Prediction of Indoor Air Circulation of Residential Room with Adaptation of Solar Chimney Using Numerical Technique

Punit Kongi, Thejaraju. R, T S Madan Kumar, Prince Joseph

Abstract - With the exponential increase in consumption of electrical power during the summer season by household, there is a great need for households to withhold sustainability. To maintain the temperature of the household a passive heating and cooling system is used i.e. Solar Chimney. Ventilation, through a natural convection process, is gaining a lot of attention to be an alternative technique for mechanical air conditioning ventilation because of its reduced power usage when compared to the external cooling devices used in residential buildings of hot regions. The present study, involve solar chimney of horizontal and vertical designs in comparison with different width and height. The following paper studies the effect of a solar chimney on the indoor thermal behavior using Numerical Technique for a prototype of a residential room. The performance on the ventilation velocity and air temperature operation inside the room with varying air gap width is studied based on multiple numerical analysis solutions. The present study deals with two different architectures of a two dimensional model and results have shown that the ventilation velocity has increased to 0.017626444 kg/s and operative air temperature has been decreased by 7.26ºC for the vertical model while the horizontal model has shown a mass flow rate of 0.018027636 kg/s and a temperature decrease of 9.15ºC. The most efficient chimney was found to be model 7 which is horizontal solar chimney 3 with an air gap width of 0.05625m and a height of 0.3175 m, when compared to the other models from model number one to six.

Keywords: Natural Convection, ventilation, solar chimney, Numerical Technique.

I. INTRODUCTION

With an increase in incomes and urbanization, India is about to see an exponential boost in the use of electrical air conditioning from 23.6 million units in 2018 to 1 billion units by 2050. If this need for active cooling methods follows the same growth pattern, cooling units alone will account to 0.5°C increase in the Earth's Temperature (Mohan, 2019) along with the doubling of India's energy consumption by cooling units alone within 2027 [2]. This calls for an alarming need for alternative methods for cooling and this paper gives a study on passive cooling in a residential building prototype by natural convection and solar energy using the numerical technique [3].

Revised Manuscript Received on September 2, 2019.

Punit Kongi , Faculty B.Tech degree in Mechanical Engineering B.V.B College of Engineering and Technology, India

Thejaraju, Assistant Professor, in the Department of Mechanical and Automobile Engineering in the area of Heat Transfer, Fluid Dynamics and Energy

T S Madan Kumar, Assistant Professor, in the Department of Mechanical and Automobile Engineering

Prince Joseph , B.Tech Degree in Mechanical Engineering from CHRIST (Deemed to Be University), Faculty of Engineering in the year 2019

A. Solar Chimney

Solar energy utilization technology is basically characterized in two ways Active and Passive solar design. To heat water/fluid with direct solar radiation is characterized as active solar design and for heating and cooling of the living area with exposure of solar energy is considered as passive solar design.

The passive cooling system design allows buildings to adapt much better with their local environment and also provide a cleaner and safer environment for building users [3]. The principle of natural convection plays a major role in the passive cooling of buildings and it serves as a major advantage since no electricity is being consumed and can help bring down the cooling load of buildings. [4]

It is a mechanical system that makes use of solar power to build buoyancy-driven airflow and naturally ventilate buildings. It is a chimney which has 1 or more of its walls made transparent by adding a glazed wall(s). This system heats up the air present in the chimney resulting in density difference between the air inside and outside the chimney. Thus, a naturally convicted airflow is produced [5].

Solar Chimneys have proved to be an extremely effective method of passive solar-induced ventilation which have been adopted from the traditional architecture. The ability to be designed as a vertical element enables it to utilize solar energy to enhance the natural stack ventilation within a room [6].

Other pros of this mechanical system would be energy efficient, cost-effective, reliable, low maintenance and environmentally friendly. Another important advantage is that the system balances itself, i.e., Higher the outside temperature, hotter the air in the chimney and faster the heat transfer.

