Building’s solar chimney: The performance of width and inlet to in tropical country

S B Abraham1,2,3 and T Z Ming1

1School of Civil Engineering and Architecture, Wuhan University of Technology, Wuhan, P.R. China
2Architecture Department, Mercu Buana University, Jakarta, Indonesia

E-mail: abraham.seno@mercubuana.ac.id

Abstract. The main purpose of the passive solar design strategy is to maximize the potential for abundance of solar radiation to improve the heating/cooling of buildings. One of them is by utilizing the stack ventilation strategy. Solar Chimney (SC) is a technology that utilizes this effect. Further simulation of Abraham's research on buildings with SC such as widening the chimney and the opening/closing strategy of the inlet to test the performance of the inlet. The results showed that changes in the variation in width and angle of the solar chimney effectively increased the value of indoor wind speed by 10%. Whereas through the inlet open/close strategy it can be seen which inlet has the best performance.

1. Introduction
In a humid tropical climate such as Jakarta, thermal comfort is mainly provided by cooling room temperature, decreasing the moisture content supplied to the room, and providing clean air. "Comfortable" conditions as defined by the Indonesian standard (SNI) are room temperature 25°C and 54% to 66% relative humidity [1]. The effect of air movement with cross ventilation is not enough to create physiological cooling of buildings. The air velocity associated with natural convection is relatively small, usually not more than 2 m/s [2].

One method of passive cooling for tropical climate areas is the stack ventilation strategy [3]. The same principle can be used with openings of different heights, where the pressure difference between the two openings is related to the vertical gradient. One possibility to increase the stack effect used by the sun is using Solar Chimney (SC) to increase convective flow with an increase in temperature difference in the system [4].

Some researchers have researched the SC. Zha et al in his research on experimental and numerical SC for building ventilation [5]. The results of his research was the air velocity can be induced by SC between 0.02-0.45 m/s. Zha concluded that the use of SC could save around 14.5% of building electricity consumption in China. Nugroho's and Hiung research about evaluating parametric development of vertical SC ventilation in hot and humid climate [6]. In this study a CFD (Computational Fluid Dynamic) simulation program was used in a simple home model using vertical solar chimney ventilation, resulting in the optimal use of vertical SC ventilation parameter design (height, width, length) to increase air velocity (0, 15 m/s) is good for cooling the temperature in the room. Others, Hassanein et al experimentally investigate the effects of the number of solar chimneys, SC height, air gap width, and SC orientation in natural ventilation in space. Hassanein concluded that using 3 SC was able to increase ventilation by 12% [7]. Ding et al examined the highlighted natural
Ding concluded that increasing the SC height made more ventilation rates and also advantageous to obtain a favorable distribution of pressure differences. Abraham discussed the effectiveness of using Double-Skin Façade (DSF) and SC on an office building prototype [9]. The three variants of using DSF and SC are used. The results show that the use of DSF and vertical SC is the best. Stably the wind velocity is maintained from the inlet, center of the room, to the gap, and in the gap itself, while the maximum wind velocity that can be induced is 2.5 m/s in the gap, while in space, due to induced DSF and SC, the speed can exceed the standard, which is above 1.5 m/s.

But none of those studies have discussed the performance of SC mainly related to changes in Heat flux, changes in SC aspects, and performance of inlets in buildings especially in humid tropical locations. In this study, various possibilities were tested to see the performance of each part such as inlet, the width of SC, and the effect of Heat flux changes on the overall ventilation performance of SC applied to an office building prototype located in one of the humid tropical, Indonesia. The main objective of this research is to re-ascertain the effectiveness of the use of SC, especially from aspects of the element changes. So, in the future, there will be a passive ventilation system prototype for office buildings.

