COMPUTATIONAL STUDY ON THE TRANSMISSION OF COVID-19 VIRUS INSIDE A SHIP

Luofeng Huang
Cranfield Univesity
Cranfield, UK
&
University College London
London, UK

Seogeng Riyadi, I Ketut Aria Pria Utama
Institut Teknologi Sepuluh Nopember
Surabaya, Indonesia

Giles Thomas
University College London
London, UK

ABSTRACT

To investigate the operational improvements of vessels under the impact of COVID-19, this work has developed a Computational Fluid Dynamics model combined with Lagrangian particles to study the airborne transmission of COVID-19 viruses inside a ship. Initially a generic model was established to enable validation against experimental results for the diffusion of flu virus in an idealised room. Following this, the room geometry was replaced by the superstructure of a full-scale crew boat. Considering the boat advancing in open water, simulations were conducted to study the particulate flow due to a person coughing and speaking, with the boat's forward door open and closed. The results have shown that, when the forward door is open, a significant airflow can carry the viruses to make extensive contacts with the passengers. This led to the suggestion of keeping the door closed. However, when the forward door is shut, face-to-face speaking can generate viruses that can float in the air for a long time, and it was found that the viruses mainly stay within a half-meter distance in front of the speaking person, before sinking to attach to the deck. Thus, a social-distancing suggestion on seat arrangement has been highlighted to minimise the risk of contagion. Overall, this work is expected to inform guidelines on hygienic and reconfiguring means for operators to counter COVID-19 and potentially the spread of similar viruses in the future.

Keywords: COVID-19; Ship; Virus; Airborne Transmission; Computational Fluid Dynamics; Particle modelling.

1. INTRODUCTION

The maritime industry has been severely affected by the COVID-19 pandemic. Vessels are currently operating with reduced capacity or restricted from leaving port. Crew and passengers can be exposed to a serious contagion risk, as the compact and crowded space of ships implies a high likelihood of COVID-19 outbreak. In this context, there is an urgent need to research the best technical solutions to ensure COVID-19 safety for ships.

COVID-19 infection has been found to be primarily induced by inhalation [1]. The virus can be transmitted via air, existing in a form of aerosols and droplets injected by humans coughing, speaking, breathing, singing and sneezing [2]. Coughing and speaking are the most likely scenarios, as coughing is one of the primary COVID-19 symptoms and speaking is almost inevitable in daily contacts which can also output a significant amount of the virus [3]. The transmission mechanism of COVID-19 virus may be referred to in previous work that studied flu, as it has a similar mechanism to SARS [4].

During ship operations, natural winds, air conditioners and ventilation systems can induce airborne transmission of virus particles inside a vessel. To investigate this problem, computational modelling has the capability to understand the virus's movement and coverage, which is essential for developing effective mitigation strategies for infection control. To date, computational studies of COVID-19 transmission has been conducted for high-risk places, such as hospitals [5] and buses [6], but relevant research has not been seen for scenarios on board ships.

In this context, the present work develops a model combining Computational Fluid Dynamics (CFD) and Lagrangian particles to analyse the potential transmission of COVID-19 virus inside a ship. The paper starts by introducing the theories and practicalities of the CFD-Lagrangian model, followed by validating the model against experimental measurements of velocity field and particle diffusion inside an idealised room. Subsequently, the room geometry is replaced by the superstructure of a crew boat that is considered to move forward in open water, and analyses were presented on the virus distribution in different scenarios, concerning a passenger coughing or speaking when the boat's forward door is open or closed. The results are used to discuss measures that may minimise the risk of COVID-19 contagion.

2. COMPUTATIONAL APPROACH

CFD is used to simulate the airflow whilst the transmission of virus particles is tracked by the Lagrangian approach. The simulations were performed using commercial software STAR-CCM+.

