MONTE CARLO RAY TRACING COUPLED CFD MODELLING AND EXPERIMENTAL TESTING OF A 1 KW SOLAR CAVITY RECEIVER RADIATED VIA 7 KW HFSS (HT2019-3541)

Cedric Ophoff

KU Leuven
Mechanical Engineering Department,
Leuven, Belgium

Nesrin Ozalp

University of Minnesota Duluth
Mechanical and Industrial Engineering
Duluth, MN, USA

David Moens

KU Leuven
Mechanical Engineering Department,
Leuven, Belgium

ABSTRACT

Current state-of-the-art development of concentrated solar power (CSP) applications target cost-effective and highly efficient processes in order to establish commercialization of these technologies. The design of solar receivers/reactors and their respective flow configuration have a direct impact on the operational performance of the solar thermochemical processes. Thermal efficiencies, reaction kinetics and other key output metrics are the intrinsic result of the chosen configuration. Therefore, reactor design optimization plays a crucial role in the development of solar thermochemical applications. In this study a computational fluid dynamics (CFD) model of a directly-irradiated cavity receiver has been developed. The CFD domain is coupled with incoming radiation that is obtained by using Monte Carlo Ray Tracing (MCRT). Experimental campaigns running the cavity receiver were carried out using a 7 kW High Flux Solar Simulator (HFSS) as radiative source. Temperature readings were obtained at different locations inside the cavity receiver for both wall and gas temperatures. In order to mimic naturally changing insolation conditions, the HFSS was run at different power levels. Heat flux at the aperture of the solar receiver was experimentally characterized. The acquired heat flux maps validated the intermediate results obtained with the MCRT method. The coupled computational model was validated against the measured temperatures at different locations inside the receiver. Computed temperature contours inside the receiver confirmed the experimentally observed non-uniformity of the axial temperature distribution. The validated analysis presented in this paper was then used as a baseline case for a parametric study. Design optimization efforts were undertaken towards obtaining temperature uniformity and achieving efficient heat transfer within the fluid domain. Enhanced flow circulation was achieved which yielded temperature uniformity of the receiver at steady state conditions. The outcome of this parametric analysis provided valuable insights in the development of thermal efficient solar cavity receivers. Hence, findings of this study will serve as a starting point for future solar reactor design. For

example, it was found that reversing flow direction has an adverse effect on the temperature uniformity inside the receiver. Similarly, increasing the inlet angle does not positively affect the temperature distribution and hence should be chosen carefully when designing a solar reactor.

Keywords: CFD, Monte Carlo Ray Tracing, Solar Receiver

NOMENCLATURE

ρ	density, kg/m^3
μ	molecular viscosity, $Pa.s$
λ	thermal conductivity, $W/m.K$
ε	turbulence kinetic dissipation rate, $J/kg.s$
ϕ_{err}	polar dispersion error, $mrad$
$\hat{n}$	normal unit vector
$\hat{s}$	ray direction unit vector
$\hat{t}$	tangential vector component of surface element
$\vec{v}$	velocity vector
θ_{err}	azimuth dispersion error, $mrad$
θ_m	Rayleigh mode, $mrad$
C	specific heat, $J/kg.K$
dS	surface element
I	unit tensor
k	turbulent kinetic energy, J/kg
L_{cyl}	length of the modeled plasma arc, mm
p	pressure, Pa
Q_{rad}	radiative source term
$\mathcal{R}$	pseudo-random number
R_{cyl}	radius of the modeled plasma arc, mm
R_{in}	inner radius of receiver cavity, mm
T	temperature, K
$I(\vec{r}, \vec{s})$	radiant intensity, W/sr

INTRODUCTION

Thorough understanding of process and reactor performance in solar thermochemical applications require extensive studies on the flow dynamics, heat transfer characteristics and chemical kinetics. With respect to iterative experimental design methodologies Computational fluid dynamics (CFD) can be considered as a cheaper and, in some cases, time-saving tool to predict the outcome of these physical interactions and, thus, can lead to reactor design optimizations. Many CFD studies on solar thermochemical applications have provided valuable insights towards improvement of the processes.

One of the earlier studies done by Steinfeld et al. showed the design of a solar chemical reactor for combined ZnO-reduction and CH₄-reforming [1]. In their study they used CFD simulations to calculate both the velocity flow field and ZnO particle trajectories. These simulations led to design optimizations where the auxiliary flow was minimized while keeping the window and compound parabolic concentrator (CPC) clean of particles. Another CFD study of a volumetric porous receiver indicated the range in which the flow rate of the inert gas should be controlled in order to maximize the temperature while preventing overheating of the receiver materials [2]. Shilapuram et al. conducted a numerical study of the residence time distribution (RTD) on an aero-shielded solar cyclone reactor for thermochemical dissociation of methane [3]. Their findings showed the impact of the screening gas type and flow rate on the RTD. Their study also showed the increasing impact of having a vortex/cyclone flow on the RTD.

