ICON Journal of Engineering Applications of Artificial Intelligence
Volume 01 | Issue 01 | 2025
e-ISSN: 3051-3820
p-ISSN: 3051-3812
Journal homepage: https://iconpublishers.com/icon-j-eng-app-ai/

DOI: 10.5281/zenodo.15857765

Research Article

Numerical Simulation of the effect of Trombe wall material on the ventilation of green building

Iman Shemshadi *
M.Sc., Department of Mechanical Engineering, Payame Noor University, Tehran, Iran

Corresponding author: Iman Shemshadi
M.Sc., Department of Mechanical Engineering, Payame Noor University, Tehran, Iran

Received Date: 05 June 2025
Published Date: 11 July 2025

Abstract

Trombe wall or solar wall is classified as one of the passive solar heating methods. In this paper, the air flow inside the thrombus wall chamber is simulated using the finite volume method, which is one of the traditional computational fluid dynamics methods. Ansys Fluent commercial software is used in the simulations. At first, a preliminary reference model and its results were presented, and then, using this reference model, parameters affecting the efficiency of Trombe wall material were studied and investigated. Finally, a comparison was made between the effect of changing the wall material on the amount of thermal storage.

Keywords: Trombe wall, solar heating, computational fluid dynamics.

INTRODUCTION

Trombe wall is designed as a passive heating element in building architecture to store and absorb solar energy. Passive solar heating refers to air heating without the use of moving and energy-consuming tools. Heating by Trombe wall is based on the absorption of solar radiation by a wall with a high heat capacity and absorption coefficient. This wall is located behind a transparent glass wall on the south side of the building (the side of the sun in the winter in the northern hemisphere).

The heat absorbed by the thrombus wall is trapped in the space between the wall and the glass wall and is transferred to the indoor air through the heat transfer phenomena of thermal conduction, displacement and radiation. The presence of two vents at the bottom and top of the vertical section of the wall creates air flow between these two sections.

In reference [15], natural smooth displacement in composite solar wall has been studied. In this article, Simpler algorithm is used in CFD solution and it is shown that the ratio of width to height does not have much effect on the efficiency of the system. In reference [16], the laminar flow equations have been solved by the finite volume method7. In this reference, the heat conduction equations in the wall have also been solved. Reference [8] has investigated the effect of buoyancy force on natural displacement in a quadrangular reservoir using fluid simulation in Fluent software. In reference [3], in addition to natural displacement modeling, the radiation equations have also been solved and it has been shown that considering the radiation causes a decrease in temperature gradients and an increase in the average Nusselt number in the domain. It has been shown in reference [19] that the most important factor affecting heat absorption is the air gap between the thrombus wall and the glass wall. Reference [17] has investigated the effect of the location of inlet and outlet valves in the solar wall by using computational fluid dynamics solution through the finite volume method. Reference [1] has simulated a solar chimney in hot and humid regions by using Fluent software and investigated the effect of air temperature on the efficiency of the wall. Analytical methods are part of very important methods in studying the study of thrombus wall. In references [5] and [4], an analytical method to solve the slow and unsteady flow for natural displacement heat transfer has been presented. These methods are based on the use of similar variables in solving the governing differential equations.

Numerical solution methods are among the other methods that have been discussed in different articles. In reference [11], one-dimensional modeling of the solar chimney and Trombe wall using the network of thermal resistances has been presented. In reference [12] also using a one-dimensional finite differential method and mass and energy balance in the control plane, the effect of different coatings on energy storage and absorption in the thrombus wall has been investigated. In reference [10], the traditional and composite solar wall has been solved using a one-dimensional finite differential method and using TRNSYS software. Comparing the results of TRNSYS software and the one-dimensional finite differential method confirms the high accuracy of the solution method.

