NUMERICAL ANALYSIS OF THE AIRFLOW INSIDE A SOLAR CHIMNEY

T. Chergui¹, M. Lebbi¹, A.H. Boualit, L. Boutina and A. Bouabdallah²

¹Unité de Recherche Appliquée en Energies Renouvelables, URAER, Centre de Développement des Energies Renouvelables, CDER, CDER, 47133, Ghardaïa, Algeria.
²LSTE Université des Sciences et de la technologie (USTHB), Alger, Algeria
tchergui@uraer.dz

Abstract: The purpose of this paper is to analysis the airflow thermo-hydrodynamic behavior in solar chimney device according to Rayleigh number variation. We attempt to improve the quality of the flow field by controlling the appearance of Rayleigh-Benard-Poiseuille vortices in the airflow initially stratified and uniform. These vortices represent dissipative phenomena by the fact that they represent fictitious obstacles and reducing significantly the massflow by reduces the air flow cross-section. The mathematical model (Naviers-Stokes and Energy Equations) was analyzed by the Finite volume Method for a series of Rayleigh numbers (Ra) reaching values up to $10^8$. Structured grid has been employed with increasing density to guarantee accurate solutions; especially near the walls for high Ra-values. This methodology allows a detailed visualization of the effects of optimal geometric and operational characteristics for such devices. Excellent agreement is obtained with previous numerical solutions.

Keywords: solar Chimney; finites volume method; free convection; Rayleigh number; Rayleigh-Benard; Poiseuille

1. INTRODUCTION

Technical solar chimneys use solar radiation for heating air under the transparent roof of the collector. Once heated, the air flows, by density gradient through the tower-chimney placed in the center of the collector. The solar energy captured by the collector is transformed into kinetic energy. The kinetic energy of the flowing air can be converted to electrical energy using wind turbines. We are talking about the Solar Chimney Power Plant (SCPP) (see Figure 1). Technical Solar Chimneys (SC) may serve in several disciplines as an flow generators, such as in air conditioning and drying. Starting with Cabanyes (1903) and Günter (1931), several projects on CCS have been communicated across the world until the completion of the first prototype in the Spanish province Manzanares (Haaf, 1983; Haaf et al. 1984). The design of this prototype was done by the consulting firm of Schlaich and Partner (Schlaich, 1994).

Since then, several studies on the evaluation of overall performance (efficiency and power output) CCS plants have been undertaken (Schlaich, 1994; Gannon and Backström, 2000; Bernardes 2003). For CFD studies who are interested in local characteristics of flow through these systems we quote the work of Pastohr et al. (2004) who used the FLUENT software (commercial code of C.F.D.) to model a SCPP power plant geometrically similar to that of Mansaranes with an aim of carrying out an analysis with more detail in the description of the operating mode and the system efficiency. Ming et al. (2008) analyzed by a numerical simulation the impact of several sizes of three different chimney configurations upon the chimney outlet air temperature and velocity, system output power and efficiency. The effect of the height-to-diameter ratio (H/D) of the cylindrical chimney on system performance was
studied as well. The authors proved that the cylindrical chimney would be the best choice among the three basic configurations, whose optimum H/D value ranges from 6 to 8.

Tingzhen et al. (2008) presented a numerical simulation method of a CSS considering a wind turbine. Based on the results obtained for the Manzanares prototype with a three-bladed wind they have proved that the increase of the rotational speed of the turbine drops the average speed at the exit of the tower and the mass flow rate, then the average temperature at the exit of the tower and the pressure drop in the turbine is increased as the maximum power available, the power output and performance of each reach a maximum value. Zhou et al. (2009) performed a simulation of the solar chimney power equipment with a self-constructed code and compared the simulated results with experimental measurements. They concluded that the airflow temperature basically increases from the opening to the collector centre since that airflow is gradually heated when forcibly driven by the thermal engine, i.e. solar chimney. However, the maximum of the mean temperatures along a collector radius is usually at a distance of 0.5 to 3-5 m away from the collector centre. As analyzed from the flow field, fluid beside the ground keeps flowing vertically to the roof and turns to the centre. Also, the circumfluence exists next the collector roof. Maia et al. (2009) performed an analytical and numerical study of the unsteady airflow inside a solar chimney. The developed model was used for airflow simulation in solar chimneys with operational and geometric configurations different from those found in the experimental prototype. Based on simple mathematical models, Larbi et al. (2010) presented an analysis of the energy performance of a SCPP designed to deliver electrical energy to isolated villages in the Algerian southwestern.