2. Model / system description

In this simulation, the 8-story office model is assumed to be located in Jakarta, or around 6°12 S, 106°49'E with medium to high-density levels. Each floor has the same room. Office buildings use SC with orientation towards north. SC has a 1000 mm gap and a clear glass collector. The building uses a passive cooling system. Air flow is induced through the inlet located on the south side of the building. The inlet is at the bottom of the room on each floor. From the inlet, air will be induced towards the SC gap through the SC gap inlet at the top of the space. After going through the SC gap, air will exit through the SC outlet. The model used is the same as the previous research. In this simulation, the building prototype has a total width of 13.2 m with a floor height of 4 m as shown in figure 1(a). The measurement points are along the SC. In this simulation, 2 line measurements are used. One is in the SC gap with 20 points; one is on each floor (room average points measurement) as shown in figure 1(b). Simulation is done in 3 stages. The first stage tries to add the width of the chimney to 1300 mm,
1500 mm and 1700 mm. The second simulation compares the performance of DSF and solar chimney in each heat flux. Heat flux compared from 100 to 800 W/m². In the third stage simulation, compared the performance of each inlet per floor and a combination of several floors. This is done to find out which combination of inlet is best for increasing overall air ventilation.

2.1. Mathematics equation
Mathematical equations include the Mass Flow equation, the Navier-Stokes equations, the energy equation, and k-ε Turbulence Model equations which are presented as follows:

Assuming one-dimensional, steady-state flow, the following mass flow rate could be valid equation:

\[
m = \rho_{co}U_{co}
\]  

Navier-Stokes equation:

\[
\frac{d\rho u}{dt} = -\frac{\partial P}{\partial r} + \frac{\partial}{\partial r}\left[2\mu \frac{\partial u}{\partial r} + \mu' \nabla \cdot \vec{u}\right] + \frac{\partial}{\partial z}\left[\mu \left(\frac{\partial u}{\partial z} + \frac{\partial v}{\partial r}\right)\right] + 2\mu \left(\frac{\partial u}{\partial r} - \frac{v}{r}\right)
\]

\[
\frac{d\rho v}{dt} = -\frac{\partial P}{\partial r} + \rho g + \frac{\partial}{\partial r}\left[2\mu \frac{\partial v}{\partial r} + \mu' \nabla \cdot \vec{u}\right] + \frac{1}{r} \frac{\partial}{\partial r}\left[\mu \left(\frac{\partial u}{\partial z} + \frac{\partial v}{\partial r}\right)\right]
\]

Energy equation:

\[
\rho c_p \left[\frac{\partial T}{\partial t} + \frac{1}{r} \frac{\partial}{\partial r}\left(r(T\rho u)\right) + \frac{\partial}{\partial z}(T\nu)\right] = \frac{\partial}{\partial z}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial z}\right] + \frac{1}{r} \frac{\partial}{\partial r}\left[r \left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial r}\right] + \frac{\partial}{\partial z}\left[\frac{\partial \rho}{\partial z}\right] + \frac{\partial P}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r}\left[r \left(\rho u e + \frac{\partial}{\partial r}\right)\right] + \phi
\]

k-ε Turbulence Model equations:

\[
\rho \left[\frac{1}{r} \frac{\partial}{\partial r}\left(rku\right) + \frac{\partial}{\partial z}(k\nu)\right] = \frac{\partial}{\partial z}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial z}\right] + \frac{1}{r} \frac{\partial}{\partial r}\left[r \left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial r}\right] + \frac{\partial}{\partial z}\left[\frac{\partial P}{\partial z}\right] + \frac{1}{r} \frac{\partial}{\partial r}\left[r \left(\rho u e + \frac{\partial}{\partial r}\right)\right] + G_k + \beta g \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial z} - \rho \varepsilon
\]

\[
\rho \left[\frac{1}{r} \frac{\partial}{\partial r}\left(r\nu u\right) + \frac{\partial}{\partial z}(\epsilon \nu)\right] = \frac{\partial}{\partial z}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right) \frac{\partial \epsilon}{\partial z}\right] + \frac{1}{r} \frac{\partial}{\partial r}\left[r \left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right) \frac{\partial \epsilon}{\partial r}\right] + C_1n \frac{\partial \rho}{\partial z} - C_2n \frac{\epsilon^2}{k}
\]