While some of these numerical studies are not concerned about the directionality and spatial distribution of the concentrated solar input power, other studies have looked at properly modeling the ray path data. Moghimi et al., for example, used the open source Soltrace code to obtain the non-uniform flux distributions from a Linear Fresnel Collector (LFC), and couple it as a volumetric heat source to a 3D fluid dynamics model of a pipe flow [4], [5]. Martinek et al. coupled the magnitude and directionality of solar energy incident on the window surface of a tube solar reactor to the finite volume (FV) radiation model that was simultaneously solved inside the CFD fluid domain [6], [7]. A similar approach was used by Dordevich, where the MCRT-based VEGAS code was used to obtain ray data which then served as the input of the computational flow domain in which radiation was solved by using the discrete ordinate radiation method (DORM) [8]. Khalsa and Ho used a similar approach where they extensively studied the transformation of radiance distributions on a solar receiver aperture into radiance boundary conditions for CFD models. They were able to validate results obtained in ANSYS FLUENT using the DORM model with results obtained from solar optics codes DELSOL and ASAP [9]–[11]. Besides the use of commercial MCRT softwares, some studies have implemented non-commercial MCRT code. He et al., for example, coupled their MCRT code for a parabolic through collector with the 3D fluid domain of the tubular flow field allowing to capture the non-uniform heat flux distribution at the outer surface of the absorber tube [12]. Kumar et al.

implemented the collision-based Monte Carlo ray-tracing technique combined with the volume fraction distribution of particles obtained from a CFD simulation to model the radiation attenuation of a particle curtain in a free-falling particle receiver [13].

In this paper, a CFD analysis of a directly irradiated solar cavity receiver is presented. Monte Carlo ray tracing is used to obtain directional information of the flux incident at the aperture of the receiver. A parametric study is conducted with the validated computational model in order to evaluate a few design optimizations.

SYSTEM DESCRIPTION

The physical setup that is used in this study consists of a cavity-type receiver that is irradiated by a high flux solar simulator (HFSS). A photo of the system configuration is shown in Figure 1. The HFSS houses a 7 kW Xenon arc bulb that is coaxially aligned with the optical axis of an ellipsoidal reflector. The reflector has a focal length ratio, i.e. f_1/f_2 , of 55/825 as seen in Figure 2 and a truncated diameter of 340 mm. Irradiation of the receiver occurs at the aperture where concentrated radiation enters the cavity through a quartz window.

Figure 1 Experimental setup of the solar cavity receiver and solar simulator

Three gas inlets with an inner diameter of 9.25 mm are tangentially positioned at an angle of 30 degrees with the outer surface normal of the receiver and connected with an inlet manifold. Hence, the main feedstock is equally distributed and develops a vortex flow pattern inside the cylindrical receiver. Cavity walls, with a thickness $t_{316L} = 15$ mm, are made of 316L stainless steel and are wrapped in a 40 mm layer of ceramic fibre insulation [14]. A cover made of aluminum sheet ($t_{Al} = 0.9$ mm) holds the insulation in position and serves as an additional thermal protective layer. A set of 11 K-type thermocouples are used to monitor the cavity wall temperature, the cavity gas temperature and the exhaust gas temperature at different locations alongside the receiver. The cylindrical fluid domain

which is depicted in blue in Figure 2 has a radius $R_{in} = 60$ mm, which is equal to the aperture radius, and a total length of $L = 216.5$ mm. Flow exits the domain through an exhaust with $R_{exh} = 10.46$ mm.

Figure 2 Schematic of the Receiver Configuration

MONTE CARLO RAY TRACING MODEL

In the current study the optical analysis is conducted independently and prior to the CFD analysis. Distinction between both analyses is emphasized in Figure 2. Tracking the ray paths with MCRT allows to extract useful information that can be used as an input for the fluid dynamics model. The MCRT code was developed according to geometrical and optical specifications provided by the reflector and Xenon bulb manufacturer, respectively. The radiation source, i.e. plasma arc between the electrodes of the bulb, was modeled as a cylinder with a radius of $R_{cyl} = 0.59$ mm and a length of $L_{cyl} = 11.7$ mm. Generation of a ray origin is constrained by the arc dimensions and a probability density function (PDF), as can be seen in Figure 3. Two pseudo-random numbers determine the position of ray origin P_0

$$P_0 = \begin{pmatrix} R_{cyl} \cdot \cos(2\pi\mathcal{R}_c) \\ R_{cyl} \cdot \sin(2\pi\mathcal{R}_c) \\ \mathcal{R}_z L_{cyl} \end{pmatrix} \quad (1)$$

where pseudo-random number $\mathcal{R}_c$ is randomly distributed between 0 and 1 and, hence, uniformly distributed the generation of P_0 in the radial direction. Pseudo-random number $\mathcal{R}_z$, on the other hand, is distributed according to a PDF and yields the z -coordinate of P_0 . The density function $PDF(Z)$, as shown in Figure 3, follows the same trend and was calculated from the brilliance distribution of the arc which was provided by the manufacturer [15]. The direction of the ray is then found by considering the normal vector of the differential surface element in P_0 tangent to the cylinder shell surface element going through P_0 . The azimuthal and polar angles are found by considering two new pseudo-random numbers and finally yield the direction vector of the ray in P_0 .

Figure 3 Generation of ray origin inside Xenon arc and probability density function (PDF) for $\mathcal{R}_z$

Once a ray origin and its direction are generated by the radiation source, the next event alongside the ray path is statistically determined. The ray then intersects with the ellipsoidal reflector or exits the reflector through either the front or back opening. In the latter case a ray can still reach the target at the focal plane, i.e. the aperture of the receiver. If a ray intersects with the reflector surface, another PDF will be considered in order to numerically account for specular errors. Figure 4 shows the reflection of a ray at the reflector surface. Direction of the incoming ray on the differential surface element dS is indicated by $\hat{s}_{in}$. Surface element dS is geometrically defined by the normal vector $\hat{n}_{id}$ and its tangential components of dS , $\hat{t}_1$ and $\hat{t}_2$. Irregularities of the reflector surface are accounted for by considering both azimuth (θ_{err}) and polar dispersion (ϕ_{err}). The normal vector $\hat{n}_{re}$ of the real surface, as illustrated in Figure 4, can be correlated to the normal vector $\hat{n}_{id}$ of an ideal reflection as

$$\hat{n}_{re} = \cos \theta_{err} \hat{n}_{id} + \sin \theta_{err} (\cos \phi_{err} \hat{t}_1 + \sin \phi_{err} \hat{t}_2) \quad (2)$$

