Numerical simulation of a solar chimney for natural ventilation of a building: Comparison of different computational domains

Trieu N Huynh and 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. Numerical simulations, particularly those based on the Computational Fluid Dynamics (CFD) Method, have been widely employed in engineering and design, such as design of green buildings. In this study, we investigated effects of the computational domain, a factor influencing the accuracy of a CFD simulation, of a solar chimney, a device for natural ventilation of buildings, on the predicted performance of a solar chimney attached to a building. Four different domains which, in turn, included both the chimney and the house, the chimney and the inlet length, the chimney with the horizontal inlet, and only the air channel of the chimney, were examined. The CFD model was based on the RANS equations and RNG $k-\epsilon$ turbulence model. The chimney had different heights, gaps, heat fluxes, and the location of the heat source in the air channel. The results show that the differences in the predicted flow rate and temperature rises through the chimney among the domains changed with the height, gap, and the location of the heat source. The simplest domain, which was a simple vertical rectangular channel, overpredicted the flow rate and underpredicted the temperature rise. The main cause of its worse performance is because of its inability to model the separation zones near the inlet of the air channel. Therefore, it is not recommended for high accuracy – simulations. Two other simpler domains, which included the inlet length and only the inlet, can be used when the required accuracy is within 10.0%. The full domain consisting of both the chimney and the building is preferred for the highest accuracy.

Keywords: solar chimney, natural ventilation, CFD, computational domain.

1. Introduction
Numerical simulations have been employed successfully in designing green buildings. They have been utilized for predicting wind load on buildings [1], designing natural ventilation solutions [2], or estimating energy consumption [3]. Among the natural ventilation solutions, solar chimneys have been examined by numerical simulations by several researchers [4–6].

Solar chimney refers to any mechanical system or building element which is based on thermal effects to induce airflow for ventilation or cooling of a building [7]. It absorbs solar radiation on a wall of an enclosed channel, which can be the cavity inside the building façade or a separate pipe or tube attached to a building. The absorbed heat warms the air in the channel and creates thermal effects to induce an air flow through the channel.
With a proper arrangement of the solar chimney, the induced air flow can ventilate or cool the building effectively. For example, it can provide sufficient ventilation rate [6] or thermal comfort conditions [8]. It can also save energy consumption of a building up to 50% [9].

For simulations of solar chimneys, the Computational Fluid Dynamics (CFD) method has been widely employed [4–6,10–13]. This technique is based on numerical approximations of the equations for conservation of mass, momentum, and energy to predict the flow and temperature patterns inside the solar chimneys and the connected buildings.

Among the factors influencing the accuracy of a CFD solution is the computational domain, which is the spatial volume or surface where the governing equations are approximated numerically. In the literature, different types of the domain have been employed for simulations of a solar chimney attached to a building. In some simulations [6,10], the domain consists of both the solar chimney and the building. In other cases [4,11], the domain only covers the solar chimney and air inlets on its side wall. A solar chimney is also simulated with a simple rectangular domain [12,13].

Gan [5] examined effects of the size of a computational domain for simulations of solar chimneys. He compared a simple rectangular domain covering only the air channel of a vertical solar chimney and an extended one which covered both the air channel and the surroundings of the solar chimney. Based on the results, he claimed that the simple domain is only suitable for solar chimneys with high ratios between the channel height and gap where the flow at the outlet of the air channel was not reversed.

As mentioned above, for simulations of a solar chimney, the employed domains are different for different studies where using simpler and smaller domains can save the computational cost. However, we have not seen a comparison among the results obtained with different domains for a solar chimney particularly for that connected to a building. Therefore, we examined effects of different types of the computational domain for simulation of the flow and temperature fields of a solar chimney attached to a building.

2. Method
2.1. The problem

Figure 1 describes a simple vertical solar chimney attached to a building in the vertical plane. The chimney comprises a cover plate placed outside a building wall to form a cavity, or channel, in the middle of the plate and the wall. The lower end of the channel is connected to the room while the upper end is open to the atmosphere. Solar radiation can be absorbed by the cover plate when the plate is opaque, or by the building wall, when the plate is transparent. The absorbed heat is transferred to the air in the channel. The air then rises due to thermal effects. Consequently, an air flow is induced through the room for ventilation.

![Solar chimney attached to a building](image)

**Figure 1.** Solar chimney attached to a building.

In a typical solar chimney, heat exchange happens in the air channel and creates the thermal effect which is the main driving force of the flow. However, the results by Gan [5] shows that the induced flow is affected not only by the boundary conditions in the air channel but also the local dynamic conditions at the inlet and outlet of the air channel. Accordingly, for the problem considered in this
work, as seen in figure 1, the window and the inlet length from the room to the air channel should also affect the induced air flow in the air channel.

