Influence of outlet/inlet area ratio on performance of a vertical solar chimney for natural ventilation of buildings

Y Q Nguyen and T N Huynh
Faculty of Engineering, Van Lang University, 45 Nguyen Khac Nhu St., Co Giang Ward, Dist. 1, Ho Chi Minh City, Viet Nam
Email: y.nq@vlu.edu.vn

Abstract. In this study, effects of the contractions of the inlet and outlet areas of a vertical solar chimney were examined with a Computational Fluid Dynamics. The results show that the flow rate decreased with both the inlet and outlet areas. The outlet area influenced more than the inlet area did. The flow rate decreased linearly with the outlet area, however, was affected significantly only when the inlet area was below half of the air gap.

Keywords: Solar chimney, natural ventilation, inlet area, outlet area, CFD.

1. Introduction
Using renewable energy resources is a key factor of the green building rating systems. For example, in the LEED rating system of the U.S. Green Building Council, renewable energy credits up to 5 over a total of 110 points. One of the applications that can be based totally on renewable energy resources in buildings is the ventilation system. Natural ventilation utilizes natural driving forces for the air flow, such as effects due to external wind or thermal effects due to indoor – outdoor temperature difference [1].

Solar chimney is the common system based on the thermal effects for natural ventilation of buildings. The simple form of a solar chimney is a tube or pipe attached to a wall of a building. The tube or pipe absorbs solar radiation on its surfaces and heats the air inside its channel. As the air temperature rises, the stack effects can withdraw air from the building and discharge to the ambient. Accordingly, the building is ventilated naturally [2].

Among the design factors of a solar chimney, previous studies showed that the size and shape of the inlet of a solar chimney strongly influence its performance. The experimental data by Mathur et al. [3] and the numerical results by Zamora and Kaiser [4] indicated that the induced flow rate increased with the inlet size. However, effects of the inlet size ceased as it was above two times the air gap [4]. For a solar chimney attached to a wall of a building, increasing the inlet height from the floor resulted in lower induced flow rate [5]. Al-Kayiem et al. [6] tested different shapes of the inlet of a roof solar chimney. They reported that the flow rate was higher when the upper wall of the air channel was longer than the lower one.

Although most of the solar chimneys examined in the literature had equal areas for inlet and outlet, tests with different areas of the inlet and the outlet were also reported. Li et al. [7] conducted numerical simulations while Susanti et al. [8] took experiments for solar chimneys with different ratios of the areas of the outlet ($A_{out}$) and inlet ($A_{in}$). Both studies showed that the induced flow rate increased with $A_{out}/A_{in}$. The results by Susanti et al. [8] furthermore showed that the outlet area influenced more than
the inlet did. For example, the case of $A_{out}/A_{in} = 312/14$ had higher flow rate than that of the case of $A_{out}/A_{in} = 14/312$. Similar conclusions were also seen in Al-Kayiem et al. [6].

In the above studies, effects of unequal area for the inlet and outlet were conducted for an inclined solar chimney [6, 8] or vertical solar chimney with different directions of the inlet and outlet [3, 4, 7]. It is practically interesting to test those effects for a vertical solar chimney with vertical inlet and outlet. In this study, we examined a vertical solar chimney with different areas for the inlet and outlet numerically. Performance of the chimney in terms of the induced flow patterns and induced flow rate was evaluated.

2. Description of the problem and the numerical model

2.1. Solar chimney

![Figure 1. Schematic of a solar chimney integrated into a building (a) and its simple model (b).](image)

A vertical solar chimney for ventilation of a building is depicted in figure 1a. The chimney is assumed on the roof of the building. Solar radiation is absorbed on the walls of the chimney. The absorbed heat is then transferred to the air in the channel. Warmed air rises up and induces an air flow through the building for ventilation.

In practical applications, an air damper for controlling the ventilation rate is placed at the inlet of the air channel, as sketched in figure 1a. At the outlet of the chimney, a cover for weatherproofing may also be required. With these additional elements, the inlet and outlet areas of the air channel are contracted and less than that of the air channel. Therefore, it is necessary to study the performance of a solar chimney when the air channel is contracted at the inlet and/or the outlet instead of a straight one.

A simple two-dimensional model of the solar chimney shown in figure 1a is presented in figure 1b. Major dimensions of the chimney are the height $H$ and the gap $G$ of the air channel. The inlet and outlet areas are denoted as $A_{in}$ and $A_{out}$, respectively. For modeling heat transfer inside the air channel, for simplicity, the heat source is assumed inside the air channel. In figure 1b, the heat source can be on either the left or the right wall.

