Enhancing induced flow rate through a solar chimney by a forced flow

Y Q Nguyen
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. Solar chimneys absorb solar radiation for natural ventilation or cooling of buildings. In typical solar chimneys, the air flow is induced by the thermal effect which causes pressure gradient inside the chimney to deviate from that of the ambient air. The induced flow is employed to ventilate the connected building naturally. In this study, we aimed to boost the induced flow rate through a typical solar chimney by a forced flow at the inlet of the air channel which is divided into two ports for the forced and the induced flows. The air flow and heat transfer were modeled with a Computational Fluid Dynamics (CFD) model with the ANSYS Fluent CFD code. Changing factors included the forced velocity, the dimensions of the inlet port for the forced air flow and the length of the partition between the ports of the naturally induced and forced air flows. It is seen that the forced flow significantly reduces the pressure in the air channel compared to that of a typical chimney; hence increasing the induced flow rate. The flow rate was significantly enhanced as the forced velocity was above 0.2 m/s; the area of the inlet port for the forced flow was 12.4% of the total inlet area; and the partition height was less than 5.0% of the total height of the air channel. In this configuration, the forced flow can be the waste exhaust air flow from other mechanical ventilation systems of the building. Therefore, no additional energy is required for this solution.

Keywords: Solar chimney, natural ventilation, forced flow, induced flow rate, CFD.

1. Introduction

Thermal or chimney effect can be utilized for ventilation or cooling of a building with enclosed cavities on the building envelope, such as double – skin facades, which connect to the rooms in the building. As solar radiation is absorbed on the surfaces of the cavity, air inside the cavity is warmed, rises, and induces an air flow through the cavity and the connected room; hence natural ventilation. Such cavities are categorized as solar chimneys [1].

The naturally induced air flow through solar chimneys has been examined by a number of researchers [1-4]. These reports show that the induced air flow rate increases with the major dimensions of the air cavity, or air channel, including the height and the gap, and the heat flux on surfaces of the channel. With proper designs, solar chimneys can provide adequate ventilation rate and thermal comfort conditions for buildings [5].

Forced air flow can also assist the naturally induced one in solar chimneys. Elghamry and Hassan [6] considered both forced and natural ventilation and cooling of a room with the combination of a solar chimney, photovoltaic, and geothermal air tube. They reported that the daily ventilated air and heat...
released from the room are enhanced with the forced flow through the geothermal tube. Liu et al. [7] utilized an indoor exhaust fan to ventilate a façade. Their results show that the forced flow reduces up to 90% of the heat flux transferred through the façade. In addition, the optimal forced air velocity is in the range of 0.5 – 1.0 m/s.

In this study, we proposed a combined system of forced and natural flows in a solar chimney for natural ventilation of a building. The forced flow assists the natural flow which ventilates the connected room. Therefore, the forced flow does not go through the room as in [6]. In addition, the forced flow can be collected from the waste flow at the exhaust of other mechanical ventilation systems; hence no additional energy is required as in [7].

The proposed system was examined numerically. The induced flow rate and temperature rise were investigated under influences of different factors including the force air velocity, the lengths of the ports of the forced and natural flows, and the height of the partition between the ports.

2. Description of the problem and the numerical model

2.1. The problem

Figure 1. Schematic of: a) A solar chimney ventilating a building, b) A simplified model of the solar chimney, and c) The computational domain and mesh.

Figure 1a shows a solar chimney for natural ventilation of a building. The chimney is attached to a building wall. Either the building wall or the cover plate of the chimney can absorb solar radiation. The lower end of the air channel is divided into two portions by a partition: The left portion is connected to the room while the right part is supplied with a forced air flow. Without the forced flow, as the air in the channel between the cover plate and the building wall is warmed by the absorbed heat, it rises and induces an air flow through the connected room. The forced flow is aimed to assist the thermal effect and enhance the induced flow through the room.