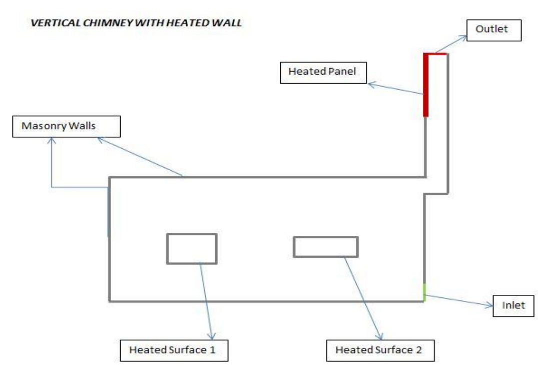


Fig 1: Schematic diagram of Solar Chimney.

B. Computational Fluid Dynamics (CFD Analysis)

Fluid Dynamics is the study of fluid flow under various stipulations following the laws of physics. The laws governing such theories and the nature of the fluids themselves are made understandable using fluid dynamics. Simulation of such stipulations is done using computational calculation [7]. CFD is the most commonly used tool to solve such sophisticated calculations and geometries, and analyzing the laws governing fluid dynamics numerically. Incorporating CFD into the process allowed the designer to get real-time results of the thermal performance of the room in response to the climatic factors.

II. RESEARCH APPROACH

A. The Objective of the Study

The ultimate objective of this study is to ensure an increase in the thermal comfort of building occupants and to evaluate the optimal performance of the chimney architecture design for two models. This study is based on CFD simulation to compute the air gap width effect on the air flow rate and the operating air temperature inside the room. This would help achieve the long term objective of reducing the usage of electro-mechanical appliances for the purpose of cooling.

B. Analysis Tool

In this study, ANSYS Workbench was used to model the prototype along with the chimney and ANSYS Fluent was chosen as the CFD simulation software [9]

C. Methodology

a) Selection of building type which has been used by the larger population and scaling down the size of the building for analysis and experiment investigation.
b) Designing the building in a 2D form for analysis purpose, two types of solar chimney design were analyzed.
c) Each solar chimney design was analyzed for three different widths of the chimney each increased by 50% from the previous design.
d) Based on mass flow rate and temperature difference results the solar chimney was selected for the building.
e) Next, for this design, an experimental setup was designed and readings were noted down for comparison.
f) Lastly based on results obtained discussion and suggestions were given.

III. NUMERICAL ANALYSIS

Numerical analysis is the method of finding approximate but accurate solutions for hard to solve problems by implementing algorithms. The required calculations are performed to simulate the free-stream flow of the fluid, and the surface-fluid interactions subjected to boundary conditions [10]. In this Thermal Simulation, the building comprises one inlet and one outlet, which is positioned at the bottom right and the upper right corners respectively. This structure is built on 30*40 square feet site, scaled down to 1/12th. The current model is constructed using Design-Builder of Length 0.635 m and height 0.254 m. Considering the exact size of this edifice, the analysis would be strenuous to complete. To achieve accurate results, 2D model of the building is taken as the object under consideration. There are numerous heat sources inside the building such as Refrigerator, Ceiling Fans, Television, Microwave Oven and many other Mechanical Systems. Additionally, the human body also releases a certain amount of heat. For the purpose of the analysis, two sources of heat are considered which will sum up all the systems in the building. The hot surfaces are given a temperature of 320K each.

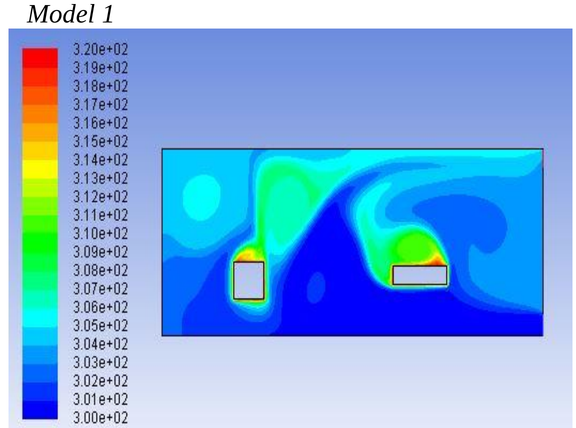
A. Case Study

A scaled down model of a residential room was taken and an analysis was performed without the presence of the solar chimney. Fig 2. Shows the change in temperature when the room has one single opening which serves as both inlet and outlet. Since multiple heat sources are naturally present in a room, two heat sources have been taken into account for simulation purposes.

