NUMERICAL INVESTIGATION OF A NOVEL DESIGN OF A SOLAR CHIMNEY COUPLED WITH A TURBINE VENTILATOR

Ke San YAM[1], Sukanta ROY[1], Aun Naa SUNG[1], and Ming Chiat LAW[1]
[1] University of Curtin Malaysia, Department of Mechanical Engineering, Faculty of Sciences and Engineering, 98009, Miri, Sarawak, Malaysia. Email: yamks@curtin.edu.my.

***ABSTRACT**: Solar chimney is a popular natural ventilation system that utilizes solar energy to generate air movement for the purpose of controlling indoor air quality and temperature in building applications. The design of solar chimney has been optimized over the years and the effectiveness has been tested in numerous experiments and real buildings. This work focusses on a new design of solar chimney that couples with a turbine ventilator. The new design combines both the wind-driven ventilation from turbine ventilator and the sun-driven ventilation from solar chimney for an enhanced ventilation rate and improved operating conditions. The effectiveness of the design is evaluated by performing high fidelity flows and heat transfer calculation in idealized scenarios using computational fluid dynamics method. The results show the design has a promising prospect to be a better ventilator than its individual components.*

***KEYWORDS**: Natural ventilation, solar chimney, turbine ventilator*

1 INTRODUCTION

Energy use in buildings has contributed towards a large part of global and regional energy demand. The advancement and increasing sophistication of the indoor climate comfort systems became a major factor behind the steady rise of energy consumption in both commercial and residential sectors. Hence there is an increasing research and interest towards developing energy efficient building climate system to mitigate building carbon footprint (Chua et al. 2013).

Solar chimney is one of the strategies that receive increasing research interest (Monghasemi and Vadiee, 2018; Zhai et al. 2011). Solar chimney for natural ventilation is excellent passive system that maximizes solar gain in order to create an enhanced buoyancy effect and appreciable air movement. A significant amount of research work either has been done on solar chimney in the past decades to understand and improve the model better. Numerous experimental investigations of solar chimney have been reported in the existing literature. Some early work focusses on the impact of the inclination angle. For instance, Mathur and Mathur (2006) found that tilted chimney has more significant ventilation rate than the vertical chimney. Their study shows that optimum absorber inclination angle varies from 40° to 60°, based on the place latitude. In more recent work, researchers have been exploring the possibility of the performance improvement through combination with other strategies and heat transfer components. For instance, Nguyen et al. (2020) proposed solar chimney that is fitted with a horizontal absorber surface in their study and showed the improvement in the flowrate. Serageldin et al. (2018) investigated the possiblity of enhancing performance of solar chimney by coupling it with an earth-to-air heat exchanger. Cheng eta l. (2018) explored the feasibility of the solar chimney to be used as smoke exhaustion and concluded that the external radiation enhances natural ventilation but has limited influence on the smoke exhaustion. Elghamry and Hassan (2020) performed experiment on a novel system that consists of combination of solar chimney, PV panel, and geothermal air tube and prove their ability to cool the room temperature up to 3.5 °C and change daily its air 42 times.

With development of fast numerical schemes, computational fluid dynamic (CFD) method becomes increasingly favourable in solar chimney analysis as CFD can provide detailed and accuate flow information throughout the domain of interest. In the case of solar chimney, CFD are used to predict the flow and thermal variation inside the chimney and the adjoining space. Using CFD method, Khanal and Lei (2012) studied the adverse reverse flow phenomena occurs inside chimney and proposed a new solar chimney design that includes an inclined passive wall to suppress the reverse flow and enhance ventilation performance. Bassiouny and Koura (2008) investigated the correlation of the geometrical parameters such as chimney inlet size and width and found that the chimney width has a more significant effect on air change rate per hour (ACH) compared to the chimney inlet size. Bhardwaj and Agrawal (2017) combined CFD analysis with Taguchi's Method to optimize the solar chimney window to wall ratio, and window orientation for the maximum exit velocity for ventilation in buildings. More recently, Kong et al. (2020) performed CFD simulations on a two-dimensional solar chimney model with inclination angles varying from 30° to 90° relative to the horizontal plane under different heat fluxes three Australian cities, and found that the optimum inclination angle varies from 45° to 60°, depending on the latitude and season of operation.