2.2. Governing equations

A two-dimensional, steady, incompressible, and turbulent flow in the vertical plane can be modeled with the Reynolds Averaged Navier–Stokes equations as follows.
\[
\frac{\partial u_j}{\partial x_j} = 0
\]

\[
\frac{\partial (u_iu_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} - g_i \beta (T - T_0) + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial u_i}{\partial x_j} - u'_i u'_j \right)
\]

\[
\frac{\partial (T u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \nu \frac{\partial T}{\partial x_j} - T' u'_j \right)
\]

where \( u \) and \( T \) stand for the time-averaged velocity and temperature; \( u' \) and \( T' \) are the fluctuating velocity and temperature; \( T_0 \) is the ambient temperature; \( p \) is the pressure; \( \nu, \rho, \beta \) are the air kinematics viscosity, density, and thermal expansion coefficient; \( Pr \) and \( g \) are the Prandtl number and gravitational acceleration; \( u'_i u'_j \) and \( T' u'_j \) are the turbulence stress and turbulence heat flux which are solved with the RNG \( k - \epsilon \) turbulence model.

### 2.3. Numerical setup

The above equations were discretized by the Finite Volume Method with the CFD (Computational Fluid Dynamics) code ANSYS Fluent (Academic version 2020R2). The computational domain and mesh are presented in figure 2.

![Figure 2. Computational domain and mesh.](image)

The computational domain (figure 2) included both the air channel and ambient air. Gan [9] claimed that the extension outside the air channel is required to obtain reliable results. In figure 2, the extension was more than 10 times of the channel gap above the outlet and 5 times of the channel gap below the inlet and to the sides of the air channel.

The mesh was structured. It was finer near the solid surfaces. Mesh – refinement tests showed the numerical results changed below 1.0% when the non-dimensional distance \( y^+ \) of the first grid points near the solid surface was less than 1.5.

On the boundary of the domain, the pressure was assumed at the atmospheric value. All solid walls applied non-slip conditions. Air entered the domain at the ambient temperature which was fixed at 20°C in all tests. Inside the air channel, a uniform heat flux was applied on one wall while the opposite wall was able to receive radiative heat transfer from the other. Two cases of the heat source location were examined: heating the left or the right wall.

For the numerical setups, radiation between two walls in the air channel was calculated with the S2S radiation model in ANSYS Fluent. The SIMPLE method for the coupling between the continuity equation (equation (1)) and the momentum equation (equation (2)), and the PRESTO! method for
interpolation of the pressure on the staggered mesh were selected. Second order upwind scheme for equations (2) and (3), and first order upwind scheme for the equations for \( k \) and \( \varepsilon \) were utilized.

2.4. Validation

Figure 3 shows the induced flow rate through a solar chimney obtained with the CFD model and the experiment by Burek and Habeb [10]. The chimney had a length of 1.025 m, a width of 1.0 m, and a gap of 40 mm. The heat flux changed in the range of \( 400 \text{ W/m}^2 \) to \( 1000 \text{ W/m}^2 \). The CFD model slightly underestimated the flow rate at \( 400 \text{ W/m}^2 \) and \( 1000 \text{ W/m}^2 \) and overestimated at \( 800 \text{ W/m}^2 \). However, the maximum difference between them was only 9.2% at \( 400 \text{ W/m}^2 \). This discrepancy should be within the uncertainties of the measurement. Therefore, the prediction of the CFD model was acceptably accurate.

3. Results and Discussions

In this section, the induced flow rate was compared among four cases:
- Without any area restriction at the inlet and outlet \( (A_{out} = A_{in} = G, \text{the base case}) \).
- With only restriction at the outlet \( (0 < A_{out} \leq G, A_{in} = G) \).
- With only restriction at the inlet \( (A_{out} = G, 0 < A_{in} \leq G) \).
- With restrictions at both outlet and inlet \( (0 < A_{out} \leq G, 0 < A_{in} \leq G) \).

In all tests, H and G were fixed to 1.0 m and 0.2 m, respectively. First, \( A_{out} \) and \( A_{in} \) were fixed to 50.0% of G, if any, while the heat flux changed from 250 to 1000 \text{ W/m}^2. The heat source was either on the left or the right wall (see figure 2). Second, the heat flux was fixed to 500 \text{ W/m}^2 while \( A_{out} \) and \( A_{in} \) changed from 0 to G.

