The effects of room length on jet momentum flux

Wei Liu[1*], and *Mats* Sandberg[2]

[1]Department of Civil and Architectural Engineering, KTH Royal Institute of Technology, Brinellvägen 23, Stockholm, Sweden
[2]Department of Building Engineering, University of Gävle, Gävle, Sweden

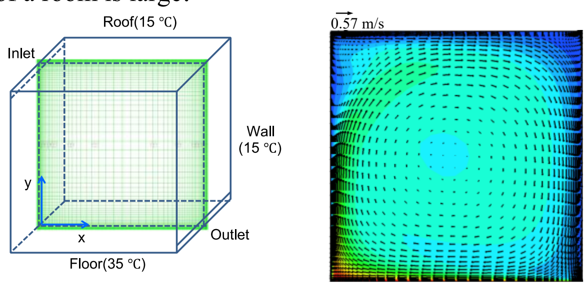
Abstract. As to mixing ventilation in indoor environments, the turbulent jet plays a major role in driving the air movement, contaminant transport, and heat transfer. The main characteristic of a turbulent jet is its momentum flux. By entrainment of air, the flow of a jet increases and may enhance the flooding of contaminant. In investing the jet's momentum flux, it is generally regarded that the supply jet collides with the opposing wall and the jet is transformed into a wall jet. However, this is not always true if a jet is not sufficiently strong, or the length of a room is large. Therefore, this study adopted computational fluid dynamics (CFD) to investigate the supply jet development and its momentum flux by varying the room length. Initially, the width of the air supply inlet was the same with that of the room. By defining n as the ratio of room length and height, when *n* = 3, there is a horizontal a vortex which is the normal behaviour. When the room length increased further, the supply jet was unable to collide with the opposing wall. This investigation got two vertical vortices at the room end which is new. The two new vertical vortices were most pronounced for *n* = 5. It is possible that increasing the length of the room introduces a gradual transition towards a flow in a rectangular duct. This flow is probably very much governed by the side walls. Therefore, this study reduced the width of the air supply inlet by half and maintained the same flow rate. However, a single vertical vortex was identified at the room end for *n* = 5. In both scenarios, the supply jet may create new vortices that would enhance the flooding of contaminants.

1 Introduction

In built environments equipped with mechanical ventilation, a turbulent jet plays a major role in driving the air movement, contaminant transport, and heat transfer. The intention of supplying air by jet is maintaining acceptable indoor air quality by diluting and removing the contaminants [1]. In some scenarios, the jet is also expected to deliver coolth or heat to ensure a thermally comfortable environment [2]. However, the jet in a room might be unable to realize the expected outcomes according to the fact of cross infections during the COVID-19 pandemic [3].

As highlighted in [4], the jet in a room plays a dual role. Besides the intended aims mentioned above, a turbulence jet may help spread indoor contaminants. This is because the created air distribution is unable to remove the contaminants immediately. In a mixing ventilation system, the air as well as the contaminants are recirculated. In a poorly ventilated space, the transmission of infectious contaminants would be enhanced [5]. Therefore, it is important to investigate the development of the jet and how it drives the flooding of contaminants.

The main characteristic of a turbulent jet is its initial momentum flux M_{in}[N] [6]. By entrainment of air, the flow of a jet increases and may enhance the flooding of contaminant. In investing the jet's momentum flux, it is generally regarded that the supply jet collides with the opposing wall. At the collision with the opposing wall, there is a reaction force R. With both supply and extract terminals located on the same wall, the normalized reaction force $\frac{R}{M_{In}}$ is between 1 and 2. With supply and extract terminals on opposite walls the normalized reaction force is between 0 1nd 1. Recorded reaction forces are reported in Figure 12 in [6]. The jet is then transformed into a wall jet. For example, Fig. 1(a) shows a cavity with a horizontal plane jet and a heated floor [7]. A validated computational fluid dynamics (CFD) simulation gives the air distribution as shown in Fig. 1(b). The supply horizontal jet collides with the right wall and creates a vertical wall jet. However, this is not always true if a jet is not sufficiently strong, or the length of a room is large.



(a) Geometry and mesh (b) Air distribution [8]

Fig. 1. A ventilated cavity with a horizontal wall jet.