A Rayleigh distribution was considered for the azimuthal specular error given its better fit for the angular dispersion error and computational superiority [16], [17]. This yields

$$\theta_{err} = \sin^{-1}(\sqrt{-2\theta_m^2 \ln(1 - \mathcal{R}_{err})}) \quad (3)$$

where θ_m is the Rayleigh mode and $\mathcal{R}_{err}$ is a pseudo-random number that is distributed uniformly between 0 and 1. The reflected ray direction $\hat{s}_{ref,id}$ can then be related to $\hat{n}_{re}$ by the following equation

$$\hat{s}_{ref} = \hat{s}_{in} - 2(\hat{s}_{ref,id} \cdot \hat{n}_{re})\hat{n}_{re} \quad (4)$$

Figure 4 Reflection of a ray on surface element dS

From this, the ray path can further be tracked until the next intersection. At each intersection, the absorption of the ray is evaluated through the use of an additional pseudo-random number. By randomly generating a number between 0 and 1 and comparing this with the absorptivity of the surface, which is a value between 0 and 1, a ray is either absorbed or reflected since materials are assumed to be opaque.

For the optical analysis in this paper, 2M rays were generated and tracked. The code was implemented and run in Matlab with an overall computational time of 66 seconds. A target with the same radius as the receiver aperture ($R_{in} = 60$ mm) was defined at the focal plane ($f_2 = 825$ mm) of the ellipsoidal reflector. Ray data were then collected, which included flux density and directionality of each ray.

COMPUTATIONAL FLUID DYNAMICS MODEL

In order to discretize the fluid domain, a solid modeler (SolidWorks®) was used to make the geometry according to the specifications of the receiver. A non-uniform unstructured mesh was generated consisting of tetrahedral cells. The geometry of the computational grid is shown with its dimensions in Figure 5. Effect of the number of cells on the flow parameters was evaluated. Results of this grid independence study are shown in Table 1. Maximum skewness, area-weighted averaged of the exhaust velocity $V_{avg,out}$ and $y+$ at the walls were computed. The maximum skewness for all grid sizes does not exceed 0.8 and can be considered to be acceptable. In the grid size range of 771,597 – 1,583,647 cells the average velocity $V_{avg,out}$ does not vary significantly which indicates that the smallest grid size (771,597 cells) can be considered. The non-dimensional $y+$ value at the walls should be lower than 1 in order to capture the viscous sublayer at the wall [18]. Hence, taking into account the increase of computational time with increasing grid size, a grid size of 771,597 cells was selected for this CFD study.

Figure 5 Computational grid of the fluid domain (771,597 cells)

The CFD model is solved using the FLUENT commercial software package, which uses the finite volume (FV) discretization method. Velocity and pressure are coupled through the SIMPLE algorithm and PRESTO scheme was used for the face pressure interpolation, while second-order upwind scheme was used for the remaining variables. The convergence criterion of the residual for continuity was set at 10^{-3} , and 10^{-6} for the imbalances of momentum and energy. Given the benefits with respect to flows involving rotation and recirculation, the realizable $k-\epsilon$ turbulence model was chosen for modelling turbulence in the flow domain [19]. Steady-state conditions were considered in this study. The governing equations are given below. For the continuity equation:

$$\nabla \cdot (\rho \vec{v}) = 0 \quad (5)$$

The momentum equation:

$$\nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \mu [(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I] \quad (6)$$

where gravitational and external body forces are neglected.

Table 1 Results of the grid independence study

Grid size [# cells]	Max. skewness [-]	$V_{avg,out}$ [m/s]	$Y+_{walls,avg}$ [-]
300,441	0.7916	0.5287	1.1719
771,597	0.7975	0.5243	0.8346
1,003,070	0.7973	0.5240	0.7585
1,583,647	0.7992	0.5243	0.6517
2,808,906	0.7953	0.5279	0.5314

Table 2 Physical properties of the gas, receiver walls, insulation and aluminum shielding

Property	Unit	Formula (temperature-dependent)	Reference
Density	kg/m^3	$\rho_g(T) = \frac{PM}{RT} \rho_{316L} = 7970 \rho_{ins} = 128 \rho_{quartz} = 2214$	[19], [20],[*]
		$C_{p,g}(T) = 947.51 + 0.26667T - 5e^{-5}T^2$	[21]
Specific heat	$J/kg.K$	$C_{p,316L}(T) = 349.145 + 0.469T - 4.007e^{-4}T^2 + 1.34e^{-7}T^3$	[20]
		$C_{p,ins} = 1130 C_{p,quartz} = 700$	[*]
Thermal conductivity	$W/m.K$	$k_g(T) = 0.0146 + 5e^{-5}T$	[21]
		$k_{316L}(T) = 11.133 + 0.013T$	[20]
		$k_{ins}(T) = 0.0504 - 1e^{-4}T + 3e^{-7}T^2$	[*]
		$k_{quartz} = 1.4$	[*]
Viscosity	$kg/m.s$	$\mu_g(T) = 4e^{-5} + 5e^{-7}T - 10^{-10}T^2$	[22]