3.1. Comparison at different heat fluxes

Figure 4 shows the induced flow rate obtained with four cases of the inlet and outlet areas at different heat fluxes. The heat source was on the left wall (LWH) of the right wall (RWH). In all cases, the flow rate increased with the heat flux. This trend has been widely reported in the literature [10 – 12] as the flow rate increased with the heat input in the air channel. The base case had the highest flow rate, as the flow did not experience any restriction of the flow area. Other cases always had lower flow rates.

Two interesting points are observed in figure 4. Firstly, with the same area of 0.5G, the case \( A_{out} = 0.5G \) had much lower flow rates than those with the case \( A_{in} = 0.5G \). For example, at 1000 \text{ W/m}^2, the cases \( A_{out} = 0.5G \) resulted in 0.0404 kg/s (LWH) while the case \( A_{in} = 0.5G \) had 0.0623 kg/s (LWH), which was almost 50.0% higher. Therefore, the restriction of the outlet area influenced more than that of the inlet did. This point is further confirmed as the case \( A_{out} = A_{in} = 0.5G \) and \( A_{out} = 0.5G \) had similar flow rates. This observation was also reported by Susanti et al. [8] and Al-Kayiem et al. [6]. Secondly, between heating either side of the channel, LWH offered higher flow rate for the case \( A_{out} = \)
0.5G but lower flow rate for the case $A_{in} = 0.5G$. Particularly, the case $A_{in} = 0.5G$ and RWH had a flow rate identical to that of the base case.

![Figure 4. Induced flow rate obtained with $A_{in} = G$ and $A_{out} = 0.5G$ (a), with $A_{in} = 0.5G$ and $A_{out} = G$ (b), and $A_{out} = A_{in} = 0.5G$ (c), for heating the left (LWH) or the right (RWH) wall. The base case is for $A_{out} = A_{in} = G$.](image)

Figure 5 displays the flow and temperature fields for the cases in figure 4. In the base case, the velocity and thermal boundary layers developed freely along walls of the air channel. When the outlet was contracted (figure 5b), the flow was obstructed. A separation zone appeared near the outlet which should increase the flow resistance; hence reduced the flow rate. The RWH had a separation zone larger than that of the LWH; hence lower flow rate, as seen in figure 4a. A separation zone near the inlet was also seen in figure 5c and caused the reduction of the flow rate compared to that of the base case. However, it is interesting to see that for RWH, the thermal boundary layer along the right wall in figure 5c was enhanced and yielded the flow field at the outlet similar to the base case. As a result, the flow rate in this case was close to that of the base case, as seen in figure 4a. When both openings were contracted (figure 5d), the flow fields near the outlet were identical to those in figure 5b; hence similar flow rates are seen between figures 4a and 4c.

![Figure 5. Flow (left) and temperature (right) fields obtained with four cases of the inlet and outlet areas.](image)

### 3.2. Comparison of different inlet and outlet areas

#### 3.2.1. Changing outlet and outlet areas

Figure 6 shows the flow rate when one end of the channel was fully open while the other end was partially restricted. It is seen that the flow rate decreased with
both $A_{in}$ and $A_{out}$. The decrease rate was almost linearly for $A_{out}$. For $A_{in}$, the decrease was significant when $A_{in}$ was smaller than 0.5G. RWH cases resulted in lower flow rate as $A_{out}$ changed but higher flow rate as $A_{in}$ changed. However, the maximum difference between the flow rates of LWH and RWH in figure 6 was only less than 12.0%.

The flow field in figures 5b and 5c may again help to explain the fact that $A_{out}$ influences the flow rate more than $A_{in}$ does. The separation zone near the outlet as $A_{out}$ decreases (figure 5b) is larger than that near the inlet as $A_{in}$ is contracted (figure 5c). Therefore, the flow with the contracted $A_{out}$ is more obstructed and accordingly, the flow rate is less.

In figure 5b, between LWH and RWH, though the separation zones are similar, the main flow field near the heated wall for RWH may experience more resistance at the outlet due to sudden change of the flow path as $A_{out}$ is contracted on the right side. Therefore, LWH offered slightly higher flow rate, as seen in figure 6a. In figure 5c, the thickness of the thermal and velocity boundary layers of the RWH case are more than those of the LWH. Accordingly, the flow rate of RWH is higher, as seen in figure 6b.

