A Numerical experimentation on fluid flow and heat transfer in a SCPP

D. K. Mandal\textsuperscript{1}, P. Goswami\textsuperscript{2}, S. Pradhan\textsuperscript{3}, R. Chakraborty\textsuperscript{4}, N. A. Khan\textsuperscript{5} and P. Bose\textsuperscript{6}

\textsuperscript{1,3,4,5,6} Dept. of Mechanical Engg., College of Engg. & Management, Kolaghat, P.O: K.T.P.P. Township, Midnapore (E) - 721 171, West Bengal, India, \textsuperscript{2} WBPDCL, KTPP, Midnapore (E) - 721 171, West Bengal, India

Corresponding author, dipkuma@yahoo.com

Abstract. Solar chimney power plant (SCPP) is a promising unit of power generation from renewable energy source in near future. Apart from climatic conditions, the performance of the unit depends on design parameters of plant. To understand the performance of the unit, fluid flow and heat transfer study is essential throughout the unit. In present work, a numerical study has been conducted to see the effect of inlet area on the performance of the unit through fluid flow and heat transfer. Numerical solutions of the steady turbulent flow are carried out by commercial software Fluent. A finite volume method has been employed to solve the governing equations. It is revealed that the considered plant having inlet height of 5cm performs better than that of having the inlet height 10cm.

Keywords: SCPP, heat flux, Natural convection, velocity

Nomenclature

\begin{align*}
A_i & \quad \text{Inlet area, m}^2 \\
A_{ch} & \quad \text{Area of chimney, m}^2 \\
C_p & \quad \text{Specific heat of air, kJ/kgK} \\
D_{ch} & \quad \text{Chimney outlet dia, m} \\
D_{col} & \quad \text{Collector dia, m} \\
H_{col} & \quad \text{height of collector from ground surface at entry, m} \\
H_{2col} & \quad \text{height of collector from ground surface at middle, m} \\
H_{ch} & \quad \text{Chimney height, m} \\
G & \quad \text{Amount of solar radiation absorbed, w/m}^2 \\
r & \quad \text{Radius of absorber plate, m} \\
T_a & \quad \text{Ambient temperature, K} \\
\eta_t & \quad \text{Turbine efficiency} \\
\eta_{coll} & \quad \text{Collector efficiency} \\
V_{ch} & \quad \text{Velocity through chimney, m/s}
\end{align*}
1 Introduction
Continuous increase in population and globalization causes huge demand within human beings that resulted in exploitation of natural resources, oil fields. Climate Change has become a burning question throughout the world and researchers around the globe are finding out ways to adopt alternative sources energy to meet the daily needs. Solar energy has become an alternative source to satisfy rising demands of huge power and electricity.

The SCPP model possesses three main units, a large solar collector made of greenhouse plastic, a chimney of long length built of PVC pipe, a generator run by a pressure-based wind turbine, converting potential and kinetic energy into mechanical energy. Convection mode heats the air above the bottom wall (absorber) in the ground and hot air is forced to rise through the chimney due to buoyancy. Increase in temperature and decrease in density of air is funneled towards the tower. Wind turbine present at the bottom of the tower converts energy of the air flow into mechanical energy and in turn converts into electrical energy with the aid of electric generators. Design optimization is possible by varying the diameter and height of chimney, area of absorber plate and inlet height. Solar radiation and incidence angle have a huge impact on quantity of reflected, absorbed and transmitted energy. Design and development of efficient solar chimney power project with appropriate material, shape and size in the working setup will be a great application of knowledge in the field of fluid thermal sciences.

