Cooling Effect of Mechanical Night Ventilation over Internal Wall Surface: A Parametric Analysis

W.H. Ji\(^1\), Q. Luo\(^1\), Z.L. Zhang\(^2\), H.H. Wang\(^1\)

\(^1\)The Faculty of Urban Construction and Environmental Engineering
Chongqing University, Chongqing, 400042, PR China
\(^2\)Department of Engineering
Aarhus University, 8000 Aarhus C, Denmark

SUMMARY
This paper presents experimental and numerical investigations on forced convection over internal wall surface of a chamber applying night-ventilation technique in a hot-humid climate in China. Ambient cool air is supplied into chamber through a longitudinal slot below the ceiling. Forced convection is then generated by air jet flowing downward over internal wall surface of west wall. One case is selected to investigate the cooling effect of ventilation, and the cooling efficiency of this case is calculated. On the other hand, a transient CFD model is established and then validated by the experimental data. This numerical model is afterwards used to optimize the ventilation schedule. The start-up time of night ventilation shows insignificant influence on the cooling effect, and the turn-off time of night ventilation is suggested as the time when lowest ambient air temperature appears.

INTRODUCTION
Nowadays, buildings are responsible for more than 40% of global energy used, and up to one third of global greenhouse gas emissions [UNEP 2009]. In China, the amount of building energy consumption during the life cycle can be as much as 43% of the total energy consumption and even 46% of the final energy consumption [Zhang et al. 2015]. The energy demand for air-conditioning of buildings will further increase due to global warming, thus, it is important to maintain the indoor environment through high efficiency strategies of buildings.

Due to the popularity of exterior insulation on walls in the office buildings in China, it is difficult to eliminate heat which stored in walls during daytime. Thus, night ventilation can be applied to remove the heat stored in walls and room furniture to achieve better indoor thermal condition and reduce energy consumption. Previous studies [Geros et al. 1999; Artmann 2007; Kubota 2009; Irulegi 2016] have shown the potential of night ventilation.

It is evident that the performance of night ventilation depends on the ambient environment as well as the ventilation parameters [Artmann 2008]. Due to the hot and humid climate condition and small temperature variation between day and night in Chongqing, China, mechanical ventilation is superior to night ventilation. Many studies have been carried out to evaluate the convection over different room surfaces. Awbi and Hatton [2000] developed the correlation of convective heat transfer of short longitudinal jet over different heated room surfaces. Jeong and Mumma [2003, 2004, 2007] estimated the enhancement of cooling capacity of cooled ceiling panels by mixed convection. In the research work by Samo Venco et al. [2014, 2015], convection caused by longitudinal air jet over thermally activated cooled wall was experimentally investigated. Results show a significant decrease of energy consumption with enhanced convection.

In the present study, a novel night ventilation system is proposed, which provides an efficient solution for enhancing the convective heat exchange between induced cold air and internal surface of external insulated wall, as well as for eliminating the accumulated heat in thermal mass in a short time.

EXPERIMENTAL STUDY
Outline of field experiment
The test room is located in Chongqing, southwest of China, which presents a typical monsoon-influenced humid subtropical climate. During summer time, Chongqing is one of the hottest and most humid cities in China. The outdoor air average highest temperature and average relative humidity in summer are 32.8 °C and 76%, respectively.

The experiments are conducted in a test chamber (3.6 m length \(\times\)3.3 m width \(\times\)3 m height) at Chongqing University, China. The test room has three windows (1.45 m width \(\times\)1.46 m height) in the east wall, north wall and west wall, respectively, while the door (0.8 m width \(\times\)2 m height) is installed in the south wall. The walls are constructed with perforated brick walls using thermal mortar as external insulation.

The mechanical ventilation system is established to supply outdoor air into the test room during night time. Air is supplied into the chamber through an opening of 0.4 \(\times\)0.4 m\(^2\) located at the north corner directly below the ceiling, and flows through...
an air duct (W × H × L = 0.25 m × 0.25 m × 3 m), as shown in Figure 1. Afterwards, the air adheres to the inner surface of vertical west wall from a long and narrow opening (W × L = 0.02 m × 2.8 m) at the bottom of the air duct. The north exhaust opening is kept open during the night time. A supply fan is mounted as the driving force of ventilation devices, which is regulated by a frequency converter.

