Applied Thermal Engineering 98 (2016) 1036–1043

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Computational analysis of a solar energy induced vortex generator

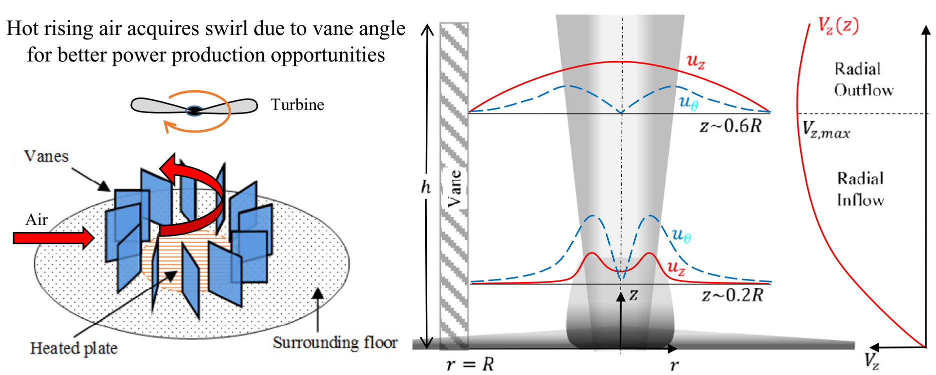
Abdullah Mohiuddin [a], Eray Uzgoren [b,*]

[a] Sustainable Environment and Energy Systems, Middle East Technical University, Northern Cyprus Campus, Mersin 10, Turkey
[b] Mechanical Engineering Program, Middle East Technical University, Northern Cyprus Campus, Mersin 10, Turkey

HIGHLIGHTS

- A novel device mimicking dust devils is proposed for solar energy applications.
- CFD model is devised to analyze influence of geometric parameters on the swirl.
- Parametric study on device geometry reveals optimal vane height and width.
- Vortex stability and torque strength increase at vane angles greater than 30°.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Received 20 October 2015
Accepted 1 January 2016
Available online 12 January 2016

Keywords:
Solar updraft tower
Free convection
OpenFOAM
Buoyancy induced vortex

ABSTRACT

This study presents a computational analysis of a device that mimics dust devils in a controlled environment in order to explore its capacity as an energy conversion apparatus in solar energy applications. Concept is built upon the buoyancy effect over a heated plate surrounded by stationary vertical thin plates (vanes), which cause swirl in the raising air. The novelty of the paper is that it is the first parametric study that investigates effects of vane width, vane height, number of vanes, and vane angle on the vertical flow rate of the proposed device. It is found that (i) the optimal vane width is 1.4*R*, (ii) reduction in spacing between the vanes improves vertical flow strength, (iii) minimum vane height is 1.4*R*, and (iv) high vane angles increase swirl and hence vortex stability to yield better torque production opportunities with the high air speeds away from the vertical centerline.

1. Introduction

The idea of harvesting energy through rising air has been explored in various renewable energy applications. One promising example is the solar updraft towers, which generate mechanical energy in terms of turbine shaft work from rising hot air that is heated by solar energy [1,2]. According to a recent review by Zhou and Xu on solar updraft towers [3], one of the fundamental challenges is related to the low energy conversion efficiency owing to limitations related to tower's height. Reasonable energy conversion efficiencies are only possible for towers with heights ranging from 100 m and 1000 m but construction of such towers bring additional challenges related to construction materials' availability, investment cost, and environmental concerns.

Zhang et al. [4] recently proposed a device to replace the tower by a vertical columnar vortex. Such a vortex can be formed with the help of vertical planes (vanes) beneath the solar panels, guiding warm air tangentially toward the center of the collectors at a certain angle so that the rising air acquires tangential and radial velocity. This idea was previously used by Fitzjarrald [5] in 1973 in an experimental setup with 20 vanes to study naturally occurring dust-devils, which are small-sized vertical columnar vortices with high

* Corresponding author. Tel.: +903926612925; fax: +903926612999.
E-mail address: uzgoren@metu.edu.tr (E. Uzgoren).