To examine effects of different parts of the system in figure 2 in the induced air flow, four different domains were considered, as presented in Figure 2. In figure 2a (domain SC+B), the domain covers both the solar chimney and the building. The domain is further extended beyond the window of the building to allow the air flow to adapt freely to the flow dynamics at the window. Air enters the domain on the open boundary (dashed lines) on the right side, receives heat in the air channel, and escapes through the upper end (outlet) of the air channel. In figure 2b (domain SC+L), the domain includes the air channel and the inlet length. Air enters the domain at the lower opening of the channel and leaves at the outlet. In figure 2c (domain SC+I), the domain is further simplified by excluding the inlet length from the domain in figure 2b. In figure 2d (domain SC), the domain is a simple rectangular channel covering only the channel height above the inlet. For consistent comparison among the domains, it is assumed that the air channel receives solar radiation along with the height above the inlet, i.e. the height h in figure 2a.

2.2. Numerical model

The governing equations for the air flow and heat transfer were predicted by RANS (Reynolds Averaged Navier – Stokes equations) method in CFD. In these equations, time – averaged quantities for velocity, pressure, and temperature are described in equations (1) – (3).

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

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

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

With these equations, it is assumed that the flow and heat transfer are two dimensional and steady, and the air flow is incompressible. As the air in the channel is warmed, its temperature increases from \(T_0\) at the inlet to \(T\) at a point in the channel. Variations of the air properties are described by the Boussinesq approximation in equation (2). Other quantities in equation (1) to (3) are as follows:

- \(u\) and \(u'\) are respectively the time – averaged and fluctuating velocity.
- \(T\) and \(T'\) are respectively the time – averaged and fluctuating temperature.
- \(p\) stands for pressure.
- \(\rho, \nu, \beta\) are the air density, kinematic viscosity, and thermal expansion, respectively.
- \(Pr\) and \(g\) are the Prandtl number and gravitational acceleration, respectively.
In equation (2) and (3), two turbulence quantities $u'_i u'_j$ and $T'u'_j$ were obtained with the RNG $k - \epsilon$ turbulence model, which has been used for simulations of solar chimneys in the literature [5,14]. 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.

Equations (1) to (3) were numerically discretized by the Finite Volume Method on staggered rectangular meshes, as seen in figure 2, by the CFD code ANSYS Fluent (Academic version 2020 R2). The meshes were clustered near the solid surfaces. For all cases in figure 2, 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 1.0. The according mesh resolution was more than 8000 cells in a solar chimney of $H=1.0$ m and $G=0.1$ m, and a heat flux of 1000 $W/m^2$. For all tests in this study, the mesh was carefully checked to satisfy that required maximum $y^+$.

Other settings of the numerical scheme included the SIMPLE method for the coupling between the continuity equation (equation (1)) and the momentum equation (equation (2)), the PRESTO! method for interpolation of the pressure on the staggered mesh, second order upwind scheme for equations (2) and (3), and first order upwind scheme for the equation for $k$ and $\epsilon$.

In figure 2, all open boundaries were applied with the atmospheric pressure. The ambient temperature $T_0$ was assigned at all inlets. All walls were assumed to be no – slip. As seen in figure 1, the heat source can be either on the left or the right wall of the channel. On the heated wall in the air channel, a uniform heat flux $I$ was distributed. The opposite wall in the air channel received radiative heat transfer from the heated wall, which was described by the S2S model in ANSYS Fluent. Other walls were adiabatic. Details of the above numerical setup are similar to those in our previous works [15, 16].

The CFD model was validated against the experiment by Burek and Habeb [17] for a vertical solar chimney whose height was 1.025 m. The channel gap varied from 40 mm to 110 mm. A heat flux of 422 $W/m^2$ was applied on a wall of the air channel. The flow rate through the chimney obtained with the CFD model was compared with the measured one and plotted in figure 3. It is seen that the CFD model slightly over – predicted the flow rate. It may be due to heat loss through the walls of the air channel in the experiment that was not considered in the CFD model. However, the maximum discrepancy was less than 10.0% and should be within the measurement uncertainties.