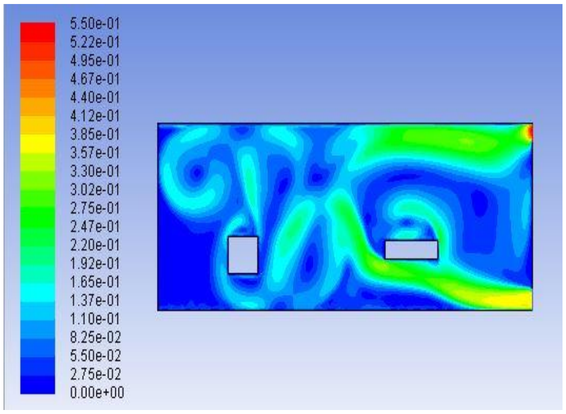
TABLE I
Description of different Solar Chimney Models

Model Name	Description	Dimension (m)
Model 1	Without Solar Chimney	NIL
Model 2	Vertical Solar Chimney 1	0.025
Model 3	Vertical Solar Chimney 2	0.0375 (Width increased by 50% of Model 2)
Model 4	Vertical Solar Chimney 3	0.05625 (Width increased by 50% of Model 3)
Model 5	Horizontal Solar Chimney 1	0.025
Model 6	Horizontal Solar Chimney 2	0.0375 (Width increased by 50% of Model 5)
Model 7	Horizontal Solar Chimney 3	0.05625 (Width increased by 50% of Model 6)

- *Model 1*



a). Temperature Contour



b). Velocity Contour

Fig 2: (a) and (b) shows the Temperature and Velocity distribution in the building without Solar Chimney.

Above shown are the Temperature and Velocity contours of the 2D building model post-thermal simulation. The 2D model was deciphered using ANSYS 16.0. To capture the accurate results, an element size of 4x10-3 is utilized to mesh the geometry. The boundary conditions set to the inlet is Pressure Inlet, and for the outlet, domain is Pressure outlet. And the other parts are considered as Wall. As mentioned earlier; the two heat sources are of 320K each. Based on the research, for analyzing such a case k-ω viscous model is used to lead the simulation to a simple and better approach. Hence the k-ω model has been implemented.

From the temperature contour, it is observed that the larger part of the area is in the temperature range of 304-312K. Hence this temperature range will cause discomfort inside the building.

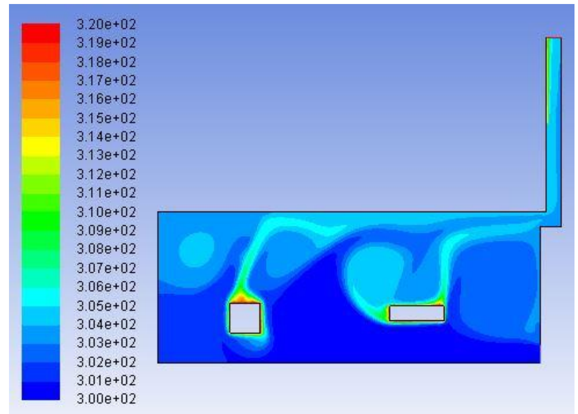
B. Solar Chimney CFD

The target of this simulation is to enhance the rate of flow and to make the conditions comfortable in a room by the application of a mechanical system, i.e., a solar chimney on the western façade. This may affect the natural ventilation rate positively and increase the comfort of the conditions inside the room. The objective of the Solar Chimney design analysis is to reduce the temperature inside the building through natural convection. Two types of designs are analyzed as follows.