If a jet is unable to reach the wall, secondary vortexes in the near-wall region might be created. Such vertices would enhance the cross infection as the

* Corresponding author: weiliu2@kth.se

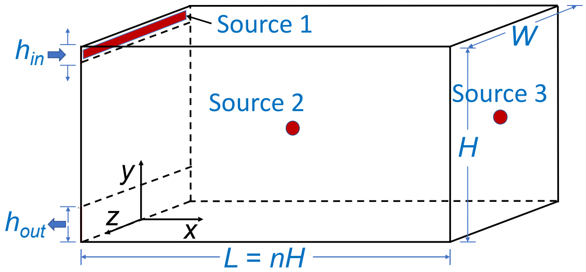
 https://doi.org/10.1051/e3sconf/202235601001

corresponding region could be a stagnation zone for the contaminants. Therefore, this study adopted CFD simulations to investigate the supply jet development, its momentum flux, and the possible creation of secondary vortexes by varying the length of a room with horizontal wall jet. The impact of the jet width and location of outlet on the jet development was also investigated.

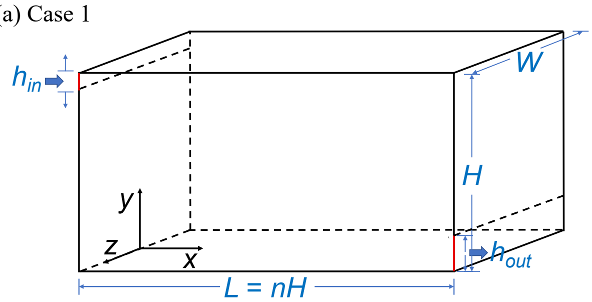
2 Methods

This section introduces the case setup and CFD models.

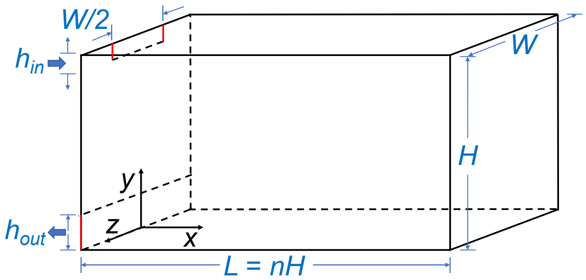
2.1 Case setup



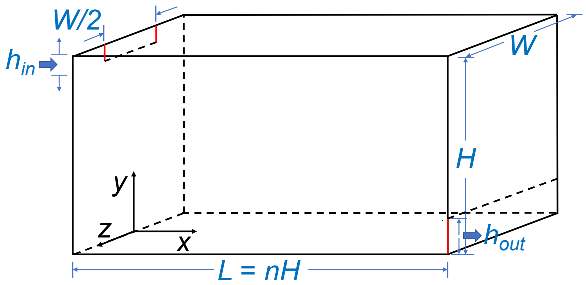
(a) Case 1



(b) Case 2



(c) Case 3



(d) Case 4

Fig. 2. A ventilated room with a horizontal wall jet.

Fig. 2 shows the geometry of four cases considered in this study. The dimension of the room was L x W x H, where $H = W = 3$ m, $L = nH$, and $n = 3, 4, 5, 6$. n was the ratio of room length and height. A greater n is not realistic. An inlet was located at the top of the left wall and an outlet was located at the bottom of the left wall in cases 1 and 3 and the right wall in cases 2 and 4. The heights of the inlet and outlet were $h_{in} = 1$ cm and $h_{out} = 4$ cm, respectively. In cases 1 and 2, the inlet width was the same with that of the room, which was W. In cases 3 and 4, the inlet width was $W/2$. The air supply velocity U_{in} was 10.8 m/s for cases 1 and 2, and 21.6 m/s for cases 3 and 4. The turbulence intensity of the supply jet was assumed to be 5%. Since no heat source was considered in this study, the flow was assumed to be isothermal. For $n = 3$ scenario, the grid-independent mesh had 1.6 million hexahedron cells. For greater n, the same grid size was used in generating the mesh.

2.2 CFD setup

This study ran all the simulations with simpleFoam solver in OpenFOAM [9], which is an open source CFD program. simpleFoam is a steady-state solver for incompressible, turbulent flow. As the name of the solver indicates, it adopts the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm to decouple the velocity and pressure. The convection terms were discretized by the second order upwind scheme. To simulate the turbulence, this investigation used the Re-Normalisation Group (RNG) k-ε model [10]. This study ran all the simulation with 10,000 iteration that ensured the residuals were less than 10^{-4} for all the variables.