The present study proposes a new design that combine the solar chimney with a turbine ventilator. Turbine ventilator is a popular passive ventilation device that relies on good winds condition to draw air up through the vent. The devices are low cost, easy to install and requires little maintenance, and hence are already widely adopted in the market. However, in low wind condition, the device has disadvantage for becoming less reliable. Therefore, the current proposal is intended to eliminate the weakness by harnessing both wind and solar potential to induce the ventilation..

2 PROBLEM DESCRIPTION

We consider a turbine ventilator with an outer diameter of 800mm that is attached to the outlet of 5m x 4.4m rectangular chimney as shown in Figure 1 below. To model the ambient environment, the turbine ventilator is considered to be situated inside a 7m long air channel with an air velocity inlet at one end and a pressure outlet at the other end. With flowing air inside the duct, the turbine ventilator can rotate and draw out the air from the solar chimney. On the other hand, the function of the solar chimney is to trap heat and increase the air temperature and velocity flowing inside the chimney. The heating of the chimney is modelled by prescribing the frontal and sides surfaces of the solar chimney with a fixed heat flux. The solar chimney has an opening at its bottom that draws air from buildings. In this case, the building space is not considered.

The objective of the study is to investigate the ventilation rate of the new model, particularly the effect of the chimney heat flux on the turbine speed and the ventilation rate, and flow physics behind the scenario.

2.1 Numerical Approach

We employed the Computational Fluid Dynamics commercial code StarCCM v15.4 to computationally analyse the problem. CFD is able to simulate detailed flows and heat transfer within the model to yield accurate predictions on the flow behaviours. We divide the model into three distinctive domains, i.e. two stationary domains (air channel and the chimney), and one rotating domain for the turbine ventilator. This division is performed in order to model the turbine angular motion. 1DOF model is prescribed to the turbine, which predicts the turbine rotation based on the pressure and shear forces on the surfaces of 6-DOF continuum bodies. The boundary conditions for the rest of the domains are summarised in Table 1 and Figure 2 below.

The flows transport and heat transfer mechanisms of the airflow in the domain are simulated using the Reynolds-Averaged Navier Stokes (RANS) model, the k-epsilon turbulence model equations, and the energy equation. Finite volume approach are employed to discretize these equations into numerical equations and SIMPLE iterative solvers are employed to solve these governing equations until specified convergence criteria are met.

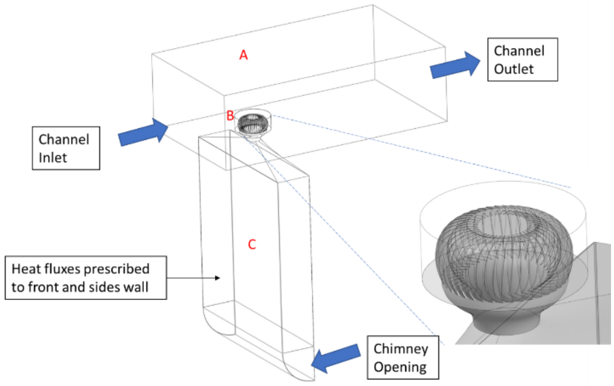

Fig 1. Schematic diagram of the model used in the present investigation (A: Air channel, B: Turbine Ventilator, C: Chimney).

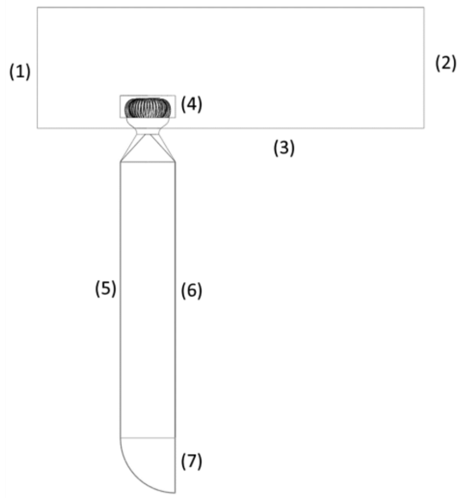

Fig 2. Numbering for the boundary in each domain.

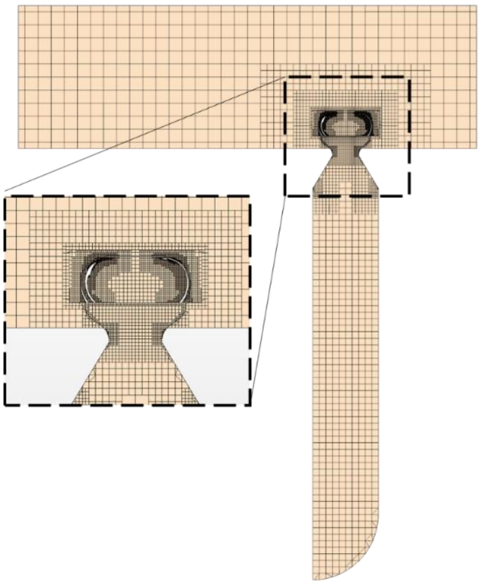

Fig 3. Meshing deployed inside the computational model.

