Renewable Energy 62 (2014) 24–30

Contents lists available at SciVerse ScienceDirect

Renewable Energy

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

Numerical simulations of solar chimney power plant with radiation model

Peng-hua Guo, Jing-yin Li*, Yuan Wang

School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an 710049, PR China

ARTICLE INFO

Article history:
Received 29 March 2013
Accepted 24 June 2013
Available online

Keywords:
Solar chimney power plant
Numerical simulation
Radiation model
Solar load model

ABSTRACT

A three-dimensional numerical approach incorporating the radiation, solar load, and turbine models proposed in this paper was first verified by the experimental data of the Spanish prototype. It then was used to investigate the effects of solar radiation, turbine pressure drop, and ambient temperature on system performance in detail. Simulation results reveal that the radiation model is essential in preventing the overestimation of energy absorbed by the solar chimney power plant (SCPP). The predictions of the maximum turbine pressure drop with the radiation model are more consistent with the experimental data than those neglecting the radiation heat transfer inside the collector. In addition, the variation of ambient temperature has little impact on air temperature rise despite its evident effect on air velocity. The power output of the SCPP within the common diurnal temperature range was also found to be insensitive to ambient temperature.

1. Introduction

A rapid increase in renewable energy utilization has been observed worldwide because of environmental and energy crisis issues. The solar chimney power plant (SCPP) system presents an interesting option for the large-scale use of solar energy. The prototype at Manzanares, Spain, which ran automatically from mid-1986 to early 1989, has proven that the SCPP is a reliable approach in generating electricity [1,2]. As shown in Fig. 1, a typical SCPP is mainly composed of a solar collector, chimney, and turbine. The collector is used to heat the incoming air through the greenhouse effect. The heated air flows into the chimney and finally escapes to the atmosphere. The chimney effect results in a large pressure difference between the chimney base and collector entrance. The pressure difference, often defined as the system driving force, makes the chimney the actual thermal engine of the system, and drives the turbine installed at the chimney base to generate electric power. The SCPP has no adverse effect on the environment, requires no cooling water, and has low maintenance costs. These distinct advantages make the SCPP a promising technique for generating electricity in vast desert regions that lack water resources but are abundant in solar energy.

Many papers on SCPP have been published since Haaf et al. [1,2] reported their fundamental studies on the Spanish prototype. Two methods are generally employed to compute SCPP characteristics: the analytical and numerical methods. The typical analytical method is mainly based on a one-dimensional thermal equilibrium analysis inside the collector. For example, Pasumarthi and Sherif [3,4] published an approximate model to investigate the effects of various parameters on air temperature and velocity distribution. Zhou et al. [5] developed a mathematical model to determine the power output for varying solar radiation, collector area, and chimney height, and validated their findings against experimental results from a small-scale pilot plant. Bernardes et al. [6] established an analytical model to predict the characteristics of a large-scale commercial SCPP. They also discussed the optimal proportion of driving force used to drive the turbine. Li et al. [7] recently proposed a comprehensive theoretical model for SCCP performance evaluation that considers the effects of flow and heat losses on SCPP performance.

The number of studies on numerical methods adopting the computational fluid dynamic (CFD) programs to predict SCPP performance has been increasing rapidly. As a pioneer, Pastohr et al. [8] performed a 2-D numerical simulation on SCPP to study the temperature and flow fields in the collector. A similar numerical method was used by Xu et al. [9], the only difference being the settings of the energy storage layer and turbine model. Ming et al. [10] conducted numerical simulations for the Spanish prototype with a three-blade turbine and presented the simulation results of a

---

* Corresponding author. Tel.: +86 29 13152181528; fax: +86 29 82668723.
E-mail address: jyli@mail.xjtu.edu.cn (J.-y. Li).

0960-1481/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.renene.2013.06.039

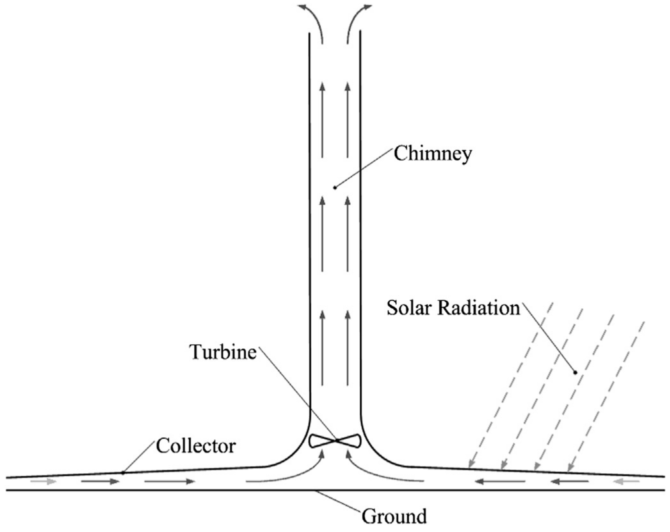

Fig. 1. Schematic diagram of solar chimney power plant.

