Systematic Approach for Design Optimization of a 3 kW Solar Cavity Receiver via Multiphysics Analysis

C. Ophoff¹, M. Abuseada², N. Ozalp^{3*}, D. Moens¹

¹ Department of Mechanical Engineering, KU Leuven, 3001 Leuven, Belgium

² Department of Mechanical and Aerospace Engineering, University of California Los Angeles, 90095 California, USA

³ Department of Mechanical and Civil Engineering, Purdue University Northwest, 46323 Indiana, USA

Abstract

Use of Computational Fluid Dynamics (CFD) modeling for solar reactor design optimization is a powerful tool in developing efficient reactors for solar thermochemical applications. CFD models can capture complex multiphase flow dynamics to explain the physics linking the heat transfer and flow dynamics on chemical conversion efficiency and continuous operation of a solar reactor. Flux map and temperature data can be numerically obtained via CFD inside the solar reactor, which is a harsh environment for penetrating measurement tools. In this paper, a CFD analysis of a solar cavity receiver operated with air flow and is directly irradiated by a 10 kW_e high flux solar simulator (HFSS) is presented. Radiative heat transfer from the HFSS is modeled by implementing Monte Carlo Ray Tracing (MCRT) for tracking the ray paths from the radiative source to the aperture plane of the receiver. Flux densities were computed at the aperture and validated by experimental heat flux characterization of the solar simulator. The validated MCRT model provided directional information on the radiative flux and was sequentially coupled to the computational fluid domain. Through experimental testing, receiver temperatures up to 1200 K were measured with solar input at the entrance of the cavity of around 3 kW radiative power. Experimentally measured non-uniform temperature distribution in the axial direction was recorded, which developed a hot spot at the backplate of the cavity. Accuracy of the numerical model was validated by comparing the temperature prediction inside the cavity receiver to experimental results. Validated model was then used to conduct a parametric study on the flow configuration to identify run conditions yielding enhanced fluid mixing and improved heat transfer characteristics. The results showed that altering the angle of the main inlet ports proved to be effective in reducing the local hot spot in the back of the receiver. Moreover, results of the parametric study identified the increase of main flow rate as a positive contribution towards temperature uniformity throughout the fluid domain.

Keywords: CFD; Monte Carlo Ray Tracing; Reactor Design; Solar Receiver; Solar Simulator

33 NOMENCLATURE

34 *Latin notations*

35	a	length of ellipsoidal semi-major axis, m
36	b	length of ellipsoidal semi-minor axis, m
37	C	specific heat, J/kg.K
38	I	unit tensor
39	$I(\vec{r}, \vec{s})$	radiant intensity, W/sr
40	k	turbulent kinetic energy, J/kg
41	L_{cyl}	length of the modeled plasma arc, mm
42	$\hat{n}$	normal unit vector
43	p	pressure, Pa
44	Q_{rad}	radiative source term, W
45	$\Re$	pseudo-random number
46	R_{ap}	aperture radius, mm
47	R_{cyl}	radius of the modeled plasma arc, mm
48	R_{in}	inner radius of receiver cavity, mm
49	$\hat{s}$	ray direction unit vector
50	dS	surface element
51	$\hat{t}$	tangential vector component of surface element
52	T	temperature, K
53	$\vec{v}$	velocity vector
54	V_{avg}	average velocity, m/s
55	y^+	dimensionless wall distance
56		

57

58

59 ***Greek symbols***

60 ρ density, kg/m³

61 μ molecular viscosity, Pa.s

62 λ thermal conductivity, W/m.K

63 ε turbulence kinetic dissipation rate, J/kg.s

64 ϕ_{err} polar dispersion error, mrad

65 θ_{err} azimuth dispersion error, mrad

66 θ_m Rayleigh mode, mrad

67

68 ***Subscripts***

69 *abs* absorbed

70 *avg* average

71 *cyl* cylinder

72 *err* error

73 *g* gas

74 *id* ideal

75 *in* incident

76 *ins* insulation

77 *re* real

78 *ref* reflected

79

80

81 ***Abbreviations***

82	AAD	Average Absolute Deviation
83	ARD	Average Relative Deviation
84	BC	Boundary Condition
85	CFD	Computation Fluid Dynamics
86	DORM	Discrete Ordinates Radiation Model
87	FEM	Finite Element Method
88	FVM	Finite Volume Method
89	HFSS	High Flux Solar Simulator
90	MCRT	Monte Carlo Ray Tracing
91	PDF	Probability Density Function
92	PRESTO	PREssure STaggering Option
93	RMSE	Root Mean Square Error
94	RTE	Radiative heat Transfer Equation
95	SIMPLE	Semi-Implicit Method for Pressure Linked Equations
96	TNU	Temperature Non-Uniformity
97	UDF	User-Defined Function

