Investigation on cross-vent design in building drainage system by numerical simulation approach

Eric Wai-Ming Lee[a], Horace Kwok-Wai Mui[b], Ling-Tim Wong[b]

[a]Department of Architecture and Civil Engineering, City University of Hong Kong, Kowloon Tong, Hong Kong; [b]Department of Building Environment and Energy Engineering, Hong Kong Polytechnic University, Hung Hom, Hong Kong

ABSTRACT

Air pressure fluctuation in building drainage system is always a problem if proper venting is not provided. Excessive positive pressure may push the foul to interior of the building through the trap seal. Excessive negative pressure may empty the trap seal. Such consideration is especially important for vertical drainage stack. In presence, secondary ventilated system as specified in British Standard BS12056-2 has been widely adopted in Hong Kong. The system adopted a vertical vent stack running in parallel with a vertical drainage system. To reduce the pressure fluctuation inside the drainage stack, cross-vents are provided to connect the drainage stack to the vent stack at every two or three floors. The purpose of the cross-vent is to relief the excess air pressure inside the drainage stack. However, investigation on the design of the cross-vent is still very limited. This study adopted computational fluid dynamics techniques to investigate the cross-vent design. This study found that, practically, cross-vent should be provided to each floor which is in line with the recommendations stated in China standard and British standard.

KEYWORDS

Building drainage; computational fluid dynamics; cross-vent

1. Introduction

Building drainage system is one of the oldest building services system. It draws the society's concerns in the recent decades since it was considered as a way for spread of bacteria. In year 2003, building drainage system was confirmed to be a way to spread the SARS (WHO 2003; Jack 2006; Zhang et al. 2021) which killed 399 peoples in Hong Kong. In recent years, building drainage system is also suspected to be a way for spreading COVID-19. The spread of bacteria is due to the ingress of foul air into a toilet space through its floor drain with empty trap seal and the negative air pressure of the toilet induced by its extraction fan.

Apart from evaporation, loss of water seal of the floor drains or other sanitary fixtures is most likely caused by the excessive negative air pressure inside the vertical drainage stack. Such negative air pressure usually occurs above the base of the stack. Air pressure at the top of the vertical drainage stack equals to atmospheric pressure. The annular water flow inside the vertical drainage stack draws air from the top of the stack. Along the dry section at the top the stack, friction between flowing air and stack reduces the air pressure. When air is passing through a section with branch discharge, the discharge flow occupies part of the cross-sectional area of the stack. The free area for air passing through at that section is reduced. It is similar to an orifice at which the pressure at the downstream is reduced. Therefore, the air pressure is further reduced. Such reduction in air pressure has been well investigated by different researchers (Verma, Chakrabarti, and Khanna 1976; Swaffield 2010). These studies also depicted the positive air pressure occurring at the base of the vertical stack. When water flowing to the base of the stack, it changes its direction from vertical to horizontal. A water curtain emerges at the beginning of the horizontal section. It restricts air coming from upstream to pass through. The air accumulates and increases its pressure until it is sufficiently high to 'blow' through the water curtain. It explains the existence of positive pressure at the base of the stack. Whenever excess positive pressure exists (i.e. air pressure is higher than the 75mm water seal) at the base of the vertical drainage stack, foul air may ingress into the toilet space via the trap of the floor drain of sanitary fixtures connected to the positive pressure section of the stack.

The industry aware of the hygienic problems of the excessive air pressure inside the vertical drainage stack. One of the common ways to prevent the excessive pressure is provision of a vertical vent stack running in parallel with the vertical drainage stack with cross-vents provided to connect the drainage stack and vent stack for every two or three floors. However, investigation on the cross-vent design is still limited.

2. Cross-vent designs and investigations

2.1. International standards on cross-vent provision

The statutory requirement on the design cross-vent in Hong Kong is specified in ArchSD (2017) that cross-vents should be provided to connect the vertical vent stack and drainage stack for every 5 floors. It is the only statutory requirement on cross-vent in Hong Kong. The statutory requirement on cross-vent in Singapore is stated in PUB (2021). The requirement comprehensively specifies the locations of the cross-vents to be provided in a building drainage system. It specifies that, for a fully ventilated system, a cross-vent should be provided at mid-height of a building up to 20 storeys. For buildings higher than 20 storeys, cross-vent should be provided for every 10 storeys. If the building is a ventilated stack system (i.e. no vent pipe is provided for any soil fitment), cross-vent should be provided at every 3 storeys. The Uniform Plumbing Code (IAPMO 2021) in United States also specifies the provision of yoke vents (i.e. cross-vents). It should be provided to connect the vertical drainage and vent stacks for every 5 floors starting from top for pressure relief. The size of the cross vent should be the same as the vertical drainage stack or the vertical vent stack whichever smaller. The national standard in China (MHURD 2019) also stated the requirement of cross-vent which should be provided for every floor or every two floors but should not be more than 8 floors. British Standard BS-12056-2 (BSI 2000) illustrates the provision of cross-vents in their secondary ventilation system configurations. Cross-vent is advised to be provided at each floor to connect the vertical vent and drainage stacks. Also, the size of the cross-vent is advised to be the same diameter as the vertical vent stack. We can see that cross-vent is an international common practice to be provided for relieving excessive air pressure inside building drainage system.

2.2. Recent researches related to cross-vent

Sharma, Chakrabarti, and Khanna (1981) investigated the venting of drainage stack with the presence of vent stack and cross-vents. An experiment was carried out and found that air is drawn from the vent stack through the cross-vents to the main drainage stack. A mathematical model was developed to correlate the water flow inside the stack and the suction created. However, this study only studied one scenario without addressing the effect caused by different cross-vent provisions. Although it did not address the effect of cross-vents' locations, it is one of the pioneer studies in study of cross-vent. Swaffield and Campbell (1995) developed a one-dimensional model to correlate the air pressures at stack suction and base of stack with the water flow rates. It was also applied on a single stack system and modified one pipe system (i.e. drainage stack with vent stack and cross-vents) to estimate the air pressure and entrained air flow at the base of the drainage stack. The results showed that the air pressure at the base of the stack was reduced by the cross-vents. The study focused only on design of cross-vent for every floor. Variation of the cross-vent provision was not included in their study. Swaffield and Jack (2004) applied AIRNET (Jack and Swaffield 1999) model to simulate air pressure transient propagation inside a modified one pipe system under different defective conditions. This study investigated the response of the system with cross-vents implemented. This study developed a very powerful model to investigate the dynamic air pressure distribution in the building drainage network. However, it was only applied to study the cross-vent provision for every 2 floors.

