- [1] WELLS W F 1934 ON AIR-BORNE INFECTION*: STUDY II. DROPLETS AND DROPLET NUCLEI. *Am. J. Epidemiol.* **20** 611–8
- [2] Gao N and Niu J 2006 Transient CFD simulation of the respiration process and inter-person exposure assessment *Build. Environ.* **41** 1214–22
- [3] Anon Transport of Droplets Expelled by Coughing in Ventilated Rooms Wei Sun, Jie Ji, 2007
- [4] Busco G, Yang S R (양세로), Seo J (서요섭) and Hassan Y A 2020 Sneezing and asymptomatic virus transmission *Phys. Fluids* **32** 073309
- [5] Aliabadi A A, Rogak S N, Green S I and Bartlett K H 2012 CFD Simulation of Human Coughs and Sneezes: A Study in Droplet Dispersion, Heat, and Mass Transfer ASME 2010 International Mechanical Engineering Congress and Exposition (American Society of Mechanical Engineers Digital Collection) pp 1051–60
- [6] Zhu S, Kato S and Yang J-H 2006 Study on transport characteristics of saliva droplets produced by coughing in a calm indoor environment *Build. Environ.* **41** 1691–702
- [7] Verma T N, Sahu A K and Sinha S L 2017 Study of Particle Dispersion on One Bed Hospital using Computational Fluid Dynamics *Mater. Today Proc.* 4 10074–9
- [8] CDC 2020 Coronavirus Disease 2019 (COVID-19) Cent. Dis. Control Prev.
- [9] Bivolarova M, Ondráček J, Melikov A and Ždímal V 2017 A comparison between tracer gas and aerosol particles distribution indoors: The impact of ventilation rate, interaction of airflows, and presence of objects *Indoor Air* **27** 1201–12
- [10] Lefebvre A 1988 Atomization and Sprays (Taylor & Francis)
- [11] O'Rourke P J and Amsden A A 1987 *The TAB method for numerical calculation of spray droplet breakup* (Los Alamos National Lab., NM (USA))
- [12] Liu F, Qian H, Zheng X, Song J, Cao G and Liu Z 2019 Evaporation and dispersion of exhaled droplets in stratified environment *IOP Conf. Ser. Mater. Sci. Eng.* **609** 042059
- [13] Gosman A D and Ioannides E 1983 Aspects of computer simulation of liquid-fueled combustors J Energy U. S. 7:6
- [14] Samiuddin S and Budaiwi I M 2018 Assessment of thermal comfort in high-occupancy spaces with relevance to air distribution schemes: A case study of mosques *Build. Serv. Eng. Res. Technol.* 39 572–89