2.1 Governing equations

In this work, Lagrangian particles are applied to model the COVID-19 virus aerosols/droplets, by which the particle movement is subjected to its gravity (***G***) and a drag force from its surrounding airflow ($\boldsymbol{F_d}$):

$$m\frac{d\overline{\boldsymbol{V_P}}}{dt} = \boldsymbol{G} + \boldsymbol{F_d} \quad (1)$$

where m denotes mass, $\overline{\boldsymbol{V_P}}$ is the particle's velocity, $\boldsymbol{G} = mg$ where g is the gravitational acceleration, set at 9.81 m/s^2. The fluid drag force is calculated through the Schiller-Naumann Correlation [7]:

$$\boldsymbol{F_d} = \frac{1}{2}C_d\rho_P A_P|V_s|\,V_s \quad (2)$$

where ρ_P is the particle density, A_P is the particle project area and V_s is the relative velocity between the particle and the air. C_d is an empirical coefficient calculated based on the particle's Reynolds number (Re_P), which is defined as follows.

$$C_d \begin{cases} \frac{24}{Re_P}\left(1 + 0.15{Re_P}^{0.687}\right), & \text{if } Re_P \leq 1000 \\ 0.44 \qquad\qquad\qquad\quad , & \text{if } Re_P > 1000 \end{cases} \quad (3)$$

The surrounding fluid flow is solved by the standard Reynolds-averaged Navier-Stokes (RANS) equations:

$$\nabla \cdot \overline{\mathbf{v}} = 0 \quad (4)$$

$$\frac{\partial(\rho\overline{\mathbf{v}})}{\partial t} + \nabla \cdot (\rho\overline{\mathbf{v}}\overline{\mathbf{v}}) = -\nabla\overline{p} + \nabla \cdot \left(\overline{\tau} - \rho\overline{\mathbf{v}'\mathbf{v}'}\right) + \rho g \quad (5)$$

where $\overline{\mathbf{v}}$ is the time-averaged velocity, $\mathbf{v}'$ is the velocity fluctuation, ρ is the fluid density (ρ_{air} = 1 kg/m^3), $\overline{p}$ denotes the time-averaged pressure, $\overline{\tau} = \mu[\nabla v + (\nabla v)^T]$ is the viscous stress term, μ is the dynamic viscosity ($\mu_{air} = 1.48\times10^{-5}$ N·s/m^2). Since the RANS equations have considered the turbulent fluid, the Shear Stress Transport (SST) $k-\omega$ model was adopted to close the equations [8,9].

In particular, a sufficiently-small particle in turbulent flow reveals a randomly-varying velocity field, which induces the microscopic particles to perform constant stochastic motions and diffuse. This behaviour is modelled by including the effect of instantaneous velocity fluctuations on the particle [10]:

$$\mathbf{v} = \overline{\mathbf{v}} + \mathbf{v}' \quad (6)$$

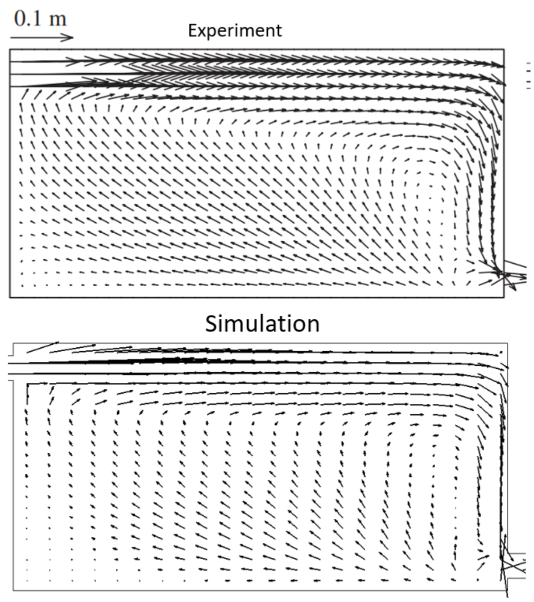
To be more specific, the applied fluid velocity in calculations is **v**, which is different from a usual RANS approach for macroscopic problems where $\overline{\mathbf{v}}$ is directly used to simplify the calculation, e.g. [11].