In reference [13], laboratory tests were performed on the thrombus wall in two low-consumption buildings and the results were checked by recording heat flux data by temperature sensors and infrared photography. In reference [6], experimental-numerical comparisons have been made between Trombe walls with a south wall of single-walled glass, double-walled glass and solar battery. Reference [18] has been made to the laboratory study of the effect of the gradual cooling of the air during the day on the indoor temperature and the results obtained have been compared with the results of the thermal resistance network solution. Considering their audience, review articles help a lot to understand the issue. In reference [14], based on the classification of the methods, it has been given a summary of different solution methods. Reference [7] has examined the types of new active and passive solar ventilation methods in the last ten years. Reference [9] also introduced and explained the types of passive solar methods, including solar chimney, solar wall, solar windows, and solar roof, and finally, in reference [2], the European standard for double walls The wall has been prepared and presented in it Standard design strategies of passive solar ventilation equipment have been developed for practical uses. In this article, the computational fluid dynamics method has been used to simulate the thrombus wall and examine different parameters affecting the system efficiency. The aim of this paper is to simulate the air flow and heat transfer in the Trombe wall container (the air gap between the wall and the glass wall) by solving the Neuer-Stokes equations. Lunet is done, in addition to Eqs Neuer-Stokes, fluid turbulent flow equations and radiation equations are also solved.

