Solar Energy 151 (2017) 95–101

Contents lists available at ScienceDirect

Solar Energy

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

The impact of the parabolic dish concentrator on the wind induced heat loss from its receiver

M. Uzair [a,b,*], T.N. Anderson [a], R.J. Nates [a]

[a] Department of Mechanical Engineering, Auckland University of Technology, Auckland, New Zealand
[b] Department of Mechanical Engineering, NED University of Engineering & Technology, Karachi, Pakistan

ARTICLE INFO

Article history:
Received 9 May 2016
Received in revised form 3 March 2017
Accepted 5 May 2017

Keywords:
Parabolic dish
Heat loss
Wind
CSP

ABSTRACT

To achieve higher operating temperatures, power output and system efficiencies in parabolic dish cavity receivers, larger dish sizes and structures are used to increase the concentration ratio. This increases capital investment and installation costs, which in turn places a much stronger emphasis on accurately predicting the performance of the system and the heat loss from it. Numerous studies have investigated the natural convection heat losses from cavity receivers, and some have examined a cavity exposed to wind. However, the effect of the dish on the wind flow and subsequently the heat loss from the receiver has not been widely considered.

In this work, computational fluid dynamics was used to model the flow of air around a parabolic dish concentrator operating at varying angles of operation. The flow fields were validated using wind tunnel testing and published data regarding the aerodynamic characteristics of parabolic dishes. The results showed that the orientation of the dish has a significant effect on the flow structure near the receiver. Subsequently, using the validated method, the convective heat loss from the receiver of a large parabolic dish system was determined for a range of operating conditions.

The results support the assertion that the flow characteristics near the cavity receiver aperture depend strongly on the orientation of the dish structure. This resulted in the calculated heat loss being up to 40% lower than previous studies where the presence of the dish was included. As such, the wind flow around the dish needs to be accounted for when analyzing the performance of parabolic dish systems to avoid an overly conservative and hence more expensive design.

© 2017 Elsevier Ltd. All rights reserved.

1. Introduction

Concentrating solar power (CSP) systems can be classified into four main types: parabolic dish systems, solar towers, parabolic troughs and Fresnel reflectors. Among these classifications, parabolic dish systems are considered to be the most efficient as a result of them achieving higher concentration ratios than the other techniques (Steinfield, 2005 and Tyner et al., 2001). To compete with conventional power generation techniques, the thermal performance of these CSP systems plays an important role. The performance of a CSP system utilizing a parabolic dish is sensitive to heat losses from the cavity receivers employed in these systems, particularly at high temperature. In particular, the heat loss from the cavity receiver is affected by the surrounding air motion, and consequently exposure to this could result in increased heat loss and decreased thermal performance (Lupfert et al., 2001).

Analytical techniques are available to determine the radiation and conduction heat losses from cavity receiver; however due to the complicated velocity and temperature field around the receiver, determination of convective heat losses is less straightforward. In this regard a significant body of research has developed with a view to understanding the convective heat losses from the cavity receiver of parabolic dish systems (Wu et al., 2010), particularly under natural convection conditions. In two of the most frequently cited works, Clausing (1981) and Clausing et al. (1987) investigated the natural convection heat losses from large cavities and proposed an analytical solution. In their implicit solution, the cavity receiver was divided into a convective zone and a stagnant zone. Similarly, Le Quere et al. (1981a, b) performed a numerical and experimental study in order to develop a relationship between the Nusselt number and tilt angle. In this vein, numerous further studies (Siebers and Kraabel, 1984; Stine and McDonald, 1989; Leibfried and Ortjohann, 1995) have attempted to quantify and deliver a rela-

* Corresponding author at: Department of Mechanical Engineering, Auckland University of Technology, Auckland, New Zealand.
E-mail address: muhammad.uzair@aut.ac.nz (M. Uzair).

http://dx.doi.org/10.1016/j.solener.2017.05.022
0038-092X/© 2017 Elsevier Ltd. All rights reserved.

tionship between the natural convection heat transfer from parabolic dish receivers and their orientation (tilt angle), aperture size and cavity geometry.

Now in reality parabolic dish receivers are likely to be exposed to some degree of forced convection, however investigations of forced convection heat loss from these receivers are relatively scarce compared to those focussing on natural convection. In his study Ma (1993) examined a parabolic dish receiver and came to the conclusion that convection losses due to wind varied strongly with the receiver's tilt angle. More recently, Paitoonsurikarn and Lovegrove (2006) and Paitoonsurikarn et al. (2004) numerically examined how variations in wind velocity near a cavity receiver influenced the heat loss from it and developed a relationship describing this.

Now in almost all studies, researchers have treated the cavity receiver as an isolated entity, decoupled from the dish/reflector structure of a real parabolic dish system. Wu et al. (2010) highlighted this issue in their review of the field, noting that there is a dearth of information relating to wind effects on heat loss and the interaction between the wind and dish, and by extension the influence on the heat loss. In this regard, the present study set out to address some of the issues highlighted by Wu et al. (2010) by examining the heat loss from a parabolic dish cavity receiver due to the wind flow around the parabolic dish.