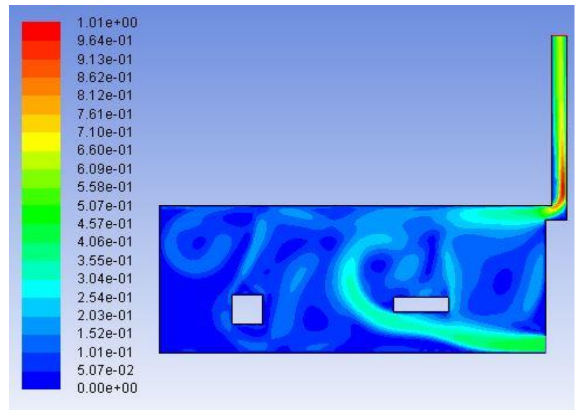
- ***Vertical Model***

An additional solar chimney is added to the case study model to observe the effect it has on the indoor temperature, outlet mass flow rate and the air velocity present in the room. In this design, the chimney is of the length L=0.3175 m and of width W=0.025 m and the Vertical Chimney's top portion is covered with aluminum for faster heat absorption.

- ***Model - 2***



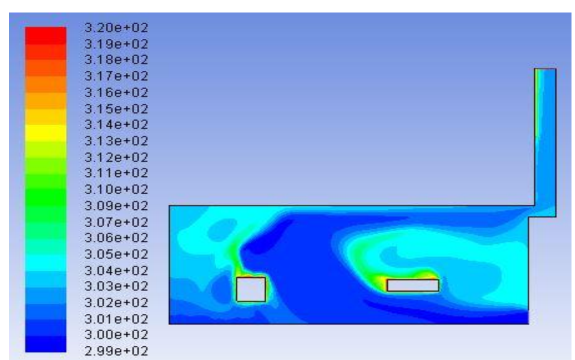
a). Temperature Contour



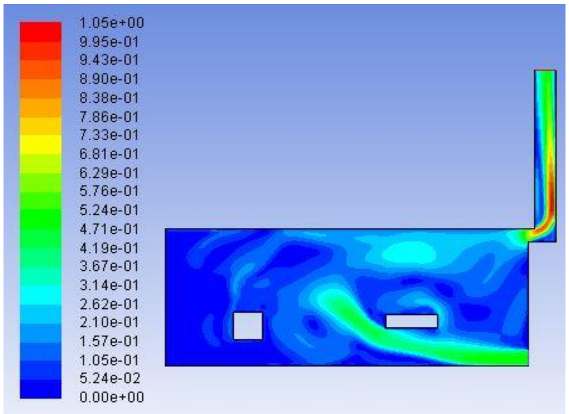
b). Velocity Contour

Fig 3: (a) and (b) shows the Temperature and Velocity distribution in the building with Vertical Solar Chimney. The Temperature and the Velocity contours clearly depict the fall in temperature and rise in the outflow air velocity as compared to Model 1.

- ***Model – 3***



a). Temperature Contour

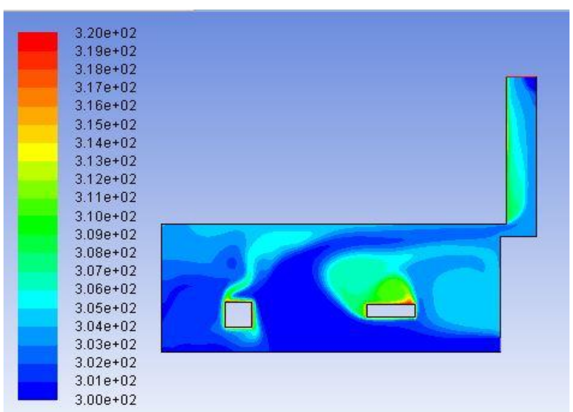


b). Velocity Contour

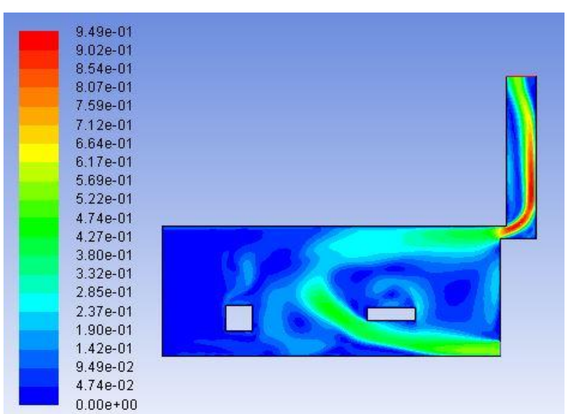
Fig 4: (a) and (b) shows the Temperature and Velocity distribution in the building with Vertical Solar Chimney, width increased by 50% of Model 2.

