Numerical investigation on heat transfer characteristics amelioration of a solar chimney power plant through passive flow control approach

Ehsan Shabahang Nia, M. Ghazikhani *

Department of Mechanical Engineering, School of Engineering, Ferdowsi University of Mashhad, P.O. Box No. 91775-1111, Mashhad, Iran

A R T I C L E I N F O

Article history:
Received 27 April 2015
Accepted 10 July 2015

Keywords:
Solar chimney power plants
CFD
Passive flow control

A B S T R A C T

In this study, potential improvements in flow field and heat transfer characteristics of a prototype solar chimney power plant through passive flow control approaches is numerically investigated. The simulations are conducted through a 2D axisymmetric incompressible steady computational fluid dynamics solver and grids with analogous characteristics are utilized for analysis of three different flow control obstacles. Analogous ameliorations in heat transfer characteristics along the absorber surface and velocity magnitude at the entrance of the chimney are obtained when flow control devices are implemented. Emergence of vorticities and fluid mixing at the downwind of the obstacles, agitations in thermal boundary layer thickening and developing, and flow pattern guidance are deemed as the three major mechanisms resulting in improved heat transfer characteristics and increased velocity magnitudes. Flow control is believed to be an expeditious tool in efficiency improvement in solar chimneys.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

The significant growth in the depletion rate of the fossil fuel sources along with the environmental predicaments inflicted by uncontrolled deployment of these sources around the world, have initiated a global movement toward more sustainable and cleaner energy sources [1]. Solar energy is arguably the most abundant within the renewable energy sources while the others usually encounter obstacles such as discontinuous availability, and also has started to provide a considerable amount of energy for various energy consuming sections [2]. The solar chimney power plant (SCPP) is a simple solar thermal power plant which is capable of converting the solar energy into electricity. A SCPP is composed of three main components: collectors, chimney and turbine. Based on the greenhouse effect, the absorbed solar energy warms the air below the transparent collector roof; the warmed air climbs up at the chimney due to density reduction and buoyancy effect, which leads to conversion of the thermal energy to its kinetic form. Employing of a wind turbo-generator on the entrance of chimney would lead to electricity generation which can be supplied to different consumers [2,3]. Solar chimney is a relatively new method to produce electrical power and has been enhanced and studied in recent years, Wibing et al. [4] developed a novel model to analyse costs and benefits of reinforced concrete solar chimneys, Ghorbani et al. [5] achieved a concept design to improve Rankine cycle efficiency through integration with a solar chimney, Suarez-Lopez et al. [6] numerically investigated the application of solar chimney concept in building ventilation, Peng-Hua et al. [7] evaluated the annual performance of a case study solar chimney in China, Fei et al. [8] assessed the design characteristics of a combined geothermal–solar chimney power plant, Fei et al. [9] carried out simulations for a sloped solar chimney power plant in China, Al-Kayiem et al. [10] conducted mathematical analysis of the influence of geometrical characteristics of a roof top solar chimney on its performance, Khanal et al. [11] presented a review on the effects of geometry an inclination angle on the ventilation performance of solar chimney. Obviously, the requirement for efficiency amelioration never ceases to increase since it would supply the utilizers with fewer expenses and would substantiate the technology as a more viable option.

Remarkable research efforts have been dedicated to experimental and numerical studies of the fluid procedure governing the energy conversion phenomena in SCPPs, which have contributed to a more profound understanding of the occurring thermal and dynamic processes. Fasel et al. [12] conducted a CFD analysis for solar chimney power plants; focusing on scale comparisons, detailed resolving of a particular case, and thermal instabilities assessing, Shahreza and Imani [13] utilized both experimental and numerical approaches in order to investigate an innovative solar chimney; the study included utilizing intensifiers in order
to increase the heat flux in the system along with a detailed rotational pattern investigation. Geometrical parameters of a solar chimney were optimized by Kasaeian et al. [14] through a hybrid numerical–analytical method. Patel et al. [15] investigated the effects of geometrical parameters of a SCPP on the performance characteristics through temperature and velocity profile monitoring utilizing a computational approach. Sangi et al. [16] employed two different flow solvers in order to obtain a more detailed numerical simulation of a SCPP which led to good quantitative agreement between numerical and experimental results. A 3D numerical simulation based upon the radiation model is performed by Peng-Hua et al. [17] providing more reasonable results for absorbed energy and turbine pressure drop. The airflow behavior through the chimney was numerically analysed by Lebbi et al. [18] in order to investigate the hydrodynamic field affected by various tower dimensions. The majority of the prior studies in the field consider geometrical characteristics of the SCPP in order to obtain more eligible flow behavior or investigate possibilities in improvements on the thermal properties of the utilized material for collector part to achieve higher rates of heat transfer.