[*] provided by manufacturer

The equation for turbulence kinetic energy k is represented by

$$\nabla \cdot (\rho k \vec{v}) = \nabla \cdot \left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k + G_k + G_b - \rho \varepsilon \quad (7)$$

and turbulence kinetic energy dissipation rate ε by

$$\nabla \cdot (\rho \varepsilon \vec{v}) = \nabla \cdot \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} C_{3\varepsilon} G_b \frac{\varepsilon}{k} \quad (8)$$

The energy conservation equation for non-reacting flow with a radiative heat source denotes as

$$\nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot (k_{eff} \nabla T + \vec{v} \mu \left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I \right] + Q_{rad} \quad (9)$$

The discrete ordinate radiation model (DORM) is used to solve the radiative transfer equation (RTE) for a finite number of discrete solid angles. For the radiation throughout the computational domain gray properties were assumed. This includes the radiation properties of the flow medium as well as the solid surfaces at the respective locations of interaction. Hence, the RTE denotes

$$\nabla \cdot (I(\vec{r}, \vec{s}) \vec{s}) + (a + \sigma_s) I(\vec{r}, \vec{s}) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s}') \Phi(\vec{s} \cdot \vec{s}') d\Omega \quad (10)$$

Given the quasi-transparent nature of the flow medium both the absorption coefficient a and scattering coefficient σ_s are considered equal to $10^{-9} m^{-1}$ while the refractive index n is equal to 1 and the phase function Φ is assumed isotropic.

Other model constants and parameters used in Equations (5) – (10) are listed in Annex A. Detailed elaboration of these equations can be found in Ref. [19]. Physical properties of the solids and fluids used in the computational domain are shown in **Error! Reference source not found..** The flow medium considered in this study is air.

A total of four boundary conditions were defined in the FLUENT solver. An inlet velocity of 0.8733 m/s was assigned to all three inlet ports and corresponds to an air flow rate of 10 slpm that was used throughout the experimental campaigns. Walls of the cavity receiver, which comprise of the cylindrical shell and back plate, were modeled as no-slip walls with conjugated conduction-convection-radiation heat transfer. The outlet was defined as a pressure outlet at atmospheric pressure. The fourth and last boundary condition, i.e. the aperture window, accounts for the irradiation input of the fluid domain. Due to the bivariate distribution of the flux density and the variable directionality of the rays, this boundary is challenging to be correctly modeled.

The flux density at the window was obtained by conducting indirect heat flux mapping experiments with the 7 kW HFSS as was presented in Ref. [23]. The resulting distribution can be seen as a Gaussian or Lorentzian distribution with a relatively small half width. The average of both the horizontal and vertical direction of the experimentally obtained flux map is shown in Figure 6. In FLUENT these flux data were assigned to the boundary by use of a user-defined function (UDF). Therefore, the experimental data were curve-fitted in order to obtain a mathematical equation that would accurately describe the spatially distributed flux density over the window boundary.

Figure 6 Cross-section of the 7 kW HFSS heat flux map and the fitted Gaussian curve

It was found that in order to capture the wide tail and narrow peak of the flux a summation of three Gaussian distributions, each having a different amplitude and standard deviation, would result in an accurate curve fit. The amplitudes and standard deviations of all three Gaussian curves were obtained through least-square method. The resulting fit is shown in Figure 6. The accuracy of the fit was quantified by an R^2 value of 0.9953 and a root mean square error (RMSE) of 37 615 W/m². Given the peak flux density going up to an order of magnitude of 10⁶ W/m², both values indicate a very good fit.

Directionality of the rays incident on the window is an important aspect to consider. Not considering the ray directions leads to the assumption that the irradiation boundary condition can be considered as a collimated beam with spatially variable flux. Hence, this will yield a different distribution of the irradiation on the inner cavity walls. Directional information was obtained from the numerical MCRT model.

RESULTS AND VALIDATION

First, the numerical ray-tracing model was compared against experimental flux data, as shown in Figure 7. Overall, both trends are very similar. A resolution of 1 by 1 mm was defined at the focal plane of the MCRT model. The resolution affects the smoothness of the curve as can be seen close to the peak flux value where a sharp cornered peak is observed. The coefficient of variance (COV) of the MC estimator is often used as a measure for its convergence which can be considered in MCRT as well. Specifically, the COV is given as:

$$COV^{MC} = \frac{\sqrt{Var^{MC}[U]}}{\sqrt{N_{sim}} E^{MC}[U]} \quad (11)$$

with $E^{MC}[U]$ is the mean and $Var^{MC}[U]$ is the variance of the computed output $U(\theta_i)$, which is a function of the statistical sample θ_i [24]. Consequently, the numerical error can be estimated as

$$\epsilon \sim O\left(\frac{1}{\sqrt{N_{sim}}}\right) \quad (12)$$

which indicates that the numerical error is inversely proportional to the square root of the number of rays traced. In the present code this equates to an order of magnitude of $O(7e^{-4})$.

The overall fit is translated into an R^2 value of 0.92963 and RMSE of 128 818 W/m² evaluated on a 240 x 240 mm² target. Compared to the Gaussian curve fit (Figure 6) these values are higher. Integral of the flux density values on the specified target size showed a total incoming power of 1736 W for the experimental flux map. This, compared to 1812 W for the numerical flux map. Nonetheless, only the directional data of the rays incident at the aperture plane will be extracted from the numerical model. Flux density values are taken from the experimental data.

Figure 7 Experimental versus numerical cross-section of flux density at the aperture plane

For every ray that is incident on the aperture plane at the entry of the receiver the unit vector representing the ray direction is tracked. The azimuth angle θ between the unit vector and the normal vector of the aperture plane is then computed. Given the discretization of the focal plane according to the abovementioned resolution, more than one ray intersects with each surface element.

