International Journal of Heat and Mass Transfer 56 (2013) 309–318

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Thermal analysis and design of a solar prototype for high-temperature processes

M. Isabel Roldán [a,*], I. Cañadas [b], J.L. Casas [c], E. Zarza [b]

[a] Convenio UAL-Plataforma Solar de Almería, Ctra. de Senés, km. 4.5, E04200 Tabernas, Almería, Spain
[b] CIEMAT-Plataforma Solar de Almería, Ctra. de Senés, km. 4.5, E04200 Tabernas, Almería, Spain
[c] Universidad de Almería, Ctra. Sacramento s/n, E04120, La Cañada de San Urbano, Almería, Spain

ARTICLE INFO

Article history:
Received 23 December 2011
Received in revised form 19 September 2012
Accepted 27 September 2012
Available online 27 October 2012

Keywords:
Solar furnace
High-temperature processes
Solar prototype
Heat transfer
Computational Fluid Dynamics (CFD)
3D thermal simulation

ABSTRACT

The thermal evaluation of different prototype configurations for a volumetric solar receiver system designed for the Plataforma Solar de Almería's Solar Furnace has been carried out by means of commercial Computational Fluid Dynamics (CFD) software for 3D numerical simulation.

Simulation results for proposed solar-prototype configurations are discussed and compared to find the optimum configuration by which the interior of the prototype is uniformly heated to 1300–1373 K by an air stream as it passes through the solar receiver. The concentrated solar radiation produces the temperature increase of the air flow. Different geometries and insulation materials with several different thicknesses have been analyzed in order to select the most appropriate prototype configuration in which high-temperature industrial processes can be carried out.

1. Introduction

The need to reduce pollution in intensive power-consuming industrial processes has led to studies of supplying thermal energy with solar-power systems [1]. Previous research has demonstrated the possibility of using solar thermal power in high-temperature industrial processes by means of high-temperature solar receivers [2–6].

Experimental research requires funding and time to build a facility that can reproduce the industrial-process conditions [7,8], this is the case of solar furnaces. In order to optimize resources and to avoid the assembly of several different prototype configurations, theoretical computational modeling is used [9–11].

The simulation and modeling of different solar-receiver configurations are essential for optimizing the receiver-configuration design and increasing the high-temperature-process efficiency. Several authors have determined the heat-transfer analysis of different high-temperature processes in order to evaluate the thermal losses and improve the solar-receiver reliability.

A chemical reactor for the steam-gasification of carbonaceous materials using high-temperature solar process heat has been modeled by applying advanced Monte-Carlo and finite-volume techniques, and a solar reactor prototype was assembled in a high-flux solar furnace. The model included a two-phase formulation that couples radiative, convective, and conductive heat transfer to the chemical kinetics for polydisperse suspensions of reacting particles [12]. This model determined the great influence of the particle size distribution over the absorption coefficient and chemical kinetics, and consequently mass and heat transfer.

The hydrogen production from methane cracking has also been analyzed using Computational Fluid Dynamics (CFD) for the modeling of a high-temperature solar chemical reactor. In this case, additionally to the experimental study, a 2D computational model coupling transport phenomena was developed to predict the mapping of reactor temperature and species concentration, and the reaction extent at the outlet [13]. Simulations results showed the temperature distribution and the locations in which the reaction takes place mainly.

Different geometries were studied for a directly irradiated solar thermochemical reactor used in solar cracking process to characterize the influence of flow behavior on the heat transfer. The radiative heat transfer from carbon particles is considered by providing global absorption and scattering coefficients in the computational domain, and a specific model is selected to enhance the accuracy for the swirl flows. The numerical results showed that the radiative heat transfer mechanism is the dominant means of heat transfer compared to the effects of conduction and convection, and that carving influences significantly the flow behavior [14].

Numerical simulation was also used for the analysis of a thermochemical solar reactor fitted with a porous ceramic foam structure in order to predict the thermal transfer of the volumetric solar

* Corresponding author. Tel.: +34 950387839.
E-mail address: mariaisabel.roldan@psa.es (M. Isabel Roldán).