As compared to Model 2, the width of the chimney is 50% more. Also, the static interior temperature is less than that of Model 1. The increase in the air speed at the outlet is also encountered**.**

- ***Model – 4***



a). Temperature Contour



b). Velocity Contour

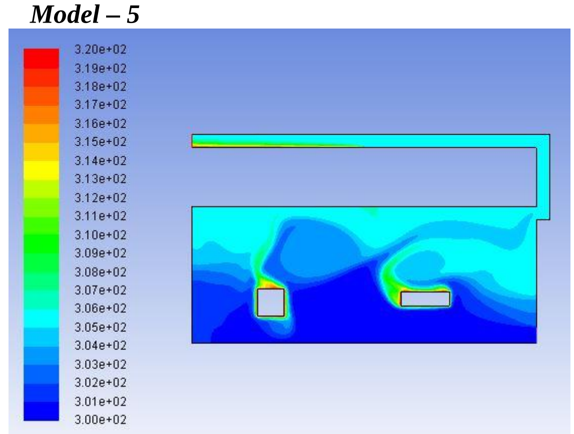
Fig 5: (a) and (b) shows the Temperature and Velocity distribution in the building with Vertical Solar Chimney, width increased by 50% of Model 3.

The temperature at the outlet is comparatively less than the Model 1. The increase in the width of the chimney by 50% to the last design allows the air to move at a faster rate.

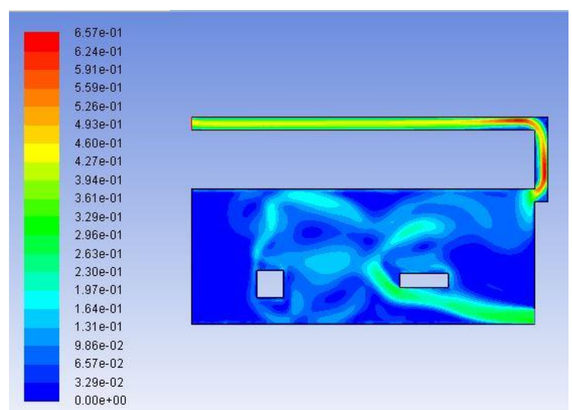
- ***Horizontal Model***

In this model the chimney design is in the horizontal direction as shown in Fig 6, the main advantage of this model is it protects the building roof by direct sunlight. The length of the chimney L=0.660 m (length of the room + width of the chimney) and of the width, W=0.025 m. The top of the chimney is covered with aluminum foil for increased heat transfer.

- ***Model – 5***



a). Temperature Contour

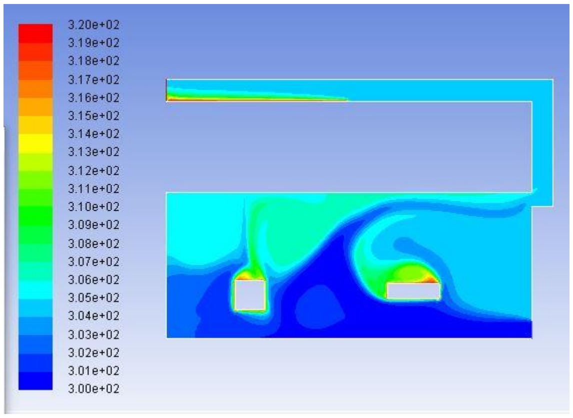


b). Velocity Contour

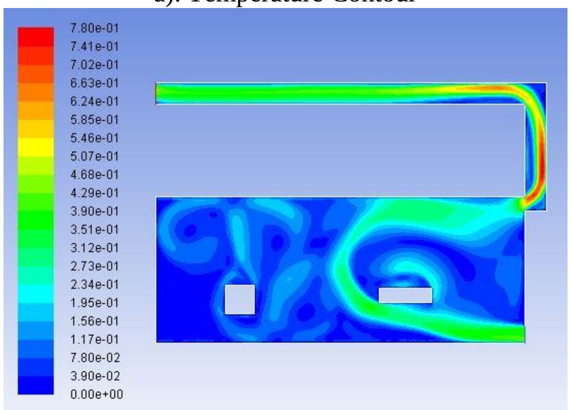
Fig 6: (a) and (b) shows the Temperature and Velocity distribution in the building with Horizontal Solar Chimney.