2.2 Validation

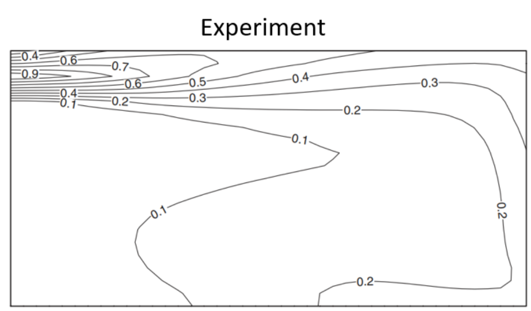
To confirm the suitability and accuracy of the applied equations for modelling the proposed problem. A validation study was conducted against experimental measurements of particle diffusion in an idealised room. As shown in Figure 1, a rectangular room is applied (0.8 m × 0.4 m × 0.4 m), where an air inlet is near the left top side and an outlet is near the right bottom side. From the inlet, an airflow is coming into the room at a constant velocity, carrying artificial particles that have a diameter of 10 μm, which is similar to the diameter of flu viruses.

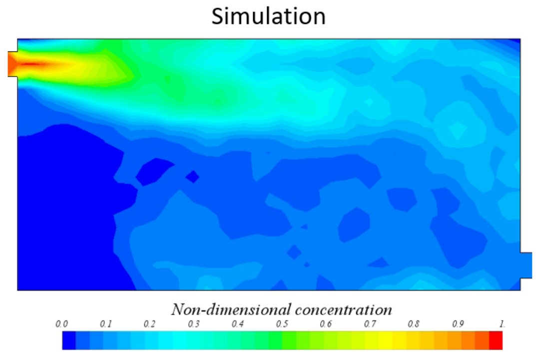
Chen et al. [12] measured the velocity profile and particle distribution inside the room, and the developed CFD-Lagrangian model was used to replicate their experiment. Figure 2 compares the experimental and computational results plotted on the central section (profile direction) of the room. It can be seen that the simulation replicated the experiments well, where the general trend and local quantities are very close; noting that the simulation could not generate exactly the same style of pictures as in Chen et al. [12], this is because the experiment used a different measurement method. The comparison demonstrates the computational approach used is suitable to study the airborne transmission of COVID-19 virus that is similar to a flu virus. This validation analysis has also indicated the suitable mesh density and timestep size for the simulation, which are respectively 0.05 m and 0.001 s.

Figure 1: A rectangular room case to validate the computational model



(a) Velocity profile





(b) Particle concentration

Figure 2: Experimental and computational comparison of results, measured on the central section (profile direction) of the room

3. RESULTS AND DISCUSSION

Upon validation of the computational model, the standard room geometry is replaced by the superstructure of a crew boat existing in real life, as shown in Figures 3 and 4. The vessel is 19.5 m long and 4.5 m wide, with an internal space of approximately 7 m long. The boat has 25 seats inside, and the computational representation of the internal space is in full scale, which is shown in Figure 5. The reason for choosing this boat in the present study is not only that its superstructure is compact but also that its average voyage time is 12 hours, which gives a hazardous environment of COVID-19 transmission where people stay in a crowded space for a fairly long time.