The simplified model of the solar chimney in figure 1a is presented in figure 1b. It consists of a straight channel with two inlets at the lower end. The induced flow and the forced flow enter the channel from the left and the right inlets, respectively. The upper end of the chimney opens to the atmosphere. The absorbed solar radiation is described by a heat source on either the left or the right wall of the air channel. This simplified model is similar to those in the literature [3,4,7].

2.2. Governing equations

The computational model was based on the Computational Fluid Dynamics technique. The air flow and heat transfer in the air channel were approximated by the governing equations which state conservation laws of mass, momentum, and energy. For the chimney model in the vertical plane shown in figure 1b, a two – dimensional, steady, incompressible, and turbulent flow can be modeled with the Reynolds Averaged Navier – Stokes equations as follows.

\[
\frac{\partial u_j}{\partial x_j} = 0
\]  

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

(2)

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

(3)

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. Other turbulence models, such as the standard \( k - \epsilon \), standard \( k - \omega \), standard \( k - \omega \) with modifications for low Reynolds number effects were also tested. The RNG \( k - \epsilon \) offered the best convergence rate. Therefore, it was selected.

### 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 1c. The domain coincided with the air channel. The mesh was structured and rectangular. The mesh density was higher toward the solid walls. By changing the number of cells and the growth rate of the mesh cells from the solid surfaces, the induced flow rate through the chimney was compared. It was seen that the difference in the flow rate was less than 1.0\% when the maximum non-dimensional distance of the first cell centre from the solid walls, or \( y^+ \), was less than 1.5. The according mesh resolution was more than 15000 cells in a solar chimney of \( H=0.9 \, \text{m} \) and \( G=0.2 \, \text{m} \), and a heat flux of 500 \( W/\text{m}^2 \). For all tests in this study, the mesh was carefully checked to satisfy that required maximum \( y^+ \).

At the inlet for the induced flow, the pressure was assumed at the atmospheric value while at the inlet for the forced flow, a uniform air velocity was assigned. At the outlet of the air channel, the atmospheric pressure was also applied. Air entered the domain at both inlets at the ambient temperature which was fixed at 20\(^{\circ}\)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 either the left or the right wall. All solid walls applied non-slip conditions.

As seen in figure 1c, the height over which the heat source was applied is denoted as \( H \). \( G \) indicates the gap of the air channel. \( L_i \) and \( h_i \) respectively are the length of the inlet for the forced velocity and the height of the partition. Radiative heat transfer was allowed between the two surfaces over the height \( H \). Other walls were adiabatic.

For the numerical setups, radiation was computed 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 \( \epsilon \) were utilized.
2.4. Validation

In figure 2, the CFD model was validated against the experiment by Burek and Habeb [8]. 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 – 1000 W/m². The CFD model slightly underestimated the flow rate at 400 W/m² and 1000 W/m² and overestimated at 800 W/m². However, the maximum difference between them was about 9.2% at 400 W/m². This discrepancy should be within the uncertainties of the measurement. Therefore, the prediction of the CFD model was acceptably accurate for the induced flow rate.

3. Results and Discussions

In the following tests, the chimney height H and the air gap G were fixed to 0.9 m and 0.2 m, respectively. Two heat fluxes of 250 W/m² and 500 W/m² were applied on either wall of the air channel. The induced air flow and temperature rise were examined as the forced air velocity \( V_f \), length \( L_i \) of the inlet for the forced air flow, and the height \( h_i \) of the partition changed.

3.1. Effects of the forced velocity

In this test, \( L_i \) and \( h_i \) were fixed to 0.025 m and 0.1 m, respectively. The forced velocity, \( V_f \), changed from 0.0 m/s to 1.0 m/s. When the forced air velocity was 0.0 m/s, the inflow along \( L_i \) was entirely induced by the thermal effect in the air channel. The heat source was on the left wall. The induced flow rate and the temperature rise through the chimney are presented in figures 3a and 3b, respectively.