2.2. Boundary condition
Gravitational acceleration was set at -9.81 m/s² at y-axis. The turbulence model is k-ε with full buoyancy effect used to model the turbulent flow [10,11]. Inlet and outlet are conditioned as pressure inlet/outlet, so the wind velocity’s input set at 0 m/s (without outside wind/temperature). While air density was set as boussinesq in 1.205 kg/m³, thermal expansion coefficient set at 3.41E-03 1/K [12]. Glass and Concrete material are use in this simulation. Table 1 below are the material properties used (table 1). The boundary conditions used are heat walls, glass walls, inlet, and outlet. Ambient temperature simulations was set at 293 K [13], while the heat-wall surface was thought to be exposed to a heat flux of 800 W/m² [14]. Glass wall set with 305 K temperature. All other surfaces are designated as adiabatic (table 1). Outside building temperatures were ignored in this research.

| Place          | Type            | Value               |
|----------------|-----------------|---------------------|
| Room inlet     | Pressure inlet  | \( P_{r,0}=0 \) Pa, \( T_0=293 \)K |
| Chimney outlet | Pressure outlet | \( P_{r,0}=0 \) Pa |
| Glass wall     | Temperature     | 305K                |
| Heat wall      | Heat flux wall  | 800 W/m²            |
| Others wall    | Adiabatic wall  | \( q=0 \) W/m²      |

3. Result and discussion
This research based on previous research on 8-story office building models. In this study, the upright
SC model was chosen with a width of 1000 mm. Then tested by changing the width of the SC. Using 800 W/m² uniform Heat flux, the simulation is carried out at 1000 mm (as a comparison from previous studies) and the gap width is 1300, 1500 and 1700 mm.

**Figure 2.** Wind velocity along the chimney with SC gap 1000-1700 mm.

**Figure 3.** Wind velocity in each room with SC gap 1000-1700 mm.

Figures 2 and 3 above are wind velocity along the chimney and wind velocity in each room every floor at Heat flux 800 W/m² for gap changes of 1000, 1300, 1500 and 1700 mm, respectively. Based on these results, the 1000 mm gap has a very good SC performance because it can induce wind velocity to close to 3 m/s, but only occurs in the final 1/3 of the SC, while the first 2/3 part, the wind velocity that is successfully induced is not significant. This is due to the minimal pressure that occurs in the lower 2/3 of the chimney. This 2/3 earlier part of SC functions to suck air from the room (from the inlet), while 1/3 the final part of the chimney only channel to the outlet chimney.

Meanwhile, based on figure 2, the use of SC with a 1300 mm gap has the best performance if we saw from the wind velocity which is successfully induced in each room on each floor. In figure 3, it can be seen that the performance of the chimney from 1st floor to 5th and 6th floors has decreased to 40%, but on the 7th and 8th floors increased again to 25%. This shows the higher floor, the worst performance will produced, but, the more towards the outlet (which is not burdened with the suction process), the SC suction performance will actually improve. However, along with the increasing wind velocity in the SC (the final 1/3 of the chimney) itself will force the process of vacuuming towards the chimney (previous analysis).

In the second simulation, the chimney was tested with uniform heat fluxes ranging from 100 to 800 W/m². This simulation will only see the effect of heat flux differences on SC performance. For this
reason, this simulation uses SC with a 1000 mm wide gap based on previous research. The difference in heat flux is based on the hourly solar radiation recorded in Jakarta, Indonesia [15,16].

![Figure 4. Wind velocity along the chimney with uniform heat flux 100-800 W/m².](image1)

![Figure 5. Wind velocity on room with uniform heat flux 100-800 W/m².](image2)

Figures 4 and 5 above are wind velocity along the chimney and wind velocity in each room every floor in chimney with a 1000 mm gap with uniform Heat flux between 100 to 800 W/m², respectively. Based on figure 4, there is indeed no difference in wind velocity along the chimney from each heat flux from 100 to 800 W/m². The difference in increase tends to be stagnant from one heat flux to higher heat flux.