Figure 3: A crew boat of ship operator PT. Pelayaran Nasional Ekalya Purnamasari (PNEP)

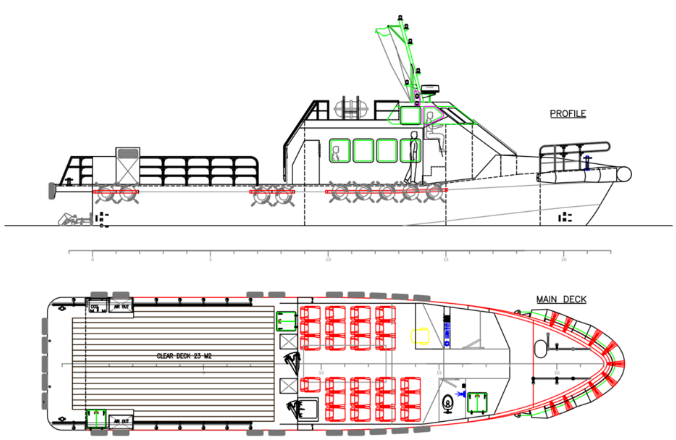


Figure 4: Drawing of the boat's external and internal design

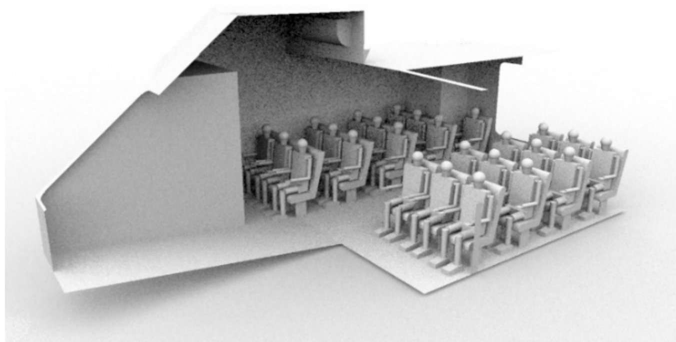


Figure 5: Computational geometry of the boat's internal space

Four case studies have been conducted, as a combination of the situations when a first-row passenger is coughing or speaking, and when the forward door is open or close. The simulation conditions of the four cases are shown in Table 1. Particles are injected after a 10s simulation of airflow, which is to ensure that the CFD solution has converged.

Table 1: Summary of simulation conditions

	Case 1	**Case 2**	**Case 3**	**Case 4**
Forward door	Open	Close	Open	Close
Virus source	Coughing		Speaking	
Injection duration	0.3s, short event		60s, long event	
Inject speed	11.7 m/s		3.1 m/s	
Particle diameter	13.5 μm		16 μm	
Inject particle number	6950 per second		443 per second	

The simulation details of coughing and speaking were set according to the measurements of Chao et al. [3]. Coughing is a short event whose duration is considered to be 0.3s and speaking is modelled to last 60s. The viruses injected through coughing has a higher concentration and initial speed than those from speaking, but overall speaking imports more viruses as its duration is much longer. After the coughing or speaking duration, the simulations were kept running to continue tracking the virus movement.

The boat is assumed to operate at a speed of 6 knots. When the boat forward door is open, a significant airflow forms in the passenger space (Figure 6), which can cause the viruses to move. By contrast, when the forward door is closed, there is no wind inside the boat, so the viruses are considered to only perform gravitational and stochastic motions.