A number of studies and research on Solar Chimney Power Plant (SCPP) have been successfully completed to increase the efficiency of the chimney. Buoyancy-nature of heated air has been utilized by Saini et al. [1] to generate solar energy by discarding the use of solar panels. They have done simulation using ANSYS Fluent to calculate heat transfer during night using Roseland radiation model. The maximum chimney height for convection by neglecting negative buoyancy has been determined by Zhou et al. [2]. Desired height for highest power output has been analyzed and compared with the measurements of the model in Manzanares. Thermal analysis for laboratory type 2-D SCPP and discrete ordinance (DO) for power generation have been studied by Lal et al. [3] in a warm climate of Kota, India and compared with experimental results. Bejalwar and Belkhode [4] have constructed a SCPP on the hill top which occupies an area of near about 23 sq. m² and along with it 2mm thickness UV reinforced plastic, crystalline plastic and polycarbonate have been installed. Rabehi et al. [5] have constructed a Solar Chimney Power Plant (SCPP) in Algeria. Numerical study has been done in ANSYS Fluent on a two- dimensional axisymmetric model of SCPP using standard k-ε turbulence model in double precision. During simulation pressure-velocity coupling have been done by SIMPLE Arithmetic and QUICK scheme to analyze the convection terms. CFD analysis on Solar Chimney Power Plant has been done by Gholamalizadeh [6] using the k-ε turbulence and radiative transfer equations. During CFD analysis of buoyancy driven flow and heat transfer air velocity has been clearly viewed about 8.9 m/s and during turbine operation the velocity is 25% lower than no-load condition as turbine operation produces a drag against air-flow. Fasel et al. [7] have carried out numerical analysis using ANSYS Fluent and analytical steps using CFD code on a different range of tower heights in SCPP. Wide time-dependent simulations of flow have given an in-depth idea about fluid dynamics and heat transfer phenomena. A SCPP with a new idea of chimney with a hyperbolic shape was developed by Nasaroui et al. [8] and did analysis experimentally and numerically for the better performance of the hyperbolic chimney. They have shown that there is a rise in velocity in chimney inlet and decrease in the outlet due to a recirculation zone in the outlet. Xu and Zhou [9] have carried simulation in divergent chimney solar power plants (DSPPs) by varying chimney outlet- inlet area ratios (COAR) using the realizable k-ε model in 3-D model. The authors have applied the model of buoyancy- driven flow where density gap is not large and analyzed the streamline vortices, curvature and rotation.

Brief review gives parametric effects in the SCPP unit. Mostly they have varied the design parameters at different climate conditions to discuss the performance of SCPP. The physics of heat and fluid flow study is not addressed by any researchers so far. Inlet height may be one of the parameter to influence the performance of SCPP. Present work focuses of details insight of fluid flow
and heat transfer study to focus on the velocity generated which in turn will drive the turbine to generate electricity. Inlet height is also varied in our simulation.

2 Numerical Modelling

2.1 Physical Prototype

The main components with dimensions of the solar chimney power project (SCPP) are depicted in Fig.1.

2.2 Numerical procedure

The mesh generation is done by Fluent commercial software, tetrahedron elements with 219803 nodes 1141776 elements is taken after grid sensitivity test. The following Fig. 2 shows the mesh of the geometry.

Numerical simulation is done by using Fluent with finite volume method to discretize the Navier-Stokes equations. SIMPLE algorithm is used with the pressure–velocity coupling, convective terms are discretized by 2nd order upwind scheme. All numerical calculations are done in double precision.
The iteration error for momentum is $10^{-5}$ for all calculations and $10^{-8}$ for the energy equations. Standard k-ε turbulence model is taken in this flow study. The Boussinesq model is chosen in this simulation due to less change in temperature in our study. Ambient temperature is taken as 300K and heat flux at absorber plate is taken as 600W/m$^2$ respectively considering sunny conditions.

2.3 Boundary conditions

The boundary conditions for our numerical simulation are given below.

1. Inlet: Pressure inlet, gauge pressure is zero, ambient temperature is 300K.
2. Collector (transparent cover): Heat transfer coefficient is 10W/m$^2$K, ambient temperature is 300K.
3. Ground surface (absorber plate): Heat flux is taken as 600W/m$^2$.
4. Outlet: Pressure outlet boundary, gauge pressure is zero, ambient temperature is 300K.
5. Chimney wall: Adiabatic wall.

To make the problem simple the following assumptions are considered:

i) Air flow is chosen as incompressible flow.
ii) Radiative heat transfer is negligible.
iii) Steady State is assumed in case of the mathematical model.
iv) The Boussinesq model is considered.

