CFD Analysis of Room Air Ventilation through Solar Chimney

Sushil Kumar[1], Dr. V. N. Bartaria[2]

[1]M. Tech. Scholar, Department of Mechanical Engineering, LNCT, Bhopal

[2]Professor & Head, Department of Mechanical Engineering, LNCT, Bhopal

Abstract: *Energy has very important role in the livelihood of human being. It is the prime agent in the social and economic development. Energy resources may be classified in two ways primary energy resources or conventional resources and Renewable sources of energy. For the healthy and comfortable conditions it is required to keep the clean and comfortable environment in the living and working places. A room to be ventilated to exchange the room air with the fresh outside air is required conventional source of energy as electricity or by mechanical means. The passive design of building approach provides us the better option in which renewable energy source of energy may be used such as the use of solar energy. This paper presents the CFD study of flow and temperature features in a typical room having solar chimney. The results in the form of contours and plots are obtained to show the temperature and velocity variations. The paper presents the CFD approach for the solution of solar air ventilation using solar chimney.*

Keywords: Passive design, solar chimney, CFD

1. Introduction

Energy has very important role in the livelihood of human being. It is the prime agent in the social and economic development. Energy resources may be classified in two ways primary energy resources or conventional resources and Renewable sources of energy. For the healthy and comfortable conditions it is required to keep the clean and comfortable environment in the living and working places. A room to be ventilated to exchange the room air with the fresh outside air is required conventional source of energy as electricity or by mechanical means.

Numerical Simulation
The technique of numerical simulation is being used by the researchers and engineers in many of the engineering and scientific analysis. Numerical analysis method is used in solving physical problems with the use of mathematical analysis and numerical approximation. It has wide applications of scientific computations in all fields of engineering and sciences. With the increase in computation power of modern computers, numerical methods are now not required to have large hand calculations.

Computational Fluid Dynamics, CFD is the numerical method of solving engineering problems involving fluid flow and heat transfer.

A 2-dimensional CFD simulation of a room with solar chimney has been investigated in this study. Commercial CFD simulation code FLUENT (version 6.3) is used to solve the conservation equations for mass, momentum and energy.

For the preparation of the geometrical model and meshing GAMBIT 2.3 has been used. For the simulation of the results in terms of the contours of temperature and velocity ANSYS FLUENT software has been used. Two test locations at a distance of 1 m are taken.

Solution Domain
The 2-dimensional solution domain used for CFD analysis has been generated as shown in Fig. 1. The solution domain is a room of typical dimensions with a solar chimney placed at the roof of the room with absorber plate at an angle of 45°. Three cases have been studied for the study of room ventilation in which the solar radiations falling on the absorber plate with the intensity of 600 W/m^2, 700 W/m^2 and 800 W/m^2 respectively.
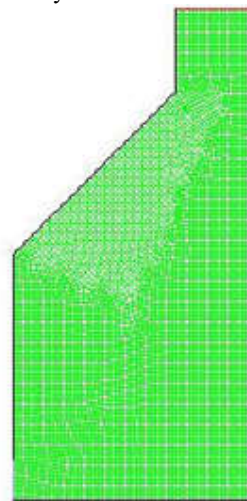
Figure 1: Schematic of two-dimensional solution domain for CFD analysis.

Meshing of the domain is done using GAMBIT software. A non-uniform mesh with very fine mesh size is used, since low- Reynolds number turbulence models are employed, the grids are generated so as to be very fine.

