Renewable Energy 63 (2014) 498–506

Contents lists available at ScienceDirect

Renewable Energy

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

Three-dimensional CFD analysis for simulating the greenhouse effect in solar chimney power plants using a two-band radiation model

Ehsan Gholamalizadeh, Man-Hoe Kim*

School of Mechanical Engineering, Kyungpook National University, Daegu, Republic of Korea

ARTICLE INFO

Article history:
Received 24 July 2013
Accepted 2 October 2013
Available online 25 October 2013

Keywords:
Solar chimney power plant
Computational fluid dynamics
Greenhouse effect
Heat transfer modeling

ABSTRACT

The greenhouse effect in the solar collector has a fundamental role to produce the upward buoyancy force in solar chimney power plant systems. This study underlines the importance of the greenhouse effect on the buoyancy-driven flow and heat transfer characteristics through the system. For this purpose, a three-dimensional unsteady model with the RNG $k-\varepsilon$ turbulence closure was developed, using computational fluid dynamics techniques. In this model, to solve the radiative transfer equation the discrete ordinates (DO) radiation model was implemented, using a two-band radiation model. To simulate radiation effects from the sun's rays, the solar ray tracing algorithm was coupled to the calculation via a source term in the energy equation. Simulations were carried out for a system with the geometry parameters of the Manzanares power plant. The effects of the solar insolation and pressure drop across the turbine on the flow and heat transfer of the system were considered. Based on the numerical results, temperature profile of the ground surface, thermal collector efficiency and power output were calculated and the results were validated by comparing with experimental data of this prototype power plant. Furthermore, enthalpy rise through the collector and energy loss from the chimney outlet between 1-band and two-band radiation model were compared. The analysis showed that simulating the greenhouse effect has an important role to accurately predict the characteristics of the flow and heat transfer in solar chimney power plant systems.

1. Introduction

Solar thermal power systems utilize the heat generated by a solar collector to convert the solar energy into electrical power. A solar chimney power plant (SCPP) converts solar energy into electrical energy by a combination of four main parts, the collector, the chimney, wind turbine and energy storage media. The air inside the collector is heated by the greenhouse effect. Therefore, a continuous updraft in the chimney is produced by the upward buoyancy force. The airflow at the base of the chimney runs a pressure-staged wind turbine. Finally, mechanical energy is converted into electrical energy by using a conventional generator. A schematic of the SCPP is shown in Fig. 1.

The basic principles and reported preliminary test results for a prototype SCPP built in Manzanares, Spain, were presented in Refs. [1–3]. By introducing the SCPP concept, several studies including fluid dynamic and thermal models to simulate the SCPP have been done increasingly. In Refs. [4–10], various one-dimensional mathematical models based on thermal equilibrium to analyze the performance of the SCPP were presented. In general, one-dimensional mathematical models cannot provide detailed distributions of the velocity, temperature and pressure inside whole the system. Hence, several numerical simulations to solve a coupled set of conservation equations of mass, momentum and energy based on computational fluid dynamics (CFD) solution have been carried out. Bernardes et al. [11] presented a CFD solution for Navier–Stokes and energy equations in an SCPP for the natural laminar convection in steady state using finite volumes method. Pastohr et al. [12] carried out a two-dimensional steady state numerical analysis on the whole solar chimney systems including the collector, chimney, energy storage layer, and turbine using CFD. The conservation equations for mass, momentum and energy were solved in a small-scale SCPP using the finite volume method by Koonsrisuk and Chitsomboon [13]. Ming et al. [14] presented numerical simulations of the airflow and the characteristics of heat transfer in the SCPP with an energy storage layer including the effect of solar radiation on heat storage on the ground. Ming et al. [15] performed steady state numerical simulations on the SCPP coupled with turbine using the Manzanares prototype plant as a

* Corresponding author.
E-mail addresses: ehsan@knu.ac.kr (E. Gholamalizadeh), manhoe.kim@knu.ac.kr (M.-H. Kim).

0960-1481/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.renene.2013.10.011

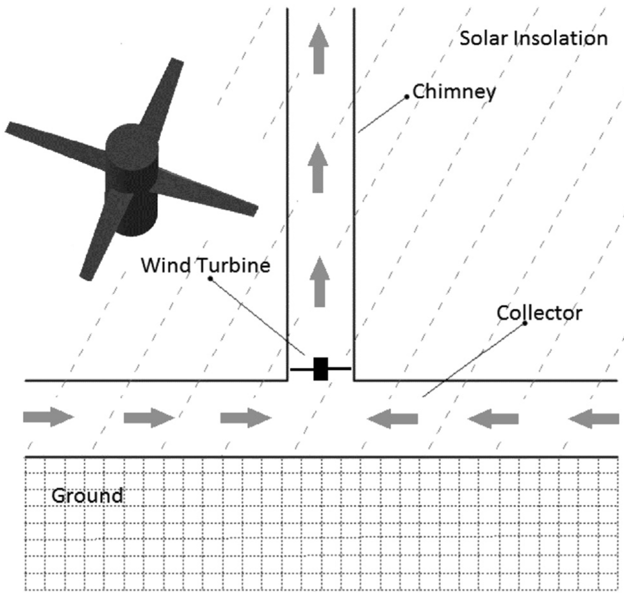

Fig. 1. A schematic of the solar chimney power plant system.