In figure 3a, the flow rate was computed at the inlet of the induced flow rate, which is the left inlet in figures 1b and 1c. It is seen that the induced flow rate increased with the forced velocity when \( V_f \) was above 0.2 m/s. The induced flow rate was slightly affected for \( V_f \leq 0.2 \) m/s with changes of within 6.0% compared to the case of \( V_f=0.0 \) m/s. At \( V_f=2.0 \) m/s, the flow rate was up to about 3.1 times of that at \( V_f=0.0 \) m/s. The flow rate also increased with the heat flux, as expected.

The temperature rise, \( \Delta T \), through the chimney is plotted in figure 3b. In contrast to the flow rate, \( \Delta T \) decreased with the forced velocity. However, similar to the flow rate, \( \Delta T \) was almost unchanged when \( V_f \) was less than 0.2 m/s. \( \Delta T \) at \( V_f = 2.0 \) m/s was about 70% of that at \( V_f = 2.0 \) m/s.

![Figure 2. Computed vs. measured flow rate through the solar chimney in the experiment by Burek and Habeb [8].](image)

![Figure 3. Induced flow rate as the forced velocity changed.](image)
To observe effects of the forced flow, the pressure, velocity, temperature distributions at $V_i = 0.0 \text{ m/s}$ and 1.0 m/s are presented in figure 4. Without the forced flow, the pressure distribution in the air channel followed the well-known thermal effect where the pressure increased along with the channel height. Near the inlet, the pressure was in the range of $-0.0131 \div -0.0115 \text{ Pa}$. With the forced flow, the pressure on the lower half of the air channel was strongly reduced down to $-0.0164 \text{ Pa}$. Lower pressure near the inlet achieved with the forced flow resulted in higher induced air velocity, as seen in figure 4c; hence higher flow rate, as seen in figure 3a.

It is seen in figure 4b for the temperature fields that in both cases of $V_i$, the thermal boundary layers along the heated (left) wall were identical. However, the thermal layer near the right wall, which received radiative heat from the left wall, was thinner with the forced flow. As a result, the temperature rise was lower for $V_i = 1.0 \text{ m/s}$, as seen in figure 3b.

3.2. Effects of inlet length
In this test, $h_i$ and $V_i$ were fixed to 0.1 m and 1.0 m/s, respectively. $L_i$ changed from 0.0 m (no forced flow) to 0.15 m, which is 75% of the air gap. Figure 5 shows the flow rate obtained at two heat fluxes of 250 $W/m^2$ and 500 $W/m^2$. The heat source was either on the left (Left) or the right (Right) wall. It is seen that the flow rate varied strongly with $L_i$. The peak flow rate was at $L_i = 0.025 \text{ m}$, which is 12.5% of the air gap. The peak flow rate was about 200% of that at $L_i = 0.0 \text{ m}$. As $L_i$ increased further, the flow rate decreased. At $L_i = 0.15 \text{ m}$, the flow rate was down to about 76% (at 250 $W/m^2$) to 64% (at 500 $W/m^2$ – Right) of that at $L_i = 0.0 \text{ m}$.

In figure 5, the flow rate is seen to increase with the heat flux. At the heat flux of 500 $W/m^2$, heating the right wall resulted in a lower flow rate which was about 6.0 to 10.0% less than that with heating the left wall. However, similar trends of variation of the data are seen for both cases of the heat flux and location of the heat source.

3.3. Effects of partition height
In this test, $L_i$ and $V_i$ were kept to 0.025 m and 1.0 m/s, respectively. The heat source was on the left wall. The partition height changed from 0.025 m to 0.3 m at two heat fluxes of 250 $W/m^2$ and 500 $W/m^2$. The induced flow rate and the temperature rise are plotted in figure 6. It is seen that the flow rate and the temperature rise increased with the applied heat flux but they were almost unchanged as $L_i$ increased from 0.025 m to 0.05 m. As $L_i$ increased further, both the flow rate and the temperature rise decreased down to 83% and 78% of those at $L_i = 0.025 \text{ m}$. Examination of the pressure field reveals that the pressure near the inlet of the air channel increased with the partition height; hence less induced flow rate. In addition, as the partition as treated as an adiabatic plate, radiative heat transfer in the air channel was blocked more as $h_i$ increased. This fact resulted in lower temperature near the right wall and less temperature rise through the air channel.