Table 1. Boundary Conditions Used in the Simulation

No.	Domain	Boundary Conditions
1	Air channel	Velocity inlet, fixed temperature
2	Air channel	Pressure outlet
3	Air channel	No-slip wall, constant temperature
4	Turbine Ventilator	1 DOF model with fixed
5	Chimney	No-slip wall, constant heat flux
6	Chimney	No-slip wall, constant temperature
7	Chimney	Pressure Inlet, constant temperature

2.2 Validation

In order to validate the accuracy of the model, we conducted a computational study with a setup similar to Jadhav et al. (2016) and compare our simulation output against theirs. Figure 4 shows the prediction on the time history of the rotational velocity of our turbine ventilator against Jadhav et al. Despite the present simulation predicts a faster increase of the turbine speed with time but the overall trend matches and the comparison on the final RPM has discrepancy less than 5%. Additional cases are performed for different inlet velocity for a comparison on the final turbine speed. As shown in Figure 5, very good comparison is achieved for the final turbine speed at different inlet velocity with error less than 5% for all three cases.

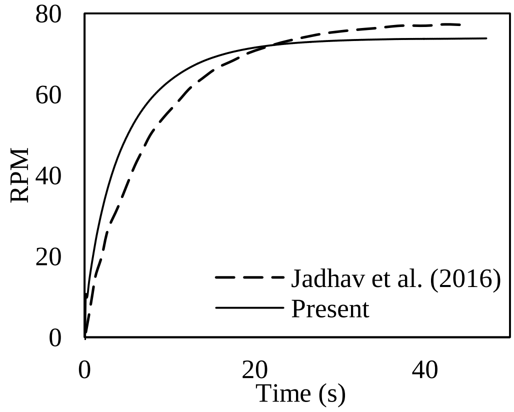

Fig 4. Prediction on the time history of the rotational velocity of the turbine ventilator

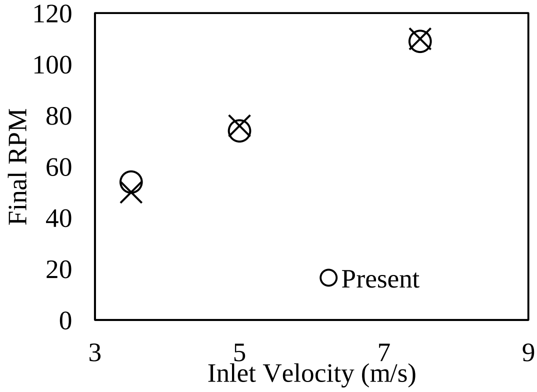

Fig 5. Comparison on the final rotational speed of the turbine ventilator for different inlet channel velocity.

3 RESULTS AND DISCUSSION

Simulations were performed for different values of chimney heat fluxes and inlet velocity to investigate the effect of the chimney heating on the turbine speed at different speeds. Figure 6 shows the time history of the turbine speed when there is no heat fluxes or heat fluxes of 1000 W/m^2. The figure clearly shows the turbine speed is greatly increased when chimney heating is present. For the case of inlet velocity of 2 m/s, the increment of 70% is achieved at the final RPM when chimney heating is present. On the other hand, for the case of inlet velocity of 3.5m/s, the increment of 15% is achieved at the final RPM. The increment is due to an added upward pressure from the air in the turbine. When the heating is prescribed, the air temperature inside the turbine is increased by 70 degrees as shown in Figure 7. The increased temperature increases the air buoyancy force and hence increases the upward force on the turbine. This apparently increases the turbine rotational speed.