MW-graded SCPP with a five-blade turbine. Koonsrisuk and Chitsomboon [11,12] used numerical simulations to test their dimensional analysis on SCPP and the validity of their proposed dimensionless variables.

Compared with the analytical method, fewer assumptions are used in numerical simulations but more detailed descriptions of temperature and flow field could be obtained. Although many numerical studies have been published, they mainly focused on the 2-D simulations. In addition, the radiation heat transfer inside the system has rarely been considered in previous numerical simulations. Radiation heat transfer, however, is an important factor in the greenhouse effect and is therefore worthy of investigation.

As an early attempt in considering radiation heat transfer in the collector, this paper first verified the 3-D numerical simulation method incorporating the radiation, solar load, and turbine models, by comparing with the experimental data obtained from the Spanish prototype. Based on the proposed numerical approach, the effects of solar radiation, turbine pressure drop, and ambient temperature on the SCPP system performance were investigated in detail.

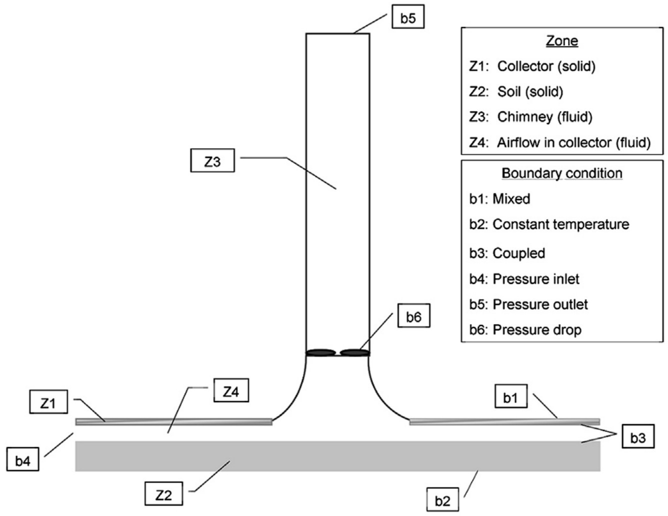

Fig. 2. Main zones and boundary conditions.

2. Numerical models

2.1. Physical model

No experimental results on a large-scale SCPP system have ever been reported, except for the prototype in Spain. Thus, the Spanish prototype was selected as the physical model to verify the numerical method. The computational domain, as shown in Fig. 2, is divided into the following zones: the collector (z1), energy storage layer (soil, z2), chimney (z3), and airflow (z4). The main dimensions of the Spanish prototype are listed in Table 1. The collector and the chimney are smoothly connected to facilitate the computation. As indicated in the measured data of the Spanish prototype [2], soil temperature at 0.5 m depth underneath the ground remained unchanged with time. A similar phenomenon was found in the study of Pretorius and Kröger [13]. Therefore, a ground thickness of 2 m is believed to be sufficiently deep to facilitate the isothermal condition of the bottom boundary of the soil layer.

2.2. Models for heat transfer and airflow in SCPP

All simulations were conducted for steady flow using the finite volume-based solver FLUENT. Heat transfer in the SCPP system involves all three modes: conduction, convection, and radiation. In simulating the flow in SCPP, computations using models that only focus on conduction or convection are the simplest, whereas those involving buoyancy-driven flow and radiation models are more complex. Radiation heat transfer mainly occurs in the collector, which is covered by different types of semi-transparent materials such as glass or plastic. The cover materials are nearly transparent for incident solar radiation but partly opaque for infrared radiation from the ground. In the present simulations, the discrete ordinate (DO) radiation model was adopted to solve the radiative transfer equation for the following reasons: (1) only the DO model can be used to model semi-transparent walls of various types, (2) only the DO model can be used to compute non-gray radiation using a gray band model, and (3) the DO model can work well across a full range of optical thicknesses.

In previous studies, incident solar radiation on the ground through the semi-transparent collector was commonly treated as an internal heat source or heat flux. For example, Pastohr et al. [8] and Xu et al. [9] considered solar radiation as an internal heat source in the ground's thin layer. In Koonsrisuk and Chitsomboon's study [12], solar heat absorption per unit volume of air is modeled as a uniform heat source within the airflow in the collector. Li et al. [14] treated the absorption of solar radiation as heat flux from the ground to the airflow. In the present study, a solar ray-tracing model provided by FLUENT was used to calculate the radiation effects of the sun's rays entering the computational domain. The ray-tracing model is an efficient and practical approach to applying solar loads as heat sources in energy equations. Solar radiation is modeled using the sun's position vector and illumination parameters, which can be specified by users or by a solar calculator utility provided by FLUENT. Solar load is available for 3-D simulation only and can be used to model steady and unsteady flows.

Table 1
Main dimensions of the Spanish prototype.

Parameter	Value
Mean collector radius	122.0 m
Mean collector height	1.85 m
Chimney height	194.6 m
Chimney radius	5.08 m
Ground thickness	2 m