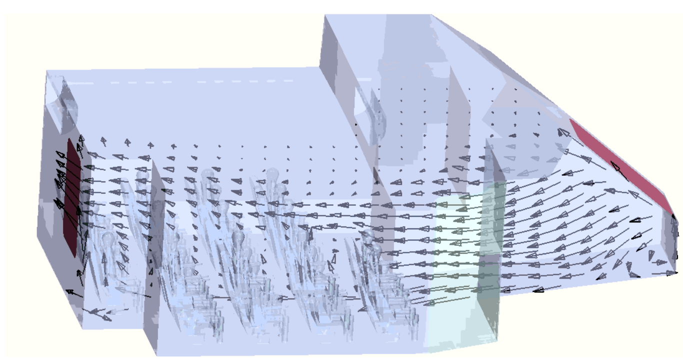
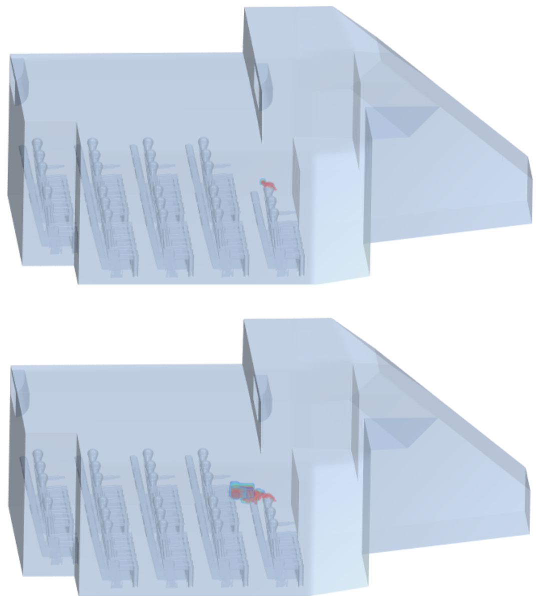
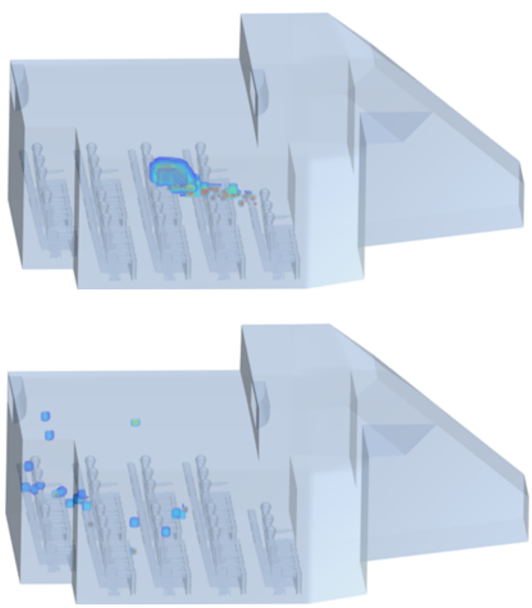


Figure 6: Velocity field inside the crew boat advancing at 6 knots, where winds coming in from the forward door

For the simulation of a passenger coughing with the forward door open, it can be seen in Figure 7 that viruses were coughed out and scattered. Along with the wind flow, the viruses moved rapidly towards the outlet. For around three seconds, the passengers behind the coughing person were extensively contacted by the viruses, although the viruses were mostly gone after 5 seconds.



(a) Upper: 0.1s during coughing; lower: 0.3s during coughing



(b) Upper: 1s after coughing; lower: 5s after coughing

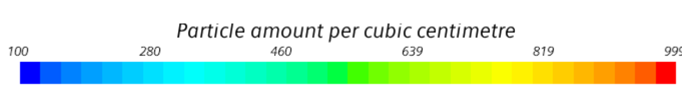
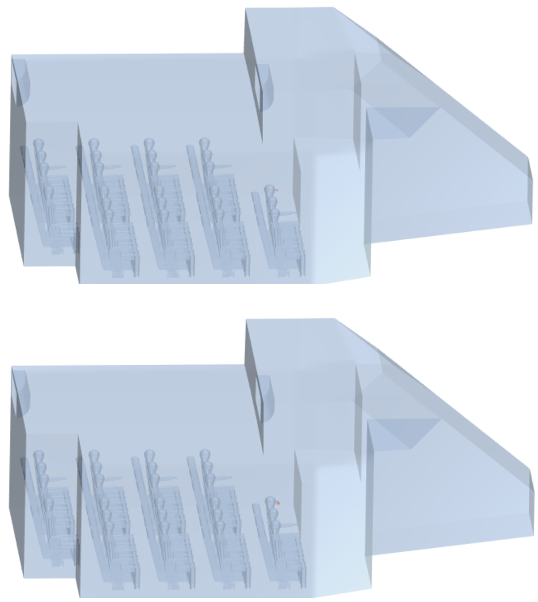


Figure 7: Case 1 - coughing with the forward door open; the contour indicating the particle concentration is applicable for all following pictures.