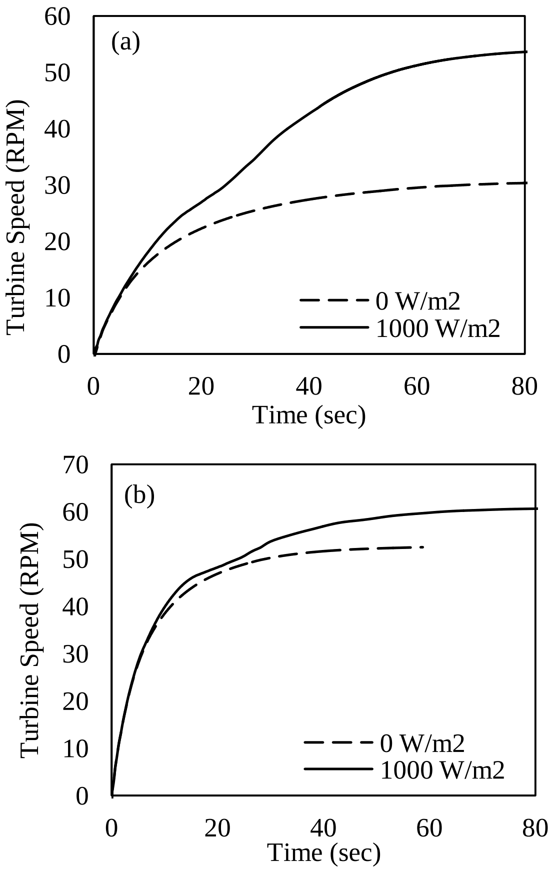

Fig 6. Comparison on time history of the turbine speed for different chamber heat flux (a) inlet velocity = 2m/s, (b) inlet velocity = 3.5m/s.

The overall effect of the heat fluxes on the turbine speed is summarized in Figure 8, which shows the final turbine speed for heat flux ranges from 0 to 1000 W/m^2 at wind speed of 2m/s and 3.5m/s. The figure shows the turbine speed increase linearly with the increase of the heat fluxes. But the increment is lesser at higher velocity of 3.5m/s. At high speed, air flows through the turbine faster, and apparently this offset the pressure added from the buoyancy effect.

The effect of the heat fluxes on the corresponding chimney mass flowrate is shown in Figure 9. Like turbine speed, the mass flowrate also increases with the increase of the heat fluxes. At zero heat fluxes, the predicted mass flowrate is 0.12 kg/s and 0.21 kg/s for inlet velocity of 2m/s and 3.5 m/s, respectively. With the heat fluxes increased, mass flow rate is greatly increased initially, i.e. by one-fold, but the increment rate can be seen reduces at higher heat fluxes (increasing heat flux to 1000 W/m2). The mass flowrate also converges to a value of about 0.48 kg/s at heat fluxes of 1000 for both inlet velocity, showing a reduced benefit the chimney heat fluxes can provide at higher inlet velocity. The observation is consistent with the turbine speed trend in Figure 9. The diminishing increment at higher heat fluxes is because the added pressure from the buoyancy has been offset by the decreasing air pressure at high speed.

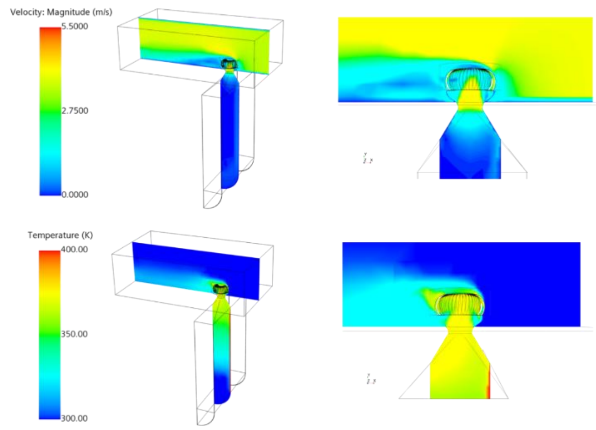

Fig 7. Velocity and temperature contour of the flow in the computational domain for different chimney heat fluxes for inlet velocity = 3.5m/s, and chimney heat flux = 1000 W/m^2