Figure 4. Pressure, velocity, temperature distributions at $V_i = 0.0 \text{ m/s}$ and 1.0 m/s.

3.3. Effects of partition height
In this test, $L_i$ and $V_i$ were kept to 0.025 m and 1.0 m/s, respectively. The heat source was on the left wall. The partition height changed from 0.025 m to 0.3 m at two heat fluxes of 250 $W/m^2$ and 500 $W/m^2$. The induced flow rate and the temperature rise are plotted in figure 6. It is seen that the flow rate and the temperature rise increased with the applied heat flux but they were almost unchanged as $L_i$ increased from 0.025 m to 0.05 m. As $L_i$ increased further, both the flow rate and the temperature rise decreased down to 83% and 78% of those at $L_i = 0.025 \text{ m}$. Examination of the pressure field reveals that the pressure near the inlet of the air channel increased with the partition height; hence less induced flow rate. In addition, as the partition as treated as an adiabatic plate, radiative heat transfer in the air channel was blocked more as $h_i$ increased. This fact resulted in lower temperature near the right wall and less temperature rise through the air channel.
4. Conclusions

From the results, the following findings are observed:

- The forced flow further decreased the pressure near the inlet of the air channel compared to that due to the thermal effect. The flow rate increased but the temperature rise decreased when the forced velocity was above 0.2 m/s. At $V_i = 2.0$ m/s, the flow rate was up to about 3.1 times but $\Delta T$ was down to 70% of those at $V_i = 0.0$ m/s. At $V_i \leq 0.2$ m/s, the flow rate and temperature rise were almost unaffected.

- As the length of the inlet for the forced flow increased, at $L_i = 0.025$ m, the flow rate increased to the peak value of about 200% of that at $L_i = 0.0$ m. Further increase of $L_i$ reduced the flow rate.

- Increasing the heat flux also resulted in an increase flow rate with similar trends at 250 W/m$^2$ and 500 W/m$^2$. At 500 W/m$^2$, heating the right wall reduced the flow rate up to 10% less than that of heating the left wall.

- Increasing the partition height did not affect the flow rate and temperature field of up to $h_i = 0.05$ m. Afterward, the flow rate and $\Delta T$ decreased down to 83% and 78% of those at $L_i = 0.025$ m.

In practical applications, exhaust air from the outlet of other mechanical ventilation systems can be collected for the forced air flow. Based on the findings in this study and depending on the required ventilation rate of the building, the dimensions of the solar chimney and the port of the forced flow, and the forced air speed can be selected. A damper for controlling the air speed at the outlet of the exhaust air may be required to achieve the optimal forced air speed according to the results in this study.

In future works, performance of the proposed system at different scales can be investigated both numerically and experimentally.
References

[1] Bansal N K, Mathur R and Bhandari M S 1993 Solar chimney for enhanced stack ventilation *Building and Environment* **28** (3) 373–377

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

[3] Khanal R and Lei C 2012 Flow reversal effects on buoyancy induced air flow in a solar chimney *Solar Energy* **86** (9) 2783–2794

[4] Nguyen Y Q and Wells J C 2020 Effects of wall proximity on the airflow in a vertical solar chimney for natural ventilation of dwellings *Journal of Building Physics* **44** (3) 225–250

[5] Tan A Y K and Wong N H 2012 Natural ventilation performance of classroom with solar chimney system *Energy and Buildings* **53** 19–27

[6] 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** 142–153

[7] Liu L, Yu Z and Zhang H 2017 Simulation study of an innovative ventilated facade utilizing indoor exhaust air *Energy Procedia* **121** 126–133

[8] 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