The temperature contour shows that there is a drop in the interior temperature as compared to that of Model 1. The air velocity has also constantly boosted up.

- ***Model – 6***



a). Temperature Contour

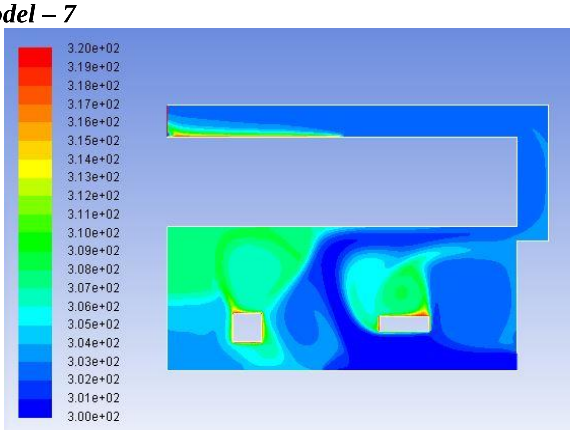


b). Velocity Contour

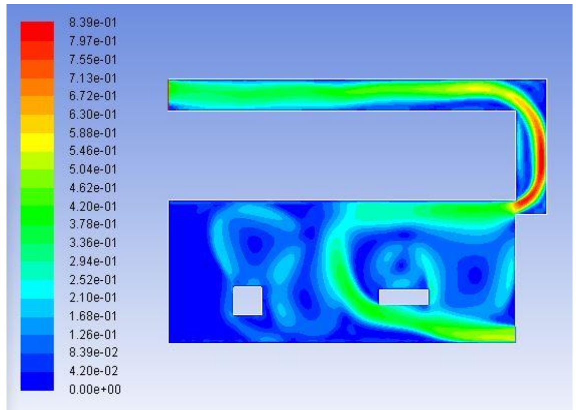
Fig 8: (a) and (b) shows the Temperature and Velocity distribution in the building with Horizontal Solar Chimney, width increased by 50% Model 6.

Here in this model, the chimney width is increased to 50% of Model 5. Analyzing the temperature and velocity contours, there is a cutback in the interior temperature and a gain in the air speed at the outlet compared to Model 1.

Model – 7



a). Temperature Contour



b). Velocity Contour

Fig 8: (a) and (b) shows the Temperature and Velocity distribution in the building with Horizontal Solar Chimney, width increased by 50% Model 6.

Above displayed are the temperature and the velocity contours of the Horizontal chimney in which the thickness is expanded to 50% of Model 6. As compared to Model 1, the temperature within the building has steadily decreased, whereas the air at the outlet has gained some momentum.

IV. EXPERIMENTAL INVESTIGATION

A scaled-down area of the 30*40 square feet site is selected and a built-up area of 161.29 m2 is used as the case study model. A single story building with one inlet and one outlet is made out of cardboard. Bulb (100W), as a heat source, is fixed to demonstrate the warmth delivered (ascend in temperature) inside the room. The model has two points of the heat source and to record the exact temperature a Surface Thermometer is utilized. The test is carried out in a detached region with the encompassing air at the surrounding condition. Under this experiment investigation, two unique solar chimneys have been advanced of which one is with a vertical model and the other with a horizontal model. The chimney is 0.3175m in length and 0.025m in width.

The air inside the test object gets warmed up because of the heat source present inside. As the air gets heated, we encounter a difference in the density of the air molecules. Hence the heated air moves upward giving way to the cold air which moves towards the heat source. Since there is additional pressure difference, this warm air moves out through the vent created at the top, thereby carrying away the heat.