2. Results

The numerical results are obtained in the form of contours of temperature and velocity to show the ventilation by flow and thermal features. The absorber plate of the solar chimney receives the radiations with the heat flux of 600 W/m^2.
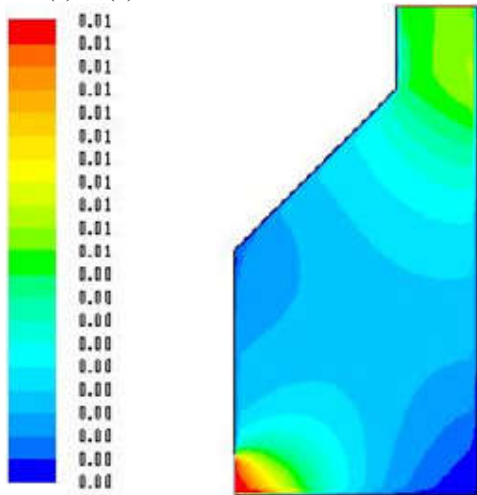
Contours of Temperature and Velocity

To show the ventilation in the room due to the solar chimney contours of Temperature and Velocity are obtained as shown in figure 2 (a) & (b).


(a) Velocity Contours
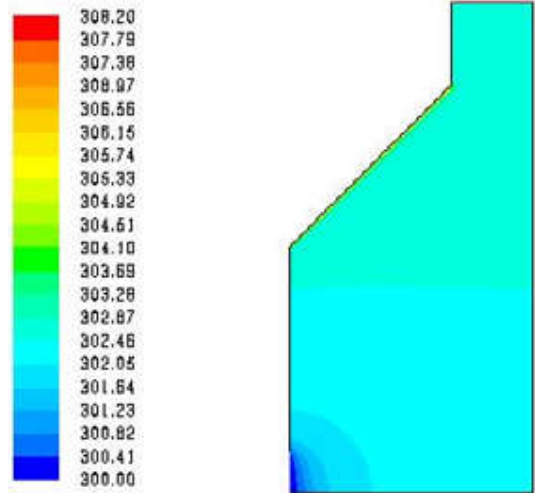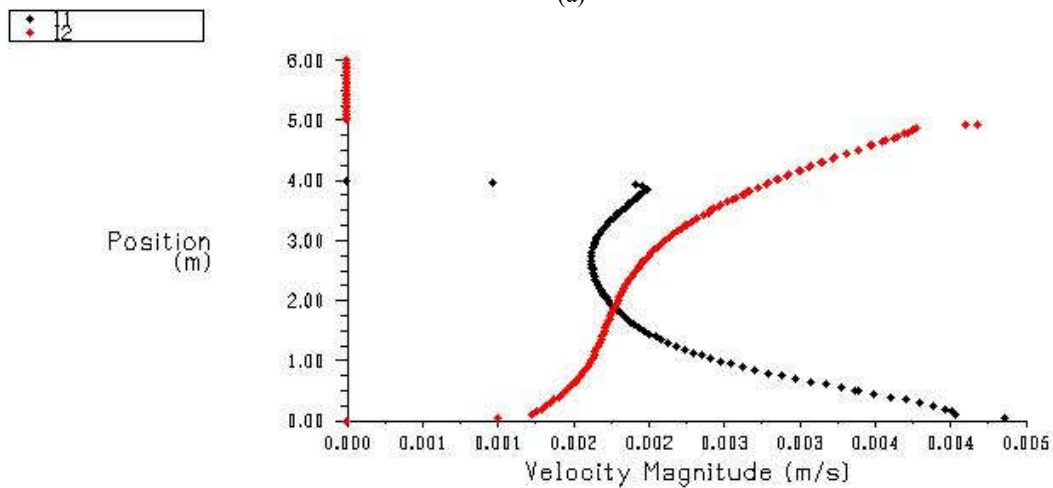