http://dx.doi.org/10.1016/j.applthermaleng.2016.01.005
1359-4311/

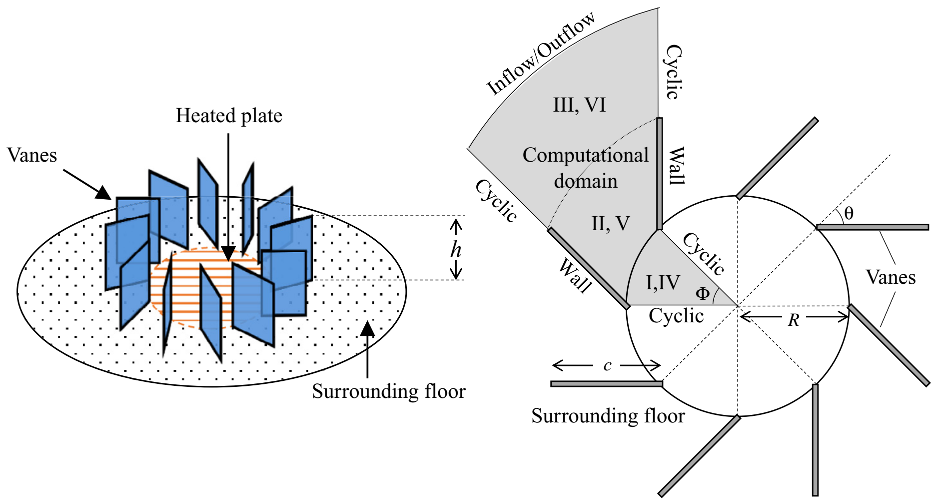


Fig. 1. Flow geometry, computational domain and boundary conditions.

concentrations of kinetic energy. Simpson et al. [6] used a similar setup to explore its power generation capabilities through a turbine placed at the top of the vanes. They used a heated square plate surrounded by 12 planar aluminum vanes, each of which was 35 cm in width and 60 cm in height. They were able to demonstrate the formation of a meter scale vortex of 4 m high and a core diameter of 5–25 cm, which was able to spin a rotor at 15–25 rpm at a plate temperature 22 °C higher than the ambient temperature. While they investigated power production opportunities at various temperature differences, they did not focus on scaling of geometrical parameters. Zhang et al. [4] investigated geometrical scaling of a similar device with 8 curved vanes through experiments on a prototype with 2 m radius and numerical analyses for 20 and 200 m radii. They observed that the maximum speed scales with Froude number and identified that the tangential velocity increased the most when the vane angle was changed from 30° to 60°. Michaud and Monrad [7,8] demonstrated the energy conversion potential based on a thermodynamic model as part of commercialization opportunities within cooling towers of already existing power plants. Natarajan [9] used CFD simulations to explore the flow induced by such a device.

Device proposed by Refs. 4–9 relies on the swirl of the uprising air owing to vanes, which direct air to flow tangentially to the convective plume. These studies have mostly reported vortex formation at meter scale and it is still not clear whether similar flow conditions for vortex growth are attainable at larger scales for cost-effective commercial applications [7]. Furthermore, such large scale applications would also require certain measures to be taken for flow control to relieve safety and environmental concerns [8]. Thus, a better understanding of flow's characteristics is needed. The present study investigates the influence of geometrical and physical parameters of such a device on the flow field using numerical analysis tools. Specifically, vertical flow features are examined through simulations of various vane widths, various vane spacing, various vane heights, and vane angles to understand their role so that better designs can be developed for power production purposes.

2. Methodology

Numerical simulations are carried out using OpenFOAM, open-source finite volume based computational fluid dynamics software [10]. Basic flow geometry is shown in Fig. 1, which consists of a circular heated plate at an elevated temperature surrounded by an unheated floor. Both circular heated plate and floor are open to the atmosphere. The heated plate is surrounded by a number of vertical thin flat plates, referred to vanes throughout the paper. Vertical vanes are considered as the core focus of this paper, as they guide the entrained flow off the center to form a vortex. Numerical experiments are carried out to investigate the influence of vane width, vane height, vane angle and heated plate's radius. Vane angle is the angle between the vane and an imaginary radial line stretched from the center of heated surface, vane width is the horizontal vane length, and vane height is the distance from the floor to the top of the vane as shown in Fig. 1.

2.1. Numerical method

The governing equations for incompressible fluid flow including the Boussinesq approximation are given as follows:

$$\nabla \mathbf{u} = 0 \tag{1}$$

$$\rho_o\left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u}\right) = -\nabla p + \mu\nabla^2\mathbf{u} + \rho_o(1-\beta\Delta T)\mathbf{g} \tag{2}$$

The scalar transport-diffusion equation for the temperature is given as follows:

$$\frac{\partial T}{\partial t} + (\mathbf{u}\cdot\nabla)T = \alpha\nabla^2 T + \varphi \tag{3}$$

Equations (1)–(3) are solved using OpenFOAM's *buoyantBoussineqPimpleFoam*, which is a precompiled solver using a pressure-based method for incompressible transient flows with Boussinesq approximation for the buoyant forces. Standard k–ε two-equation turbulence model is used in simulations. Specifics of simulation setup are summarized in Table 1, while readers can refer to Ref. 11 for additional details.

Flow domain is defined by a truncated circular plate in shape of a piece of pie (sector) with a span characterized by the span angle as shown in Fig. 1. All vertical sides of the sector except vanes are modeled using cyclic boundary condition for all transport variables. Vertical vanes, heated plate and surrounding wall are considered as isothermal walls. Outer circumferential vertical surface switches between constant pressure/temperature for inflow and zero pressure/temperature gradient for outflow conditions. Simulations are initialized with zero velocity field and ambient temperature and carried out until reasonable steady state conditions are satisfied,