3 Results

Using case 1 as the baseline, this study shows the predicted streamlines for this case in Fig. 3 first. When $n = 3$ or 4, there was a horizontal vortex which was the normal behavior that could also be observed in Fig. 1(b). When the room length was increased further, the supply jet was not sufficiently strong to collide with the opposing wall. For $n = 5$ and 6, this investigation got two vertical vortices at the room end which were new. The two new vertical vortices were most pronounced for $n = 5$. When $n = 6$, the flow close to the right wall was rather low-speed and complicated.

This study further plots the normalized static pressure $p^* = p/(\rho U_{in}^2/2)$ along the inlet jet at $y = H - h_{in}/2$, $z = 0$ in Fig. 4. $\rho = 1.225$ kg/m^3 is the air density. Along with the major horizontal vortex, the air pressure varied a lot. At the minimum static pressure, the air had the highest speed. When n > 3, locations of the minimum static pressure did not change much, indicating the minor change of the major horizontal vortex. When x/H > 3.65, where the secondary vertical vortices showed, the air pressure was close to a constant value. A possible reason was the low-speed airflow in the secondary vertical vortices.

 https://doi.org/10.1051/e3sconf/202235601001

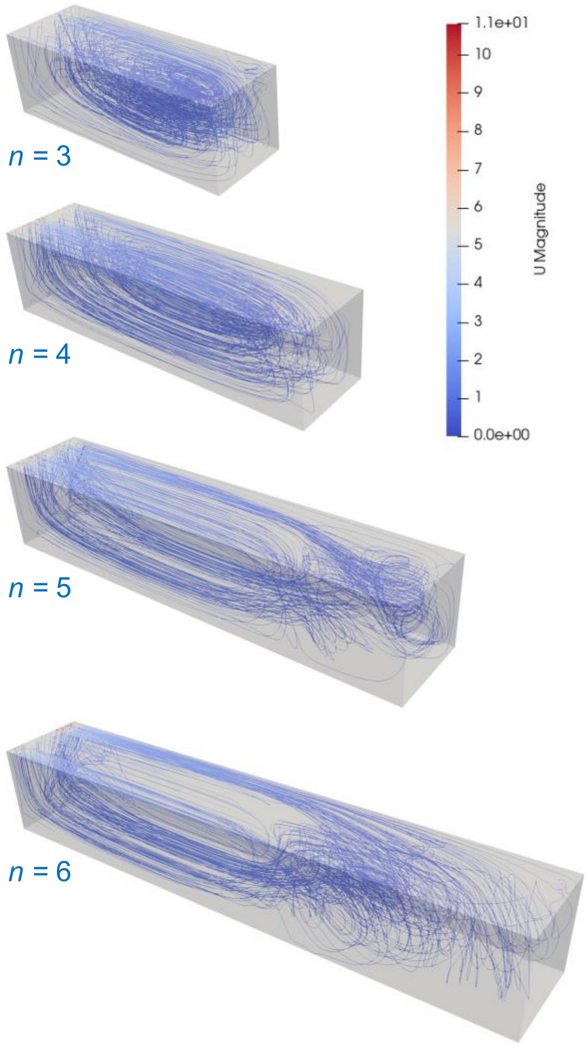


Fig. 3. Predicted streamlines for case 1.

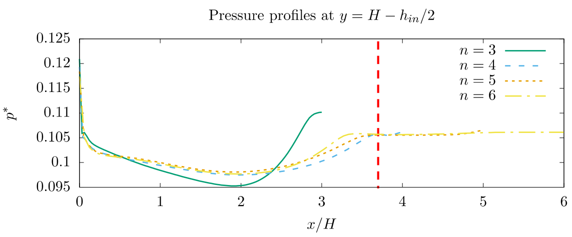


Fig. 4. Normalized air pressure along the inlet jet at at $y = H - h_{in}/2$, $z = 0$ for case 1.

Case 2 had the similar setup with case 1 except that the outlet was located at the bottom of the right wall. When $n = 3$, the flow of case 2 was quite similar with that of case 1. However, two vertical vortices at the room end were evident when $n = 4$ of case 2 as shown in Fig. 5. Therefore, placing the inlet and outlet at opposite walls would increase the possibility of secondary vertices.

This investigation again plots the normalized static pressure $p^* = p/(\rho U_{in}^2/2)$ along the inlet jet at $y = H - h_{in}/2$, $z = 0$ in Fig. 6 for case 2. In those regions close to the right wall, the fluctuations of pressure were evident, which indicates higher speed airflow than that in case 1. There were also multiple local minimums of the static pressure. The global minimum indicated the major horizontal votex and the other local minimums indicated secondary vortices.