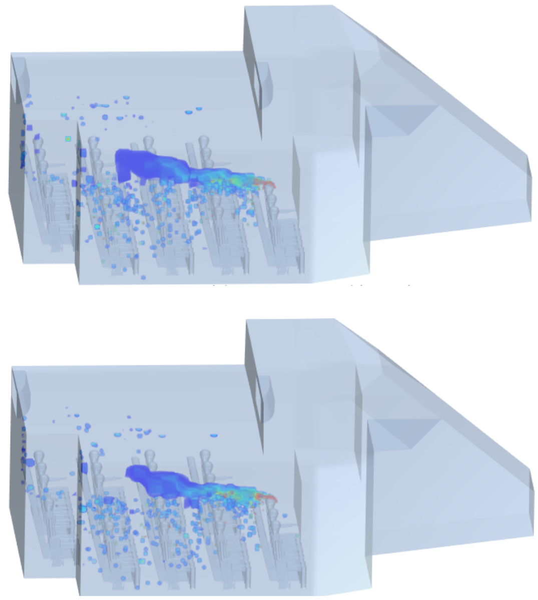
For the simulation of a passenger coughing with the forward door closed, it can be seen in Figure 8 that viruses were coughed out and then only stayed in the area where the passenger sits. The virus movement was mainly a slow sinking due to gravity, alongside a small diffusion due to stochastic motions. Although the viruses stayed in the air for around 100s, the contagion risk appears to be minimal, based on the assumption that coughing is usually not made directly towards other people.



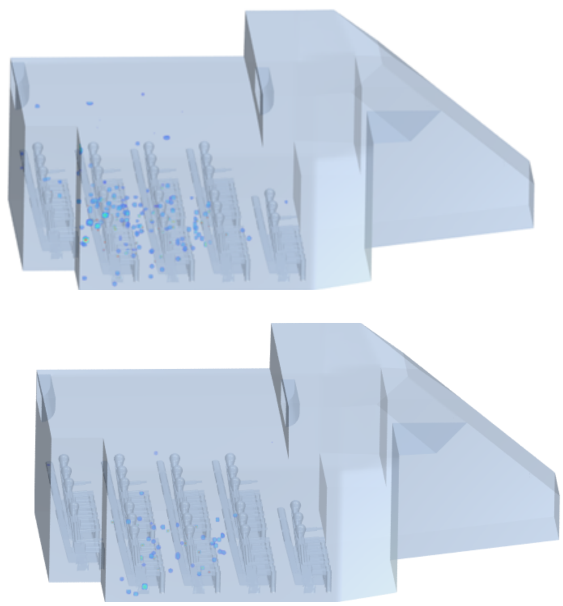
(a) Upper: 0.1s during coughing; lower: 0.3s during coughing

(b) Upper: 50s after coughing; lower: 100s after coughing
Figure 8: Case 2 - coughing with the forward door closed

For the simulation of a passenger speaking with the forward door open, it can be seen in Figure 9 that a wake of viruses was created due to the wind and severely affected the passengers behind for the whole speaking duration, in particular affecting the next two rows. When the speaking stopped, the viruses were carried away from the boat in 10s.