The difference of heat flux can be seen more clearly from the wind velocity that was successfully induced by the chimney in the space on each floor. The best performance of the chimney in inducing wind velocity is best seen in the heat flux 800 W/m². However, it can be seen that between the heat flux 200-250 W/m² and the heat flux 500-600 W/m² the difference is very small. Moreover, as mentioned before, from the 1st floor up to the 5th and 6th floors of chimney performance, it decreased by around 40%, but on the 7th and 8th floors, the performance of chimney increased again by about 25% (figure 5). This phenomenon due to an increase in wind velocity in the SC section on the 7th and 8th floors.

The third simulation is to test the open/close of the inlet. This configuration means that in the real condition, it might happen the floor cannot be used/closed. This configuration is inlet closure on one
floor, up to seven floor, compared to all open inlets on the heat flux 800 W/m².

Figure 6 above is the graphic of maximum wind velocity induce by solar chimney with the open/close inlet strategy on the heat flux 800 W/m². The Y-axis shows the maximum wind velocity which can be generated from each configuration. Axis X shows the number of configurations of the Open/close inlet strategy that can be used. For example, for one open inlet, only inlet on 1st, 2nd, until the 8th floor. The configuration two inlet can be the 1st and 2nd floor, 1st and 3rd floor, and others. All the configurations can be seen in table 2 below.

| Configuration of Open Chimney | 1 open inlet | 2 open inlet | 3 open inlet | 4 open inlet | 5 open inlet | 6 open inlet | 7 open inlet | All Open Inlet |
|-----------------------------|--------------|--------------|--------------|--------------|--------------|--------------|--------------|---------------|
| 1 Open Inlet              |              |              |              |              |              |              |              | 1 config      |
| 2 Open Inlet              |              |              |              |              |              |              |              | 2 config      |
| 3 Open Inlet              |              |              |              |              |              |              |              | 3 config      |
| 4 Open Inlet              |              |              |              |              |              |              |              | 4 config      |
| 5 Open Inlet              |              |              |              |              |              |              |              | 5 config      |
| 6 Open Inlet              |              |              |              |              |              |              |              | 6 config      |
| 7 Open Inlet              |              |              |              |              |              |              |              | 7 config      |
| All Open Inlet            |              |              |              |              |              |              |              | 8 config      |

Based on figure 6 above, it can be seen that the more inlets that are supposed to be opened, the more air flow that will be induced. The peak reaches 3.06 m/s in the chimney section. If using the open/close inlet strategy, the most effective way to close the inlet is only to close 1-4 inlets. Opening a minimum of 4 inlets can be induced the wind velocity on each floor of more than 2.5 m/s. Open the inlet less than 4 can be induces the wind velocity on each floor not more than 2.5 m/s.

4. Conclusion

Simulation of several variations related to the use of vertical solar chimney on office building prototypes has been carried out. Through the simulation also proves that the use of vertical solar chimney can induce air to the room/floor. In some simulations, the wind velocity in the SC gap does not have significant changes, but in the room/floor section, have a significant change especially in wind velocity. This becomes very important because the emphasis of this research is on how to increase the comfort of building occupants through increasing wind velocity.

Widening the SC’s length is indeed one solution, but does not significantly affect the performance of SC. Widening the SC to 1300 mm only increases wind velocity by 10%. While the best chimney performance remains when the maximum daylight is exposed to the heat wall or the heat flux on 800 W/m². Through the Strategy Open/close inlet, it can be seen how minimal the inlet can be closed, to anticipate natural ventilation of the building if there is a room on closed. In general, the use of the
open/close inlet strategy, the performance of the chimney itself is still running, although there is a small reduction of performance depending on the number of closed inlets.