Trombe wall

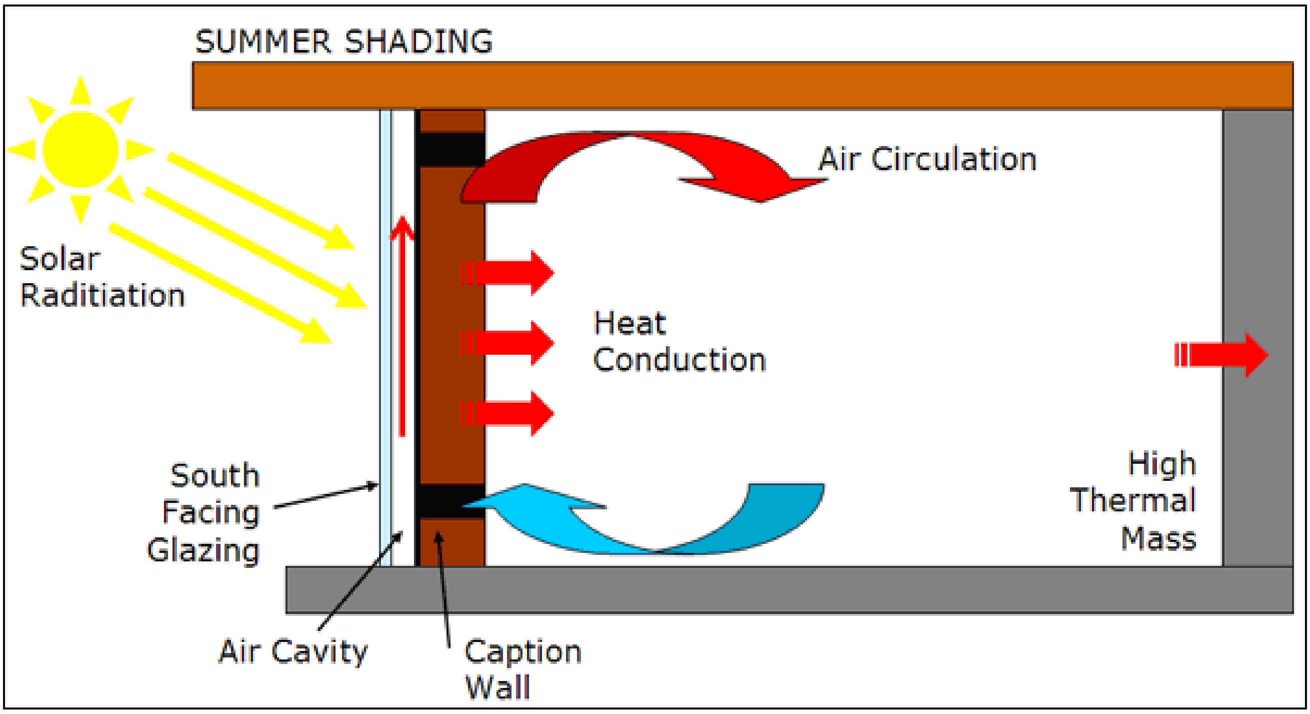


Figure 1: An overview of the Trombe wall and how it works

The Trombe wall is built in the direction of the sun (south in the northern hemisphere) and has a high heat capacity that absorbs and stores energy. Dark materials with high radiation absorption coefficient are usually used to increase heat absorption. Also, in many of the designs, the glasses that are glazed and polished from the inside are used. This causes a decrease in the reflection of energy from the air gap to outer space. After absorbing the heat, the solar wall starts to reflect the heat. The advantage of the wall is the reflection in the infrared light range, which causes the proper and desirable distribution of heat in the room. Another advantage of the Trombe wall is its high heat capacity and ability to store heat during the day and release it. The temperature gradually increases at night. The aim of the simulation is to solve the domain of fluid flow and heat transfer in the thrombus wall. Heat transfer in the thrombus wall takes place in three ways: 1) natural displacement due to buoyancy forces, 2) infrared reflection on the walls and 3) heat conduction in the wall.

Computational fluid dynamics

Computational fluid dynamics (CFD) is the analysis of systems including fluid flow, heat transfer and related phenomena, such as chemical reactions, based on computer simulation. CFD is a very powerful method, so that it includes a wide range of industrial and non-industrial applications. The advantages of CFD compared to experimental methods are as follows:

- Basic reduction in the time and price of new designs: Of course, the cost of performing CFD calculations is lower than the cost of performing an experiment and building the corresponding laboratory, on the other hand, the speed of performing the calculation is also faster than the speed of performing the experiment.
- The ability to study and simulate issues that are difficult or impossible to conduct experiments on (such as large systems or very high temperatures, etc.)
- The ability to simulate and study problems in special and critical conditions: that is, by using CFD analysis, conditions of the device that are impossible to reach in laboratory conditions can also be studied.
- Obtaining complete information and very precise details of the solution: in the CFD problem, all the desired quantities such as speed, temperature, density, energy, etc. can be calculated at any point of the flow, but in the experiment only in a few The point is that limited information can be obtained.
- Experimental fluid mechanics can provide the required information for a specific flow field. In any case, due to equipment limitations such as the size of the test samples and the size of the wind tunnel, as well as the problems caused by the lack of complete similarity with the real flow field, it is impractical to obtain laboratory information in most of the flow fields.

ANSYS FLUENT

FLUENT software is the pinnacle of programming for modeling fluid flow and heat transfer in complex geometries. This software provides the possibility to change the network completely and analyze the flow with unstructured networks for complex geometries. The types of meshes that can be generated and received by that software group include meshes with triangular and quadrilateral elements (for 2D geometries) and tetrahedral, hexahedral, pyramidal or wedge elements (for 3D geometries). Also, FLUENT allows the user to improve the network (for example, to make the network smaller or larger in the boundaries and necessary places in the geometry).

Governing Equations

Reynolds Averaged Navier Stokes (RANS) equations, derived from the general steady-state form of the governing equations for the conservation of mass:

$$\nabla \cdot (\rho \vec{v}) = 0$$

and the conservation of momentum

$$\nabla \cdot (\rho \, \vec{v} \, \vec{v}) = - \nabla P + \nabla \cdot (\bar{\tau}) + \rho g$$

where ρ is density, v is velocity, P is pressure, τ is the stress tensor, and g is the gravitational acceleration, were solved using a finite volume based commercial CFD software package. In RANS simulants, The Time-Avenged Momentum Transport Equations are Closed by Modeling the Momentum flux Terms (Reynolds Stresses) Using a Turbulence Model.

Methodology

The governing equations for solving the problem include continuity equations, momentum equation, energy equation, turbulent flow equations of the ε-k RNG model and radiation equation of the P1 model. The simulations are reliable. The reason for choosing reliable and time-independent calculations in the above calculations is that the time of changes in the temperature of the outside air (environment) and the amount of received radiation are higher than the changes in the temperature of the inside air, meaning that the time format of the changes in the temperature of the outside air and Shasha was received within an hour While the range of indoor air temperature changes is within a few minutes. On the other hand, in this article, the aim is to investigate the effect of different parameters on the efficiency of the thrombus wall. Therefore, many parameters (especially the amount of radiation) should be simulated in a controlled manner. Examining the effects of variable temperature and radiation and its effect during the day is an independent discussion that should be addressed in other articles.