2.4 Turbulence modelling

The continuity equation, Navier-Stokes equations, energy equation, and k-ε equations are shown below:

**Continuity Equation**

$$\frac{1}{r} \frac{\partial (ru_r)}{\partial r} + \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

**Momentum Equation**

$$\rho \left( u_r \frac{\partial u_r}{\partial r} + \frac{u_r}{r} \frac{\partial u_r}{\partial \theta} + u_z \frac{\partial u_r}{\partial z} - \frac{u_r^2}{r} \right) = \rho g_r - \frac{1}{r} \frac{\partial p}{\partial r} + \mu \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_r}{\partial r} + \frac{1}{r^2} \frac{\partial u_r}{\partial \theta^2} + \frac{\partial^2 u_r}{\partial z^2} - 2 \frac{\partial u_r}{\partial \theta} \frac{u_r}{r} \right) \right]$$

$$\rho \left( u_\theta \frac{\partial u_\theta}{\partial r} + \frac{u_\theta}{r} \frac{\partial u_\theta}{\partial \theta} + u_z \frac{\partial u_\theta}{\partial z} - \frac{u_\theta^2}{r} \right) = \rho g_\theta - \frac{1}{r} \frac{\partial p}{\partial \theta} + \mu \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_\theta}{\partial r} + \frac{1}{r^2} \frac{\partial u_\theta}{\partial \theta^2} + \frac{\partial^2 u_\theta}{\partial z^2} - 2 \frac{\partial u_\theta}{\partial \theta} \frac{u_\theta}{r} \right) \right]$$

$$\rho \left( u_z \frac{\partial u_z}{\partial r} + \frac{u_z}{r} \frac{\partial u_z}{\partial \theta} + u_z \frac{\partial u_z}{\partial z} \right) = \rho g_z - \frac{1}{r} \frac{\partial p}{\partial z} + \mu \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_z}{\partial r} + \frac{1}{r^2} \frac{\partial u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2} \right) \right]$$

**Energy Equation**

$$u_r \frac{\partial T}{\partial r} + \frac{u_\theta}{r} \frac{\partial T}{\partial \theta} + u_z \frac{\partial T}{\partial z} = \frac{q_s}{c_p} + \alpha \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} + \frac{1}{r^2} \frac{\partial T}{\partial \theta^2} + \frac{\partial^2 T}{\partial z^2} \right) \right] + \frac{\varphi}{\rho C_p}$$

Where $\varphi$=viscous dissipation rate

K-ε turbulence model based on computations of kinetic energy and dissipation rates
\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu_s + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu_s + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\mu \varepsilon} \frac{\varepsilon}{k} (G_k + G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon
\]

\(G_k\) is the kinetic energy generation because of turbulence and mean velocity gradients, and \(G_b\) the kinetic energy generation due to turbulence and buoyancy. \(\sigma_p, \sigma_k\) and \(\sigma_\varepsilon\) are the turbulent Prandtl numbers for \(T, k\), and \(\varepsilon\) respectively (\(\sigma_p = 0.9, \sigma_k = 1.0, \sigma_\varepsilon = 1.3\)). \(C_1\) and \(C_2\) are two constants for turbulent model: \(C_1 = 1.44\), \(C_2 = 1.92\) and \(C_\mu = 0.09\).

There are many factors which affect the electric power generation, geometrical configurations like, collector area, chimney diameter and height, climatic conditions like amount of solar radiation absorbed and ambient temperature. The expression for electric power generated by the plant is obtained by Sangi [10]

\[P_e = \frac{2}{3} \eta_{coll} h_i \frac{g}{C_p T_u} H_0 \pi^2 G\]

Efficiency of the turbine is considered in between of 80% to 90%.

Collector efficiency can be obtained from

\[\eta_{coll} = \rho_{air} V_{ch} A_h C_p \Delta T \quad \pi^2 G\]

3 Results and Discussion

The phenomenon in solar chimney power plant (SCPP) may be discussed in such a way that air in contact to bottom wall (absorber) is heated up by the heat flux of absorber plate. The heat transfer to air takes place mainly due to convection force. Initially there is no flow in SCPP. At inlet and outlet, pressure is atmospheric and temperature is 300K. The temperature of air above absorber plate increases and density of air drops. Due to this, air starts to move upward the plant for natural convection and finally lighter air finds easy way to go out through chimney. Hot air in chimney develops negative pressure inside the chimney. So, air flow begins through inlet due to pressure difference and natural convection created between inlet and outlet of the plant. However, this phenomenon in SCPP behaves similar to forced convection although there is no external equipment. Since, the main physics for the performance of SCPP is heat transfer and fluid flow through the system, therefore, an attempt has been made to study the flow and heat transfer between transparent plate and absorber plate in SCPP. This may definitely help for designing and achieving the better performance of SCPP. Present study gives insight of variation of velocity and temperature in between the transparent plate and absorber plate of SCPP. Apart from variation of velocity and temperature at different sections, variation of mean temperature and mean velocity along the flow direction is also presented in this study. During numerical investigation, the parameters considered are as follows: inlet height is 10 cm and 5 cm; absorber radius is 125 cm; height of the chimney is 300 cm; radius of chimney is 5 cm; heat flux of absorber plate is 600 W/m².