Flow control has been the subject of the major research areas in fluid mechanics for the recent years [19,20]. It offers new solutions for manipulating boundary layer growth, separation, and attachment, mitigation of shock strength, drag minimization [21], vibrations and fluctuations, preventing the flow mixture from blending, stall control [22], and the performance maximization of existing designs to meet the increasing requirements of the aircraft industries [23]. Energy systems dealing with moving fluids possess the potential to deploy this concept in order to enhance their efficiency rates through flow field and heat transfer characteristics betterments. Since SCPPs perform merely based on convective heat transfer and kinetic energy extraction mechanisms, they can be considered as promising energy systems to benefit from flow control ideas, where passive flow control devices (PFCD) can be implemented to desirably alter the flow field and conclusively energy efficiency rates at a reasonable cost.

In this study a prototype SCPP, presented in [24], with experimental data on temperature field along the collector, is numerically investigated to assess the capability of three different PFCDs in improving the flow field and heat transfer features of the system. The first step of the methodology includes numerical simulation of the prototype through a 2D axisymmetric incompressible steady computational fluid dynamics (CFD) solver to concretize the validity of the employed CFD modeling approach; then, by maintaining the same grid characteristics, the aforementioned PFCDs are integrated to the existing geometry of the system to evaluate the resultant flow fields. Illustrating contours and stream lines are provided to clarify the effect of the devices on the temperature and velocity profiles of the understudy SCPP; moreover, comparisons of the influential characteristic between the base case, and the PFCDs are presented to provide a conclusive cognition of the mightiness of implementing flow control concept for performance improvement of SCPPs.

2. Case study description

2.1. Base case

According to [24,25], the solar chimney, the schematic of which is depicted in Fig. 1, was built in University of Zanjan, Zanjan, Iran. Dimensions of this prototype are included in Table 1 [24]. The temperature is sensed at a 10 cm distance above the absorber surface through 5 sensors with 1 m distance between each one. For purpose of measuring air flow velocity, a speedometer propeller is located at the chimney entrance [24]. In order to investigate the performance of the PFCDs the data set belonging to hour of 11:00 in the date 7th September is utilized to evaluate the performance of the CFD approach. Since the height of the collector entrance is no more than 15 centimetres and the effect of probable wind velocities can be neglected in this case the inlet velocity in this region is considered to be zero; in addition environmental temperature is considered equal to the temperature value obtained via the first sensor since it is placed only 25 centimetres after the collector entrance and no significant temperature variation can occur in this span.

2.2. Controlled cases

The flow control mechanism to be utilized for a SCPP has to possess certain characteristics to prevent any disturbance in the energy production process. There are various commonly utilized flow control mechanisms in aerospace industry, these mechanisms generally manipulate the boundary layer in a manner to elude flow separation phenomena; the broadly used methods include suction, injection, and vortex generators. Due to the fact that natural convection phenomena dominates the heat transfer and energy production of a SCPP, the flow control strategy should be capable of enhancing the convective heat transfer rates along the absorber as well as increasing the vertical velocity at the entrance of the chimney.

With all the above being discussed, three different ring shaped PFCDs with rectangular, triangular, and semicircle profiles are considered to be placed in the path of the flow in collector part of the solar chimney. In order to maintain the effect of the mentioned devises, they are implemented in couples along the collector area, Fig. 2 delineate each case. The dimensions of the devises are selected in a way not to entirely cease the momentum of the flow neither to leave the flow field unaffected, the selection is based upon observations from different numerical simulations with various dimensions assigned to devises.

3. Governing equations

3.1. Navire–Stokes equations

The mathematical model for the study of fluid dynamics problems is based on the fundamental mass, momentum and energy
conversation principles. The approach used in this study is called Reynolds Averaged Navier–Stokes (RANS) modeling. These equations govern transport of the averaged flow quantities, and can be used for the entire flow field.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m
\]  

(1)

\[
\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla \cdot p + \nabla \cdot (\bar{\tau}) + \rho \bar{g} + \bar{F}
\]  

(2)

\[
\frac{\partial (\rho H)}{\partial t} + \nabla \cdot (\vec{v}(\rho H + p)) = \nabla \cdot (k_{eff} \nabla T) + S
\]  

(3)

\[
\bar{\tau} = \mu \left[ (\nabla \vec{v} + \nabla \vec{v}^T) - \left( \frac{2\nabla}{3} \right) \cdot \vec{v} \right]
\]  

(4)

In order to predict the natural convection flow properties different turbulence models have been developed. In this study the shear stress transport \( k-\varepsilon \) model is employed, the detailed description of these turbulence models are given in [26].