![Figure 3. Computed (CFD) and measured (Expt.) flow rate through the solar chimney with a heat flux of 422 $W/m^2$ in the experiment by Burek and Habeb [17].](image)

3. Results and Discussions
Comparisons of four computational domains shown in figure 2 are presented in this section in terms of the induced flow rate and temperature rise through the air channel. Changing factors included the chimney $H$, the heat flux applied on the heated wall, and the gap $G$ of the air channel. The location of the heat source was either on the left (LWH) or the right wall (RWH) of the air channel.
3.1. Induced flow rate

The induced flow rate for different heights is presented in figure 4. Three heights of H=1.0 m, 2.0 m, and 3.0 m were tested. The heat flux varied from 200 to 1000 W/m$^2$. The air gap, G, and the inlet height, $h_i$, were fixed to 0.1 m in all tests.

For LWH (figure 4a), it is seen that at H=1.0m, all four domains offered flow rates similar to each other with the maximum difference of less than 5.0%. However, as the height increased further, at H=2.0 m and 3.0 m, the SC domain predicted the highest flow rate. Three other domains (SC+B, SC+I, and SC+L) had similar flow rates. At H= 3.0 m, the maximum difference between flow rates obtained with SC and with SC + B was up to 14.0% and the differences among three domains (SC+B, SC+I, and SC+L) were within 3.0%.

Similar trends were also seen for RWH in figure 4b. However, at H=1.0 m and 2.0 m, the differences in the flow rates among the domains were higher than those for LWH in figure 4a. The highest flow rate was with the SC+I at H=1.0m and with the SC at H=2.0 m. In both cases, the highest flow rate was 10.0% higher than that with the SC+B. At H=3.0 m, the SC also offered the highest flow rate, similar to figure 4a, and up to 13.0% higher than that with the SC+B.

From the observations for both LWH and RWH, it is concluded that the differences among the flow rates obtained with four domains depended on both the channel height and the location of the heat source for H=1.0 m and 2.0 m. However, at H=3.0 m, heating either side had very similar results.

![Figure 4. Induced mass flow rate at different heights for heating: a) Left wall, b) Right wall](image)
domains are seen to be close to each other. The differences among them were within 5.0%. At H=3.0 m, at all gaps, three domains SC+B, SC+L, and SC+I had similar flow rates with the maximum difference of less than 5.0%. The flow rates with SC were higher than those with three other domains. The difference between the flow rates with SC and SC+B increased with the gap. That difference was about 2.0% at G=2.5 cm but was up to 12.0% at G=10 cm. Therefore, the difference was magnified as the gap increased.

Examination of the flow fields for the cases in figure 6 reveals that the separation zones near the inlet were similar to those in figure 5. However, their sizes increased with the gap and led to more flow resistance. Consequently, the difference in the flow rates obtained with the SC and the others were enhanced as the gap increased.

Figure 5. Flow fields for H=3.0 m, G=0.1 m, and the heat flux of 600 W/m² on the left wall (LWH).

Figure 6. Induced mass flow rate at different gaps and heights for heating left (a) and right (b) walls.

3.2. Temperature rise

Figure 7 shows the temperature difference between the inlet and the outlet of the air channel obtained with four domains for both cases of heating (H=1.0 m and 3.0 m, G=0.1 m). Similar trends and values for a given height are seen from figures 7a and 7b. In both cases of heating and both heights, the SC offered the lowest temperature rise while the SC+B offered the highest.

At H=3.0 m, three domains SC+B, SC+L, and SC+I had identical temperature rise which was up to 3.1 K higher than that of the SC. Figure 8 shows the temperature fields and confirms that difference. The temperature field near the outlet of SC is seen to be lower than those of the other cases, which are similar to each other.

The temperature fields near the inlet of the air channel may explain the difference in the temperature rise obtained with SC and with three other domains. For three other domains except for the SC, the separation zones contracted the flow field and enhanced the mixing and the expansion of the thermal
boundary layers near both walls of the air channel. As a result, at the outlet, the temperature field is more uniform and the temperature rise was higher for three domains SC+B, SC+L, and SC+I.

Figure 7 shows that less difference in the temperature rise is seen among the domains for RWH. For example, at H=3.0 m, the difference between the SC and SC+B was 2.3 K for RWH but 3.1 K for LWH. It may be because when the right wall was heated, part of it was embedded in the separation zone (i) (figure 5), which should exchange heat worse than a straight wall did. Therefore, the effects of the separation zone (i) on enhancing the mixing of the temperature field should be less than those when the left wall was heated.

