Energy 165 (2018) 984–998

Contents lists available at ScienceDirect

Energy

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

An advanced modeling and experimental study to improve temperature uniformity of a solar receiver

Hamed Abedini Najafabadi [a], Nesrin Ozalp [b, *]

[a] School of Chemical Engineering, Iran University of Science and Technology, Tehran, Iran
[b] Mechanical and Industrial Engineering Department, University of Minnesota Duluth, 55812, Duluth, MN, USA

ARTICLE INFO

Article history:
Received 8 June 2018
Received in revised form
19 August 2018
Accepted 6 October 2018
Available online 9 October 2018

Keywords:
Solar receiver
Radiation model
Fluid flow
Temperature distribution
Hot spot

ABSTRACT

Harnessing solar energy for thermochemical processing is an exciting and fast emerging research area with significant potential for reducing CO_2 emissions. However, maintenance of a uniform temperature distribution as well as avoidance of hot spots in solar cavity receivers are challenges of present technology which are adversely affecting the process efficiency. This study presents a model based methodology as a design tool for iterative creation of optimum solar receiver geometry. Several discrete solutions are demonstrated as case studies via experimental testing of a solar receiver radiated by a 7 kW solar simulator. Experimental observations are compared with the results of the numerical analysis based on two-dimensional (2D) numerical model that couples the fluid flow and heat transfer mechanisms in the solar receiver. A Monte-Carlo ray tracing method was used to model the incoming radiation from the solar simulator and radiative exchange between the inner surfaces of the cavity receiver. Comparison of the simulation results to experimentally measured steady state temperatures at different points of the solar receiver shows 6.68% average absolute error confirming appreciable accuracy of the model. The results also show that reversing the gas flow direction and increasing the insulation layer do not improve the temperature distribution in the receiver. However, reducing the front flange dimensions and decreasing the inner receiver radius do enhance the temperature distribution and increase the average receiver temperature. Numerical results show that these changes can increase the average temperature of the inner cavity cylinder walls by 27%, and increase the temperature uniformity index by 58%. These findings provide essential insight for solar reactor design to reduce hot spot problems and improve temperature uniformity.

1. Introduction

Solar energy is featured prominently among renewable energy sources. The primary focus of solar energy utilization has been efficient generation of electricity via photovoltaics or solar powered Rankine cycles. With the increasing concern of carbon dioxide emissions and climate change, more focus should be on solar technologies for producing storable fuels, so-called “solar fuels”. Solar fuel production relies on concentrated solar energy as the heat source for direct decomposition of a chemical feedstock via solar thermochemical processes, which significantly reduce emissions in comparison to traditional processes heated via fossil fuel combustion [1,2]. The heart of solar thermochemical processes is a solar reactor, which typically features a cavity-type receiver for effective capture of concentrated solar radiation entering the reactor through a small aperture.

The basic principle of solar thermochemical processes is that the reactants inside a solar reactor absorb and distribute thermal energy via radiation and convection. Concentrated solar energy raises the reactants' temperature to the dissociation temperature and drives the intensive endothermic decomposition process inside the solar reactor. Therefore, it is important to monitor and control the temperature of the solar reactor. In particular, a uniform temperature distribution without hot spots inside the solar reactor is essential for higher overall efficiency, which can be achieved through meticulous solar reactor design.

Development of validated modeling paradigms can provide a basis for model-based strategies to guide solar reactor design and optimization of solar reactor operating conditions. Comprehensive modeling of solar receivers requires coupling the fluid flow and heat transfer mechanisms in an iterative solar reactor design process. Such models can also be incorporated in dynamic simulation

---

* Corresponding author.
E-mail address: nozalp@d.umn.edu (N. Ozalp).

https://doi.org/10.1016/j.energy.2018.10.033

of the system to design control structures, stress assessment, safety analysis, and process optimization [3,4]. For example, Li et al. designed a 10 kW_{th} horizontal cavity-receiver with an internal array of tubular absorbers for hydrogen and syngas production [5]. Heat transfer within the tubular absorbers was simulated using a Lattice Boltzmann (LB) model which was then coupled with a concentrated solar radiation model of a high flux solar simulator. Their simulation results were used as a basis to design a solar reactor and optimize the operating conditions. Bhatta et al. designed a solar receiver-reactor with the aim to promote the photo-thermochemical conversion of carbon dioxide below 1000 °C [6]. They used computational fluid dynamics (CFD) to analyze their design complemented by experimental results.

Development of validated modeling paradigms can also provide basis for nonlinear model-based control strategies to optimize operation of solar reactors under real-world conditions. For example, Abedini Najafabadi and Ozalp developed a dynamic numerical model for a cylindrical cavity receiver by coupling a radiation model based on the method of Monte Carlo ray tracing (MCRT) with energy conservation equations of the system components [7]. The model was used to simulate the dynamic behavior of the receiver radiated by a parabolic dish concentrator. The model was also used to design a basic control system to regulate the temperature of the receiver.

These numerical models can also be used for troubleshooting problems, such as *hot spots*. Hot spot formation inside a solar reactor reduces the overall efficiency and system safety due to thermal stress and thermal deformation [8,9]. Radiation modelling is a key tool for accurate identification of hot spots in high temperature solar reactor/receiver systems [10].