Fig 9: Experimental Setup

V. RESULTS AND DISCUSSION

In this study of analysis and experimental investigation of the building, the main objective is to check the temperature difference with the addition to the mass flow rate of the air at different points in the selected region.

Based on the simulation results from the temperature contour, a set of mass flow rates and the temperature differences is displayed in the list below.

TABLE II
Mass Flow Rate

Model No	Outlet (Kg/s)	Percentage change (%)
1	0.012588314	0
2	0.014740752	17.0986
3	0.016273987	27.2785
4	0.017626444	40.0222
5	0.014027645	11.4397
6	0.015003277	19.1841
7	0.018027636	43.2092

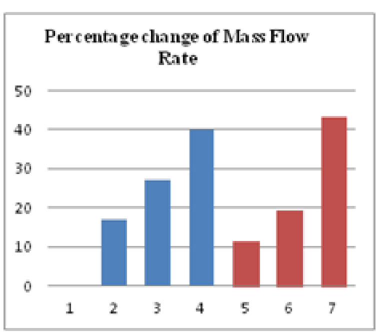


Chart 1: Percentage change in Mass Flow Rate of different models in comparison to Model 1.

Based on the above chart, we come across an increase in the Mass flow rate at each model outlet compared to the Case Study model. Here in the chart presented, Serial No. 1 represents the Case study model, similarly, numbers from 2-4 depict the Vertical solar chimney and Numbers from 5-7 shows the Mass flow for Horizontal solar chimney.

TABLE III
The temperature at the outlet (k)

Model No	Temperature (k)
1	308.201
2	307.691
3	305.157
4	302.941
5	306.255
6	304.171
7	301.051

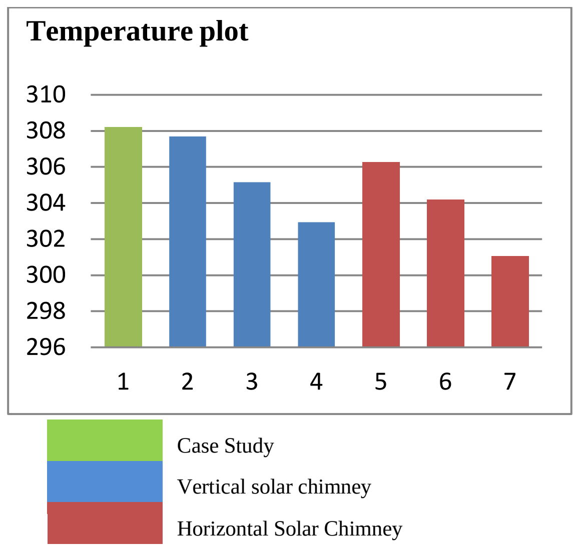


Chart 2: Difference in the temperature at the outlet vent for different models.

Above shown are the Temperature chart of the case study, Vertical solar chimney and Horizontal solar chimney with the Serial numbers 1, 2-4 and 5-7 respectively.

It is found that temperature has progressively decreased

VI. CONCLUSION

Numerical model has successfully developed for solar chimney with vertical and horizontal models, the results shows satisfactory improvement in the air circulation and temperature distribution with the addition of solar chimney.

The solar chimney helps in reducing the hotter areas of the room and decreases the working temperature through natural ventilation and hence increases the comfort of building occupants.

The observations show that the addition of solar chimney can reduce the average working temperature from 310.2 K to 302.9 K for the vertical model with an air gap width of 0.05625m and a vertical height of 0.3175m, and from 310.2 K to 301.1 K for the horizontal model with an air gap width of .05625m and a horizontal length of 0.635m.

In short, though the mass flow rate and the temperature do not reach the required comfort condition, a significant change is observed in the thermal performance when compared to outdoors.

Improvements in the present study have to be made through further experiments to find the optimum features of solar chimney architecture to achieve standard comfort conditions.