For temperature measurement, 37 thermocouples of type T are used. Wall surface temperatures are detected by temperature sensors with their locations shown in Figure 2. An additional temperature sensor is located in the centre of the test chamber at the height of 1.5m above the floor to monitor the room air temperature. Air velocity and temperature of supply air are detected by five sensors positioned at different locations of the supply air duct.

![Figure 1. Test chamber layout](image)

![Figure 2. Locations of thermocouples for measuring wall surface temperatures](image)

**Experimental results**

All tests are conducted in July with similar climate conditions, so a typical case is selected and presented in this paper. In this case, the highest and lowest outdoor air temperature are 33 °C and 24 °C, respectively. The ventilation schedule in this case is from 22:00 to 8:00 in the next morning, and the air supply velocity is 3 m/s. The recorded results of wall surface temperatures and room air temperature are illustrated in Figure 3.

The outer surface temperature of west wall presents a sharp increase from 14:00 to 18:00, as a consequence of solar radiation. After sunset, the outer surface temperatures of four walls decrease gradually. At the meantime, due to the heat release of the walls and other thermal mass in the room, both the inner surface temperatures of four walls and the room air temperature increase gradually. At 22:00, the measured results of inner surface temperatures of four walls and the room air temperature show a significant decline when night-time ventilation begins, revealing that the cooling effect of convective night ventilation on the wall surface and room air is significant. Due to the heat dissipation of supply fan and pipeline heat loss, the temperature difference between cool ambient environment and indoor environment is about 2 °C during night-time.

![Figure 3. Temperature profile of the experiment results](image)

For evaluating the impact of mechanical night ventilation, the cooling efficiency of night ventilation $\epsilon$ is described as:

$$\epsilon = \frac{\Delta T_{\text{inner}}}{\Delta T_{\text{outer}}}$$

where $\Delta T_{\text{inner}}$ is the temperature variation of west wall inner surface during the first ventilated hour, namely, from 22:00 to 23:00. Accordingly, $\Delta T_{\text{outer}}$ is the temperature variation of west wall outer surface during the first ventilated hour.

According to Equation (1), $\Delta T_{\text{inner}}$ and $\Delta T_{\text{outer}}$ for this studied experiment are 0.94 °C and 0.40 °C respectively. The night ventilation efficiency of this case is 2.35.

**Numerical study**

**Numerical model**

In order to simulate the night ventilation chamber bounded by wall surface convective heat transfer conditions, a computational fluid dynamics (CFD) model is established in Fluent 15. The simulation model is built by mimicking the main geometrical characteristic of the test chamber setup. The air duct and outlet have exactly the same size of the openings in the test. On the other hand, the model is also simplified by omitting some geometrical features such as the very details of the walls and windows. As the data acquisition instrument is the only heat load of the test chamber and its heat release rate can be ignored compared to the cooling capacity of ventilation, the internal heat source of the CFD model is set to be zero.

Non-structural meshes are generated using ICEM and mesh refinement are performed for the inlet, the outlet and neat wall section. A grid independency study is carefully performed. The resulting grids have 4,749,080 cells. Figure 4 shows the non-structural meshes.

As the ambient temperature changes periodically, the simulation model is conducted transiently with 3D model. There is a general consensus [Zhai Z. et al. 2007; Zhang Z. et al. 2007; Posner J.D. et al. 2003] that the Re-normalized
Group (RNG) $k - \varepsilon$ turbulence model is the best choice for simulations involving forced flows, taking into account both the accuracy and computational efficiency. Therefore, the RNG $k - \varepsilon$ turbulence model is employed in this study. For the downward flow over inner wall surface, non-slip enhanced wall functions are used. The finite volume differencing scheme is used in the simulation model. Body-force weighted discretization scheme is employed for pressure equation, and second order upwind scheme is selected for momentum, density and energy equations. The SIMPLE algorithm is adopted for the pressure-velocity coupling. Furthermore, the default FLUENT residual convergence criteria are employed for all simulations.

Due to the vagaries weather condition of the test day and the mean evaluation of 5 sensors on west wall outer surface temperature, the measured outer wall surface temperatures exhibit considerably large fluctuation. A specific UDF file is created and smoothed from measured weather data to represent the ambient thermal environment. Initial conditions of the simulations are set to be the boundary conditions recorded at 10:00 AM. The fixed airflow velocity is same as the velocity of constant airflow in the test. All the other relevant inputs to the CFD model are in accordance with the measured experimental data.