Examples of notable research done on the improvement of the temperature distribution and elimination of hot spots include a study by Furler and Steinfeld that focused on heat transfer enhancement in a solar reactor containing a reticulated porous ceramic (RPC) foam made of CeO_2 for solar thermal splitting of H_2O and CO_2 [11]. To simulate the heat transfer inside the solar reactor, they presented a transient model by coupling MCRT with CFD techniques. After validating the model by comparing its predictions with the experimental results, the model was used to investigate the effects of several modifications on the performance of the process. An alternative reactor design featuring a conical cavity shape was suggested to improve temperature distribution and avoid hot spot formation. Their simulation results of the new design exhibited more uniform temperature and less heat loss with improvement to the solar-to-fuel efficiency. Wei et al. performed a CFD analysis to simulate a pressurized-air solar receiver comprising 45 parallel tubes subjected to a non-uniform net heat flux [12]. Their results showed that optimization of the flow field can minimize the peak temperature inside the solar receiver. In another study done by Qiu et al., a numerical model was developed to find an optimum solution to increase the temperature in an air tube-cavity solar receiver [13]. Their simulation results showed that changing the reactor geometry can increase the temperature and thermal efficiency by about 35% and 15%, respectively.

This paper demonstrates a methodology that can be used as a design tool to enhance heat transfer inside reactors and receivers. An example is given of a "*hot spot*" problem experienced inside a solar cavity receiver that was tested by our research group using a 7 kW High Flux Solar Simulator (HFSS) [14,15]. Several solutions to improve the temperature distribution and avoid hot spots are provided by modifying the reactor design via a detailed model based on two-dimensional (2D) fluid flow dynamics and heat transfer simulations of the solar receiver. Validation of the model predictions was done at different gas flow rates and different solar receiver aperture sizes by comparing the simulation results with experimental data. The model was then used to evaluate the effect of solar reactor design on temperature distribution and elimination of hot spots. The methodology given in this paper provides guidance on how to develop a custom model of thermal-fluid behavior of the system for optimization of reactor or receiver geometry per specific process needs and creativity.

2. Experimental setup

Fig. 1 shows the configuration of the experimental setup with the solar receiver radiated by a 7 kW HFSS. The solar receiver consists of three main parts: cylindrical cavity, front flange, and back plate, all of which are made of stainless steel 316 L. The cylindrical cavity has a length of 200 mm, inner radius of 60 mm, and outer radius of 75 mm. The air flow is regulated by an Omega FMAA2411 flow rate controller [16], and the air enters the receiver at 20 °C through three tangential inlets symmetrically located on the cavity cylinder wall. The front flange has a thickness of 25 mm and radius of 160 mm. The receiver front flange holds a circular quartz aperture with radius of 60 mm to allow solar radiation entry into the cavity. The receiver back plate has a thickness of 15 mm and radius of 75 mm. An exit port with radius of 10 mm is located at the middle of the back plate for air outflow.

The cylindrical cavity and back plate are covered by Cerablanket ceramic insulation [17] with 40 mm thickness, as shown in Fig. 2. The insulation layer is enveloped by a thin Aluminum shield with thickness of 1 mm. The temperature was measured using *K* type thermocouples with 1 K precision [18] at different locations of the cavity cylinder and the Aluminum shielding. Locations of these thermocouples are shown in Fig. 2-a. Temperature measurements were monitored via LabVIEW after being processed through a NI compact-RIO (cRIO) with an M12 NI9214 module. In order to adjust the amount of radiation entry into the receiver, a simple variable rotary aperture mechanism was used. This mechanism consists of a rotating disc with 12 circular aperture holes with radii in the range of 10 mm–45 mm. As seen in Fig. 1, different size apertures come in place as the circular main disc is rotated by a stepper motor. Details of this mechanism can be found in Ref. [19].

A 7 kW HFSS was used as the radiative heat source which consists of a lamphouse, an ellipsoidal reflector coated by polished Alumina, and a 7 kW Xenon short arc lamp as the light source (Fig. 1). The Xenon arc bulb is located at the first focal point of the reflector, and its radiative power is reflected to the second focal point where the aperture of the solar receiver is located. The focal length of the reflector is 760 mm. The power level of the HFSS is adjusted by changing the input current in the range of 115 A–155 A, mimicking fluctuating solar irradiance at different times of the day from sunrise to sunset.

Experimental determination of the energy entry into the receiver per flux intercepted by each aperture size was characterized via a camera-target method described in Refs. [19,20]. This method includes measurement of flux at the second focal point using a heat gage embedded in a Lambertian target and correlating it with the grayscale values of images captured by a complementary-symmetry metal-oxide-semiconductor (CMOS) camera. In the present study, a Gardon-type heat gage (Vatell Corporation, TG-1000 [21]) with accuracy of ±3% and repeatability of 1% was used. The radiative power input through the aperture was obtained by integration of the radiative flux over the aperture area.

3. Numerical analysis

3.1. Radiation modeling

Distribution of the radiative heat flux at the focal plane of the HFSS and solar cavity walls was determined using an in-house code