Gormley et al. (2021) recently investigated the current building drainage system design for tall buildings. It was found that, for a tall building, the vent pipe and cross-vents should be at least the same diameter as the main stack. This study draws attention of engineers and scientists that the current design practice on building drainage system is required to be reviewed. Cheng et al. (2008) carried out full-scale experiment on a fully-ventilated drainage system (i.e. one drainage stack and one vent stack with cross-vents connected) to investigate the peak air pressure inside the system. The experiment was carefully designed to collect air pressure along the vertical drainage stack under different flow rates in few cross-vent configurations. Although there was only one branch discharging water into the drainage stack, the result showed that, in general, the air pressure fluctuation along the drainage stack is minimum when cross-vent was provided to every floor. Their experiment allowed only one floor discharge to the stack and the flow rate achieved almost the design flow rate of the 100mm diameter vertical stack (i.e., ¼ full-bore flow at around 4 litres/s). This scenario is not common. Usually, wastewater is discharging from different floors. It inspired the design of our methodology. Cheng et al. (2010) conducted another comprehensive experiment to determine the characteristics of air pressure fluctuations along vertical drainage stack. The test rig consists of a 38 m high vertical drainage stack running in parallel with a vent stacks from roof, 12th floor, 11th floor down to 1st floor then turning to horizontal run. Cross-vents connecting the two stacks were provided in every floor. Water was discharged from a single inlet in different levels with different flow rates. The experimental results showed failure of water seal at traps may likely to occur between a quarter and half of the stack height from its base. The experiment provides data to this study for model verification.

The above reviews summarize that cross-vent is a common practice in building drainage system design when fully ventilated one-pipe system is adopted especially for high-rise buildings. Although the above researches included cross-vents into their studies, investigation on the effect of cross-vents with different configuration and different scenarios of branches discharge is still very limited.

3. Methodology

3.1. General

This study is to investigate the optimum provision of cross-vents to the fully ventilated one-pipe system (i.e. a pair of vertical drainage and vent stacks with cross-vents connected) for achieving the minimum variation of air pressure along the length of the vertical drainage stack. Since different scenarios will be studied, computational fluid dynamics (CFD) is adopted to simulate the flow of air and water inside the drainage system and the air pressure along the drainage stack will be captured for analysis. The following two parameters are varied to observe the change of the system performance.

a. Number of floors between two consecutive cross-vents

Referring to the statutory requirements summarized in section 2.1, the number of floors between two consecutive cross-vents varies in different countries.

b. The pattern of the discharge branches

Referring to the pioneer researches in section 2.2, the experiments were usually carried out with single branch discharge. The approach deviates from the actual discharge pattern of a building in which multi-branches discharge more likely happens.

3.2. Model geometry

We adopted Ansys' Fluent (2016) to carry out the CFD simulation in this study. A full-scale CFD model is established according to the geometry of the experimental setup of the full-scaled drainage test carried out by Cheng et al. (2008); Cheng et al. (2010). The test rig was constructed in a tower. A 100 mm diameter drainage stack runs from 13/F with open vent down to low level 1/F and turn to horizontal run. A 75 mm diameter vertical vent stack is provided in parallel with the drainage stack. It connects to the drainage stack at the high level of 1/F and also at high level of 12/F. Starting from 12/F down to 2/F, 50 mm diameter cross vent is provided at low level of each floor. A schematic diagram as shown in Figure 1 illustrates the CFD geometry of the model. The total number of meshes of different cases ranged from 2.1 million to 4.4 million. We adopted FLUENT's feature to convert the meshes from tetrahedral to polyhedral for improving the speed of computation and the accuracy of simulation. Sensitivity study was conducted to justify the adopted mesh size. It was found that, generally, there was less than 1% in the simulation result by reducing the mesh size to 90%.

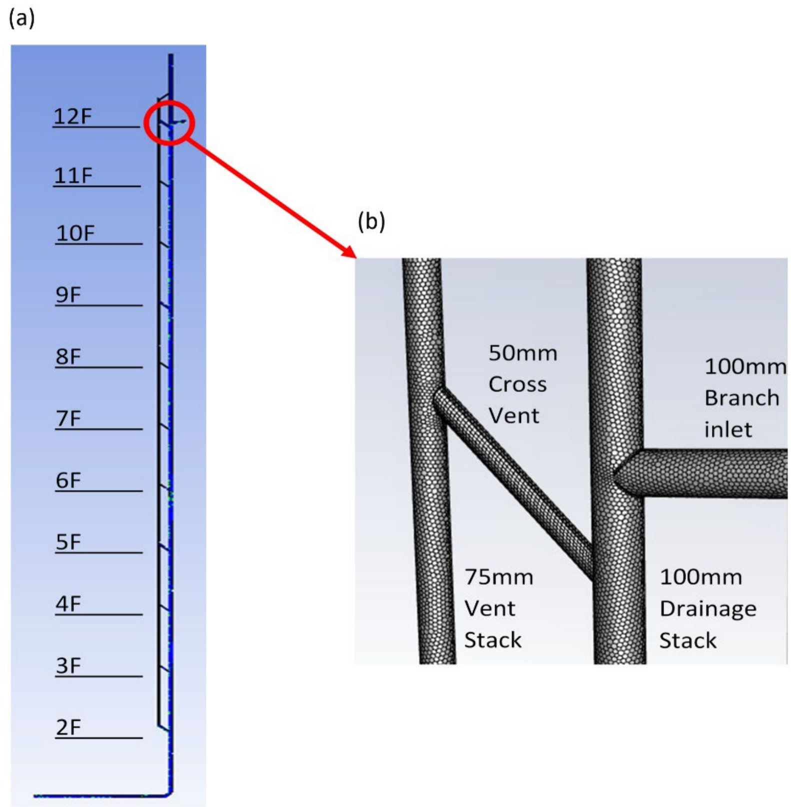

Figure 1. Geometry of the constructed CFD model of the building drainage system. (a) Geometry of the drainage system; (b) Blow-up detail of pipe connections.