Chergui et al. (2010) developed a finite volume method code to simulate a thermo-hydrodynamic behavior of the airflow through an axisymmetric system, such as solar chimneys, with defined boundary conditions. They show for different Rayleigh numbers the dimensionless isothermal and velocity lines for two rapport of the height collector at entrance and tower height (e/H = 0.01 and e/H = 0.1). They notice that the velocity magnitude increases with Rayleigh numbers and its maximum is located approximately at the inlet of the chimney. Chergui et al. (2011) determined the temperature distribution and the velocity field in the system by solving the energy and momentum equations using the finite element method. The concept of entropy generation minimization was investigated for an optimization purpose in order to look for the optimal geometrical configuration. Chergui et al. (2012)
predicted numerically local and global entropy generation rates in natural air convection through solar chimney heated at uniform heat flux are reported. Results of entropy generation analysis are obtained by solving the entropy generation equation based on the velocity and temperature data. Cao et al. (2013) developed a program based on TRNSYS to evaluate the performance of CCS. They checked the influence of meteorological parameters on the performance of CCS and therefore they could perform a technical and economic analysis on the same commercial CCS for sites that are not included in the TRNSYS database. Lebbi et al. (2014) analyzed numerically the influence of some geometric parameters on the thermal hydrodynamic behavior of the airflow in the solar chimneys. The results obtained shows that the solar tower dimensions allow a direct control of the hydrodynamic field via the flow velocity. Lebbi et al. (2015) analyzed the effects of the tower outlet / inlet radii ratio on the flow performances in a solar chimney. They investigate the optimal point which presents the mass flow rate. They conclude that the radii ratio variation has an optimum value that should not be exceeded. Recently Chergui et al. (2015) were particularly interested in the effect of the shape of some parts of solar chimney devise like the shape of the cover-tower junction, or the convergent-divergent tronconical tower. For this purpose the flow transport equations were modeled and solved numerically for different geometrical shapes using the Finite Volume Method in Generalized Coordinates. We find that a few geometric configurations generated flow perturbations, which reflected on the thermo-hydrodynamic behaviors. Geometrical shape modifications can eliminate this dissipative perturbation. As it has been shown that the most important solar chimney dimensions in the hydrodynamic field control was the tower size, we gave special importance to the tower shape. We tested the influence of the tronconic convergent and divergent tower respecting the flow cross section. The divergence shape increase the mass flow rate but for a certain divergence angle the mass flow drop. The convergent tower shape gives inverse results by decreasing mass flow but the velocity at the outlet is greater due to the contract flow cross section.

II. MATHEMATICAL MODEL

II.1 Physical Problem and Boundary Conditions

The physical problem and its boundary conditions are prescribed in Figure 2. Since the flow is produced by buoyancy forces then air enters in the collector with zero axial velocity, and an unknown radial velocity to calculate using a mass balance at each iteration until convergence of results. At the outlet of the tower, it is assumed that the flow is fully developed. Since an axisymmetric flow is assumed.

![Figure 2. Study domain and boundary conditions.](image-url)
At the walls, the no-slip and impermeable walls were considered. These conditions were applied to the cover at the junction, to the walls of the tower and to the ground surface. However, the air enters the collector at the ambient temperature. The walls of the tower and junction are assumed adiabatic. The transparent collector roof temperature and the soil temperature, considered as the system hot temperature, were calculated by heat balance based on the incident radiation.

II.2 Governing Equations

The airflow governing equations in a solar chimney are: the continuity equation, the equations of momentum, the energy equation in addition to two other transport equations necessary to calculate the turbulent quantities as the eddy viscosity contained in the equations of the mean flow. For the turbulent calculations, the standard model of Launder and Spalding (1974) with wall functions was selected. In this model, the turbulent viscosity is evaluated from the turbulent kinetic energy \( \kappa \) and its dissipation \( \varepsilon \):

\[
\mu_t = \rho c \mu f \frac{\kappa^2}{\varepsilon}
\]

The low governing equations are then (Pasthor et al. 2003)

- The continuity equation

\[
\frac{\partial}{\partial x_i}(\rho u_i) = 0
\]

- The momentum equations

\[
\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i
\]

- The Energy Equation

\[
\frac{\partial}{\partial x_j}(\rho c_p u_i T) = \frac{\partial}{\partial x_j}\left(\frac{k}{\kappa} \frac{\partial T}{\partial x_i}\right) + \tau_{ij} \frac{\partial u_i}{\partial x_j} + \beta T \left(u_j \frac{\partial p}{\partial x_j}\right)
\]

- Turbulent Kinetic Energy Equation (\( \kappa \)):

\[
\frac{\partial}{\partial x_j}(\rho u_i \kappa) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial \kappa}{\partial x_i}\right] + G_k + \beta g_i \frac{\mu_t}{Pr_t} - \rho \varepsilon
\]

