Solar Energy 174 (2018) 463–473

Contents lists available at ScienceDirect

Solar Energy

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

An experimental and numerical study of granular flows through a perforated square lattice for central solar receiver applications

Matthew Sandlin*, S.I. Abdel-Khalik

Woodruff School of Mechanical Engineering, Georgia Institute of Technology, Atlanta, GA 30332, United States

ARTICLE INFO

Keywords:
Concentrated solar power
Particle heating receiver
Granular flow
Discrete element method
Computational fluid dynamics

ABSTRACT

A proposed design for a concentrating solar power (CSP) receiver uses a granular material - such as sand - as the heat transfer and energy storage medium. Early designs of particle heating receivers (PHR) utilize a falling curtain of particles which directly absorbs the concentrated solar radiation. However, falling curtain receivers have several disadvantages, including significant heat and particle losses, and a short residence time within the irradiation zone. One design proposal which overcomes these challenges is the so called "impeded flow PHR design", in which the particles flow over, around, or through a series of obstacles in the flow path. This reduces the average velocity of the particles, thereby increasing residence time in the irradiation zone of the receiver. It also reduces heat and particle losses from the receiver. However, granular flows through complex structures are not well understood, rendering a priori design of impeded flow PHR geometries difficult. To better understand these flows, lab scale models of a PHR design variant using a perforated square lattice at an oblique angle have been constructed, allowing granular flows through the receiver geometry to be experimentally analyzed. In addition, two different numerical modeling approaches - the discrete element method (DEM) model, and a two-fluid computational fluid dynamics (CFD) model - have been developed to model the flow of particles through the specified receiver geometry. The results of the DEM model are in reasonable agreement with the experimental data with respect to mass flux, and better matches the experimental data than the CFD model.

1. Introduction

The use of concentrated solar power (CSP) for electricity production has received increasing amounts of research and public interest in recent years (Behar et al., 2013). The renewable nature of the energy source and the lack of pollution during operation are the two chief advantages of CSP.

CSP takes on several forms, including parabolic trough designs, Fresnel lens reflectors, and solar power towers (Zhang et al., 2013). All CSP technologies operate using the same principles: using mirrors, the incident solar radiation is focused onto a receiver where a heat transport medium, such as water, pressurized air, or a liquid salt solution, is used as a coolant. This medium can then be used to generate electricity using a typical power cycle, or can store that heat to enable power generation during periods of no solar input.

Many current forms of CSP which include heat storage use oil or a molten salt solution to store the solar energy. While these designs enable the use of current technology to store and move the heat transfer fluid, there are several drawbacks: the molten salt solution itself is relatively expensive, corrosive, and has a relatively narrow range of operating temperatures of roughly 250–620 °C (Zhang et al., 2013).

One proposed approach uses a granular material - such as sand - as the heat transfer and storage medium. Not only would such a system be cheaper and safer, but would potentially offer much higher temperature limits, increasing thermodynamic efficiency.

The use of solid particles for a heat transfer and storage medium has been under investigation since the early 1980s (Martín and Vitko, 1982). Extensive research in the area of falling curtain solid particle receivers has been performed (Falcone et al., 1985; Chen et al., 2007; Kim et al., 2009, 2010; Khalsa et al., 2011; Röger et al., 2011; Ho et al., 2014b; Gobereit et al., 2015; Zanino et al., 2016), as the design is relatively simple, yet allows for direct heating of the particles by concentrated solar irradiation. However, falling particle curtain designs share a common disadvantage: particle hydrodynamics play a large role in receiver design and particle selection, not only from increased convective heat losses from the particle curtain, but from particle loss from the receiver as well. In fact, particle aerodynamics and heat transfer performance may be at odds, as smaller particles absorb heat faster via irradiation, but lose heat faster due to convective losses, and are more easily blown around by air currents in the receiver. Larger particles can

* Corresponding author.
E-mail addresses: msandlin6@gatech.edu (M. Sandlin), said.abdelkhalik@me.gatech.edu (S.I. Abdel-Khalik).

https://doi.org/10.1016/j.solener.2018.09.029
Received 15 January 2018; Received in revised form 31 July 2018; Accepted 12 September 2018
0038-092X/

be used to offset some of the particle loss issues, but their higher terminal velocity means that residence time in the receiver is shorter, leading to lower heat absorption (i.e. a lower temperature rise across the receiver). Some of the proposed falling particle designs use particle recirculation systems to overcome this issue, but that adds mechanical complexity and parasitic load. The recirculation system also needs to be carefully controlled to ensure high efficiency.

One proposed receiver design to overcome these issues is a so called impeded flow particle heating receiver (PHR). In this receiver design, particles flow over a series of obstacles in the flow path. In one rendering of such a design, the particles flow through a porous ceramic foam structure (Al-Ansary et al., 2017). This design should greatly reduce the issues related to particle loss and convective loss by controlling maximum particle velocities and an overall reduction in air velocity within the particle flow zone in the receiver cavity. The reduction in maximum particle velocity also increases the residence time of particles in the irradiated zone, allowing high particle temperatures to be achieved without complicated particle recirculation systems. On-sun testing of a design variant of an impeded flow PHR which uses chevron shaped wire mesh screen structures has shown that it offers advantages over simple falling particle curtains with respect to particle temperature rise and overall receiver efficiency, without the need to incorporate complex recirculation systems (Ho et al., 2016a,b). However, the testing has also revealed challenges in materials selection, and highlighted the limited understanding of the granular flow through these complex structures. While there have been numerous studies on predicting the rate of granular flow through simple structures such as hoppers with a single opening (Fowler and Glastonbury, 1959; Beverloo et al., 1961; Jenike, 1961; Brown and Richards, 1965; Savage, 1965; Crewdson et al., 1977; Williams, 1977; Nedderman et al., 1982; Gu et al., 1992, 1993; Weir, 2005; Tighe and Sperl, 2007; Hilton and Cleary, 2011; Janda et al., 2012; Oldal et al., 2012), no general continuum analysis of granular flows, such as through complex structures such as those envisioned for impeded flow PHRs, appears to exist at present (Abrahamsson et al., 2014).