Table 1 shows three types of Trompe wall models along with the material specifications and amount of heat flux absorbed by the Trompe wall.

Table (1)

	Trompe wall materials	heat flux w/m2
Model 1	(concrete with clay) and glass	500
Model2	(concrete or clay) and copper sheet and glass	700
Model3	Paraffin wax material, copper sheet and porous glass	900

Result:

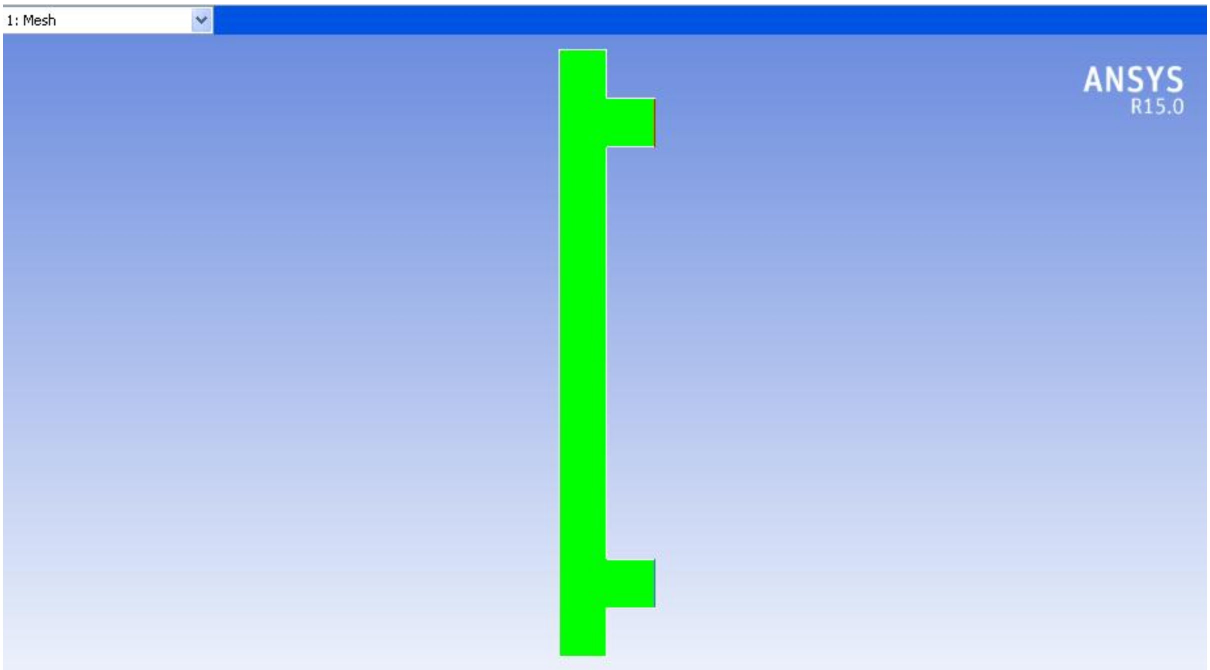
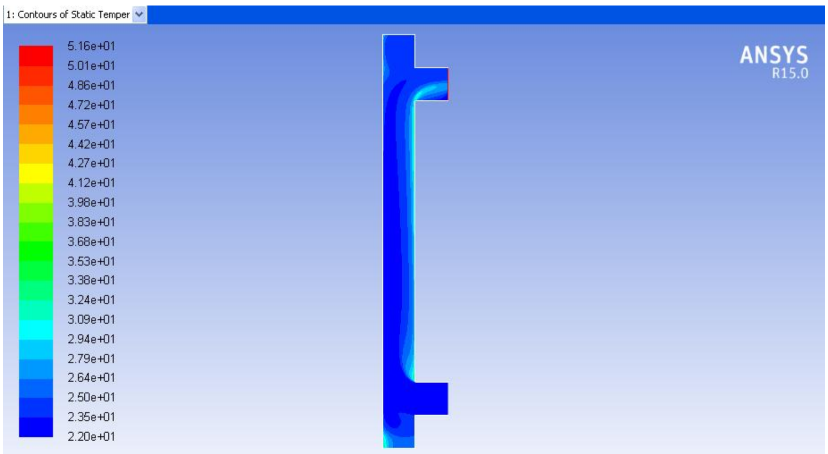


