Numerical study for the ventilation with solar chimney under effect of different location and the shape of the section opening window

Ahamed Q. Mohamed¹, Ahmed K Alshara², Hussein M. Mitalaik¹
¹Middle Technical University, Baghdad Technical College, ²University of Maisan, Engineering college

Abstract: Heat transfer process and fluid flow in a solar chimney used for natural ventilation are investigated numerically in the present work. Solar chimney was tested by selecting different positions and shape of window namely: at the up, middle, and bottom side, as well as for horizontal and vertical and square shape windows. CFD analysis based on finite volume method is used to predict the thermal performance, and air flow in three dimensional solar chimney under steady state condition, to identify the effect of different parameters such as solar radiation. Results show that a solar chimney with horizontal at the bottom of the Room better ventilation performance.

1. Introduction
Building components like walls, roof fenestrations and floors can be built and designed to get more benefit of solar radiation in the form of heat in the winter and reject solar heat in the summer to increment the requirements of building ventilation. This is called passive solar design or climatic design. Passive solar technique is one of the most effective methods used for natural ventilation. It can greatly increase the energy efficiency of a building and can supply 100% of a home ventilation in many cases. Passive solar houses include a wide variety from those ventilation almost entirely by the sun to those with East facing windows when α= 145° (α) sun angle at evening that provide a fraction of the heating load. One of the most common designs of the solar passive techniques which are incorporating best ventilation and delivery system called a natural ventilation by solar chimney. The solar chimney is a smart device for collecting heat from the sun during the daytime and induces the hot air into a building space. [1]. Many works have studied the performance of the solar chimney at many locations and applications including power generation, natural ventilation of buildings etc. Khedari et al. (2000) [2] have achieved experiments on an installed school building structure using two roof solar chimneys, a modified Trombe wall and a metallic solar wall to determine their combined effects on thermal comfort. The building volumes of 25 m³and 27 different positions were measured to obtain the interior air temperatures and speeds. Results showed that with a combined surface area of 6 to 9m², the temperature difference between the interior and ambient was 3°C this vale was lower than the case without any ventilation by about 6°C. The average air change per hour (ACH) by 15ACH while the air speed within the room ranged between 0.02 to 0.09 m/s. Although results did not satisfy thermal comfort, it gave the possibility of combining various solar chimney designs to achieve better performance. Bunnag.et.al.(2004)[3].Investigated the influences of the tilt angles chimney (15°, 30°, 45°, 60° and 75°), air gaps (0.14m and 0.19m) and heat
flux $(262 \text{W/m}^2, 408 \text{W/m}^2 \text{and} 574 \text{W/m}^2)$ on the convective heat transfer coefficient of a roof solar collector measuring $1.00 \text{m}$ wide by $1.50 \text{m}$ long under steady state condition. Results showed that the temperature difference between the heated and unheated surfaces remained almost constant with different values of heat fluxes. Furthermore, increasing the heat fluxes or decreasing the tilt angles increased the air temperature. These results were further correlated and none dimensionalized as relationships between Nusselt number, Rayleigh number and Reynolds number. Bansal et al. (1989) [4] using a wind tower coupled with a solar chimney a mathematical model was used to calculate the system performance. The equations of the energy balance for the air flow through the chimney were used to predict the unit performance. It was found that for the low wind speed coupled with the maximum solar radiation increased the air mass flow rate up to 50%. The proposed models show that the generated mass flow rate was doubled as compared with that for the traditional wind chimney. Haghighi & Maerefat (2014) [5] studied numerically natural ventilation of a room which uses solar chimney as a heating source to determine the effects of sizes of air gap, openings and environmental ambient condition on air change per hour (ACH) and indoor air temperature. As a result, it was found that the maximum ACH can be achieved when the air gap size was 0.2m. Therefore; it was concluded that the use of solar chimney with outlet size of less than 0.3m can provide thermal comfort and avoid undesirable vertical temperature gradient. Harris and Helwig (2007) [6] have introduced a CFD modeling for the design of the solar chimney for building induced ventilation, the effect of the chimney tilt angle, number of glazing cover, and the emissivity on the chimney performance were investigated. The results showed that inclined chimney improved the ventilation by 11% as compared with that for the vertical chimney. The effect of absorber height to the air gap of the chimney that used for building ventilation was studied by Mathur et.al (2006) [7], the study showed that maximum air flow rate through the $27 \text{m}^3$ room was 5.6 air change per hour when the incident solar radiation was 700 W/m².