3.3. CFD model

Boundary conditions of the CFD model are set as follows. Regarding the setting of boundary conditions, atmospheric pressure is applied to the top of the vertical drainage stack to simulate the open end and the discharge of the horizontal drain. A 100 mm branch drain is connected to the drainage stack at low level of each floor from 12/F down to 2/F. If a branch has no discharge to the vertical drainage stack in a scenario, the branch is blanked to simulate the trap seal of the connected sanitary fitments. Since the design flow rate of a 100 mm diameter vertical drainage stack is 4.0 l/s, in a scenario with discharges from N no. of branches, the flow rate of each branch is determined to be 4/N l/s. At the inlet of each branch, it is assumed to be partially flow with velocity linearly reduced to zero from free surface to pipe invert. Boundary of the branch above the free water surface is set to be atmospheric pressure. The length of the branch drain is taken to be 1m long to allow sufficient length for the flow development from the boundary condition to the connection to the vertical stack.

Since water and air are flowing inside a drainage system, in order to investigate the flow behaviours of both fluids, multiphase flow with Euler-Euler approach is adopted. Different phases of fluids are treated mathematically as continua. The volume fraction (VF) of the fluids are assumed to be continuous function in space and time and the sum of the VFs of all phases equals to one. Volume-of-fluid (VOF) model is used in this study for tracking the interface between air and water. It is purposed for simulating the interface between immiscible fluids of the multiphase flow condition. The VFs of all fluids are tracked throughout the space and time domains and one momentum equation is shared by the fluids

The VOF (Gopala and van Wachem 2008) model, which attempts to track the interface based on the mass conservation, has been widely adopted. Nevertheless, it suffers from the well-known numerical diffusion problem causing significant difficulty to track and retain high curvature or shape interface. In the VOF model, the sum of the VFs of all fluids inside any cell of the building drainage system (i.e. air, water and sediment) equals to one as described in equation (1) where α_q is the VF of the qth fluid inside a control volume.

$$\sum_{q=1}^{n} \alpha_q = 1. \tag{1}$$

If the VF of the qth fluid in a cell is denoted as α_q, the following scenarios are possible.

- Scenario 1 $(\alpha_q = 0)$ represents the cell is empty of the q^{th} fluid;
- Scenario 2 $(\alpha_q = 1)$ represents the cell is full of the q^{th} fluid; and
- Scenario 3 $(0 < \alpha_q < 1)$ represens the cell contains the interface between the q^{th} fluid and one or more other fluids.

The interface(s) between the phases is tracked by the solution of the continuity equation for the VF of one or more phases. The VF equation of the qth fluid can be written as equation (2) where ρ_q is the density of the q^{th} fluid. The $\vec{v}_q$ and α_q are the velocity vector and VF of the q^{th} fluid and $p = 1,2,\cdots n$ is the suffix of the fluids in the multiphase flow. The $\dot{m}_{pq}(\dot{m}_{qp})$ is the mass transfer from phase $q(p)$ to phase $p(q)$. Equation (2) describes the volume transfer of the qth fluid in space and time domains of a control volume.

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) = \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp})\right]. \tag{2}$$

The material property of the control volume is approximated by the material properties of the fluids weighted averaged by their VFs as shown in equation (3) where ξ_q and ξ are respectively the material property of the q^{th} fluid and the representative material property of the control volume.

$$\xi = \sum_{q=1}^{n} \alpha_q \xi_q. \tag{3}$$

Similar to the material property, the velocity field is also shared among the fluids within the control volume. It is solved by a single momentum equation as shown in equation (4) where p

is the pressure and $\vec{g}$ is the gravitational force. It describes that the change of momentum in space and time domains equals to the sum of the forces including the differential pressure, shear stress and gravitational force acting on the control volume.

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + [\mu(\nabla\vec{v} + \nabla\vec{v}^T)] + \rho\vec{g}. \tag{4}$$

A standard finite-difference interpolation schemes and the explicit approach are used for the VF equation. The face values of the q^{th} VF are interpolated by using geometric reconstruction scheme. A realizable k-ε model is used to model the turbulent viscosity. A fractional step method is used as a velocity-coupling scheme. Secondary order spatial discretization scheme is used for momentum and turbulent kinetic energy equations. Since this study focuses on the effects of different cross-vent configurations under different discharge patterns to the air pressure distribution inside the vertical drainage stack, simulation on steady state is adopted.

3.4. Cases to be investigated

According to the results of the previous researches in section 2.2, reduction in flow rate will incur less pressure variation. Therefore, we would adopt the design flow of a stack in the CFD simulations. In this study, 100mm diameter drainage stack is adopted. According to Table 11 of BS12056-2 (BSI 2000), the design flow rate of 100 mm diameter drainage stack is 4.0 l/s and also widely adopted as an industrial design practice. Therefore, this flow rate is adopted throughout this study. Discharge pattern from branches is also considered in this study. In each discharge pattern, the sum of total discharge rates from branches should not exceed the design flow rate of the stack. That is, in case of multiple branches discharge, the design flow rate is equally shared between the branches. Table 1 summarizes the cases to be simulated in this study. A cross-vent is provided for every N no. of floors. The cases with N=0 mean that no cross vent is provided. Four different discharging patterns with single, two, three and four branches discharge are studied. It covers most of the possible discharge patterns from branches. In order to cover the recommendations of international standards as shown in section 2.1, the number of floors between two consecutive floors varies from zero floor to 4 floors as shown in Table 1.

Table 1. Summary of the cases to be investigated by CFD simulations.

N = no. of floor per cross-vent	Branch(es) discharging water into drainage stack			
	12/F (Scenario 1)	12/F, 7/F (Scenario 2)	12/F, 8/F, 4/F (Scenario 3)	12/F, 9/F, 6/F, 3/F (Scenario 4)
0	Case 1-0	Case 2-0	Case 3-0	Case 4-0
1	Case 1-1	Case 2-1	Case 3-1	Case 4-1
2	Case 1-2	Case 2-2	Case 3-2	Case 4-2
3	Case 1-3	Case 2-3	Case 3-3	Case 4-3
4	Case 1-4	Case 2-4	Case 3-4	Case 4-4

3.5. Assessment on air pressure profile

In general, the air pressure is negative near the top of the drainage stack and gradually changes to positive when approaching the base of the stack. To achieve minimum impact on the connected trap seals, the pressure variation along the stack should be minimized. Since root-mean-square (RMS) value is in fact measuring the variance from zero mean, we propose to adopt RMS value of the air pressures along the stack to represent the performance.