4. CFD simulations

4.1. Computational domain

Since the geometry and all boundary conditions of the problems are axially symmetric, half of a 2-D model of the power plant is investigated. Fig. 3 shows the 2-D computational domain of the base case, the initial grid is comprised of 219,417 quadratic elements with high resolution grid employed near the collector, the center line, and absorber surface due to the fact that these regions represent great importance in boundary layer and maximum velocity investigation. Note that this initial grid around the collector and PFCDs will undergo adaption (refinement) in the process of the simulation, in order to match the convergence criteria.

4.2. Solving methods

Since the natural convection is the phenomena to assess in this study, the Boussinesq assumption has been deployed, which deems that the difference in inertia is negligible but gravity is sufficiently mighty to make the specific weight appreciably different between the two fluids [27]. The low Reynolds flow inside the solar
chimney is considered as incompressible; as a result, the “pressure-based” method is used. The pressure–velocity coupling algorithm is “Coupled” since it would lead to faster solution convergence; the discretization of the convection terms in the transport equations is selected to be “second order upwind”. A detailed description of these methods can be found in [28].

Two main criteria are majorly considered in order to guarantee the convergence of the solution.

1. Equity of the mass flow rates for the collector and the chimney.
2. Proper wall $y^+$. The first criterion is considered as a substantial factor illustrating solution convergence; since it would guarantee that the probable reverse flows detected at the chimney outlet would not affect the final solution severely. Turbulence wall $y^+$ is a factor of correct boundary layer modeling; which can be employed to maintain analogous grid characteristics for simulation of different cases [29]. When the grid is refined during the calculations to obtain $y^+$ between 1 and 6, for each and every simulation case a distinct grid independency study is conducted, which refines the initial general grid for different geometry to fulfill a coherent $y^+$ requirement.

The boundary conditions considered for base case simulation are as shown in Table 2.

5. Results and discussion

5.1. Base case – validation

As mentioned earlier, simulations with characteristics discussed in the previous section are conducted to enable validation of the numerical approach against the experimental data recorded for the SCPP under study. The experimental data includes temperature values at the points at which the aforementioned sensors are located. In order to validate the numerical approach, the static temperature values in the length of the collector, obtained via CFD is compared against the experimental values, this comparison is depicted in Fig. 4. As can be observed from this figure, temperature values obtained through CFD simulation is in acceptable agreement with the experimental data. The values of the nodes generating this figure are presented in Table 3; the table also includes calculated errors based on both Celsius degrees and Kelvin degrees. The assessed errors show a maximum of 12% difference between numerically and experimentally obtained values which can be considered as sufficient agreement between numerical and experimental results. Furthermore, the reason of temperature decrement perceivable in the last node for experimental data could be traced in the possibility of shade formation during the time of data recording.

In addition, a validation approach deployed in previous studies [30,31] involves utilization of fluid properties due to the idea that these properties will experience variation as the temperature changes along the collector; however, since in this study temperature is the sole numerical parameter available, and the change in temperature is not considered severe the mentioned approach would not be an option.

In order to obtain a perspective of the flow filed in the base case contours of velocity magnitude and static temperature in the solar chimney is depicted in Fig. 5, the observations of which is actually used to select the locations for the PFCDs in order to improve these characteristics more efficiently. Moreover, the reversed flow

![Fig. 3. The initial grid utilized for base case simulation.](image)

![Fig. 4. Plots of temperature distribution along the collector, CFD against experimental data.](image)

<table>
<thead>
<tr>
<th>Place</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Centerline</td>
<td>Axis</td>
<td>Symmetry</td>
</tr>
<tr>
<td>Absorber</td>
<td>Wall</td>
<td>$Q$</td>
</tr>
<tr>
<td>Roof</td>
<td>Wall</td>
<td>$Q = 0$</td>
</tr>
<tr>
<td>Chimney</td>
<td>Wall</td>
<td>$Q = 0$</td>
</tr>
<tr>
<td>Inlet</td>
<td>Pressure inlet</td>
<td>$P_{inm}$</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet</td>
<td>$P_{out}$</td>
</tr>
</tbody>
</table>
detectable in figure is believed to be mainly caused by the diffusive
geometry of the collector and the generated large vortex region
which causes a low pressure region forcing a fraction of the flow
backwards.