Zamora and Kaiser 2009, [8] studied, through numerical investigation, the laminar and turbulent flows induced by natural convection in channels, with side wall solar chimney configuration, for a wide range of Rayleigh number and several values of the relative wall-to-wall spacing. The open from the room to the chimney gap is located at the bottom. They recommended an optimum gap-to-height ratio in correlation format, as function of Ra number ranging from 105 to 1012. Tan and Wong 2013 [9], through numerical simulation, studied the effect of many parameters on the flow and temperature on a solar chimney connected to the side of a room. Among the investigated parameter is the location of the side inlet from the room to the chimney. There simulation results are different compared with those obtained by Tan and Wong 2012 [10] performed experimentally on a classroom. Lowering the inlet position to the middle position leads to significant increase in the output air velocity. The inlet position is mainly affecting the air speed in the localized region around the solar chimney inlet. In the present work, it is found that the shape of the inlet also affects the air velocity, and the chimney performance, as well. However, solar chimney geometries, inlet size and shape, construction materials and the surrounding weather conditions are all inter-dependent. Optimization of individual parameter at certain condition may not be optimum if combined with others. Since the solar is changing on a daily base, experimental investigations are not practical if they are considered as unique.

2. Mathematical Model

Figure 1 shows the mathematical model for the chimney showing the natural buoyancy-driven fluid flow and heat transfer, several assumptions were used to solve the present flow and heat transfer cases which is listed as the following:

- Steady-state conditions.
- Turbulent flow.
- Three dimensional.
- Boussinesq approximation is utilized.
• Incompressible flow.
• No-slip conditions between the fluid and wall.

![Figure 1. Configuration of the solar chimney under study](image)

2.1 The Governing Equations
The flow of air in the solar chimney is caused by natural convection due to solar radiation. The governing equations used in the simulation consist of the Reynolds averaged Navier – Stokes (RANS) equations along with continuity, energy, turbulence, and radiation transfer equation [8]. The increases in mass within a control volume must be equal to the mass inflow minus the mass out flow through the control volumes surface. This may be expressed mathematically for an incompressible flow the continuity equation for:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$  \hspace{1cm} (1)

The other fundamental set of equations that govern the flow of a fluid are derived from Newton’s second law (the conservation of momentum). The equations are called the Navier-Stokes equations, and for incompressible fluid flow are:

$$\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho vu)}{\partial y} + \frac{\partial (\rho wu)}{\partial z} = -\frac{\partial p}{\partial x} + \mu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right]$$  \hspace{1cm} (2)

$$\frac{\partial (\rho v)}{\partial x} + \frac{\partial (\rho vu)}{\partial y} + \frac{\partial (\rho wv)}{\partial z} = -\frac{\partial p}{\partial y} + \mu \left[ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] + S_{bj}$$  \hspace{1cm} (3)

$$\frac{\partial (\rho w)}{\partial x} + \frac{\partial (\rho vw)}{\partial y} + \frac{\partial (\rho ww)}{\partial z} = -\frac{\partial p}{\partial z} + \mu \left[ \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right]$$  \hspace{1cm} (4)

Where \(u\), \(v\) and \(w\) are stream-wise, lateral and vertical velocity component respectively. \(x\), \(y\) and \(z\) are the corresponding directions.

\(S_{bj}\) is the buoyancy source or sink term.

The following energy equation represents the transport of heat within the flow field.

$$\rho \frac{\partial}{\partial x} (uT) + \rho \frac{\partial}{\partial y} (vT) + \rho \frac{\partial}{\partial z} (wT) = \frac{\partial}{\partial x} (\Gamma_{eff.h} \frac{\partial T}{\partial x}) + \frac{\partial}{\partial y} (\Gamma_{eff.h} \frac{\partial T}{\partial y}) + \frac{\partial}{\partial z} (\Gamma_{eff.h} \frac{\partial T}{\partial z}) + S_t$$  \hspace{1cm} (5)

Where

\(\Gamma_{eff.h}\) : Is effective diffusion coefficient.

\(S_t\) : Is a source term is computed from the ray tracing algorithm

2.2. Mesh Generation
In the present CFD Simulation, the unstructured mesh of the arithmetic field is used by controlling a finite-scale scheme where the arithmetic field is divided into a number of control units. The model was meshed by ANSYS Fluent software. Various meshes were formed for the 3-D model of the solar chimney.
for different models. The mesh dependency was tested for one model, that two parameters taken into account for mesh accuracy were room temperature and ACH of the system. Thus the optimum mesh selected for the models ranged about (3,400,000 to 4,400,000) as listed with details in the standard model from the Table 1 and viewed in Figure 2.