4. Results and discussion

The model has been established. We acquired the experimental result of Case 1-1 (Cheng et al. 2010) from the authors of [4]) for calibrating our CFD model by adjusting the mesh size to compromise between the computational speed and the accuracy. The maximum mesh size was determined to be 10mm after the calibration. The experimental result and the CFD simulation result after calibration are plotted in Figure 2. It shows that the coefficient correlation between them is 0.97. We may conclude that the CFD simulation result achieves a reasonable agreement with the experimental result. This calibrated CFD model is used to evaluate the system performance in the following studies.

(a) (b)

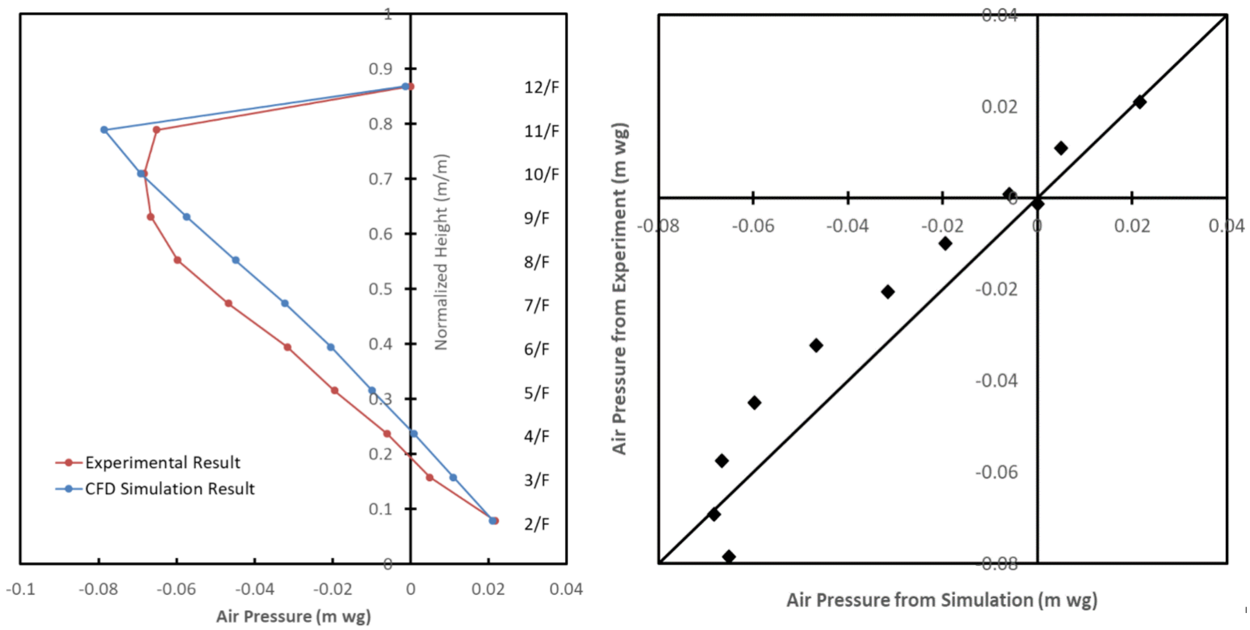

Figure 2. Experimental result and simulation results of Case 1-1. (a) Air pressure distribution inside the vertical stack. (b) Plot of experimental and simulation results air pressure along the vertical stack. The coefficient of correlation is 0.97.

4.1. Scenario 1 – Single branch discharge from 12/F

In this scenario, total 4.0 l/s water is discharging from the branch at 12/F. The results are shown in Figure 3. Pressure variations in all cases fall into $(-0.15, 0.12)$ m-wg. Figure 3(a) shows that Case 1-0 (i.e. no corss-vent provided) achieved the maximum negative pressure among all. In this case, the vent stack was only connected to the drainage stack at the bottom and top. Although the vent connection at the bottom of the drainage stack released part of the air pressure to the atmospheric via the vent stack, there is still around 0.03 m-wg pressure at 2/F. We can see the smooth air pressure change from to maximum negative at 11/F (i.e.; -0.15m-wg) to positive at 2/F since there was no other branch inlet and cross-vent in between. Since the extreme negative pressure of case 1-0 is -0.15m-wg which is lower than the trap seal (i.e. 0.075m-wg), without the provision of cross-vent, the water seal of the trap at 11/F may be emptied by the extreme negative pressure. For other cases, with the provision of cross-vent, excessive negative pressure along the vertical drainage stack can be alleviated.