REFERENCE

1. K. Ong and C. Chow, "Performance of a solar chimney," Solar Energy, vol. 74, no. 1, 2003, pp. 1–17.
2. Matheos Santamouris, "PASSIVE COOLING OF BUILDINGS" Advances of Solar Energy, ISES, 2005.
3. A. Ramesh Kumar, KCK. Vijayakumar and PSS. Srinivasan, "Passive Cooling Practices in Residential Buildings – A Review" International Journal of Mathematical Sciences and Engineering (IJMSE), Volume – I, Issue – I, March 2014.
4. B. Givoni, "Climate considerations in building and urban design", 1998, p. 1-15.
5. Mekkawi, Gehad & Elgendy, Rana, "Solar Chimney for Enhanced Natural Ventilation Based on CFD-Simulation for a Housing Prototype in Alexandria, Egypt" International Journal of Advances in Mechanical and Civil Engineering (IJAMCE), Volume 3, Issue 6, November 2016.
6. Vijay Mali, "What is Computational Fluid Dynamics?" Internet: www.quora.com/What-is-computational-fluid-dynamics, Jun. 10, 2015.
7. S. M. Metev and V. P. Veiko, Laser Assisted Microtechnology, 2nd ed., R. M. Osgood, Jr., Ed. Berlin, Germany: Springer-Verlag, 1998
8. Naci Kalkan, Ihsan Dagtekin, "CFD Analysis of Passive Cooling Building by Using Solar Chimney System", International Journal of Mechanical and Mechatronics Engineering Vol:9, 2015, No:1.,

AUTHORS PROFILE

Punit Kongi currently working as resident faculty at The Automotive Research Association of India, Pune. He received his M.Tech degree in Automobile Engineering from Vellore Institution of Technology, India in 2016 and B.Tech degree in Mechanical Engineering from B.V.B College of Engineering and Technology, India. He has given a technical presentation on Automotive Aerodynamics at Sri Lanka invited by The Institution of Automotive Engineering, Sri Lanka. He his passionate in automobile sector and papers/talk presented at different forums on topics related to electrical tricycle and electrical wheelchair. His researches are concentrated in Computational Fluid Dynamics, Electric Vehicles, Heat Exchanger, etc.

Thejaraju R, born in Bangalore, INDIA in 1987, obtained his M.Tech. in Thermal Power Engineering from the Visvesvaraya Technological University, Belgaum in 2011, and his engineering degree in Mechanical Engineering from the Visvesvaraya Technological University, Belgaum in 2009. He works as a tenured Assistant professor at the CHRIST (Deemed to be University) Faculty of Engineering as Assistant Professor, in the Department of Mechanical and Automobile Engineering in the area of Heat Transfer, Fluid Dynamics and Energy, His research areas are Computational Fluid Dynamics, Fluid flow analysis , Design of heat exchangers, Turbo machines, Refrigeration, heating, ventilation and air-conditioning.

T S Madan Kumar received his B.Tech Degree in Mechanical Engineering from CHRIST (Deemed to Be University), Faculty of Engineering in the year 2019. He is passionate about Automotive Transmission System and Material science and has attended various conferences and workshops. His researches are mainly concentrated on Computational Fluid Dynamics, Active and Passive Cooling, Refrigeration and air-conditioning, etc. He currently wishes to pursue his Masters of Technology in Renewable Resources or Power Engineering in Germany. He has attempted his A2 levels in the German language from Goethe Institute. He possesses a lot of leadership and interpersonal skills which he wants to put into practice by starting his own business after his masters.

Prince Joseph received his B.Tech Degree in Mechanical Engineering from CHRIST (Deemed to Be University), Faculty of Engineering in the year 2019. He also completed his Diploma and Advanced Diploma in Management Accounting from CIMA, UK in 2018 and 2019 respectively. He has been involved in many of projects revolving around mechanical engineering including Mini CNC Plotter, Economic Portable Washing Machine, Solid Waste Management System, Biped Robotics, Electric All Terrain Vehicle for SAEINDIA BAJA 2018 etc. He currently works in Hewlett Packard Enterprise Financial Services as an Analyst Intern. He wishes to pursue his CGMA degree from CIMA, UK. He has attended few national and international level conferences.