Several numerical studies have been reported in the literature on the use of computational fluid dynamics (CFD) simulations of falling curtain PHR designs (Chen et al., 2007; Kim et al., 2009, 2010; Khalsa et al., 2011; Ho et al., 2014b; Gobereit et al., 2015). However, the granular flow conditions present in a falling curtain are expected to be very different than in flow through complex geometric structures such as in impeded flow PHR designs, especially with respect to local solids volume fractions, and solids-structure interactions. In particular, several of the previous studies have used a discrete phase model (DPM). A DPM simulation allows the modeling of particles within a typical CFD simulation. However, most DPM simulations typically assume very low particulate volume fractions ($< 10\%$), and particle interactions are often ignored (Dickenson and Sansalone, 2009). Both of those conditions are expected to be locally violated in an impeded flow PHR.

Numerical simulation of an impeded flow PHR has only begun very recently. Lee et al. (2015) performed a two-fluid study of granular flow through a complex porous foam structure. However, instead of simulating the foam structure itself, the structure was replaced with a simpler packed bed geometry, meaning that the particle scale flow characteristics were not maintained in the simulation.

The discrete element method (DEM) (Cundall and Strack, 1979) is a Lagrangian method for modeling granular flow in that it tracks individual particles. It has been used to model granular flows through complex geometries, including wire mesh screens (Cleary and Sawley, 2002; Delaney et al., 2012; Dong et al., 2013). However, the DEM method is computationally limited in how many particles it can simulate. Recently, its application to a falling curtain design has been investigated, but the simulated particles were much larger than the actual particles in order to ease computational requirements (Zanino et al., 2016). More recent studies of impeded flow PHR design variants using commercially available wire mesh screens in the particle flow path highlight some of the strengths and weaknesses of both a two fluid CFD approach and a DEM approach, and have shown that DEM models may better capture granular hydrodynamics of these flows through particle scale geometric features (Sandlin, 2017; Sandlin and Abdel-Khalik, 2018, in press).

With no general analytic way to predict granular flow through complex structures, and no numerical modeling of the granular flows of interest, the goals of this study were twofold. First, lab scale test sections of representative impeded flow PHR design variants were constructed and flow tested to determine the particle flow characteristics. Second, numerical models of the experimental granular flows were conducted using two different methods - the discrete element method (DEM), and a CFD/finite volume method - to not only validate one or both of the numerical models, but to determine the suitability of one or both models for future use in more complicated designs. This paper presents the design and execution of experimental studies for a PHR design variant using a square lattice structure and the associated numerical models are presented. Then, the details of the two numerical models are briefly reviewed, including the underlying equations and the input parameters. Then, the experimental results are presented, analyzed, and used in an attempt to validate the different numerical models. The focus of this paper is on the experimental and numerical mass flux through the impeded flow PHR geometry, and on qualitative flow behaviors; further analysis is given in Sandlin (2017). Finally, the results are summarized, conclusions drawn, and suggestions for future research are given.

2. Experimental setup

The main goals for the experimental setup were to allow rapid changes in test section configuration, and to enable the measurement of any experimental data that could also be evaluated in the numerical simulations. The main features of the experimental setup included a particle storage hopper, the main test section, a collection bucket, and the necessary control valving and instrumentation.

To investigate the hydrodynamic performance of the specific PHR design variant, a lab scale test section was constructed to allow qualitative and quantitative evaluation of the granular flow through the square lattice geometry. Fig. 1 shows a schematic of the major dimensions used in the test section. The test section is a novel concept consisting of square tubes set at an oblique angle to the flow path. In addition, each face of the square tube is perforated, allowing granular material to flow both in the inter-square channels, and inside the cross section of the tube. In theory, the perforated faces would allow the creation of a more or less uniform curtain of particles. In the initial configuration, 51 mm square aluminum tube with six 6.35 mm holes drilled in each face in a regular pattern was used. A variable width channel, ranging from 1.59 mm to 6.35 mm in width, was maintained between adjacent squares with the use of spacers to investigate the relative effect that inter-square spacing had on overall mass flux. In another configuration, sections of 20 gauge (0.91 mm thick) perforated steel plate that were bent into square cross sections were inserted into the flow path. These were considered the limiting case of a square tube section with a high percentage of open area on each face. In the case of the perforated plates, the channel width between each squares was left at 6.35 mm, and plates with three different perforation dimensions were used (see Table 1). The main test section consisted of six squares down the centerline, and five half squares down each sidewall.

In addition, both test section variants consisted of two different sub-configurations. In the first sub-configuration, the square cross section was left intact. While this is an easier design from a manufacturing standpoint, it was found that this configuration allowed material to accumulate within each individual square tube. However, a goal for this PHR design variant was to maintain a relatively uniform particle distribution within each square, which will allow more uniform particle heating and the minimization of PHR overheating. Therefore, in the