![Figure 2. Mesh generation](image)

**Table 1.** the mesh required for the standard model

| n  | Domain   | Nod  | Elements         |
|----|----------|------|-----------------|
| 1  | absorber | 4400 | 2106            |
| 2  | domain   | 62589 | 3258895        |
| 3  | glass    | 6552 | 3175            |
| 4  | insulation | 19760 | 14118       |
| 5  | wood     | 47224 | 152174         |
| 6  | All Domains | 703,825 | 3,430,801     |

**2.3 Boundary Conditions**

The boundary condition description thermal changing on the boundary of the numerical model. Therefore, this is an important component of Ansys Fluent simulations. A symmetry boundary condition is done which the problem is symmetric as the heat and the air flow at any side of the plane is the same image of the flow on the opposite side the ambient conditions at the outermost boundaries were represented using temperature and inlet velocity values from field measurements and assuming a pressure of 1 atm. Further, the acceleration of gravity was considered; its value was set equal to 9.81 m/s². A solar load model was used to model the direction of solar beam entering the computational domain. The incident solar radiation solar and a solar location angle is selective at listed in the Table. 4. A pressure inlet with atmosphere pressure boundary condition was prescribed for the room inlet. The incoming air was assumed to be at the measured ambient temperature. At the chimney exit, a pressure outlet boundary condition was prescribed in which the fluid pressure was assumed to be equal to the ambient pressure. The Boundary conditions of one of the models is describe as shown in Figure 3. The computational domain, in Table 2 shows the material properties.
Table 2. The material properties

| n  | Materials         | Thickness (mm) | ρ (kg/m³) | Cp (kJ/kg·K) | K (W/m·K) | α  | ε  | τ  |
|----|-------------------|----------------|-----------|--------------|-----------|----|----|----|
| 1  | Glass [11]        | 4              | 2220      | 0.83         | 1.15      | 0.06 | 0.95 | 0.84 |
| 2  | Absorber plate [12]| 1              | 2700      | 0.9          | 237       | 0.95 | 0.95 | -   |
| 3  | Wood [13]         | 8              | 400       | 1.8          | 0.06      | 0.5  | 0.5 | -   |
| 4  | Insulation[14]    | 50             | 52        | 0.657        | 0.038     | 0.4  | 0.4 | --  |

Table 3. The summery of the boundary conditions

| Boundary Condition                          | Details |
|---------------------------------------------|---------|
| The inlet boundary condition                | Tin = Tamb. = 30°C, velocity inlet as pressure inlet P = Zero gauge pressure |
| The outlet boundary condition               | Pressure outlet boundary at atmospheric pressure |
| Symmetry boundary condition                 | Specify the location of the plane |
| Solar intensity (I)                         | Ambient flux of 300, 500 and 700 W/m² |
| Solar radiation angle                       | To the outlet domain of 55°, 90° and 145° |

2.4 Air Change per Hour (ACH)

One of the important factors for the ventilation performance is Air Change per Hour (ACH) which refer to the amount of air ventilated and replaced by fresh air. The hourly air change rate can be calculated using the following equation:

\[ \text{ACH} = \frac{3600 Q_{vent}}{V_r} \]  

Where:

- \( V_r \) is the volume of a room taken as \((1 \times 1 \times 1) \text{m}\)
- \( Q_{vent} = \rho \text{avg} \times A \)

Where:

- \( \rho \text{avg} \) is the average velocity at the exit cross section of the chimney \((\text{m/s})\)
- \( A \) is the inlet cross section area of the chimney \((0.15 \times 0.4) \text{m}^2\)
- \( Q_{vent} \) is the flow rate of the hot air \((\text{m}^3/\text{s})\).

The weather data such as (ambient temperature, solar radiation and wind speed) was adopted for Maysan city located at 44.34° longitude East and 32° latitude North.

3. The Numerical Results

This section studies several parameters of the solar room ventilation to identify the impacts of each parameter on enhancing ventilation rate and room temperature that will be discussed in detail. All the models cases subjected to the ambient domain pressure is atmospheric pressure and the temperature of air is 27 °C. Two rooms models are studied in order to find the enhance the ventilation rate in each one. One model is the absorber inside the room and the other outside. The following disused this concept.
3.1 Effect of the window position and Geometry of the Model