5.2. Controlled cases

Grids with $y^+$ characteristics analogous to the grid employed in
base case simulation are deployed to solve the flow fields for the
controlled cases, where $y^+$ requirements are satisfied and captured
within the proper range. Fig. 6 includes plots of this factor along
the length of absorber for the base case and three PFCDs under
study.

Stream lines of the flow along with velocity magnitude of the
controlled cases are presented in Figs. 7–9 to illuminate the effect
of the PFCDs on the behavior of the flow field; as can be observed
from these figures; different shape for the ring obstacle will impose
different manipulation on the air velocity in the solar chimney
collector, for instance it is perceivable that dissimilar vorticities
are shaped in the downwind of each obstacle and the air flow is
guided in an anomalous manner after each PFCD.

As can be perceived from Fig. 7 the heated air in the vicinity of
the absorber tends to move upward due to buoyancy effect; how-
ever, due to higher density of the colder air above (i.e.
Temperature inversion) and also due to continuity this heated air
moves toward the chimney aperture. The shear stress caused by
this movement is believed to be the major reason for forming the
large vortex detectable in the base case. In Fig. 8 employing the
first rectangular obstacle leads to mixing of the cold and heated
air flows which will allow the flow in the vicinity of the absorber
to move upward to some extent and deformation of the large
vortex which appeared in the base case; moreover, a similar large
vortex is still detectable before the first obstacle mainly caused by
the reasons discussed for the base case. The presence of the second
obstacle guides the flow toward the chimney aperture which is
believed to cause the third large vortex and the smaller vortex
region after the obstacle itself. The mechanism for flow patterns
developed in Figs. 9 and 10 are perceived to be analogous to those
discussed for the case with rectangular obstacles, the difference

Table 3
The temperature data for CFD evaluation.

<table>
<thead>
<tr>
<th>Sensor number</th>
<th>CFD (K)</th>
<th>Experimental (K)</th>
<th>Variation value (K)</th>
<th>Error (Celsius based) (%)</th>
<th>Error (Kelvin based) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>298.70</td>
<td>296</td>
<td>2.70</td>
<td>12</td>
<td>0.9</td>
</tr>
<tr>
<td>2</td>
<td>308.94</td>
<td>311</td>
<td>2.06</td>
<td>5.4</td>
<td>0.6</td>
</tr>
<tr>
<td>3</td>
<td>312.56</td>
<td>315.5</td>
<td>2.94</td>
<td>7.0</td>
<td>0.9</td>
</tr>
<tr>
<td>4</td>
<td>317.26</td>
<td>317</td>
<td>0.26</td>
<td>0.5</td>
<td>0.08</td>
</tr>
<tr>
<td>5</td>
<td>320.5</td>
<td>315.5</td>
<td>5</td>
<td>11</td>
<td>1.5</td>
</tr>
</tbody>
</table>

Fig. 5. Contours of velocity magnitude and static pressure in the base case.

Fig. 6. Plots of turbulence wall $y^+$ for the base and controlled cases.

Fig. 7. Stream lines of velocity, base case.
can be traced in different levels of fluid mixing and flow guidance provided by half-circle and triangular obstacles.

Plots of turbulent kinetic energy (TKE) for each case is depicted in Fig. 11, the magnitude of this parameter can be interpreted as the intensity of flow agitation and mixing characteristics throughout the collector. As can be observed from this figure the rectangular PFCDs will cause the minor increase in the TKE; thus, less favorable performance elevation is expected. The half-circle PFCDs possess the topmost TKE increment effect which is located approximately near the chimney entrance region; such a high level of fluid mixing is expected to deteriorate the air velocity condition in the chimney where the energy is intended to be harnessed. The triangular PFCDs have the capability of providing desirable average TKE increment throughout the collector which does not interfere with the original pattern of flow near the chimney aperture region.