0017-9310/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.09.063

Nomenclature

c_p	specific heat capacity (J/kg K)
D	diameter (m)
E	energy transfer (J/kg)
F	external body force (N)
g	gravitational acceleration (m/s^2)
h	sensible enthalpy (J/kg)
J	diffusion flux (kg/s m^2)
k_{eff}	effective thermal conductivity (W/m K)
p	static pressure (N/m^2)
Q_{AR}	aspect ratio (dimensionless)
Q_{EAS}	equiangle skew (dimensionless)
Re	Reynolds number (dimensionless)
S_h	volumetric heat source (W/m^3)
S_m	mass source (kg)
t	time (s)
T	temperature (K)
v	velocity (m/s)
x	position in axis x (m)

Greek symbols

α	absorption coefficient (m^{-1})
λ	thermal conductivity (W/m K)
μ	viscosity (kg/m s)
ρ	density (kg/m^3)
τ	stress tensor (N/m^2)

Subscripts

a	air
f	frame
h	hydraulic
is	insulating
j	species
r	refractory

receiver. It was used a local thermal non-equilibrium model to determine the temperature distributions, and it was validated with experimental results. Simulations illustrated that the thermal non-equilibrium phenomena are locally important, and the mean cell size has a dominant effect on the temperature field [15].

Other studies were carried out, in which the reactor modeling is considered as a tool that permits the reactor design improvement. CFD application for modeling fluid flow and mass transfer phenomena in commercial-type annular reactors allows for an in-depth analysis of several different physicochemical processes occurring in chemical reactors in order to achieve an improved performance, a better reliability, and a more confident scale-up of the equipment [16]. Numerical modeling also permits to analyze in detail phenomena such as the kinetics of the catalytic partial oxidation of methane in a square microchannel. These simulations were used to study the laminar flow, and, to determine an approximate laminar flow Nusselt correlation, the heat transport was also solved analytically for plug flow conditions [17]. These examples show that computational modeling can be considered as a versatile and useful tool to simulate and study several different processes.

This work focuses on the use of CFD to analyze fluid mechanics and heat transfer properties in the design and thermal characterization of a solar receiver which can reach high temperatures suitable for industrial-process applications. With this purpose it has been designed a concentrated-solar-energy driven prototype to supply heat at high-temperature ranges. Consequently, a three-dimensional CFD model has been developed by FEM package Fluent to analyze the thermal profiles of several different prototype configurations. The prototype design selected is going to be an element of an open-volumetric-receiver configuration installed at the Plataforma Solar de Almería's Solar Furnace (PSA Solar Furnace) [18].

2. Methodology

2.1. Procedure

The procedure used to carry out this work includes a numerical analysis and the thermal characterization of a prototype for a solar receiver system designed for the PSA Solar Furnace using commercial CFD software (Fluent 6.2.16) for 3D numerical simulation.

The aim of the simulation is the thermal analysis of proposed receiver designs to find the optimal configuration by which the interior of the device is uniformly heated to 1300–1373 K by a hot-air stream as it passes through the setup. Concentrated solar radiation heats the air stream and the inlet temperature is implemented by a user-defined function (UDF) that simulates the air thermal distribution. The hot-air velocity is produced and controlled by a volumetric-receiver blower, therefore forced convection must be considered additionally to conduction and radiation heat transfer.

Different geometries and materials with several different thicknesses have been analyzed to determine the optimal thermal distribution for heating the air stream up to 1300–1373 K using conservation equations for mass, momentum, and energy with the physical properties defined for the fluid and each material in a non-isothermal process. These geometries are made of three layers of material (refractory, insulating and frame) and the simulations compare a tubular design with a square one. The proposed configurations are summarized in Table 1, in which the simulations are divided into three groups:

- The first group was used to select the most appropriate refractory material.
- The best geometry was obtained from the second group of simulations, which considers three different geometries (tubular, square, and tubular with a baffle plate).
- The third group determined the most suitable thickness for the layers of refractory and insulating materials.

The thermal profile and heat flux of every configuration are simulated in 3D to describe them in detail. The 3D simulated-element geometry is created by a pre-processor to define the best mesh for discrete node-per-node calculation.

The results indicate the most suitable geometry, wall-thickness, and material for the process conditions. Based on this preliminary study, an experimental prototype configuration with the best heated-air flux is selected for its construction and installation in the PSA Solar Furnace.

2.2. Preliminary analysis for the inlet thermal distribution

The inlet-temperature profile has been obtained from the analysis of infrared-image sequences by means of Matlab® software. The infrared images considered come from previous tests [5] which have been carried out using the same absorber configuration at a defined focal position. The analysis consists of a thermal-profile