The room walls and roof for the first model are made for wood. The solar angle and ambient temperature are fixed to $\alpha = 55^\circ$ and $27^\circ C$ respectively. For the effect of windows position (P) and the solar intensity in flow field for the first model are illustrated in Figure 5 and 6, that show the velocity contours with streamline in symmetry plane at three different window position cases ((u) upper position, (m) middle position and (d) down position) for the square and horizontal rectangular window. The velocity contours show a red region (indicate highest velocity magnitude in the legend to the lower magnitude (blue
region). Generally, the highest velocity magnitudes occur at the inlet opening of the window. The magnitude of velocity is higher at high solar intensity that indicates the solar intensity plays an importance factor for air flow. For squ. Window, there is no large difference in velocity magnitude for both low and high solar intensity, but the difference is clear as compared in h.req. Window. All the inlet velocity magnitudes for the h.req. Window is higher than squ. Even low intensity that expected to give a high ventilation rate. For h.req. window at high solar intensity $I=700 \text{ W/m}^2$, the velocity is high in cases u and d while less in case m, that the red and bright blue colors zone is widely in these cases. In both figure at window position m, a very low velocity contours between the absorber and the class as compared with other position.

![Figure 5. Velocity with streamline contours for different square window positions (a) P=u, (b) P=m and (c) P=d at two solar intensity $I=300$ and 700 w/m²](image)

Moreover, a weaken streamlines at the upper room zone of the room and a more circulation occurred in case m at the room and to the chimney that indicate that the airflow patterns is weak since the reverse flow at the chimney decrease the air sucking from the chimney. Hence, one can anticipate the flow pattern is the best in case d for h.req window than others and this explained that the ACH in is dominant in this case. In order to describe thermal field of these cases, the temperatures contours are plotted in the Figures 7 and 8 for both suq. And h.req windows respectively. Generally, the room temperature is low in low solar intensity although the less ventilation occurred that expected the solar radiation and heat transfer through the wood have a major effect for increase the room temperature at high intensity. The thermal plume near the absorber tend to move out the chimney for high intensity while less for low intensity due to the high absorber temperature lead to encourage the Buoyancy force. The high temperature recoded at the upper zone for the room for case d and more for case m because the roof facing the solar intensity that has a major influence for transfer heat to the room which the hot air accumulates at the upper side. Its look that thermal stratification occurred at case m for h.req window in high intensity. The warm air is enclosing at
the upper zone since warm air arises and no venting at the upper zone. The case u gives lowest temperature since the opening located near the ceiling. The temperature of the wood is high at high flux solar intensity that effect on the heat transfer to the room. In addition to that the solar radiation has a major effect since the solar radiation transmit through the glass to the floor, then reflect the room walls and ceiling as show in the Figure 9. The projection radiation increases as intensity increases.

**Figure 6.** Velocity with streamline contours for different horizontal window positions (a) P=u, (b) P=m and (c) P=d at two solar intensity I=300 and 700 w/m²
Figure 7. Temperature contours for different square window positions (a) P=u, (b) P=m and (c) P=d at two solar intensity I=300 and 700 w/m².

Figure 8. Temperature contour for different horizontal rectangular window positions (a) P=u, (b) P=m and (c) P=d at two solar intensity I=300 and 700 w/m²
Figure 9. The irradiation contours absorbed solar heat flux on room for difference solar intensity on absorber.

For investigation show the temperature at horizontal plane inside the room for different window geometry, Figure 10 illustrates the temperature contour with streamlines in lower horizontal planes at three windows geometries. The cold air zone is low at the v. req. since narrow pithing of cold air in the lower zone of room. Generally, there is a large circulation in hot region at the room corner which indicate a poor ventilation in this region. The circulation intensity is a bit lower at the h. req. that expected a ventilation is slightly high than other. Figures 11 and 12 illustrate the ACH with different solar intensity at three window position and window types. Generally, all the cases increase with the solar intensity. From all the window types, the down position shows high ventilation rate which reach to the maximum at window type h.req. This conclude window geometry and window position have significant effect of enhancing the ventilation rate. As can be seen the ventilation rate is mainly depending on position on ventilation which is required for proper ventilation. For knowledge the best thermal condition of the room the average air room temperatures for lower and higher solar flux are summarized in the Figures 13 and (14) respectively. The middle position shows the worst thermal condition for all windows types. The lower room air temperature achieved by square. Window placed at the upper position.

Figure 10. The temperature with streamlines for horizontal plane at three windows types
Figure 11. The variation ventilation rate ACH with heat flux at two different window geometry (v.req. and squ. Window type).

Figure 12. The variation ventilation rate ACH with heat flux at window geometry (h.req. window type).

Figure 13. Average room temperature for different window types at the solar intensity 300 w/m²
To investigate the effect of solar radiation angle $\alpha = 145^\circ$ on ventilation rate, a window shading is additive to the room model $\beta = 60^\circ$ and the temperature counters clarify in Figure 15. As seen the radiation effect is removed from the floor as compared to the Figure 15b and the average temperature and ACH are slightly decrease to $31^\circ$ C and 6 respectively.