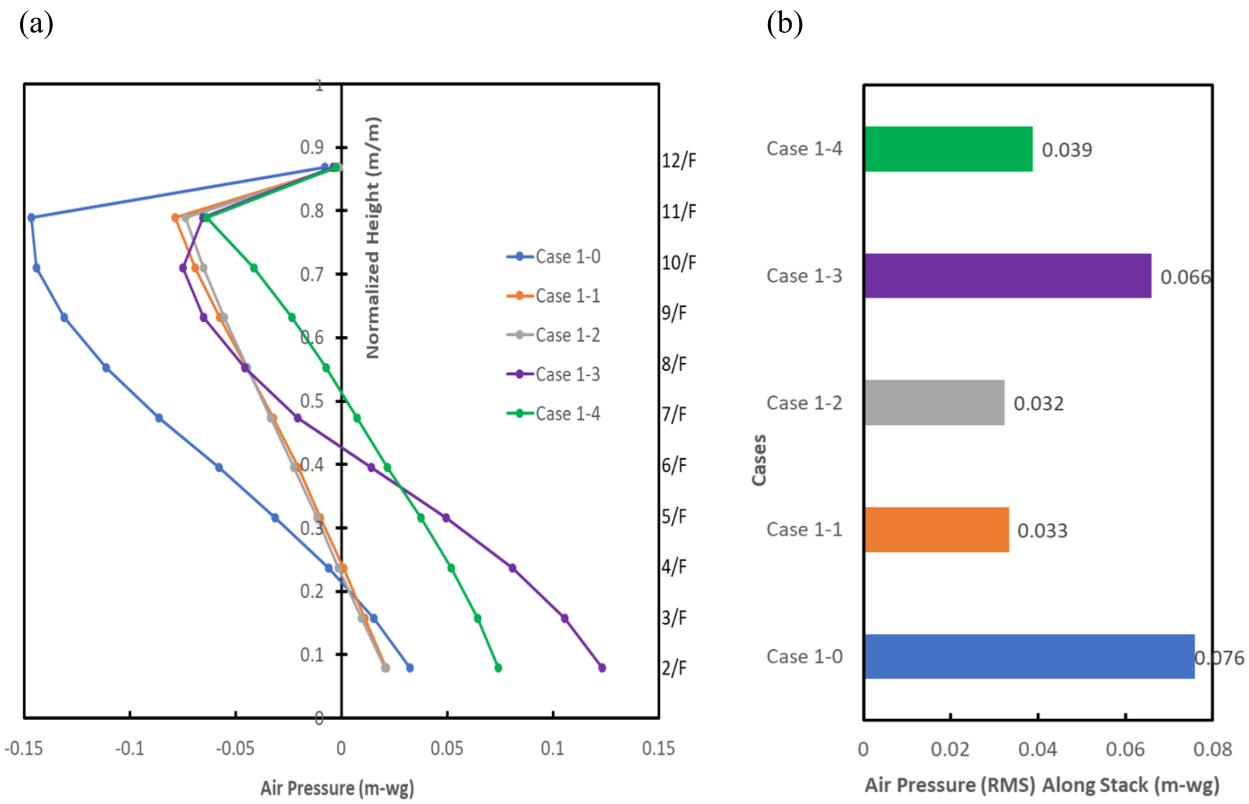

Figure 3. (a) Simulation results of air pressures at different levels along the drainage stack with single branch discharge from 12/F. (b) The RMS values of air pressures of different cases.

Another observation in case 1-3 shows that the positive air pressures at 4/F, 3/F and 2/F are higher than 0.075m-wg exceeding the 75mm water seal of the traps. Therefore, foul water may pass through the trap seals and ingress into the indoor environment from the building drainage system. The result shows that large single discharge from topmost, comparing to other discharge patterns as shown in Figure 4-6, should be prevented in system design since it may result excessive positive pressure at the base of the stack even cross-vent is provided. It should be noted from virus transmission point of view that the excessive negative or positive pressure along the stack may cause either ingress of foul air into indoor environment or loss of water seal (which eventually may also lead the ingress of foul air through the dried trap). When the foul air is contaminated by virus, such system design may cause the spread of disease to the indoor environment through the drainage system. In order to compare the results in different cases quantitatively, we adopt room-mean-square (RMS) value for the analysis. The RMS values of the 5 cases are shown in Figure 3(b). It is proved by the results of Cases 1-1, 1-2, 1-3 and 1-4 that all RMS pressures are lower than that of Case 1-0. It may conclude that the cross-vent does alleviate the pressure variation along the vertical drainage stack. We may summarise that, in this scenario, providing cross-vents at each floor or every two floors may reduce the air pressure variation along the vertical drainage stack and prevent the existence of excessive negative air pressure.

4.2. Scenario 2 – Two branches discharging from 12/F and 7/F (discharge at every 5 floors)

This scenario is with 2 nos. discharges at 12/F and 7/F. The discharge rate of each branch is 2 lit/s. We can observe from the results shown in Figure 4(a) that the air pressure variation is significantly reduced to the range $(-0.03, +0.02)$ m-wg comparing to the last scenario with single branch discharge. We can see that the flow rate of each branch has significant impact of the induced air pressure inside the drainage stack. If the water seals of the water traps are 75mm, the negative air pressure (i.e., -0.03 m-wg) should draw the water seal to the stack. The water seal resume when the negative pressure disappears and the water trap of the trap will be less than 75mm but it can be replenished by the next usage of next flushing. The variations in Cases 2-0, 2-3 and 2-4 are quite similar to each other and the air pressures of these 3 cases are all negative from 12/F down to 2/F. The two Cases 2-1 and 2-2 performed similar with positive pressure at 4/F, 3/F and 2/F. The pressure variations of different cases represented by their RMS values are shown in Figure 4(b). Case 2-1 (i.e. cross-vent is provided at each floor) achieves the best performance among them. It quite agrees with the results in last scenario.