Figure (1). Computing domain networking with network and organization

Computational domain

The two-dimensional domain of the solution only includes the air gap between the thrombus wall and the glass wall. The height of the domain is 2.5 meters and its width is 0.2 meters. The height of the valves is 0.2 meters and the width of the walls of the valves is also 0.2 meters. Air valves are considered as inlet and outlet boundaries. The produced grid is structured type 3 regular square 4. The geometry, generated grid and the boundaries of the solution domain are shown in figure (1).



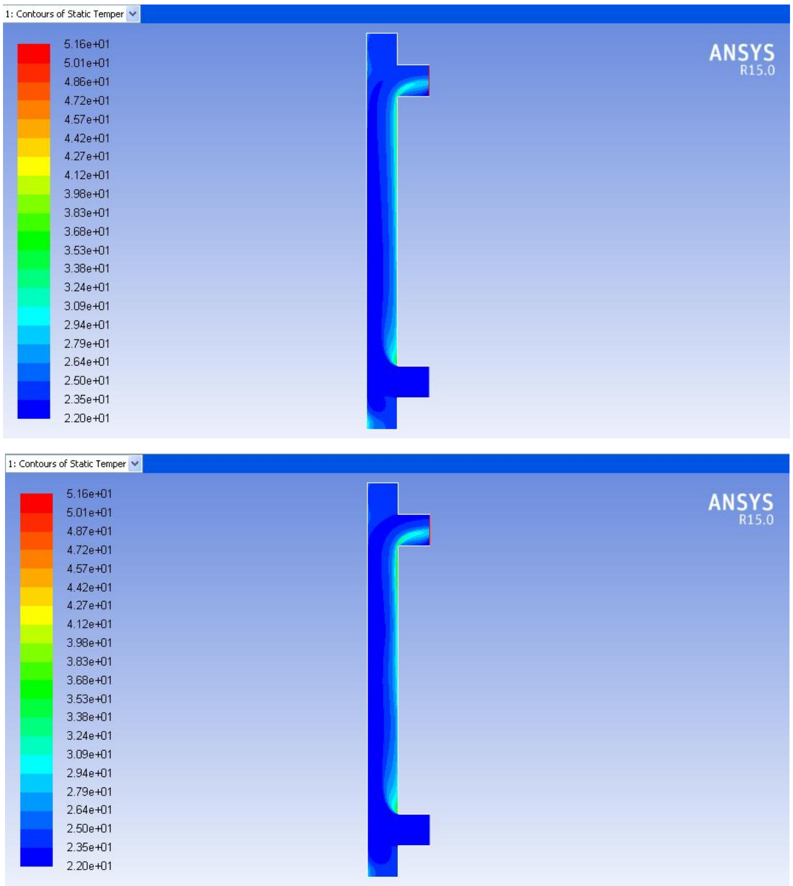
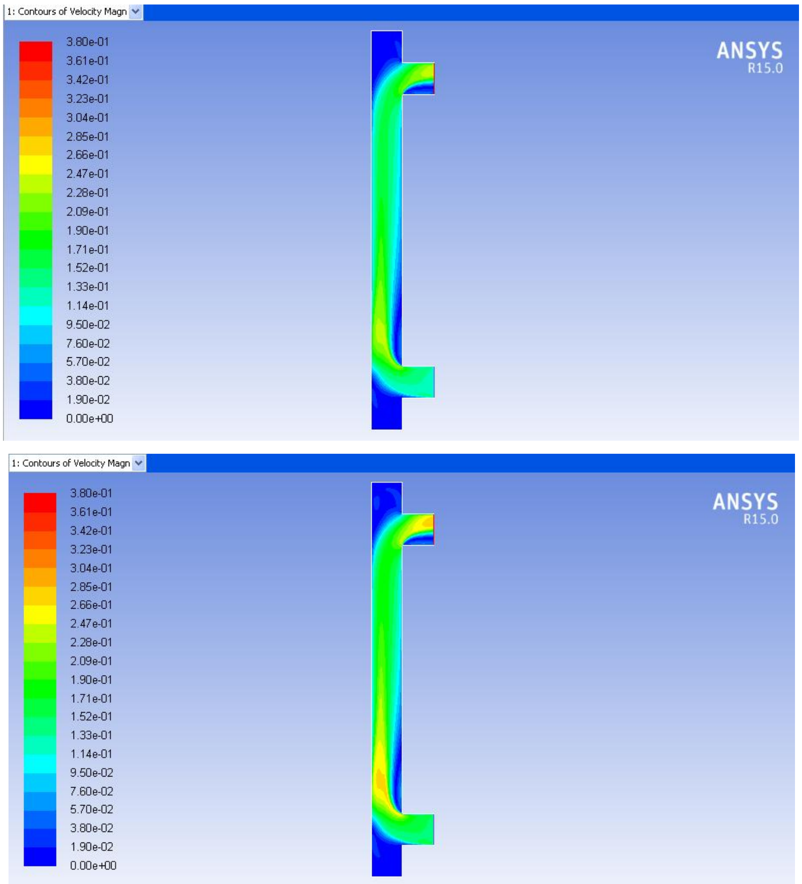