Figure 15. The effect of window shading at the chimney $\beta = 60^\circ$ (a) the irradiations counters and (b) Temperature contour

4. Conclusion
The use of the solar chimney to induce natural ventilation was studied numerically. The following conclusions were extracted
- The orientation of the suction opening entrances affects the performance of the chimney. Solar chimney at any inclination angle on the horizontal bottom suction opening produces the best thermal performances in Maysan.
- There is remarkable value in the ACH which is 165 at solar radiation $700 \text{ W/m}^2$ at the horizontal bottom inlet.
- The affects shape Window of the Room only slightly its thermal performance when average weather prevails, and it is designed and used in an “optimum” way. Rectangular but compact design with the
longer axis pointing East and West is preferable because it is less sensitive to changes in weather and in design decisions than a rectangular but longer shape.

Appendix- Nomenclature

| Symbol | Description |
|--------|-------------|
| A      | surface area (m$^2$) |
| CP     | Specific heat (J/kg. K) |
| g      | Gravitational acceleration(m/s$^2$) |
| h      | heat transfer coefficient (W/m$^2$.K) |
| I      | solar radiation (W/m$^2$) |
| kt     | thermal conductivity (W/m.K) |
| w      | air gap (m) |
| P      | pressure (Pa) |
| µ      | dynamic viscosity (N.s/m$^2$) |
| Q      | Air discharge (m$^3$/s) |
| t      | time (s) |
| T      | Temperature (K) |
| vent   | Air discharge (m$^3$/s) |
| u, v   | Velocity components in the x, y direction (m/s) |
| ACH    | Air change per hour |
| µt     | turbulent viscosity (N.s/m$^3$) |
| µ eff  | effective kinematics viscosity (N.s/m2) |
| ρ      | Air density (kg/m3). |
| k      | turbulent kinetic energy (m$^2$/s$^2$) |
| L      | chimney’s height (m) |
| x, y   | Cartesian coordinate (m) |
| d      | down |

Greek Symbols

| Symbol | Description |
|--------|-------------|
| α      | absorptivity |
| ε      | emissivity |
| ε      | rate of dissipation of kinetic energy (m$^2$/s$^2$) |
| τ      | transitivity |
| α      | absorptivity |

References

[1] Al Mossowi A.N., 2011 Turbulent developing flow and heat transfer in a porous square duct, MSc. Thesis, Mechanical and Construction Department, University of Technology, Baghdad, Iraq, (written in Arabic)
[2] Khedari, J., Boonsri, B. and Hirunlabh, J., 2000. Ventilation impact of a solar chimney on indoor temperature fluctuation and air change in a school building. Energy and buildings, 32(1), pp.89-93.
[3] Bunnag, T., Khedari, J., Hirunlabh, J. and Zeghmati, B., 2004. Experimental investigation of free convection in an open-ended inclined rectangular channel heated from the top. International journal of ambient energy, 25(3), pp.151-162.
[4] Bansal, N.K., Mathur, R. and Bhandari, M.S., 1993. Solar chimney for enhanced stack ventilation. Building and environment, 28(3), pp.373-377
[5] Haghighi, A.P. and Maerefat, M., 2014. Solar ventilation and heating of buildings in sunny winter days using solar chimney. Sustainable Cities and Society, 10, pp.72-79
[6] Harris D.J., Helwig N., 2007 Solar chimney and building ventilation, Applied Energy 84, pp. 135–146
[7] 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
[8] 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), pp.762-769.

[9] Tan, A.Y.K. and Wong, N.H., 2013. Parameterization studies of solar chimneys in the tropics. *Energies*, 6(1), pp.145-163.

[10] Tan, A.Y.K. and Wong, N.H., 2012. Natural ventilation performance of classroom with solar chimney system. *Energy and Buildings*, 53, pp.19-27.

[11] Park, D., 2016. *The application of the solar chimney for ventilating buildings* (Doctoral dissertation, Virginia Tech).

[12] Baxevanou, C. and Fidaros, D., 2017. Numerical Study of Solar Chimney Operation in a two-story Building. *Procedia Environmental Sciences*, 38, pp.68-76.

[13] H. Keshka, 2006 Thermal Insulation Coefficient *Arab Uniform Code for Building Design and Implementation*.

[14] Martinez, G.E.R.A.R.D., 1980. Optical properties of solids. In *Handbook on semiconductors* (p. 181). North-Holland Amsterdam.