Figure 8 Average azimuth angle of incoming rays at the receiver window

Consequently, the average direction of all the rays on every surface element are considered. Figure 8 shows the distribution of this angle as a function of the position along the horizontal axis of the aperture. Due to the ellipsoidal geometry of the optical setup, the average angle increases towards the center of the aperture. The distribution is considered to be axi-symmetrical. This directional information is then passed on to the fluid domain through the receiver window boundary condition. Solving the RTE equation by adopting the DORM model then transfers both the directional and flux density information from the window boundary into the fluid domain.

Preliminary simulation results were evaluated prior to the validation of the experimental data. One of the key points of interest is the propagation of the ray directionality through the fluid domain, especially since two different radiation models are used and coupled with one another. The MCRT method has insignificant limitations on computing accurate directionality of the rays. The DORM on the other hand heavily relies on the chosen angular discretization, as was studied in more detail in Ref. [8]. The angular discretization determines the directional subdivision of every octant in a three-dimensional angular space (4π). This in addition to the pixelation parameter, which defines the amount of control volume faces per discretized control angle [19]. Increasing these variable parameters has an exponentially increasing effect on the computation time and, thus, should be kept as small as possible while preserving the directional information throughout the computational flow domain.

Figure 9 Effect of DORM discretization on incident radiation along longitudinal axis of the receiver: (a) DO5x5x5x5 and (b) DO10x10x5x5

An extensive series of simulations with different DORM angular discretizations and pixelations were conducted. Overall,

the simulations showed a more pronounced effect of changing the angular discretization compared to changing the pixelation. For the sake of brevity, two results are shown in Figure 9. These results clearly show the superior performance of choosing an angular discretization of 10x10 and pixelation of 5x5 (DO10x10x5x5) instead of an angular discretization of 5x5 (DO5x5x5x5). In the latter case 4 lobes can be seen as the incoming radiation at the window propagates along the length of the receiver. The DO10x10x5x5 case shows how axi-symmetry is preserved and, therefore, was chosen for the DORM.

Figure 10 shows the velocity streamlines of the air flow medium inside the receiver. From this visualization of the flow behavior the effect of the three tangential inlets is evident. Strong swirl is observed at the front of the receiver as the vortex flow is generated by the inlet configuration. As the flow moves downstream towards the back of the receiver swirl becomes weaker and the flow behaves more like a plug flow. Given the 30° angle with the surface normal on the outer shell of the receiver, the inlets can be configured more tangentially to the surface. This way, vortex flow can be increased which will be beneficial for the heat transfer. Increasing the overall swirl of the flow will lead to more intense mixing and more turbulence which will be beneficial for the convective heat transfer between the fluid and the cavity walls.

Figure 10 Velocity streamlines of the air flow

Axial and tangential velocity were determined at four different positions along the optical axis of the receiver and are plotted in Figure 11a and b, respectively. The vortex runs between the main flow inlets and outlet and yields a maximum axial velocity at $y/L = 0.23$. As the flow approaches the exhaust of the receiver and velocity magnitude increases, the axial velocity peaks shift towards the center. The negative axial velocity observed in the profiles causes reversed flow in the core region which allows to maintain the vortex structure [25]. The tangential velocity profiles in Figure 11b confirm the gradual decrease of swirl strength from back to front, with a final

increase close to the exhaust port. The observed velocity profiles indicate the vortex characteristic of a Rankine vortex, which consists of a forced vortex in the central region surrounded by a free vortex which is free of vorticity.

Figure 11 Flow dynamics of the cavity receiver in terms of (a) axial velocity distribution and (b) tangential velocity distribution in the radial direction and at different longitudinal locations

Monitoring of the receiver temperature is carried out by using a set of eleven K-type thermocouples positioned at different circumferential and axial positions, as can be seen in Figure 12. Distinction is made between gas-temperature-measuring thermocouples and thermocouples reading the receiver wall temperatures. In Figure 12 these are indicated by 'G' for gas thermocouple and 'W' for wall thermocouple.

Figure 12 Thermocouple configuration of the cavity receiver

Symmetry can be considered for the configuration with respect to top-bottom and right-left, respectively. Hence, temperature of the receiver is measured by two gas thermocouples at the front, two gas plus two wall thermocouples at the middle of the receiver length, and two gas plus two wall thermocouples at the back of the reactor. One additional thermocouple monitors the temperature of the gas at the exhaust.

The experimentally measured temperatures at the respective thermocouple locations from Figure 12 are plotted in Figure 13 and compared with the numerical results. The uncertainty within the experimental measurements can be determined following the method defined by the American National Standard Institute (ANSI) in [26]. This divides the uncertainty into two types: *Type A* which is due to the process unsteadiness and repetitiveness of the results, and *Type B* which is due to the uncertainty of the data acquisition system and thermocouples. The uncertainty of each type have been determined to be approximately $\pm 1.6^\circ\text{C}$ and $\pm 3.9^\circ\text{C}$, respectively, at the maximum temperature of 977°C . This gives an overall maximum uncertainty of the experimental results of $\pm 8.5^\circ\text{C}$ with a 95% confidence level. The uncertainty intervals for other temperature measurements are shown in Figure 13, where the absolute uncertainty values decrease with decreasing temperatures and vary within the range of $\pm 5.4^\circ\text{C}$ up to $\pm 8.3^\circ\text{C}$. Uncertainty on the data acquisition equipment is found to be $\pm 0.6\text{ K}$ while the thermocouple uncertainty with respect to temperature relates as $\pm 0.75\% \times T$ (in $^\circ\text{C}$).