![Figure 7](image1.png)

**Figure 7.** Temperature rise at different heights for heating left (a) and right (b) walls.

![Figure 8](image2.png)

**Figure 8.** Temperature fields for H=3.0 m, G=0.1 m, and the heat flux of 600 W/m² on the left wall (LWH).

4. **Conclusions**

The results show that the differences in the induced flow rate and temperature rise through the solar chimney attached to a building simulated with four different domains changed with the height, gap, and location of the heat source, as follows:

- At H=1.0 m, all domains had similar flow rates with differences of within 5.0% for LWH but up to 10.0% for RWH.
- At H=2.0 m and 3.0 m, the SC always offered the highest flowrate while three other domains had similar flow rates of within 5.0% difference. The flow rate obtained with SC was up to 14.0% higher than those with the SC+B. LWH and RWH had similar performance.

The existence of the separation zones near the inlet in three domains SC+B, SC+L, and SC+I, which are similar to each other, may increase the flow resistance in those three domains and reduce their flow rate compared to that with the SC.

The SC always offered the lowest temperature rise while the SC+B offered the highest. The difference was up to 3.1 K at H=3.0 m for LWH and 2.3 K for RWH. Besides SC, three other domains
had similar temperature rises with differences of within less than 1.0 K. The separation zones in three domains SC+B, SC+L, and SC+I may enhance the mixing of the temperature field and result in higher temperature rise in those three domains.

From the results, it is concluded that the SC domain should not be employed for this problem as it over-predicted the flow rate and under-predicted the temperature rise. The other two domains, i.e. SC+L and SC+I, may be used when the required accuracy is within 10.0%. For eliminating the effects of the domain on the results, the full domain, i.e. SC+B, should always be used.

The results from this study can serve as a reference for architects and building engineers using CFD for building design. Depending on the required accuracy of the solutions, a particular type of the computational domain can be referred. For example, for the first stage of the design, when testing different scenarios of a solar chimney, they may employ the domain SC for a quick comparison among the design scenarios. At later stages, such as for calculating energy and thermal comfort performance of the building, the full domain SC+B can be used for more accurate and detailed results.

References
[1] Thordal M S, Bennetsen J Chr and Koss H H H 2019 Review for practical application of CFD for the determination of wind load on high-rise buildings Journal of Wind Engineering and Industrial Aerodynamics 186 155–168
[2] van Hooff T, Blocken B and Tominaga Y 2017 On the accuracy of CFD simulations of cross-ventilation flows for a generic isolated building: Comparison of RANS, LES and experiments Building and Environment 114 148–165
[3] Al Touma A and Ouahrani D 2018 Performance assessment of evaporatively-cooled window driven by solar chimney in hot and humid climates Solar Energy 169 187–195
[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] Gan G 2010 Impact of computational domain on the prediction of buoyancy-driven ventilation cooling Building and Environment 45 (5) 1173–1183
[6] Shi L, Ziem A, Zhang G, Li J and Setunge S 2020 Solar chimney for a real building considering both energy-saving and fire safety – a case study Energy and Buildings 221 110016
[7] Bansal N K, Mathur R and Bhandari M S 1993 Solar chimney for enhanced stack ventilation Building and Environment 28 (3) 373–377
[8] Tan A Y K and Wong N H 2012 Natural ventilation performance of classroom with solar chimney system Energy and Buildings 53 19–27
[9] Miyazaki T, Akisawa A and Kashiwagi T 2006 The effects of solar chimneys on thermal load mitigation of office buildings under the Japanese climate Renewable Energy 31 (7) 987–1010
[10] Bassiouney R and Korah N S A 2009 Effect of solar chimney inclination angle on space flow pattern and ventilation rate Energy and Buildings 41 (2) 190–196
[11] Khanal R and Lei C 2012 Flow reversal effects on buoyancy induced air flow in a solar chimney Solar Energy 86 (9) 2783–2794
[12] Zavala-Guillén I, Xamán J, Hernández-Pérez I, Hernández-Lopéz I, Gijón-Rivera M and Chávez Y 2018 Numerical study of the optimum width of 2a diurnal double air-channel solar chimney Energy 147 403–417
[13] Zhang T and Yang H 2019 Flow and heat transfer characteristics of natural convection in vertical air channels of double-skin solar façades Applied Energy 242 107–120
[14] Zhang T, Zhao Y and Wang S 2014 Prediction of airflow rate through a ventilated wall module Energy and Buildings 82 651–659
[15] 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
[16] 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** 101050

[17] Burek S A M and Habeb A 2007 Air flow and thermal efficiency characteristics in solar chimneys and Trombe Walls *Energy and Buildings* **39** (2) 128–135.