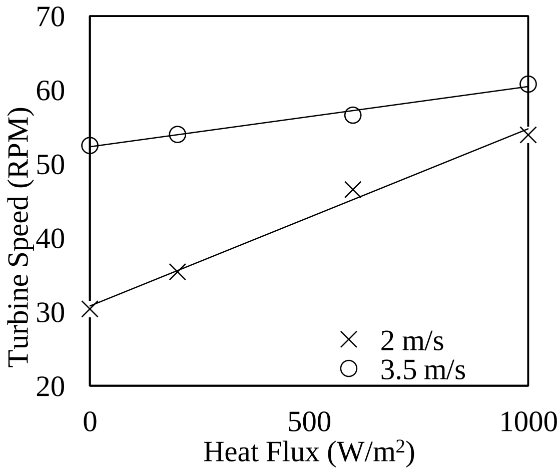

Fig 8. Final turbine speed for different chamber heat flux and inlet velocity

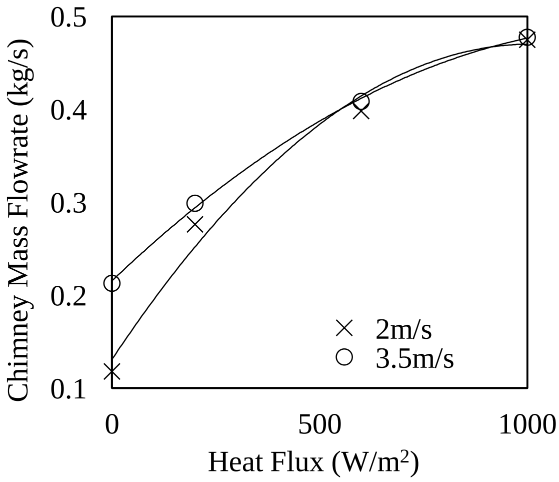

Fig 9. Mass flow rate through the Turbine Ventilator

4 CONCLUDING REMARKS

This study presents a numerical investigation on a new design of solar chimney coupled with a turbine ventilator. The study shows that the ventilation rate is greatly enhanced in new design when heat fluxes are included in the chimney. The improvement is the result of an added buoyancy force from the air due to the heating from the chimney. The enhancement on the ventilation rate is the greatest at low inlet velocity or low turbine speed. At high inlet velocity or high turbine speed, the improvement diminishes.

5 REFERENCES

●Bassiouny, R. and Koura, N.S., 2008. An analytical and numerical study of solar chimney use for room natural ventilation. Energy and buildings, 40(5), pp.865-873.

●Bhardwaj, M. and Agrawal, G.D., 2017, December. Optimization of solar chimney for ventilation in buildings using Taguchi method. In 2017 International Conference on Technological Advancements in Power and Energy (TAP Energy) (pp. 1-6). IEEE.

●Chua, K.J., Chou, S.K., Yang, W.M. and Yan, J., 2013. Achieving better energy-efficient air conditioning–a review of technologies and strategies. Applied Energy, 104, pp.87-104.

●Cheng, X., Shi, L., Dai, P., Zhang, G., Yang, H. and Li, J., 2018. Study on optimizing design of solar chimney for natural ventilation and smoke exhaustion. Energy and Buildings, 170, pp.145-156.

●Elghamry, R. and Hassan, H., 2020. An experimental work on the impact of new combinations of solar chimney, photovoltaic and geothermal air tube on building cooling and ventilation. Solar Energy, 205, pp.142-153.

●Jadhav, G.K., Ghanegaonkar, P.M. and Garg, S., 2016. Experimental and CFD analysis of turbo ventilator. Journal of Building Engineering, 6, pp.196-202.

●Khanal, R. and Lei, C., 2012. Flow reversal effects on buoyancy induced air flow in a solar chimney. Solar Energy, 86(9), pp.2783-2794.

●Kong, J., Niu, J. and Lei, C., 2020. A CFD based approach for determining the optimum inclination angle of a roof-top solar chimney for building ventilation. Solar Energy, 198, pp.555-569.

●Mathur, J., Bansal, N.K., Mathur, S. and Jain, M., 2006. Experimental investigations on solar chimney for room ventilation. Solar Energy, 80(8), pp.927-935.

●Monghasemi, N. and Vadiee, A., 2018. A review of solar chimney integrated systems for space heating and cooling application. Renewable and Sustainable Energy Reviews, 81, pp.2714-2730.

●Serageldin, A.A., Abdelrahman, A.K. and Ookawara, S., 2018. Parametric study and optimization of a solar chimney passive ventilation system coupled with an earth-to-air heat exchanger. Sustainable Energy Technologies and Assessments, 30, pp.263-278.

●Nguyen, Y.Q. and Wells, J.C., 2020. A numerical study on induced flowrate and thermal efficiency of a solar chimney with horizontal absorber surface for ventilation of buildings. Journal of Building Engineering, 28, p.101050.

●Zhai, X.Q., Song, Z.P. and Wang, R.Z., 2011. A review for the applications of solar chimneys in buildings. Renewable and Sustainable Energy Reviews, 15(8), pp.3757-3767.