Figure 3 shows the velocity profiles at different sections between absorber and transparent plate from inlet towards the centre of absorber plate. In the figures, notation H0 indicates inlet, H20 represents 20 cm from inlet towards the centre of absorber plate. Similarly H40, H60, H80, H100 and H120 are 40 cm, 60 cm, 80 cm, 100 cm and 120 cm distance from inlet respectively. Inlet velocity profile shows plug type turbulent flow. When the flow moves from inlet to the centre of chimney over the absorber plate, the middle zone of velocity profile is observed to become more flatten. The magnitude of velocity in the core section also increases. This increase in velocity is due to flow over heated bottom wall. Initiation of velocity begins at inlet due to negative draft created by chimney; wall
heat flux at absorber plate aggravates the velocity. The density of air in contact to bottom wall decreases and thereby increases specific volume. Therefore, the velocity of air rises. The asymmetric shaped profile in each case is noticed. It may be due to one side wall heat flux and density difference. Velocity profile indicates that near the centre of chimney (i.e., at H120) magnitude of velocity suddenly increases towards upper zone because of change of flow direction and phenomenon related to flow through sudden contraction. Area of flow increases from inlet to the centre over the absorber plate as the transparent plate remains at an angle. This increase in area decreases the velocity to satisfy the continuity. At the same time, velocity of air increases due to wall heat flux. The rise of velocity dominates over the decrease in velocity which finally leads to increase in velocity. The inclination in the transparent cover is allowed for smooth flow. Smooth joint between transparent cover and chimney is also provided for reduction in losses at the said joint. From the above discussion, it can be said that the average velocity should increase along the flow direction from inlet towards the centre of the chimney which has been obtained during investigation and presented in Fig. 4.

Fig. 3. Velocity profiles along the flow directions at different cross sections over absorber plate at 10 cm inlet height.

Fig. 4. Variation of average velocity along the flow directions at different cross sections over absorber plate at 10 cm inlet height.
From the temperature profiles (Fig. 5) at similar cross sections used during numerical experimentation on velocity profiles, it is evident that wall heat flux diffuses the heat transfer and increases the wall temperature from the inlet. This temperature increases along the horizontal direction of absorber plate from inlet to centre of plate. Above the thermal boundary thickness, the temperature also increases along the flow direction. This is common to attain more diffusion to the bulk fluid with the flow on absorber plate. The rate of increase in wall temperature and average temperature are shown in Fig. 6 and Fig. 7 respectively. This helps to generate velocity by buoyancy force.

**Fig. 5.** Temperature profiles along the flow directions at different cross sections over absorber plate at 10 cm inlet height.

**Fig. 6.** Variation of temperature over absorber plate along the flow directions at 10 cm inlet height.

**Fig. 7.** Variation of average velocity over absorber plate along the flow directions at 10 cm inlet height.
From the review of literature, it is noted that all the researchers have considered diameter of chimney, height of chimney, and absorber plate area as design parameters for assessing the performance of a SCPP. But, the considered design parameters should be those which ultimately dictate fluid flow and heat transfer between transparent plate and absorber plate for the generation of air velocity through the plant. At this juncture, authors feel that effect of inlet height, as another parameter, should also have impact on the performance of the plant, which has not been investigated so far by anybody. Therefore, an attempt has also been made to study the effect of inlet height, by changing the height from 10 cm to 5 cm, on average velocity and average temperature of air along the absorber plate, keeping same inclination in the transparent plate and same wall heat flux in the absorber plate. It is clear that area of flow is more at the inlet of the plant and is less at the outlet of the plant for both the cases. For 1st case of 10 cm inlet height and 5 cm chimney radius, area at outlet becomes 100 times less than that of at inlet; in case of 2nd case of 5 cm inlet height and 5 cm chimney radius, area at outlet becomes 50 times less than that of at inlet. It is also evident that the inlet area of 2nd case becomes 0.5 times less in comparison to 1st case. After simulation for 2nd case, it has been observed that average velocity in case of 2nd case over the plate increases more than the 1st case of having 10 cm inlet height (Fig.8) and mean temperature over the absorber plate is less than that of 1st case (Fig.9).