(a) Upper: 30s during speaking; lower: 60s during speaking



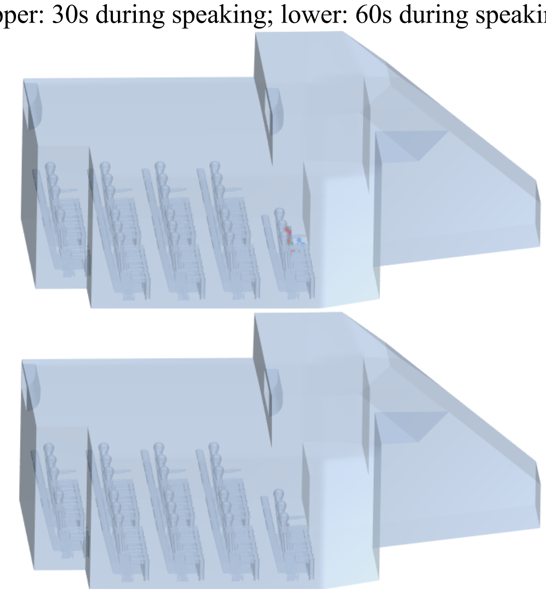
(b) Upper: 5s after speaking; lower: 10s after speaking

Figure 9: Case 3 - speaking with the forward door open

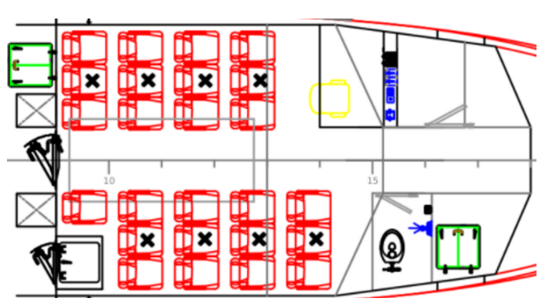
For the simulation of a passenger speaking with the forward door closed, it can be seen in Figure 10 that a significant number of viruses were output, and the viruses remain in a small area in front of the speaking person. However, since speaking usually occurs as one person facing another, a contagion risk remains if the conversating people are sitting too close. Based on the virus coverage observed from the simulation, a safety distance should be at least 0.5m. This confirms that it is necessary to keep social distancing in seats.

Figure 11 suggests two potential seat arrangements for social distancing. The aligned arrangement shown in Figure 11(a) reduces the capacity by 33%, while the crossed arrangement shown in Figure 11(a) reduced 50%, but the former setting cannot avoid the risk of a front passenger who turns around and speaks the passenger behind.

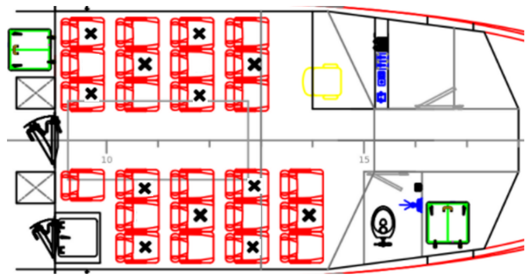
(a) Upper: 30s during speaking; lower: 60s during speaking



(b) Upper: 30s after speaking; lower: 60s after speaking
Figure 10: Case 4 - speaking with the forward door closed



(a) Aligned arrangement



(b) Crossed arrangement (recommended)

Figure 11: Social distancing in seat arrangement – black crosses indicate the seats that should not be used.

4. CONCLUSIONS

A computational model has been established to investigate the airborne transmission of the COVID-19 virus in the superstructure of a crew boat. Initially a generic model was established to enable validation against experimental results measured in a standard room. Then, the room geometry was replaced by the superstructure of a crew boat advancing in open water. Simulations were conducted to study the particulate flow due to a person coughing and speaking, including the scenarios of whether the boat's forward door is open or closed.

It was found that, when the boat's forward door is open, a significant airflow forms in the boat, which carries the viruses to make extensive contacts with the passengers in back rows. When the forward door is closed, the viruses mainly sink due to gravity and the diffusion is limited in a small area, despite that they can stay in the air for more than one minute. The study was conducted for only one ship-speed condition of 6 knots, but the conclusion is expected to be similar for higher speed conditions. Overall, it is suggested to keep the forward door closed to minimise the virus's diffusion, although this might be counter-intuitive.