![Figure 6](image)

**Figure 6.** Induced flow rate obtained with $A_{in} = G$ and changing $A_{out}$ (a) and with $A_{out} = G$ and changing $A_{in}$ (b)

3.2.2 Changing the ratio of $A_{out}/A_{in}$ In figure 7, the flow rate is plotted for different $A_{out}/A_{in}$ ratios. Figure 7a shows the results for equal area cases. The flow rate also decreased as both areas were reduced, similar to figure 6a. Both cases of heating location resulted in identical flow rates, with the differences of within 11.0%. Figure 7b presents the flow rates for different areas. It is seen that for two cases $A_{out}/A_{in} = 0.75/0.5$ and $A_{out}/A_{in} = 0.5/0.75$, though the flow experienced the same areas of the two ends of the channel, the case of larger outlet ($A_{out}/A_{in} = 0.75/0.5$) had a higher flow rate, which is 17.5% for LWH and 6.7% for RWH. Similar points are also seen for two cases of $A_{out}/A_{in} = 0.75/0.25$ and $A_{out}/A_{in} = 0.25/0.75$, however, the differences are enhanced with 28.0% for LWH and 42.0% for RWH. Therefore, it is again confirmed that the outlet area influenced more than the inlet area did, as seen in figures 4 and 7.
4. Conclusions

From the results, the following points are observed:
- The flow rate decreased with both the inlet and outlet areas. While the flow rate decreased linearly with the outlet area alone, it was affected significantly when $A_{\text{in}} < 0.5G$.
- The outlet area influenced more than the inlet area did. The difference was significant as the areas decreased. As $A_{\text{out}} = 0.5G$, the flow rate was about 57.0% of that of the base case while $A_{\text{in}} = 0.5G$, the flow rate was about 87.0% of that of the base case (for LWH).
- When either the outlet or the inlet area was reduced, reducing $A_{\text{out}}$ had higher flow rate with LWH but reducing $A_{\text{in}}$ had higher flow rate with RWH.
- When both areas are reduced, both LWH and RWH had similar flow rates with the maximum difference was less than 11.0%.

References

[1] Awbi H 2003 Ventilation of Buildings Second Edition (Spon Press)
[2] Bansal N K, Mathur R and Bhandari M S 1993 Solar chimney for enhanced stack ventilation *Building and Environment* **28** (3) 373–377
[3] Mathur J, Bansal N K, Mathur S, Jain M and Anupma 2006 Experimental investigations on solar chimney for room ventilation *Solar Energy* **80** (8) 927–935
[4] Zamora B and Kaiser A S 2009 Optimum wall-to-wall spacing in solar chimney shaped channels in natural convection by numerical investigation *Applied Thermal Engineering* **29** (4) 762–769
[5] Shi L, Zhang G, Yang W, Huang D, Cheng X and Setunge S Determining the influencing factors on the performance of solar chimney in buildings *Renewable and Sustainable Energy Reviews* **88** (2018) 223–238.
[6] Al-Kayiem H H, Sreejaya K V and Chikere A O Experimental and numerical analysis of the influence of inlet configuration on the performance of a roof top solar chimney *Energy and Buildings* **159** (2018) 89–98.
[7] Li A, Jones P, Zhao P and Wang L 2004 Heat Transfer and Natural Ventilation Airflow Rates from Single-sided Heated Solar Chimney for Buildings *Journal of Asian Architecture and Building Engineering* **3** (2) 233-238
[8] Susanti L, Homma H, Matsumoto H, Suzuki Y and Shimizu M 2008 A laboratory experiment on natural ventilation through a roof cavity for reduction of solar heat gain *Energy and Buildings* **40** (12) 2196–2206
[9] Gan G 2010 Impact of computational domain on the prediction of buoyancy-driven ventilation cooling *Building and Environment* **45** (5) 1173–1183
[10] Burek S A M and Habe A 2007 Air flow and thermal efficiency characteristics in solar chimneys
and Trombe Walls Energy and Buildings 39 (2) 128–135

[11] Chen Z D, Bandopadhayay P, Halldorsson J, Byrjalsen C, Heiselberg P and Li Y An experimental investigation of a solar chimney model with uniform wall heat flux Building and Environment 38 (7) (2003) 893–906.

[12] Shi L, Zhang G, Yang W, Huang D, Cheng X and Setunge S Determining the influencing factors on the performance of solar chimney in buildings Renewable and Sustainable Energy Reviews 88 (2018) 223–238.