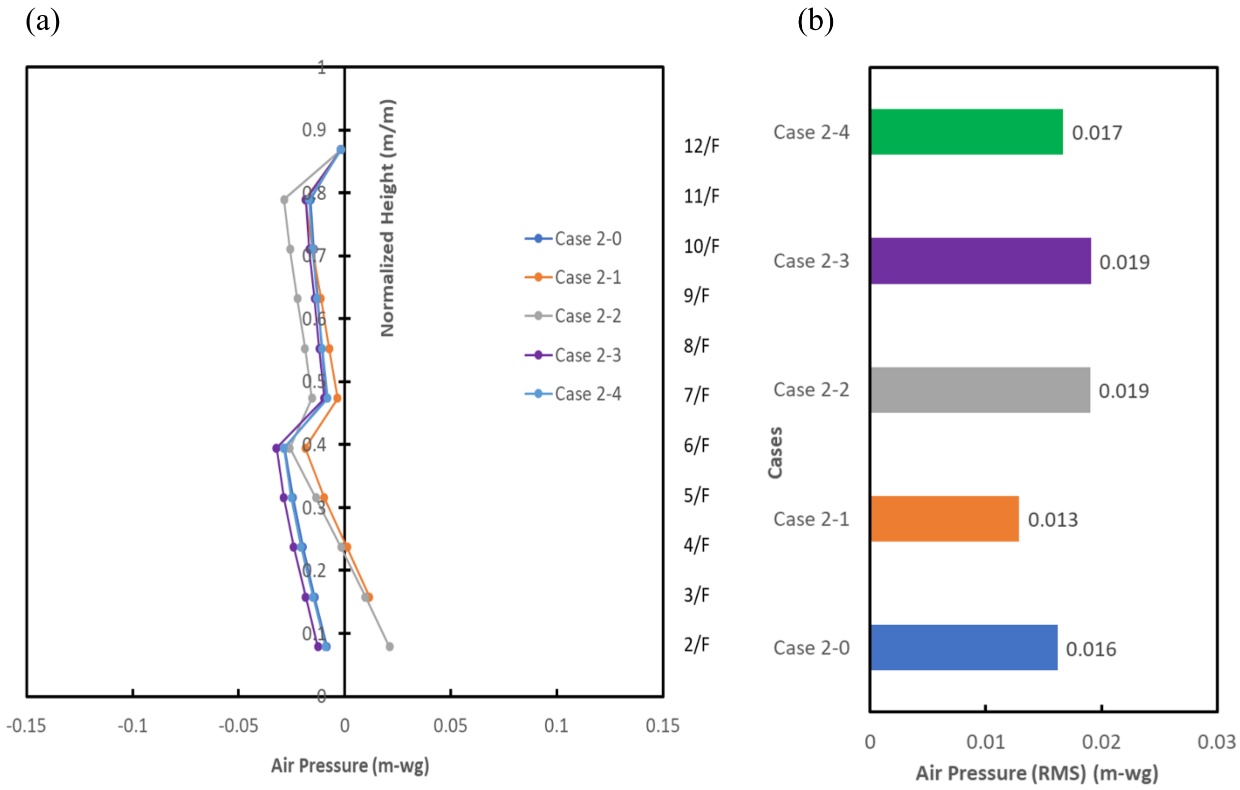

Figure 4. (a) Simulation results of air pressures at different levels along the drainage stack with two branch discharge from 12/F and 7/F; (b) RMS values of air pressures of different cases.

4.3. Scenario 3 – Three branches discharging from 12/F, 8/F and 4/F (discharge at every 4 floors)

This scenario is with 3 nos. discharges at 12/F, 8/F and 4/F. The discharge rate of each branch is 1.33 lit/s. We can observe from the results shown in Figure 5(a) that the air pressure variation is significantly reduced to the range $(-0.03, +0.02)$ m-wg. We can see that the reduced flow rate of each branch has less impact of the induced air pressure inside the drainage stack. The impact of the negative air pressure effect on the water seal of the connected trap is similar to the last scenario. The variations in all cases are quite similar to each other. The air pressures of cases 3-0, 3-1 and 3-4 are all negative at the base of the drainage stack. The two cases 3-2 and 3-3 achieved positive pressure at the base of the stack. The pressure variations of different cases represented by their RMS values are shown in Figure 5(b). Case 3-4 (i.e. cross-vent is provided at every four floors) achieves the best performance then cases 3-1 and 3-2. We can still observe that provided cross-vents at every level or every 2 levels can still achieve a good result.

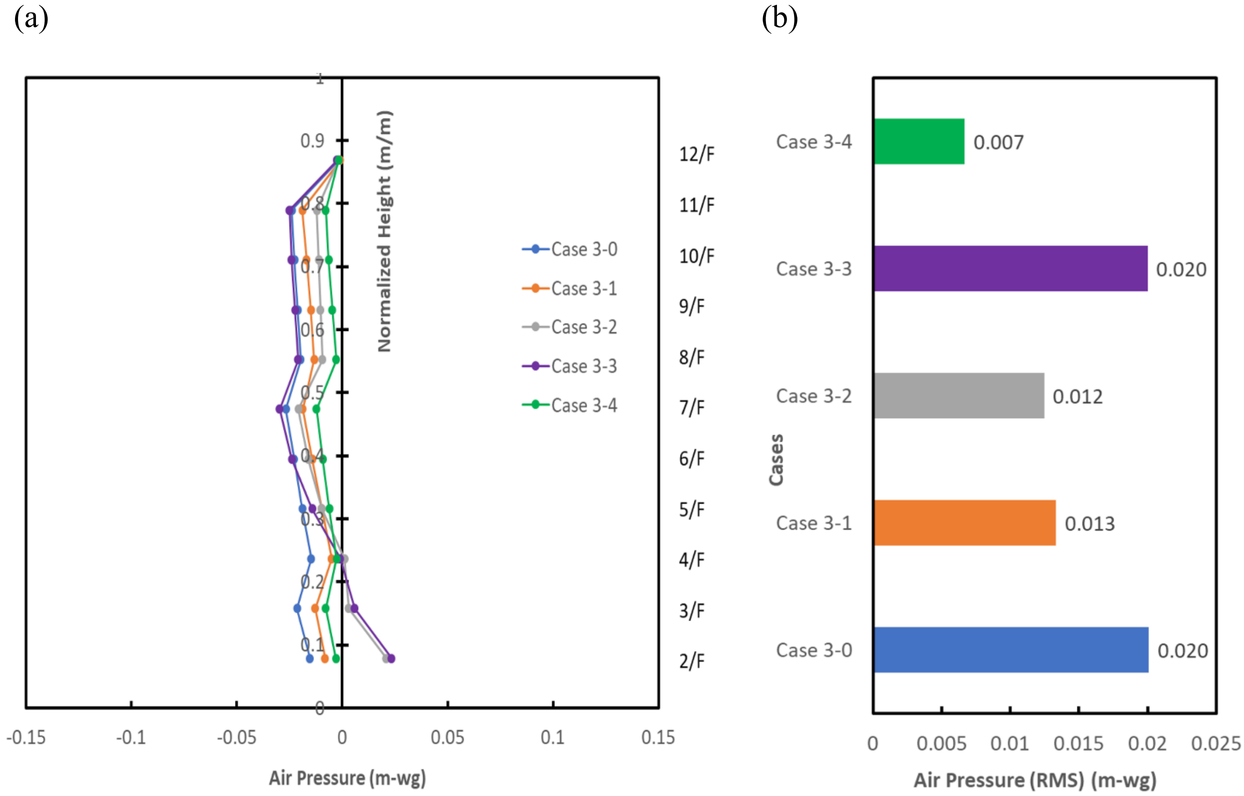

Figure 5. (a) Simulation results of air pressures at different levels along the drainage stack with two branch discharge from 12/F, 8/F and 4/F; (b) RMS values of air pressures of different cases.