practical example. Chergui et al. [16] solved the mathematical model represented by the Navier–Stokes equations in steady state, continuity and energy equations for a natural laminar convective heat transfer, through an axis symmetric system in a dimensionless form. Numerical simulations were performed by Xu et al. [17] to analyze the influences of solar radiation and pressure drop across the turbine on the steady state flow and heat transfer, power output and energy loss of an SCPP similar to the Manzanares prototype plant. Sangi et al. [18] presented a numerical analysis using the CFD method to simulate a steady state two-dimensional axis symmetric model of an SCPP for different solar radiations.

In the above CFD investigations, the effects of solar and thermal radiations were considered by setting imposing boundary conditions at the physical boundaries of the collector such as a heat source for the ground thin layer, a uniform heat source within the air, specific wall temperatures or heat fluxes. In spite of good results, in order to induce buoyancy-driven flow the temperatures or the fluxes inside the computational domain should normally be inferred from convective and radiative balances along the physical boundaries. Also, only a few CFD simulations have been carried out by solving the radiative transfer equation coupled to the convective transfer equation. In addition, the optical properties of the semi-transparent collector cover (e.g. glass) vary for different wavelength bands. The literature review indicates that none of the previous studies have been distinguished the interchange of short and long wavelength contributions of solar radiation and radiation emitted by the ground in the collector. Furthermore, the assumption of a steady state solution is not sufficient to correctly obtain the heat storage inside the ground. Therefore, it is necessary to solve the energy equation inside the ground, using an unsteady solution.

The objective of this study is to accurately analyze the SCPP system by simulating the greenhouse effect inside the collector. For this purpose, a three-dimensional unsteady CFD model is developed. It is based on the solution of the Navier–Stokes equation for turbulent flow using the RNG k–ε model. The convective and radiative transfer equations are simultaneously solved. The discrete ordinates (DO) radiation model is used, in which the non-gray radiation model is implemented using a two-band radiation for visible and infrared bands. Solar radiation reaches on the semi-transparent collector cover as an irradiation beam from outside the computational domain that is simulated using the solar ray tracing algorithm. Based on the numerical results, velocity and temperature distributions of an SCPP by the geometry parameters of the Manzanares power plant are considered. Also, enthalpy rise through the collector and energy loss from the chimney outlet for 1-band and two-band radiation model are compared. In addition, temperature profile of the ground surface and thermal collector efficiency of Manzanares prototype are calculated.

2. Mathematical modeling

2.1. Natural convection and buoyancy-driven flow

When heat is added to a fluid and the fluid density varies with temperature, a flow can be induced due to the force of gravity acting on the density variations. Such buoyancy-driven flows are termed natural-convection flows. In natural convection, the strength of the buoyancy-induced flow is measured by the Rayleigh number as follows:

$$Ra = \frac{g\beta\Delta T L^3 \rho}{\mu\alpha} \tag{1}$$

where ΔT and L are the maximum temperature difference of the airflow in the system and mean collector height. Rayleigh numbers less than 10^8 indicate a buoyancy-induced laminar flow, with transition to turbulence occurring over the range of $10^8 < Ra < 10^{10}$.

In a natural convection model inside a closed domain, the solution will depend on the mass inside the domain. Since this mass will not be known unless the density is known, in this work to model the flow a transient calculation is performed. In this approach, using the ideal gas law the initial density will be computed from the initial pressure and temperature, so the initial mass is known. As the solution progresses over time, this mass will be properly conserved.

The conservation equations for mass and momentum in an inertial reference frame can be described as follows [19]:

Continuity equation:

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\cdot\vec{v}) = 0 \tag{2}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\rho\cdot\vec{v}) + \nabla\cdot(\rho\cdot\vec{v}\cdot\vec{v}) = -\nabla p + \nabla\cdot\left(\mu\left[\left(\nabla\vec{v} + \nabla\vec{v}^T\right) - \frac{2}{3}\nabla\cdot\vec{v}I\right]\right) + \rho\vec{g} \tag{3}$$

Modeling turbulence:

The analysis simulated in this paper shows that the Rayleigh number in the system at all simulations is high ($Ra > 10^{10}$) and therefore, flow through the solar chimney system is associated with turbulent flow. In order to simulate the turbulent flow the RNG k–ε model [19] is used. In this model, the turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations, respectively:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho\varepsilon - Y_M \tag{4}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + G_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon \tag{5}$$