Validation of the CFD model

The CFD model needs to be validated by the experimental results. Figure 5 illustrates the comparison between simulated and measured temperature profile of ventilated west wall. In general, there is an acceptable agreement between the experiment and simulation results. The maximum temperature differences between the calculated and measured data of wall inner surface temperature and room air temperature are 1.52 K and 1.5 K respectively. The discrepancy mainly appears during day time, so the CFD model can be employed in follow-up work focusing on the night ventilation.

As can be seen, the wall outer surface temperature varies almost periodically and sharply, while the wall inner surface temperature changed much more gently. Comparing with the measured wall outer surface temperature, the corresponding simulation result is more smooth and less oscillatory, but the changing trend of them keeps consistent with each other.

remarkable decline of the wall inner surface temperatures takes place at 22:00 when night ventilation starts, revealing the cooling effect of night ventilation in both experimental and simulated results. However, CFD model seems overestimate the decreasing magnitude of wall inner surface temperature at the beginning of night-time ventilation.

![Figure 4: Non-structural meshes](image)

**Figure 4. Non-structural meshes**

**Figure 5. Comparison between the measured and simulated temperature results**

**Cooling effect of night ventilation**

![Figure 6: Temperature contour across the central plane of the test chamber at different time](image)

**Figure 6. Temperature contour across the central plane of the test chamber at different time (a) 21:00 (b) 22:00 (c) 23:00 (d) 00:00**

The verified CFD model is afterwards used to simulate the influence of convective regime on room temperature and velocity, as illustrated in Figures 6 and 7. Figure 6 shows that...
the induced outdoor air flowing over the inner wall surface can gradually cool down the walls and remove the heat stored in brick walls during day time. The predicted air flow pattern presented in Figure 7 indicates that the supply air continues over the floor towards the back side of the chamber. Well air circulation can be observed. The maximum air velocity of the plenum occurs near the ventilated wall.

Figure 7. Velocity contour across the central plane of the test chamber

PARAMETRICAL STUDY

Based on the simulative setup mentioned above, the CFD model is used to discuss the cooling effect of ventilation schedule and velocity. In the parametric study, the night ventilation is simulated with the same measured weather data in all simulation cases.

Start-up time of night ventilation

The original start-up time of the experiment is 22:00. This parametrical study investigates three start-up time: 22:00 (as in the test), 23:00 and 24:00, with the same ambient environment and ventilation velocity. The thermal performance of different ventilation start-up time is plotted in Figure 8.

Actually, the changed start-up time will affect the temperature difference between the ambient environment and indoor environment. The temperature results presented in Figure 8 show indistinctive temperature variation when adopting different start-up time. Thus, for a certain climate condition, night ventilation can start at any time as long as the ambient air temperature is lower than the room air temperature.

Turn-off time of night ventilation

According to the common work schedule, the time of turning off the ventilation system in the test is set to be 8:00 in the next morning. However, both experimental and simulated results indicate the ambient temperature will increase after 6:00. Thus, the influence of three different turn-off time of night ventilation is calculated and analysed, as shown in Figure 9.

Continuous night ventilation after the lowest outdoor air temperature will lead to unpleasant temperature rise of both wall surface and room air, as inferred in the case when night ventilation ends up at 8:00. The best cooling effect is achieved when night ventilation is shut down at 6:00, which is exactly the time when the ambient air temperature reaches the lowest point. Considering the energy consumption of the ventilation system, turning off the fan at 4:00 is also an acceptable choice.

CONCLUSIONS

The objective of this paper is to investigate the performance of mechanical night ventilation on inner surfaces of vertical walls in convective regime. Both experimental and numerical studies have been carried out. In the experimental study, the measured temperature results show a significant decline when night-time ventilation begins. The cooling efficiency of night ventilation in the first ventilated hour is 2.35. The experimental results reveal that this air flow pattern of mechanical night ventilation can cool the walls and room air in a short time.
In the numerical study, different ventilation schedules are carefully investigated. It turns out that the performance of ventilation depends on the closing time. The start-up time of night ventilation has less contribution to the cooling effect of night ventilation.

ACKNOWLEDGEMENT

The research described in this paper is carried out within the project “State key laboratory of building safety and environment” (project BSBE2016-07), co-financed by “National Natural Science Foundation of China” (project 51578086).

REFERENCES