Figure (2). The effect of changing the specifications of the Trompe wall on the static temperature contour



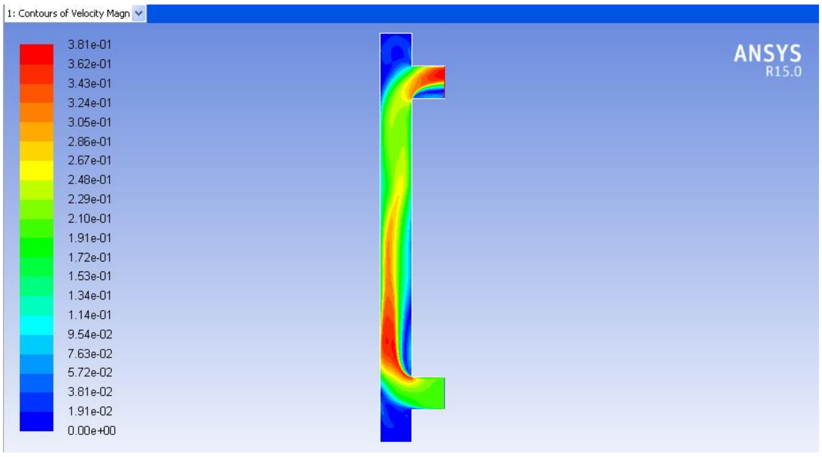


Figure (3). The effect of changing the specifications of the Trumpet wall on the velocity contour