4.4. Scenario 4 – Four branches discharging from 12/F, 9/F, 6/F and 3/F (discharge at every 3 floors)

This scenario is with 4 nos. discharges at 12/F, 9/F, 6/F and 3/F. The discharge rate of each branch is 1.0 lit/s. We can observe from the results shown in Figure 6(a) that the air pressure variation is significantly reduced to the range $(-0.04, +0.04)$ m-wg. We can see that the reduced flow rate of each branch has less impact of the induced air pressure inside the drainage stack. The impact of the negative air pressure effect on the water seal of the connected trap is similar to the last scenario. The variations in all cases are quite similar to each other at the upper portion of the stack but vary at the bottom of the stack. The air pressures of cases 4-3 and 4-4 are all negative along the stack down to 2/F. The other three cases achieve positive pressure at the base of the drainage stack. The pressure variations of different cases represented by their RMS values are shown in Figure 6(b). Case 4-3 (i.e. cross-vent is provided for every three floors) achieves the best performance then cases 4-4. We can still observe in case 4-1 that provided cross-vents at every level can still achieve a reasonably good result since the RMS value (i.e., 0.019) is still quite similar to the RMS values of case 2-1 and 3-1 (i.e., 0.013).

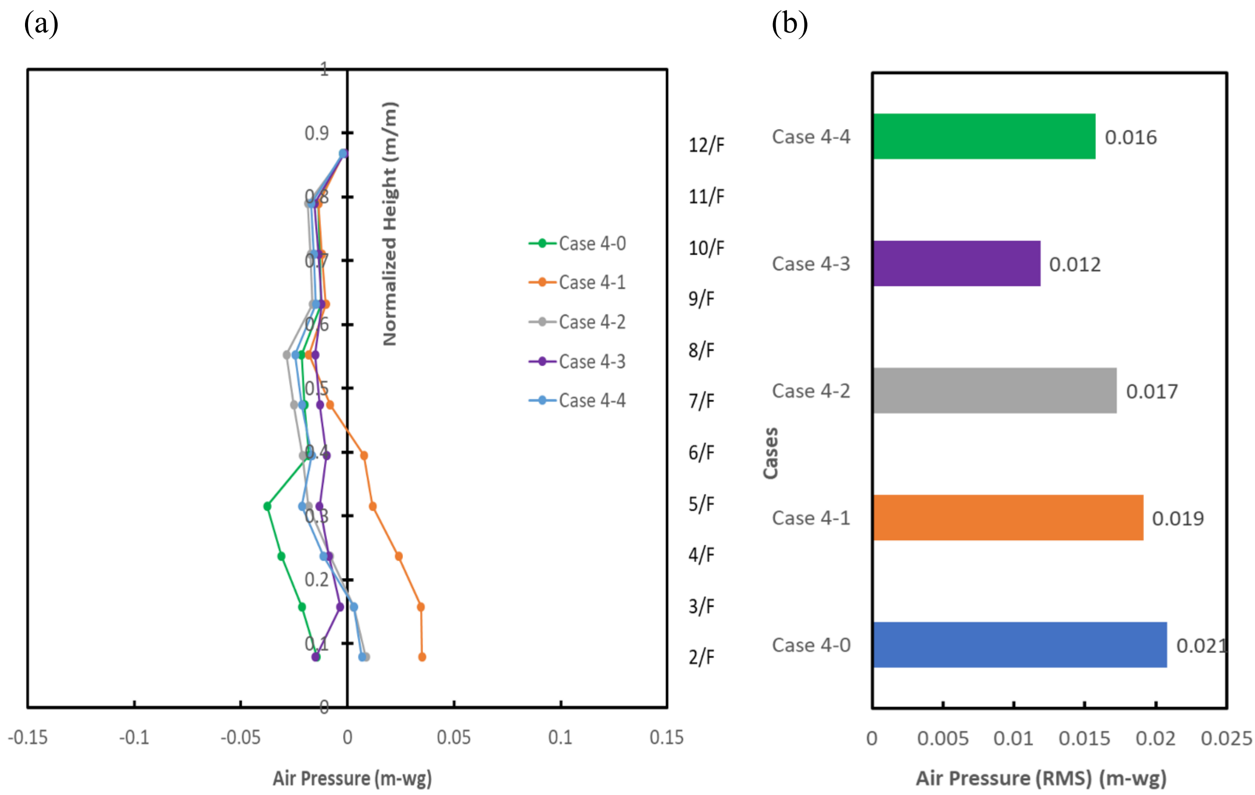

Figure 6. (a) Simulation results of air pressures at different levels along the drainage stack with two branch discharge from 12/F, 9/F, 6/F and 3/F; (b) RMS values of air pressures of different cases.

4.5. Comparison of air pressure inside drainage stack with different discharge patterns

Section 4.1 to 4.4 show the CFD simulation results of different cases. We would discuss the effect of discharge pattern to the air pressure along the vertical drainage stack. Table 2 summarizes the results.

Table 1. Summary of maximum and minimum air pressure along vertical drainage stack with different discharge patters.

Scenario	Floor of discharging branches	Air pressure (m-wg)	
		Minimum	Maximum
1	12/F	-0.15	+0.12
2	12/F, 7/F	-0.03	+0.02
3	12/F, 8/F, 4/F	-0.03	+0.02
4	12/F, 9/F, 6/F, 3/F	-0.04	+0.04

Table 2 shows the discharge pattern in scenario 1 has the largest air pressure range. The air pressure range of other scenarios are quite similar to each other. The same can be observed from Figure 3(a). It indicates that single discharge from top of stack may incur excessive negative and positive air pressure along the stack. This discharge scenario should be avoided in system design.

5. Conclusions

This study adopted CFD to study the performances of cross-vent provided between the vertical drainage stack and the vertical vent stack. The CFD simulation of case 1-1 was compared to the published experimental result and found a good agreement. We then carried out systematic analysis on the effectiveness of different cross-vent provisions by consideration the spacing between the cross-vents and also the number of branches discharging to the drainage stack. According to our observations on the simulation results, cross-vent can prevent excessive negative air pressure along the drainage stack. However, large single water discharge should be prevented since the excessive air pressure at the bottom of the stack may cause ingress of foul air into indoor environment through the 75mm water seals of the trap which may assist the virus transmission. However, in general, the provision of cross-vents can prevent the existence of excessing negative or positive pressure along the stack.