Figure 13 Comparison of experimental wall (W) and gas (G) thermocouple readings and numerical results at steady state conditions

Both trends in Figure 13 are significantly similar and indicate the accuracy of the CFD model. A maximum temperature difference of 24 °C (3.1%) was observed for thermocouple G6 while the overall averaged receiver temperature difference is equal to 14.6 °C. The average gas temperature obtained by the numerical model was found to be 291 °C and overestimates the experimental average gas temperature by 1.1%. Similarly the numerical model computed an average wall temperature of 450 °C and overestimates the experimental average wall temperature by only 0.3%. The temperature for both solid (wall) and fluid (gas) gradually increase towards the back of the fluid domain. The radiative flux incident on the walls inherently affects the wall temperature distribution. Moreover, temperatures measured and computed for every axial position along the cavity receiver differ by 2 - 5 °C. Overall, taking into account the uncertainties, the model can be assumed validated.

Based on the validated temperatures at different spatial locations of the receiver the two-dimensional temperature distribution inside the receiver can be obtained. Figure 14 shows the temperature contours of the XY and ZY cross-sectional plane. Both contour planes confirm the axi-symmetrical nature of the temperature distribution. A clear hot spot can be distinguished at the backside of the receiver. This hot spot is mainly caused by the optical configuration of the solar simulator and the resulting directionality of the incoming rays. The flux density incident at the receiver window propagates through the fluid domain and its initial Gaussian-like distribution is gradually attenuated towards the back of the receiver, as seen in Figure 9.

Figure 14 Static temperature contours for the XY-plane and ZY-plane

Nonetheless, incident radiation profile on the back plate of the chamber still contains higher flux density values at the center, which can be observed in Figure 14. The contours can directly be related to the flow pattern observed in Figure 9. The amount of swirl generated by the tangential inlets at the front reduces gradually towards the outlet and turns into plug flow. Hence, at the gas-solid interface of the back plate and rear of the shell the flow is less turbulent and lower velocities have a negative effect on the convective heat transfer between the wall and the gas. At the semi-transparent window boundary heat is lost through re-radiation and convection with the ambient air. While the core region of the fluid domain shows relatively uniform temperature, the current receiver design allows room for optimization towards enhanced heat transfer.

A parametric study was conducted on the current receiver design by considering alteration of the flow configuration. The previously presented CFD model and results depict the base case to which the results of the new flow configurations are compared. Two effects are investigated: since the base case demonstrated a hot spot at the rear end of the receiver, reversing the flow in- and outlet is presumably beneficial for both the temperature uniformity of the cavity walls as well as for the absorption of heat by the gas. The second effect studied in this parametric analysis is the angle of the tangential inlets. As it was shown in Figure 10, stronger swirl is observed at the front of the receiver but tends to diminish towards the back of the receiver. Therefore, increasing the tangentiality of the inlet ports might have an amplifying effect on the vortex generated at the front side of the receiver. Hence, the base case is compared with two different inlet angles. In order to observe the effect of reversing the flow on the temperature distribution in the receiver's

longitudinal y-direction, the gas temperature along the centerline of the fluid domain is plotted.

In order to observe the effect of reversing the flow on the temperature in the receiver, the combined temperature contour and velocity vector fields for both the base case and the reversed flow case are shown in Figure 15a and b. Reversing the in- and outlets of the fluid domain changes the overall flow regime. While the flow conditions in the base case yield a vortex core throughout the domain with minor recirculation zones between the inlets and the window, the inverse flow can be described as a plug-like flow propagating towards the front of the receiver. Two major recirculation zones can be observed between the centerline and the walls of the chamber, which positively affects re-heating of the flow medium. In addition, in Figure 15b it can be seen how the flow advects the heat from the highly irradiated exhaust port, considering Figure 15 as a reference. The comparison of both cases also shows that the colder region close the window is diminished when the flow is reversed.

Figure 15 Temperature contours and velocity vectors in the ZY-plane for (a) the base case and (b) the reversed flow case

To better compare the temperature distribution in the longitudinal direction of the flow domain, the gas temperature along the centerline of the fluid domain is plotted and compared with the reference base case in Figure 16. As aforementioned and observed in the static temperature contours for the base case in Figure 15a the temperature increases towards the back of the receiver. This trend is indirectly caused by the optical setup which redirects the rays from the bulb through the aperture of the cavity where the maximum average angle of a ray with the surface normal of the focal plane equals approximately 7.5° as was shown in Figure 8. This results in a relatively narrow beam irradiating the receiver where the biggest portion of irradiance occurs at the back of the receiver. Consequently, a maximum temperature is observed at $y/L = 1$ (back plate). In the reversed flow case the temperature distribution tends to inverse. Hence, a

maximum gas temperature of 290°C is observed at $y/L = 0.2$. A stronger gradient is observed in the latter case.

Figure 16 Gas temperature distribution along centerline of the cavity receiver

The highly directional nature of the concentrated radiation in the present study has caused the presence of thermal hot spots in the cavity receiver. The uneven temperature distribution inside the cavity not only causes local overheating of receiver components but also yields non-uniform temperature distributions inside the bulk fluid flow. In case of chemically reacting flow, e.g. methane cracking, this might further lead to negatively affecting the reaction kinetics. Therefore, prior to considering chemically reacting flow, the present study considers improvement of heat transfer in terms of temperature uniformity inside the receiver as the primary objective.