Fig.8. Variation of average velocity in between transparent cover and absorber plate.

Fig. 9. Variation of average temperature in between transparent cover and absorber plate.

The simulation results of the magnitude of velocity at inlet and at outlet of the plant for two cases have been shown in table 1. Gain in velocity of air depends on velocity due to pressure drop by the chimney and due to the effect of natural convection over the absorber plate to air. In case of 2nd case having 5 cm inlet height, mean temperature is less. It may be due to insufficient space for diffusion. It reduces the chimney draft which consequently decreases the velocity of air in comparison to the 1st case with 10 cm inlet height. At the same time, large temperature difference between absorber plate and air helps to enhance the velocity due to convective force. Gain in velocity also depends on inlet area. By reducing half inlet area (from case 1 to case 2) velocity at inlet increases about 5 times (Table
1). This is due to small inlet area and the dominating convective force over absorber plate. Velocity in 1st case increases approximately 100 times from inlet to outlet, whereas it increases almost 50 times in 2nd case. As the initial velocity increases 5 times for 5 cm inlet height compared to 1st case, therefore outlet velocity increases approximately 2.5 times of outlet velocity of 10 cm inlet height, noted in table.1. Therefore, it may be said that the plant of having 5 cm inlet height performs better.

**Table-1. Velocity at inlet and outlet**

| Case  | Inlet height | Velocity at inlet | Velocity at outlet |
|-------|--------------|-------------------|-------------------|
| Case-1 | 10 cm        | 0.097 m/s         | 9.85 m/s          |
| Case-2 | 5 cm         | 0.4918 m/s        | 24.99 m/s         |

4 Conclusions
Present study is focused to study the flow and heat transfer in solar chimney power plant (SCPP). Inlet height is varied from 10 cm to 5 cm. Comparison is made with mean velocity and mean temperature in each case. Lowering inlet height increases the outlet velocity thereby increases the power generation by SCPP. Further study is needed to identify the optimum height at inlet for achieving maximum power generation in a SCPP.

Acknowledgment
The authors are thankful to The Institution of Engineers (India) for their financial support under R&D grant-in-aid scheme to complete, (Ref. R.4/2/UG/2019-20/UG2020036).

5 References
[1] Saini D L, Singh A and Bhargava M 2017 Design, optimization and performance analysis of Solar chimney using CFD technique *Journal of Emerging Technologies and Innovative Research (JETIR)* 4(10):52-55.
[2] Zhou X, Yang J, Xiao B, Hou G and Xing F 2009 Analysis of chimney height for Solar chimney power plant *Applied Thermal Engineering* 29:178-185.
[3] Lal S, Kaushik S C and Hans R 2016 Experimental investigation and CFD simulation studies of a laboratory scale Solar chimney for power generation *Sustainable Energy Technologies and Assessments* 13:13-22.
[4] Bejalwar A and Belkhode P 2018 Analysis of experimental setup of a small Solar chimney power plant *Procedia Manufacturing* 20: 481-486.
[5] Rabehi R, Chaker A, Aouachria Z and Tingzhen M 2017 CFD analysis on the performance of a Solar chimney power plant system: case study in Algeria *International Journal of Green Energy* 14(12): 971-982.
[6] Gholamalizadeh E 2018 CFD simulation of a Solar chimney power plant* International Journal of Mechanical and Production Engineering* 6(4):1-5.
[7] Fasel H F, Meng F, Shams E and Gross A 2013 CFD analysis of solar chimney power plants, *Solar Energy* 98:12-22.
[8] Nasraoui H, Driss Z, Ayedi A and Kchaou H 2019 Numerical and experimental study of the aerothermal characteristics in Solar chimney power plant with hyperbolic chimney shape *Arabian Journal for Science and Engineering* 44(9):7491-7504.
[9] Xu Y and Zhou X 2018 Performance of divergent-chimney Solar power plants *Solar Energy* 170:379-387.
[10] Sangi R 2012 Performance evaluation of solar chimney power plants in Iran *Renewable and Sustainable Energy Reviews* 16 (1):704-710.