In this study, we investigated the effect of different cross-vent provisions and different flow patterns. The CFD simulation results show that the best cross-vent provision varies in different scenarios. The RMS values of case 1-1, case 2-1 and case 3-1 show that provision of cross-vent in every floor, although it is not the best option in the three discharge scenarios, is still reasonably good. Case 4-1 is not the best among the its discharge scenario but the RMS value is quite similar to case 2-1 and case 3-1. We consider case 4-1 is still practically acceptable. Therefore, we would propose a practical guide that, cross-vents should be provided at every floor levels connecting the vertical drainage stack and the vertical vent stack. This recommendation in line with MHURD (2019) and BSI (2000).

Nomenclature

COVID-19	coronavirus disease
m-wg	water depth in meter
SARS	severe acute respiratory syndrome
VF	volume fraction
VOF	volume of fluid
α_q	volume fraction of the q^{th} fluid
ρ_q	density of the q^{th} fluid
$\vec{v}_q$	velocity vector of the q^{th} fluid
$\dot{m}_{pq}$	mass transfer from phase p to phase q
$\dot{m}_{qp}$	mass transfer from phase q to phase p
ξ_q	material property of the q^{th} fluid
ξ	material property of the control volume
p	pressure
μ	dynamic viscosity
$\vec{g}$	gravitational acceleration vector

Acknowledgement

The work described in this paper was fully supported by Collaborative Research Fund (CRF) COVID-19 and Novel Infectious Disease (NID) Research Exercise, Research Grants Council of the Hong Kong Special Administrative Region, China (Project No. PolyU P0033675/C5018-20G).

References

1. Ansys. 2016. *ANSYS Fluent User's Guide, Release 17.2.*
2. ArchSD (Architectural Services Department). 2017. *General Specification for Drainage Installation in Government Buildings of The Hong Kong Special Administrative Region*. Hong Kong Standard. DIGS2017. Hong Kong: Architectural Services Department.
3. BSI (British Standards Institution). 2000. Gravity Drainage Systems Inside Buildings – Part 2: Sanitary pipework, layout and calculation. European Standard EN12056-2. London: British Standard Institution.
4. Cheng, C. L., K. W. Mui, L. T. Wong, C. J. Yen, and K. C. He. 2010. "Characteristics of Air Pressure Fluctuations in High-Rise Drainage Stacks." *Building and Environment* 45(3): 684-690. doi:10.1016/j.buildenv.2009.08.006.
5. Cheung, C. L., C. J. Yen, W. H. Lu, and K. C. He. 2008. "An Empirical Approach to Determine Peak Air Pressure Within the 2-Pipe Vertical Drainage Stack." *Journal of the Chinese Institute of Engineers* 31 (2): 199-213. doi:10.1080/02533839.2008.9671374.
6. Gopala, V. R., and B. G. M. van Wachem. 2008. "Volume of Fluid Methods for Immiscible-Fluid and Free-Surface Flows." *Chemical Engineering Journal* 141 (1-3): 204-221. doi:10.1016/j.cej.2007.12.035.
7. Gormley, M., D. Kelly, D. Campbell, Y. Xue, and C. Stewart. 2021. "Building Drainage System Design for Tall Buildings: Current Limitations and Public Health Implications." *Buildings* 11: 70. doi:10.3390/buildings11020070.
8. IAPMO (The International Association of Plumbing and Mechanical Officials). 2021. *2021 Uniform Plumbing Code*. American National Standard. ANSI UPC 1-2021. United States: International Association of Plumbing and Mechanical Officials.
9. Jack, L. 2006. "Drainage design: Factors contributing to SARS transmission, *Proceedings of the Institution of Civil Engineers*" *Municipal Engineer* 159 (1): 43-48.
10. Jack, L. B., and J. A. Swaffield. 1999. "Developments in the Simulation of the Air Pressure Transient Regime Within Single-Stack Building Drainage Systems." Paper presented at CIBW62 International Symposium Water Supply and Drainage for Buildings, Porto, United Kingdom, 18-20 September 2011.
11. MHURD (Ministry of Housing and Urban-Rural Development). 2019. *Code for Design of Building Water Supply and Drainage*. China Standard. China: Ministry of Housing and Urban-Rural Development.
12. PUB (Public Utilities Board). 2021. *Code of Practice on Sewerage and Sanitary Works (2 edition).* Singapore Standard. LEX-FAOC182165. Singapore: Public Utilities Board.
13. Sharma, S. K., S. P. Chakrabarti and P. Khanna. 1981. "Analytical Model for Partially-vented Drainage System in Buildings." *Building and Environment* 16 (4): 301-306. doi:10.1016/0360-1323(81)90008-1.
14. Swaffield, J. 2010. *Transient Airflow in Building Drainage System*. London: Routledge. doi:10.4324/9780203879757.
15. Swaffield, J. A, and D. P. Campbell. 1995. "The Simulation of Air Pressure Propagation in Building Drainage and Vent Systems." *Building and Environment* 30 (1): 115-127. doi:10.1016/0360-1323(93)E0006-Y.

16. Swaffield, J. A., and L. B. Jack. 2004. "Simulation and Analysis of Airborne Cross-Contamination Routes Due to the Operation of Building Drainage and Vent Systems." *Building esearch and Information* 32 (6): 451-467. doi: 10.1080/0961321042000259178.

17. Verma, N. K., S. P. Chakrabarti and P. Khanna, 1976. "Modified One-pipe System of Drainage for Tall Buildings." *Building and Environment* 11(3): 197-201. doi:10.1016/0360-1323(76)90006-8.

18. WHO (World Health Organization). 2003. "Inadequate plumbing system likely contributed to SARS transmission." Weekly Epidemiological Record (WER) 78(42): 371-372.

19. Zhang, D., X. Zhang, Y. Yang, X. Huang, J. Jiang, M. Li, H. Ling, et al. 2021. "SARS-CoV-2 Spillover Into Hospital Outdoor Environments." *Journal of Hazardous Materials Letters* 2: 100027.