Since the effect of reversing the flow should be quantified in terms of improvement on the temperature (non-)uniformity, following equation is used to express the temperature non-uniformity (TNU) of the distributions depicted in Figure 16:

$$\text{TNU} = \frac{\sum |T - \bar{T}|}{n} * 100 \quad (11)$$

where T is the observed temperature for every position y , $\bar{T}$ is the average temperature over the full range $0 \leq y \leq L$ and n is total amount of temperature data points observed. A perfectly uniform temperature distribution would result in $\text{TNU} = 0\%$. For the base case the computed TNU would result to be 19.8% . This compares to a TNU value of 34.6% for the case of reversed flow and thus, the results contradict the presumption of improvement on the temperature uniformity. Since reversing the flow configuration has a major effect on the core region of the fluid domain which translates in the significant difference of both temperature distributions shown above, the effect of the flow closer to the walls should be observed as well.

Figure 17 Wall temperature distribution of the cavity receiver

Figure 17 shows how the wall temperature distribution changes when flow is reversed. Contrary to the gas temperature distribution the difference of both wall temperature trends is less pronounced. The resulting temperature non-uniformities are equal to 19.1% and 16.9% for the base case and reversed flow case, respectively. Thus, reversing the flow has a positive effect on the wall temperature uniformity.

The second effect studied in these parametric analyses is the effect of inlet angles of all three inlet ports. The base case has three inlet ports at an angle of 30° with the normal of the surface tangential to the outer shell. Increasing the angle will increase the strength of the vortex generated at the front of the receiver as well as the turbulence which is beneficial to the convective heat transfer rate between the fluid and the solid wall. Therefore, two larger inlet angles are considered: 45° and 60° .

Figure 18 Velocity streamlines for two alternative inlet angles: 45 degrees (top) and 60 degrees (bottom)

The streamlines for both cases are shown in Figure 18. Difference between both cases is not significantly notable in Figure 18 although more swirl is observed with respect to the base case (Figure 10). Similarly to the reversed flow case, the temperature distribution in the longitudinal direction can be plotted. The distributions are depicted in Figure 19 and show no significant improvement on the temperature uniformity. The TNU was computed for the increased inlet angle cases of 45 degrees and 60 degrees and equal 31.0% and 39.0%, respectively.

Figure 19 Gas temperature distribution for different inlet angles

Furthermore, the wall temperature distributions showed no significant difference with the base case and was therefore omitted in this paper. The temperature non-uniformity for the wall temperature was found to be 19.6% and 19.7% for the 45 degree and 60 degree case, respectively. The effect of more tangential inlet ports has therefore no favorable effect on the temperature uniformity inside the cavity receiver.

CONCLUSIONS

In this paper a computational fluid dynamics study was presented for a solar cavity receiver irradiated by a 7 kW_e high flux solar simulator. Radiation intensity incident at the aperture was obtained via indirect heat flux mapping yielding spatially distributed flux densities. A validated optical model adopting Monte Carlo ray tracing was used to acquire directional information of this irradiation profile and was coupled as a radiation boundary of the computational domain. Air was used as the flow medium and was fed through three tangential inlets at the front of the receiver. The numerical model was compared with experimentally obtained temperature readings at ten different locations and thus, validating the spatial distribution of temperature throughout the cavity with 0.3% and 1.1% error for the average wall temperature and average gas temperature, respectively.

Since a hot spot was observed in both the experimental campaigns and numerical model a preliminary parametric study was conducted in order to evaluate possible improvements of the flow configurations. The effect of reversing the flow direction

and the effect of using different inlet port angles were investigated and were evaluated in terms of temperature uniformity. Following conclusions can be drawn:

- Reversing the in- and outlet of the fluid domain showed a significant effect on the gas temperature distribution while the temperature uniformity along the longitudinal direction of the cavity was not improved.
- Changing the inlet port angles to increase generation of vortex flow and, consequently, improve the heat transfer rate at the fluid-solid interface had a detrimental effect on the gas temperature uniformity.

The results of this study provided insights on optimization of receiver design and the respective flow configuration towards improvement of heat transfer and temperature uniformity. The conclusions obtained from this study can be taken into consideration for the design of directly irradiated cavity receivers.

ACKNOWLEDGMENTS

This research has been funded by Impulse Fund Project No. IMP/14/049 of KU Leuven. The authors would also like to express their gratitude to Dr. M. Helal Uddin for his guidance in the development of the CFD model.