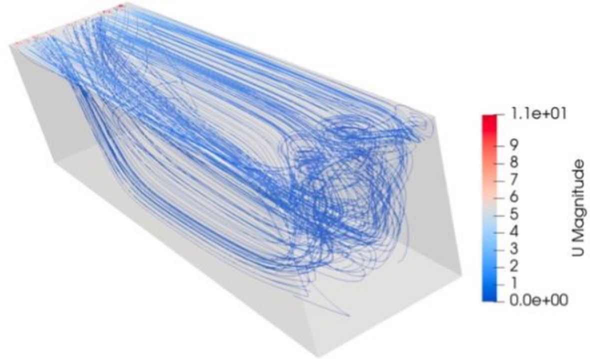


Fig. 5. Predicted streamlines for case 2, $n = 4$.

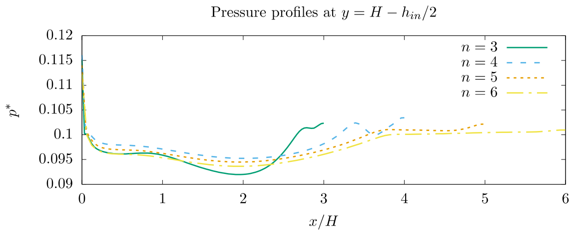


Fig. 6. Normalized air pressure along the inlet jet at at $y = H - h_{in}/2$, $z = 0$ for case 2.

For cases 1 and 2, it was possible that increasing the length of the room introduced a gradual transition towards a flow in a rectangular duct. This flow was probably very much governed by the side walls. Therefore, this study reduced the width of the air supply inlet by half and maintained the same flow rate. Then air supply velocity of cases 3 and 4 was $U_{in} = 21.6$ m/s. In case 3, the inlet and outlet were on the same side wall, however, the supply jet created evident secondary vortices even for $n = 4$, as shown in Fig. 7. In contrast, for case 4, when the inlet and outlet were on the opposite side walls, this study got a minor secondary vortex in the scenario n = 5 as shown in Fig. 8. Despite of the different setups in all the cases, the supply jet was very likely to produce secondary vertices if the ratio of room length and height was greater than 3.

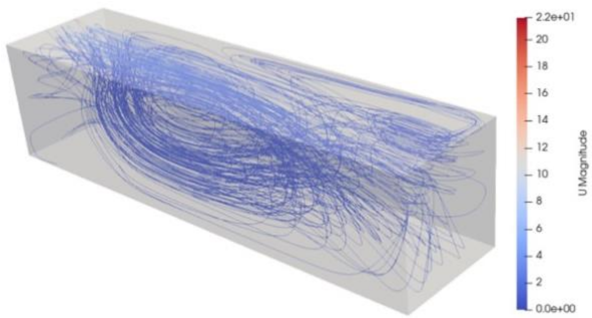


Fig. 7. Predicted streamlines for case 3, $n = 4$.

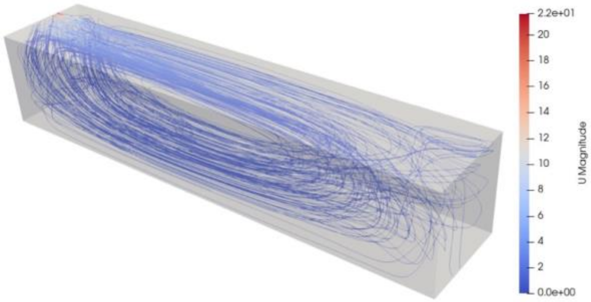
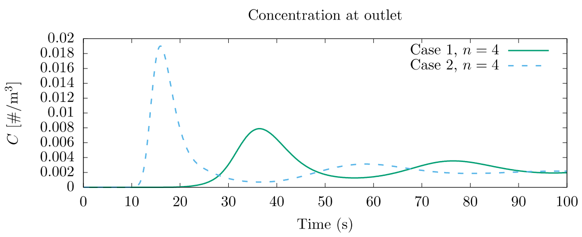


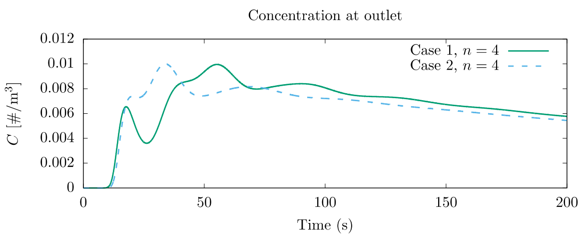
Fig. 8. Predicted streamlines for case 4, *n* = 5.