In the future, further research is still needed, especially how the effect of obstacles (building density) in front of the inlet. Also, it is also necessary to examine the effect of the crosswind on the overall SC performance. Furthermore, it also needs to be considered whether this SC is capable as pollution suction device as good as its function as passive ventilation device.

References
[1] Kholiq A and Hidayat M S 2016 Pengaruh Bentuk Atap Terhadap Karakteristik Thermal Pada Rumah Tinggal Tiga Lantai Vitruvian 5 1-7 Purwaningsih T and Hidayat M S Studi kenamannya termal ruang kelas tk tunas muda X ikkt jakarta barat
[2] Andersen K T 1995 Theoretical considerations on natural ventilation by thermal buoyancy American Society of Heating, Refrigerating and Air-Conditioning Engineers Theoretical considerations on natural ventilation by thermal buoyancy. United States: N. p., 1995. Web.
[3] Vidiyanti C Kajian Retrofit Bangunan sebagai Upaya Mereduksi Konsumsi Energi Operasional Studi Kasus: Campus Centre (Cc) Barat Itb Vitruvian 5 Kajian Retrofit Bangunan sebagai Upaya Mereduksi Konsumsi Energi Operasional Studi Kasus: Campus Centre Barat ITB Vitruvian 5 1-9
[4] Andreozzi A, Buonomo B and Manca O 2010 Thermal and fluid dynamic behaviors in symmetrical heated channel-chimney systems International Journal of Numerical Methods for Heat & Fluid Flow Int. J. Numer. Method. H. 20 811-33
[5] Zha X, Zhang J and Qin M 2017 Experimental and numerical studies of solar chimney for ventilation in low energy buildings Procedia Engineering Procedia Engineer. 205 1612-9
[6] Nugroho A M, Ahmad M H and Hiung T J 2006 Evaluation of parametrics for the development of vertical solar chimney ventilation in hot and humid climate The 2nd Int. Network For Tropical Architecture Conf. (Christian Wacana University, Jogjakarta)
[7] Hassanein S and Fadeel W 2012 Improvement of natural ventilation in building using multi solar chimneys at different location Journal of Engineering Sciences J. Eng. Sci. 40 1661-77
[8] Ding W, Hasemi Y and Yamada T 2005 Natural ventilation performance of a double-skin façade with a solar chimney Energy and Buildings Energ. Buildings 37 411-8
[9] Abraham S and Ming T 2018 Numerical analysis on the thermal performance of a building with solar chimney and double skin façade in tropical country IOP Conf. Series: Materials Science and Engineering (IOP Publishing) p 012030
[10] Ming T Z, Liu W, Xu G L, Xiong Y B, Guan X H and Pan Y 2008 Numerical simulation of the solar chimney power plant systems coupled with turbine Renewable Energy Renew. Energ. 33 897-905
[11] Guo P, Li J, Wang Y and Liu Y 2013 Numerical analysis of the optimal turbine pressure drop ratio in a solar chimney power plant Solar Energy Sol. Energ.98 42-8
[12] Guo P H, Li J Y and Wang Y 2014 Numerical simulations of solar chimney power plant with radiation model Renewable Energy Renew. Energ. 62 24-30
[13] Ming T, de Richter R K, Meng F, Pan Y and Liu W 2013 Chimney shape numerical study for solar chimney power generating systems International Journal of Energy Research Int. J. Energ. Res. 37 310-22
[14] Ahmad S, Badshah S and Chohan G 2014 Modeling and simulation of natural ventilation of building using solar chimney World Applied Sciences Journal World Appl. Sci. J. 32 741-6
[15] Budiyanto M A and Lubis M H 2018 Hourly solar radiation in Depok, West Java, Indonesia (106.7942 Longitude,-6.4025 Latitude) IOP Conference Series: Earth and Environmental Science (IOP Publishing) p 012088
[16] Purnama M S S and Sutanto D 2018 Dynamic facade module prototype development for solar radiation prevention in high rise building IOP Conference Series: Earth and Environmental Science (IOP Publishing) p 012023