It is also shown that speaking generally creates a higher risk than coughing, as speaking usually lasts a much longer duration, thereby more viruses. Whilst speaking may have been less altered than coughing which is more likely to be treated as a red flag, the research suggests that more attention should be paid to the COVID-19 risk in daily conversations. When there is no distribution from external airflow, the modelling indicated that a conversation between two people should keep a distance of at least half a meter. Based on this, it is advised for the crew boat that the seats between every two people should be kept empty.

Ongoing further work is to analyse the virus transmission including the influence of air conditioners or ventilation systems. Also, more ship types and operation scenarios should be investigated.

ACKNOWLEDGEMENTS

This work is part of a project that has received funding from the British Council under the Newton Institutional Links Grants - Ensuring the safety of Indonesian seafarers and fishers in the time of COVID-19 and beyond (agreement No. 623457938), in conjunction with the Indonesian Governmental Funding from the Ministry of Education, Culture, and Higher Education (agreement No. 2242/PKS/ITS/2021). The authors appreciate the ship company PT. Pelayaran Nasional Ekalya Purnamasari (PNEP) for sharing the crew boat data.

REFERENCES

[1] Y. Jin, H. Yang, W. Ji, W. Wu, S. Chen, W. Zhang, G. Duan, Virology, epidemiology, pathogenesis, and control of COVID-19, Viruses. 12 (2020) 372.

[2] V. Vuorinen, M. Aarnio, M. Alava, V. Alopaeus, N. Atanasova, M. Auvinen, N. Balasubramanian, H. Bordbar, P. Erästö, R. Grande, Modelling aerosol transport and virus exposure with numerical simulations in relation to SARS-CoV-2 transmission by inhalation indoors, Safety Science. 130 (2020) 104866.

[3] C.Y.H. Chao, M.P. Wan, L. Morawska, G.R. Johnson, Z.D. Ristovski, M. Hargreaves, K. Mengersen, S. Corbett, Y. Li, X. Xie, Characterization of expiration air jets and droplet size distributions immediately at the mouth opening, Journal of Aerosol Science. 40 (2009) 122–133.

[4] N.P. Gao, J.L. Niu, Modeling particle dispersion and deposition in indoor environments, Atmospheric Environment. 41 (2007) 3862–3876.

[5] L. Guo, R. Torii, R. Epstein, J. Rubin, J.P. Reid, H. Li, A. Ducci, R. Balachandran, M.K. Tiwari, Y. Ventikos, L.B. Lovat, Using portable air purifiers to reduce airborne transmission of infectious respiratory viruses – a computational fluid dynamics study, MedRxiv. (2021) 2021.11.01.21265775. https://doi.org/10.1101/2021.11.01.21265775.

[6] Z. Zhang, J. Capecelatro, K. Maki, On the utility of a well-mixed model for predicting disease transmission on an urban bus, AIP Advances. 11 (2021) 085229.

[7] A.B. Liu, D. Mather, R.D. Reitz, Modeling the effects of drop drag and breakup on fuel sprays, SAE Transactions. (1993) 83–95.

[8] F. Menter, Zonal two equation kw turbulence models for aerodynamic flows, in: 23rd Fluid Dynamics, Plasmadynamics, and Lasers Conference, 1993: p. 2906.
[9] B. Pena, L. Huang, A review on the turbulence modelling strategy for ship hydrodynamic simulations, Ocean Engineering. 241 (2021) 110082.
[10] A.D. Gosman, E. Loannides, Aspects of computer simulation of liquid-fueled combustors, Journal of Energy. 7 (1983) 482–490.
[11] L. Huang, J. Tuhkuri, B. Igrec, M. Li, D. Stagonas, A. Toffoli, P. Cardiff, G. Thomas, Ship resistance when operating in floating ice floes: A combined CFD&DEM approach, Marine Structures. 74 (2020) 102817.
[12] F. Chen, C.M. Simon, A.C. Lai, Modeling particle distribution and deposition in indoor environments with a new drift–flux model, Atmospheric Environment. 40 (2006) 357–367.