4 Discussions

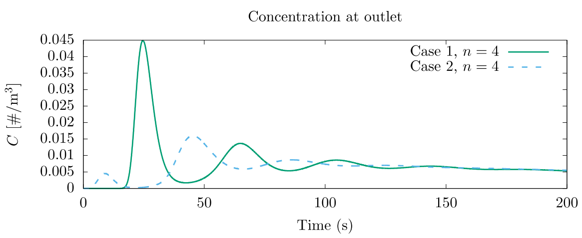
To test the efficiency of removing contaminants in the room, this study placed contaminant sources at the inlet, centre of the room, and end of the room for case 1, n = 4 and case 2, n = 4 (refer to Fig. 2). The major differences of these two scenarios were the location of outlet and further the secondary vortices. The source at the inlet was created by injecting contaminant with the jet for one second at concentration 1 #/m^3. The sources at the room centre ($x = 2H$, $y = H/2$, $z = 0$) and end ($x = 3.83H$, $y = H/2$, $z = 0$) are volumetric sources with strength 1 #/s for one second.



(a) Source from the inlet jet at 1 #/m^3 (source 1)



(b) Volumetric source 1 #/s at the room centre (source 2)



(c) Volumetric source 1 #/s at the room end (source 3)

Fig. 9. Predicted contaminant concentration at the outlet.

Fig. 9 shows the monitored contaminant concentrations at the outlet. When the contaminant was injected at the inlet (Fig. 9(a)), the contaminant concentration at the outlet of case 2, *n* = 4 had a greater and earlier spike than that of case 1, *n* = 4, which implied that the flow in case 2, *n* = 4 had better performance in removing the contaminant. It was possibly due to the location of outlet. When the contaminant was injected at the room centre, the two scenarios had very similar performance. The flow in case 2, *n* = 4 was slightly better. In the last scenario, the flow in case 1, *n* = 4 had significant better performance in removing the contaminant than that of case 2, *n* = 4. It confirmed that the secondary vortices would trap the contaminant and decrease the efficient of the ventilation.

5 Conclusions

This study numerically tested the effect of the room length/height ratio on creating secondary vertices. It was found that when the ratio was greater than 3, the air supply jet could possibly create secondary vertical vortices at the room end. The creation of the secondary vortices was also affected by the location of the outlet and the width of inlet.

By placing contaminant sources in the room, this investigation observed that the secondary vortices could trap the contaminant and reduce the ventilation efficiency especially when the source is located within the secondary vortices. Otherwise, the performance of ventilation in removing the contaminants was also affected by the location of the outlet and flow path.

This study was partially supported by the Digital Futures, C3.ai Digital Transformation Institute.

References

1. Sundell, J., 2004. Indoor air, 14(s 7), pp.51-58.
2. Zuazua-Ros, A., Martín-Gómez, C., Ibañez-Puy, E., Vidaurre-Arbizu, M. and Gelbstein, Y., 2019. Renewable Energy, 131, pp.229-239.
3. Srivastava, S., Zhao, X., Manay, A. and Chen, Q., 2021. Sustainable Cities and Society, 75, p.103408.
4. Sandberg, M., 2022. Frontiers in Built Environment, 7.
5. Li, Y., Qian, H., Hang, J., Chen, X., Cheng, P., Ling, H., Wang, S., Liang, P., Li, J., Xiao, S. and Wei, J., 2021. Building and Environment, 196, p.107788.
6. Karimipanah, T. and Sandberg, M., 2014. International Journal of Ventilation, 13(3), pp.285-298.
7. Blay, D., 1992. HTD Vol. 213, Fundamentals of Mixed Convection.
8. Liu, W., Jin, M., Chen, C., You, R. and Chen, Q., 2016. Numerical Heat Transfer, Part A: Applications, 69(7), pp.748-762.
9. Jasak, H., Jemcov, A. and Tukovic, Z., 2007, September. In International workshop on coupled methods in numerical dynamics (Vol. 1000, pp. 1-20). IUC Dubrovnik Croatia.
10. Yakhot, V.S.A.S.T.B.C.G., Orszag, S.A., Thangam, S., Gatski, T.B. and Speziale, C., 1992. Physics of Fluids A: Fluid Dynamics, 4(7), pp.1510-1520.