2. Method

2.1. Numerical setup

In order to examine the effect of wind flow on the heat loss from parabolic dish receivers it was decided to undertake a computational fluid dynamics (CFD) analysis of the flow around the dish and receiver at varying angles of attack. For this study, the geometry chosen was that of the Australian National University's 20 m^2 dish and frustum-shaped receiver described by Paitoonsurikarn and Lovegrove (2003). This parabolic dish has a focal length of 1.84 m and an aperture diameter of 5 m with a rim angle of approximately 70°; the receiver has dimensions as shown in Fig. 1.

In undertaking the analysis, the computational domain around the dish-receiver system was extended 75 m upstream of the dish,

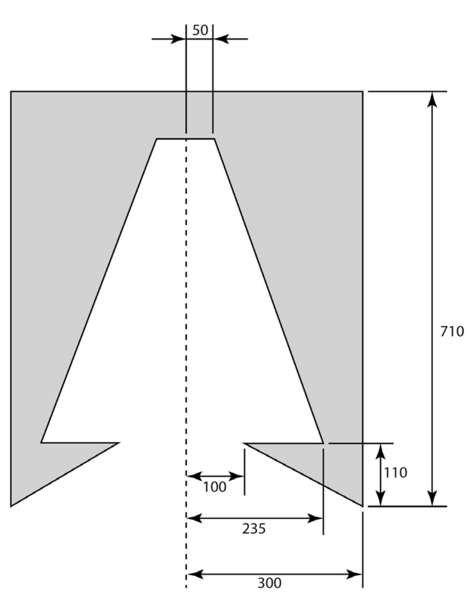

Fig. 1. Frustum shaped receiver and dimensions.

105 m downstream and 30 m in the lateral direction (Fig. 2) to ensure all flow features were adequately captured, and walls were modelled using a no-slip boundary condition. A mesh sensitivity analysis was undertaken to ensure that the grid size did not influence the heat loss from the receiver. This resulted in a highly refined mesh of approximately 2 million elements being used to perform a steady state simulation of the flow around the dish. In constructing the mesh, finer grid elements were used near the receiver and in the wake region of the dish in order to predict the flow behaviour accurately. Regions further from the dish were meshed with larger grid sizes to improve computational speed.

The wind flow over the system was modelled using the commercial CFD program ANSYS CFX 15.0.7 and the Shear Stress Transport (SST) turbulence model. The SST model has been shown to be one of the most accurate two-equation models for separation prediction and has been successfully used for studies of wind flow over parabolic troughs (ANSYS, 2013; Paetzold et al., 2015). In determining the heat loss it was assumed the system was subject to a free stream wind velocity of 5 m/s and an ambient temperature of 25 °C (a Reynolds number of $\sim 1.9 \times 10^6$ using the dish diameter as the characteristic length). Further, the internal cavity walls of the receiver were modelled as isothermal walls with a temperature of 600 °C with buoyancy effects included in the calculation, to account for natural convection. Finally, steady state simulations were performed by changing the angle of the dish relative to the wind, from 90° (direct impingement of the wind on the mirrored surface) to 0° (dish aperture facing directly upwards) to −90° (wind impinging on the back surface of the mirrors).

2.2. Experimental setup

To observe the behaviour of turbulent flow around the dish a series of wind tunnel experiments were performed using a scaled down version of the large dish, with a model diameter of 150 mm ($Re \sim 4 \times 10^4$). In order to get a smooth finish a nylon model of the dish was manufactured using a three dimensional printing system. The model was mounted on a stand in the test section by means of 180 mm long steel rod that allowed for rotation around the x-axis (Fig. 2). The test section of the wind tunnel was 500 mm × 500 mm in cross section, and extended 500 mm upstream and 1000 mm downstream of the model.

After testing of the wind tunnel to ensure homogenous flow, a series of smoke visualization experiments were performed to qualitatively validate the flow fields observed in the simulations. In order to diffuse the smoke in parallel lines inside the wind tunnel, a smoke rake was used and images of the smoke flow were captured in the presence of a green laser light sheet by a digital SLR camera. A schematic representation of the experimental setup is shown in Fig. 3.

3. Results and discussion

3.1. Examination of flow features

As mentioned previously, CFD simulations were performed for the parabolic dish system at different pitch angles while visualization of the turbulent flow around a parabolic dish structure was performed using smoke thus allowing the simulation results to be verified qualitatively. For different tilt angles of the dish, velocity streamlines along the center plane of the CFD simulation domain (with flow moving from left to right) were compared with the smoke streak lines as shown in Figs. 4–12.

The flow around the dish shows markedly different flow structures with different tilt angles. Starting from the case when flow is perpendicular to the aperture plane of the dish, i.e. 90° tilt angle;

Download English Version:

https://daneshyari.com/en/article/5450960

Download Persian Version:

https://daneshyari.com/article/5450960

Daneshyari.com