REFERENCES

- [1] A. Steinfeld, M. Brack, A. Meier, A. Weidenkaff, and D. Wullemmin, "A solar chemical reactor for co-production of zinc and synthesis gas," *Energy*, vol. 23, no. 10, pp. 803–814, 1998.
- [2] H. I. Villafán-Vidales, S. Abanades, C. Caliot, and H. Romero-Paredes, "Heat transfer simulation in a thermochemical solar reactor based on a volumetric porous receiver," *Appl. Therm. Eng.*, vol. 31, no. 16, pp. 3377–3386, 2011.
- [3] V. Shilapuram, D. Jaya Krishna, and N. Ozalp, "Residence time distribution and flow field study of aero-shielded solar cyclone reactor for emission-free generation of hydrogen," *Int. J. Hydrogen Energy*, vol. 36, no. 21, pp. 13488–13500, 2011.
- [4] M. A. Moghimi, K. J. Craig, and J. P. Meyer, "A novel computational approach to combine the optical and thermal modelling of Linear Fresnel Collectors using the finite volume method," *Sol. Energy*, vol. 116, pp. 407–427, 2015.
- [5] M. A. Moghimi, K. J. Craig, and J. P. Meyer, "Optimization of a trapezoidal cavity absorber for the Linear Fresnel Reflector," *Sol. Energy*, vol. 119, pp. 343–361, 2015.
- [6] J. Martinek, C. Bingham, and A. W. Weimer, "Computational modeling and on-sun model validation for a multiple tube solar reactor with specularly reflective cavity walls. Part 1: Heat transfer model," *Chem. Eng. Sci.*, vol. 81, pp. 298–310, 2012.
- [7] J. Martinek and A. W. Weimer, "Evaluation of finite volume solutions for radiative heat transfer in a closed cavity solar receiver for high temperature solar thermal processes," *Int. J. Heat Mass Transf.*, vol. 58, no. 1–2, pp. 585–596, 2012.
- [8] M. C. W. Dordevich, "Heat Transfer Analysis of a Lab Scale Solar Receiver," 2013.
- [9] S. S. S. Khalsa and C. K. Ho, "Radiation Boundary Conditions for Computational Fluid Dynamics Models of High-Temperature Cavity Receivers," *J. Sol. Energy Eng.*, vol. 133, no. 3, p. 031020, 2011.
- [10] B. L. Kistler, "A User's Manual for DELSOL3: A Computer Code for Calculating the Optical Performance and Optimal System Design for Solar Thermal Central Receiver Plants." Sandia National Laboratories, Albuquerque, NM, 1986.
- [11] Breault Research Organization Inc., "ASAP NextGen." Tucson, AZ, 2018.
- [12] Y. L. He, J. Xiao, Z. D. Cheng, and Y. B. Tao, "A MCRT and FVM coupled simulation method for energy conversion process in parabolic trough solar collector," *Renew. Energy*, vol. 36, no. 3, pp. 976–985, 2011.
- [13] J. Kim, "RADIATION CHARACTERISTICS OF A PARTICLE CURTAIN IN A FREE-FALLING PARTICLE RECEIVER," pp. 1–11, 2017.
- [14] M. T. Ceramics, "Cerablanket AC2," 2016. [Online]. Available: www.morganthermalceramics.com/media/2510/cerablanket-ac2-_english.pdf.
- [15] Osram, "QM Central Laboratory for Light Measurements OSRAM XBO 7000W HS XL ofr No. 0012 i868," Munich, Germany, 2008.
- [16] G. Johnston, "On the Analysis of Surface Error Distributions on Concentrated Solar Collectors," *Trans. ASME*, vol. 117, no. November 1995, pp. 294–296, 2013.
- [17] T. Cooper and A. Steinfeld, "Derivation of the Angular Dispersion Error Distribution of Mirror Surfaces for Monte Carlo Ray-Tracing Applications," *J. Sol. Energy Eng.*, vol. 133, no. 4, p. 044501, 2011.
- [18] B. Blocken, T. Defraeye, D. Derome, and J. Carmeliet, "High-resolution CFD simulations for forced convective heat transfer coefficients at the facade of a low-rise building," *Build. Environ.*, vol. 44, no. 12, pp. 2396–2412, 2009.
- [19] ANSYS, "Fluent Theory Guide 17.0." 2016.
- [20] VGB Powertech, "Design Report M218H," 1984.
- [21] M. W. Chase, "NIST-JANAF Thermochemical Tables," *J. Phys. Chem. Ref. Data*, vol. Monograph, 1998.
- [22] D. W. Green and R. H. Perry, *Perry's Chemical Engineers' Handbook*, 8th ed. McGraw-Hill, USA, 2008.
- [23] C. Ophoff, S. Korotunov, and N. Ozalp, "Optimization

of Design and Process Parameters for Maximized and Stable Solar Receiver Efficiency TFEC-IWHT2017-18225,” in *Proceedings of the 2nd Thermal and Fluid Engineering Conference*, 2017.

- [24] B. Sudret and D. Kiureghian, “Stochastic Finite Element Methods and Reliability: A State-of-the-Art Report,” *Univ. Calif. Berkeley*, no. November, p. 189, 2000.
- [25] N. Ozalp, M.-H. Chien, and G. Morrison, “Computational Fluid Dynamics and Particle Image Velocimetry Characterization of a Solar Cyclone Reactor,” *J. Sol. Energy Eng.*, vol. 135, no. 3, p. 031003, 2013.
- [26] ANSI/NCSL-Z540-2-1997, “American national standard for expressing uncertainty - U.S. guide to the expression of uncertainty in measurement.” .

ANNEX A

CONSTANTS AND PARAMETERS OF THE GOVERNING EQUATIONS

Turbulent viscosity

with

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

and

$$C_\mu = \frac{1}{A_0 + A_s \frac{k U^*}{\varepsilon}}$$

$$U^* \equiv \sqrt{S_{ij} S_{ij} + \tilde{\Omega}_{ij} \tilde{\Omega}_{ij}}$$

$$\tilde{\Omega}_{ij} = \Omega_{ij} - 2\varepsilon_{ijk} \omega_k$$

$$\Omega_{ij} = \overline{\Omega_{ij}} - \varepsilon_{ijk} \omega_k$$

$$A_0 = 4.04$$

$$A_s = \sqrt{6} \cos \phi$$

$$\phi = \frac{1}{3} \cos^{-1}(\sqrt{6} W)$$

$$W = \frac{S_{ij} S_{jk} S_{ki}}{\tilde{S}^3}$$

$$\tilde{S} = \sqrt{S_{ij} S_{ij}}$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$$

Turbulence dissipation rate constants

with

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right]$$

$$\eta = S \frac{k}{\varepsilon}$$

$$S = \sqrt{2 S_{ij} S_{ij}}$$

and

$$C_{1\varepsilon} = 1.44$$

$$C_2 = 1.9$$

$$\sigma_k = 1.0$$

$$\sigma_\varepsilon = 1.2$$

$$C_{3\varepsilon} = \tanh \left| \frac{\nu}{u} \right|$$