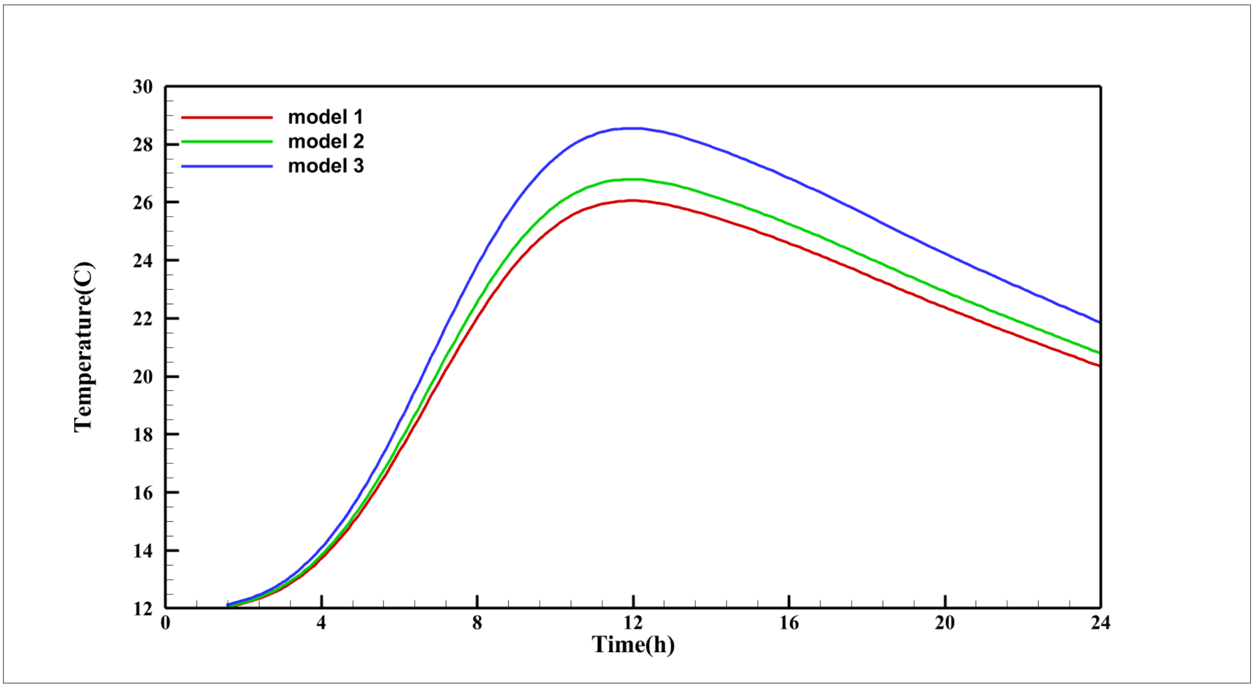


Figure (4). Thermal storage wall temperature of Trombe wall system in comparison of three modes

Figure (2). It shows the effect of changing the characteristics of the Trompe wall on the temperature contour. The average temperature in the solution domain also increases with increasing height and approaching the thrombus wall in the cross section. By changing the materials from model 1 to 3, the outlet temperature at the top of the Trumpet wall increases. In figure (3). The effect of changing the characteristics of the trompe wall on the speed contour has been investigated. By changing the materials from model 1 to 3, the speed of the air exit above the wall of the Trumpet increases. As shown in the velocity contour, the airflow is a swirling flow in the cross section of the fluid domain. The air flow is heated in the vicinity of the solar wall and moves upwards due to the increase in temperature, then when it hits the roof, some of this air leaves the outlet and a small amount is cooled again by passing through the glass wall. The velocity in the cross section starts from zero velocity in the cold wall and near the hot wall to reaches its maximum. Figure (4). It shows the temperature of the thermal storage wall of the Trombe wall system compared to three modes .In this case, the air temperature of the room reaches its maximum value around 12 o'clock, and you can see the time delay in the heat transferred to the room at night. To increase the efficiency, the effect of changing the material of the thrombus wall was investigated. Compared to models 2 and 1, model 3 has the highest amount of thermal storage for night time.

Conclusion:

In this article, the air flow inside the thrombus brush wall chamber was simulated by computational fluid dynamics using Ansys Fluent commercial software. At first, a preliminary reference model and its results were presented, and then using this reference model, parameters affecting the wall efficiency were presented. Trombe was studied and investigated. It was shown that the air temperature of the room in this state reaches its maximum value around 11 o'clock and the time delay in the heat transferred to the room can be seen at night. To increase the efficiency, the effect of changing the material of the thrombus wall was investigated. Finally, it was concluded that model 3 has the highest amount of thermal storage for night time compared to models 2 and 1.

References:

1. Tan, A. Y. K., & Wong, N. H. (2013). Parameterization studies of solar chimneys in the tropics. *Energies, 6*(1), 145–163. https://doi.org/10.3390/en6010145
2. Ruiz-Pardo, Á., Álvarez Domínguez, S., & Sanz Fernández, J. A. (2010). Revision of the Trombe wall calculation method proposed by UNE-EN ISO 13790. *Energy and Buildings, 42*(5), 763–773. https://doi.org/10.1016/j.enbuild.2009.11.010
3. Mezrhab, A., & Rabhi, M. (2008). Modeling of the thermal transfers in an enclosure of the Trombe wall type. *Thermodynamic Analysis in Renewable Energy, 10*(62), [pages not specified].
4. Mansour, A. R., Jubran, B. A., & Tashtoush, B. (1991). An approximate analytical solution to convective laminar heat transfer flow within the Trombe wall channel. *International Communications in Heat and Mass Transfer, 18*(2), 153–159. https://doi.org/10.1016/0735-1933(91)90064-C
5. Jubran, B. A., Hamdan, M. A., Tashtoush, B., & Mansour, A. R. (1993). An approximate analytical solution for the prediction of transient response of the Trombe wall. *International Communications in Heat and Mass Transfer, 20*(4), 567–577. https://doi.org/10.1016/0735-1933(93)90072-D
6. Koyunbaba, B. K., & Yilmaz, Z. (2012). The comparison of Trombe wall systems with single glass, double glass and PV panels. *Renewable Energy, 45*, 111–118. https://doi.org/10.1016/j.renene.2012.02.001
7. Quesada, G., Rousse, D., Dutil, Y., Badache, M., & Halle, S. (2012). A comprehensive review of solar facades. Opaque solar facades. *Renewable and Sustainable Energy Reviews, 16*(4), 2820–2832. https://doi.org/10.1016/j.rser.2012.01.060
8. Gan, G. (2006). Simulation of buoyancy-induced flow in open cavities for natural ventilation. *Energy and Buildings, 38*(4), 410–420. https://doi.org/10.1016/j.enbuild.2005.07.004
9. Chan, H.-Y., Riffat, S. B., & Zhu, J. (2010). Review of passive solar heating and cooling technologies. *Renewable and Sustainable Energy Reviews, 14*(3), 781–789. https://doi.org/10.1016/j.rser.2009.10.013
10. Shen, J., Lassue, S., Zalewski, L., & Huang, D. (2007). Numerical study on thermal behavior of classical or composite Trombe solar walls. *Energy and Buildings, 39*(8), 962–974. https://doi.org/10.1016/j.enbuild.2006.11.009
11. Ong, K. S. (2003). A mathematical model of a solar chimney. *Renewable Energy, 28*(7), 1047–1060. https://doi.org/10.1016/S0960-1481(02)00136-X
12. Nwachukwu, N. P., & Okonkwo, W. I. (2008). Effect of an absorptive coating on solar energy storage in a Trombe wall system. *Energy and Buildings, 40*(3), 371–374. https://doi.org/10.1016/j.enbuild.2007.03.010
13. Torcellini, P., & Pless, S. (2004). *Trombe walls in low-energy buildings: Practical experiences*. National Renewable Energy Laboratory. https://www.nrel.gov
14. Khanal, R., & Lei, C. (2011). Solar chimneyA passive strategy for natural ventilation. *Energy and Buildings, 43*(8), 1811–1819. https://doi.org/10.1016/j.enbuild.2011.03.036
15. Ben Yedder, R., Du, Z. G., & Bilgen, E. (1990). Numerical study of laminar natural convection in composite Trombe wall systems. *Solar & Wind Technology, 7*(6), 675–683. https://doi.org/10.1016/0741-983X(90)90092-G
16. Ben Yedder, R., & Bilgen, E. (1991). Natural convection and conduction in Trombe wall systems. *International Journal of Heat and Mass Transfer, 34*(4-5), 1237–1248. https://doi.org/10.1016/0017-9310(91)90265-8
17. Kim, S., & Seo, J. (2012). An influence of the opening location of the Trombe wall system on indoor airflow and thermal environment. *National Research Foundation of Korea, Gwangju*.
18. Hernández, V., Morillón, D., Best, R., Fernández, J., Almanza, R., & Chargoy, N. (2006). Experimental and numerical model of wall-like solar heat discharge passive system. *Applied Thermal Engineering, 26*(17–18), 2464–2469. https://doi.org/10.1016/j.applthermaleng.2006.03.017
19. Li, Y., Duanmu, X., Sun, Y., Li, J., & Jia, H. (2007). Study on the air movement character in solar wall system. *Building Simulation 2007*.
20. Ansys Inc. (n.d.). *ANSYS Fluent user guide*. ANSYS Inc.