(b) Turbulence Contours
Figure 2: Contours of Temperature & Velocity

Vertical Temperature and Velocity plots

Vertical temperature and velocity variations are obtained for the two test locations at one meter apart. These comparative vertical temperature and velocity plots show the temperature and velocity variations and are shown in figure 3 (a) and (b).
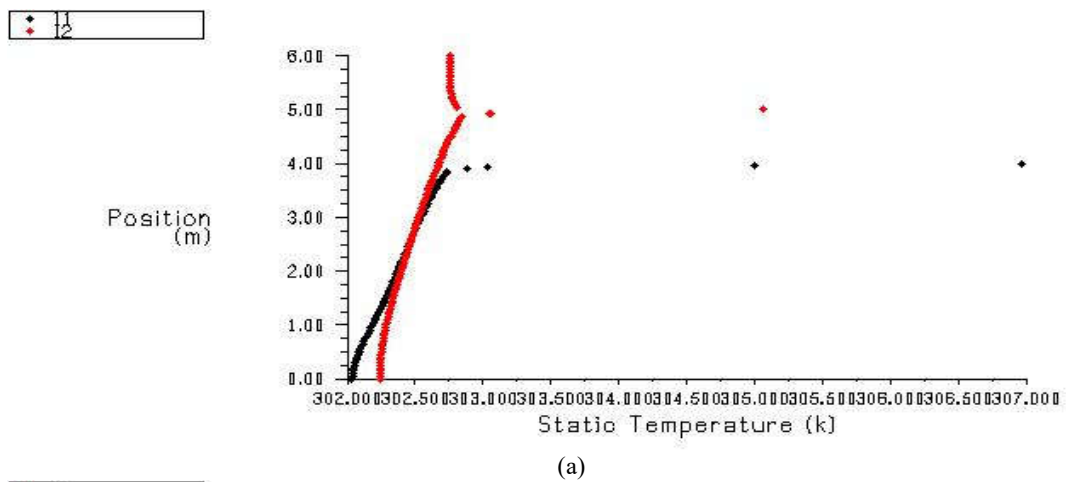
(a)

(b)
Figure 3 (a) & (b): Vertical Temperature and Velocity plots

Volume 7 Issue 8, August 2018

3. Conclusion

To improve the ventilation in the room passive building design approach with solar energy is presented in this paper. This approach of ventilation using solar chimney is useful in reducing the energy consumption in ventilation of spaces. A Solar chimney fitted in a typical room is used in the numerical simulation. The results are obtained in the form of contours of temperature and velocity to show the natural ventilation establishing in the room because of the temperature difference at the inlet at exit. The results also show that the flow is established in the room due to the solar chimney which is absorbing solar radiations and warming the air.

References

[1] Afonso, Clito; Oliveira, Armando (June 2000). "Solar chimneys: Simulation and experiment". Energy and Buildings. IOP Publishing Limited. 32 (1): 71–79.

[2] Bansal, N.K., Mathur, J. andBhandari, M.S. (1993),"Solar chimney for enhanced stack ventilation", Building and Environment, Vol. 28 No. 3, pp. 373-377.

[3] Bassiouny, R. and Korah, N.S.A. (2009), "Effect of solar chimney inclination angle on space flow pattern and ventilation rate", Energy and Buildings, Vol. 41 No. 2, pp. 190-196.

[4] Trombe, A.; Serres, L. (1994). "Air-earth exchanger study in real site experimentation and simulation". Energy and Buildings. 21 (2): 155–162.

[5] Gnielinski, V., 1976. "New equations for heat and mass transfer in turbulent pipe and channel flow". International chemical engineering, vol. 16, pp. 359-368.

[6] Shao, L., S.B. Riffat, and G. Gan, Heat recovery with low pressure loss for natural veltilation. Energy and Buildings, 1998. 28 (2): p. 179 – 184.

[7] Golder K. (2003), Combined solar pond and solar chimney. Final year Mechanical Engineering Project. School of Aerospace, Mechanical and Manufacturing Engineering, Bundoora Campus, RMIT University, Melbourne, Australia.

[8] Pearlmutter, D.; E. Erell; Y. Etzion; I. A. Meir; H. Di (March 1996). "Refining the use of evaporation in an experimental down-draft cool tower". Energy and Buildings. Elsevier. 23 (3): 191–197

[9] Chen, R.Y., 1973. "Flow in the entrance region at low Reynolds numbers". Journal of Fluids Engineering, vol. 95, pp. 153-158.