Obviously, the objective of the utilization of these devices is to enhance the heat transfer characteristics of the plant, which will consequently lead to higher velocity rates in the entrance of the chimney. Therefore, plots of Nusselt number on the absorber surface for the base case and the controlled ones is delineated in Fig. 12. As can be observed from this figure, amelioration in this deciding factor is achieved when PFCDs are utilized in the solar chimney collector. The first reason can be traced in emergence of the recirculation zones; the idea states that these obstacles can create vortices and secondary flows which lead to higher fluid mixing. In other words, more cold fluid is transported toward the absorber surface which is deemed to have a considerable increasing effect on heat transfer characteristics; the aforementioned vorticities are detectable in Figs. 7–10, and the claimed mechanism for fluid mixing can be detected from the stream lines illustrated in these figures. Moreover; for the base case the heat transfer coefficient tends to decrease along the absorber due to the development of the thermal boundary layer in the region; however, when flow control devices are implemented this boundary layer is agitated by the presences of PFCDs which makes the boundary layer thinner and improves the heat transfer coefficient. Plots of Fig. 12 provide demonstration for this idea, since it is shown that there are sudden increases in Nusselt number as the flow reaches the obstacles, this phenomena contributes to achieving higher average heat transfer rates through the collector which is the influential factor in obtaining higher velocity rates at the entrance of the chimney.

According to the absorber average Nusselt numbers, the obstacle with triangular profile possesses the topmost increment, obstacles with half circle and rectangular profiles present the second and the third best heat transfer characteristics respectively. The reason could be traced in stream lines depicted in Figs. 7–10, and turbulent kinetic plots previously presented. As can be observed from these figures; the shape of the obstacle with triangular profile does not affect the original pattern of the flow from the collector entrance toward the chimney and guides the flow in a more prolific manner while it is capable of expeditiously creating fluid mixing patterns and thermal boundary layer agitation. On the other hand, the other two obstacles result in larger circulation zones due to larger surfaces encountering the air flow which technically provides more severe fluid mixing; however, as it is detectable from the stream lines, the original path of the flow toward the chimney which is caused by the pressure difference is manipulated in an undesirable manner that prevents the flow from gaining more velocity and experiencing more heat transfer throughout the collector.

Velocity magnitude in the chimney region is evaluated for each case to investigate the effect of implementation of PFCDs on this parameter which directly affects power generation rates of the
plant, the corresponding observed values are plotted in Fig. 13 and the maximum values are tabulated in Table 4. As it is expected due to the phenomena dominating the energy conversion process, the obstacle with triangular profile possesses the topmost observed air velocity within the investigated cases.

The values of allowable mass flow rate as an indicator of flow resistance caused by each PFCD is presented in Table 5, as can be seen from this table the triangular PFCD allows more air to be sucked into the collector since it possesses the characteristics with less flow resistance; these results further demonstrate the trends observed via comparison between turbulent kinetic energy values.

All of the discussed above demonstrate that there is a third conducive factor in performance elevation which is deemed to be the flow guidance of the obstacle, as discussed earlier the shape should possess three main characteristics, boundary layer agitation, creation of secondary flows and vorticities, and enhancing the original flow pattern.

In addition, from the results in Table 4 it is perceivable that even a small increase in chimney air velocity can provide significant uplifting to power generation capacity of the plant since the production is proportional with the air velocity to the power of three.

6. Conclusion

In this study heat transfer and fluid flow manipulation in a SCPP via three different PFCDs is numerically investigated. The scope of
the research includes validation of the employed CFD approach through simulation of a prototype case against experimental data on temperature distribution along the collector, after that the validated CFD approach is utilized to investigate the flow and thermal fields resulted from implementation of flow control devices. The following points stand out as the findings of this study:

- It is shown that in all cases, the Nusselt number increases as the PFCDs are placed in the collector due to the amelioration of the synergy between the velocity and temperature fields in comparison with the base case.
- Improved hot and cold fluid mixing through secondary flows and vorticities can be deemed as a phenomenon contributor in heat transfer amelioration in the solar chimney.
- Agitation in thermal boundary layer thickening and development, caused by the presence of PFCDs, is considered as the factor maintaining a higher average absorber surface Nusselt number.
- Investigation on the shape of the obstacles shows that all the different shapes provide boundary layer agitation and fluid mixing; however, obstacle with triangular profile supplies more thermal performance enhancement since not only the flow pattern throughout the chimney is not blocked but also guidance toward the chimney is supplied to the flow.
- Evaluation of velocity magnitudes at the entrance of the chimney reveals that improvements in collector’s heat transfer characteristics results in higher air velocity rates as expected; since in the understudy SCPP free convective heat transfer acts as the initiative for the process of power conversion.
- Passive or active flow control strategies appear to possess the capability of improving the energy output of SCPPs (obtained up to 41.2% in this study) and conclusively reduce the cost of energy production making it more affordable in comparison with conventional energy technologies.

References