Energy Conversion and Management 105 (2015) 588–595

Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

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

ARTICLE INFO

Article history:
Received 27 April 2015
Accepted 10 July 2015

Keywords:
Solar chimney power plants
CFD
Passive flow control

ABSTRACT

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.

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

* Corresponding author. Tel.: +98 511 8673304x209; fax: +98 511 8673304.
E-mail addresses: e.shabahang@stu.um.ac.ir (E.S. Nia), ghazikhani@ferdowsi.um.ac.ir (M. Ghazikhani).

http://dx.doi.org/10.1016/j.enconman.2015.07.030
0196-8904/

Nomenclature

$\vec{F}$	external body force (N)
H	energy
I	unit tensor
S	source term
T	temperature (K)
$\vec{v}$	velocity component (m/s)
y^+	turbulence wall Y plus
Greek letters	
μ	dynamic viscosity (kg/ms)
ρ	density (kg/m^3)
τ	shear stress (N/m^2)

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 concrete 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