- Dissipation of Kinetic Energy Equation (\( \varepsilon \)):

\[
\frac{\partial}{\partial x_j}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma\varepsilon}\right) \frac{\partial \varepsilon}{\partial x_i}\right] + C_1 \varepsilon \frac{\kappa}{\kappa} \left(G_k + C_{3\varepsilon} \beta g_i \frac{\mu_t}{Pr_t} \frac{T}{\partial x_i}\right) - C_2 \varepsilon \frac{\varepsilon^2}{\kappa}
\]

III. Numerical Approach

The governing equations are discretized in general two-dimensional curvilinear coordinate system by the finite volume procedure. Non-uniform grid sizes with more significant concentration on the inlet-outlet and junction regions were used. A \( 35 \times 380 = 13300 \) cells (see figure 3) was adopted for grid-free solution throughout the calculations in the
present study since it has shown a negligible deviation in the respective solutions. In order to validated our numerical methodology we presented on figure 4 two temperature profiles in a cross-section of the collector, for the non-dimensional radius of \( r/R_c = 0.15 \). Those temperature profiles are related to the present and the Maia et al. (2009) studies.

**Figure 3.** Mesh of the cavity (35 x 380 cells)

**Figure 4.** Temperature profiles in a cross-section of the collector (*) this study (+) study Reported by Maia et al. (2009).

**IV. RESULTS AND DISCUSSION**

In order to study the main thermal and hydrodynamic characteristics of the airflow and to evaluate the quality of our numerical methodology, the basic values of geometric parameters adopted in the simulation were given by the following values: collector radius (\( R_c = 12.5 \) m); height of the cover at the entrance (\( H_{C2} = 0.05 \) m); height of the cover (\( H_{C1} = 0.5 \) m); chimney radius (\( R_t = 5 \) m); height of the chimney (\( H_t = 12.3 \) m). In this part of the study,
the numerical simulation of the solar chimney is presented. For a preliminary qualitative analysis we illustrate in figure 7 (a-c) the stream function distributions in the solar chimney for different thermal Rayleigh number which take value from $10^4$ to $10^8$. We can observe the appearance of the vortices of Rayleigh-Benard-Poiseuille instability for an $\text{Ra}_c$, before they disappear. By varying some important flow parameters listed in the Rayleigh number, we can delay or advance the appearance of the Rayleigh-Benard-Poiseuille vortices.

![Figure 7. Stream function lines vs. Rayleigh number](image)

Figures 8 shows, the evolution of the temperature along the collector at $y = 0.025$ m for different Rayleigh number. We note that from Rayleigh $\text{Ra} = 10^7$ the velocity increases rapidly at the outlet of the collector. It is observed that the average temperature of the flow increases with the increase of Rayleigh number, reflecting the increase of the temperature gradient and therefore the increase in temperature of the heat source.
Figure 8. Influence of the Rayleigh number on the temperature evolution along the collector, $y=0.025m$.

Figure 9. Effects of the Rayleigh number on the mass flow rate.
For each Rayleigh number, we observe that a relatively elevated velocity at the entrance of the collector decreases going towards the center of the system until it has stabilized before starting to grow to reach his second maximum. This is due to the variation of the collector flow passage section towards the center. As may be seen that the velocity in the collector increases with the increase of Rayleigh reflecting an increase in driving force which is the temperature gradient. The progressive increasing of the Rayleigh number is accompanied by the appearance of Rayleigh-Bénard-Poiseuille instability of beyond a certain thresholds critical value with \(R_{ac} = 5.8 \times 10^4\). These vortices created by the temperature gradient between the ground and collector continued to appear until its complete disappearance from \(Ra = 10^5\).

The evolution of the mass flow rate as function of Rayleigh number is plotted in Figure 9. Before taking an asymptotic form as a result of hydrodynamic equilibrium between the driving buoyancy forces resistive viscous forces, the produced mass flow rate increases proportionally with the Rayleigh number values until \(Ra = 10^5\). Which is a result of the superiority of the buoyant forces on the viscous forces. This superiority is sensed in the Rayleigh number increasing

V. Conclusion

In this work, a two-dimensional numerical study has been investigated using \(\kappa – \varepsilon\) turbulence model. The laminar and turbulent flows of free convection via Rayleigh number are analyzed. The progressive increasing of the Rayleigh number is accompanied by the appearance of Rayleigh-Bénard-Poiseuille instability in the collector beyond a certain critical value of the Rayleigh number. both transversal and longitudinal convection rolls are identified in the collector, indicating the presence of a Rayleigh-Bénard-Poiseuille instability.

Thereby we can delay or advance the appearance of the Rayleigh-Benard-Poiseuille vortices by varying some important flow parameters listed in the Rayleigh number. As a result, the quality of the flow field can be improved by controlling the manifestation of these dissipative flow field instabilities what is very important for some solar device working in a specific interval of velocities.

VI. References