98

1. INTRODUCTION

Efforts towards decarbonization of fossil fuels and mitigation of CO₂ emissions via solar thermochemical processes have significantly increased over the past few decades. This is not only based on the recent developments of high-flux optical systems [1], but is also supported by the increasing number of new research ideas and inventions [2]. Despite the important progress so far, further development of the current technologies is still required in order to be considered as sustainable replacement for conventional industrial processes. Nevertheless, many high-temperature industrial processes have already been investigated to use concentrated solar heat as the input process heat [3]. However, the endothermic chemical reactions of solar-to-fuel conversion processes require optical concentrators that ensure sufficiently high concentration factors to achieve the desired high temperatures. In addition, other technical challenges include the ability of the solar reactor to operate at and withstand extremely high temperatures, as well as maintaining ideal flow conditions for high efficiency while coping with the transient nature of solar radiation. Computational Fluid Dynamics (CFD) analysis has been an essential design tool for the optimization of solar reactors and receivers to address these engineering challenges prior to manufacturing. Combined use of in-house developed numerical modelling [4] and commercially available computational tools have proven their effectiveness in predicting and optimizing performance of solar thermochemical systems.

One of the notable research on examining the reactor design parameters and optimization was done by Christian and Ho to study the effect of solar receiver dimensions and optical properties of the walls on the thermal efficiency of the system [5]. Their results showed that reducing the aperture size, making the inner walls of the receiver more specular and other similar design alterations increased the thermal efficiency of their baseline case from 72.3% to a maximum efficiency of 86.8%. Many directly irradiated windowed cavity type receivers, which are typically used for thermal dissociation of methane or metal oxides, have also been numerically studied over the past few decades. Costandy et al., for example, compared a cylindrical windowed cavity with a sphere-shaped geometry in terms of heat transfer and temperature distribution [6]. Their results demonstrated that temperature uniformity increased inside the fluid domain when the spherical cavity geometry was used. Their numerical model included the reaction kinetics for solar thermal dissociation of methane, which showed that the enhanced uniformity resulted in a higher hydrogen yield. Schunk et al. developed a solar receiver-reactor for the thermal dissociation of zinc oxide [7]. Here, CFD was used to optimize the flow configuration of the argon window screening gas to protect their cavity window from condensation of the gaseous products.

Many numerical studies in the solar thermal and solar thermochemical field have implemented optical analyses through ray tracing to extract the non-uniform flux data at the focal plane of solar concentrator setups. Several studies have used the Monte Carlo ray tracing method (MCRT) to assess the optical

performance of solar receivers [13–16]. Other studies coupled the results of the optical analysis to a computational fluid domain. For example, Przenzak et al. used the radiative output simulated for their helioenergetic installation as an input to their CFD model in order to optimize the geometry and flow conditions of the receiver [12]. Another study performed a CFD analysis on a directly irradiated solar cavity receiver using an in-house developed model. MCRT method was implemented to obtain the radiative flux coming from the solar simulator inside the cavity receiver. Changes on the design and flow parameters were made to obtain more uniform temperature inside [13]. Zhang and Smith studied the effect of geometry factors of a partition-cavity solar thermochemical reactor. In their study they used solar flux data obtained from SolTrace and coupled it to their numerical model in the commercial software package STAR-CCM using a radiative flux boundary condition (BC) [14].

In literature, several studies have combined MCRT with CFD analyses. Costandy et al. used TracePro to obtain ray-tracing data of the concentrated radiation [6]. However, neither the variable flux distribution nor ray directionality was accounted for and the flux was modeled as a collimated beam. This assumption leads to inaccurate irradiation profiles at the inner walls inside the fluid domain, especially when directly irradiated receivers are involved. In a different study about the development of a solid particle receiver a method was presented to transform large radiance distributions, originating from a heliostat field, to a discretized radiance boundary condition. Discretization of the solar flux BC led to the use of macro elements subdivided into smaller micro-elements [15]. The optical software tool DELSOL was used to obtain the flux distribution over the macro elements while the micro elements would account for the solid angle that represents the irradiation from the heliostat field. This same methodology was also applied in the numerical CFD model of a small particle heat exchange receiver [16]. The BC at the dome-shaped window of the receiver was subdivided into 54 smaller boundary conditions, each being assigned an average flux value and beam direction.

While the latter studies considered the importance of radiation directionality involved in solar concentrators, their *discretization*-method involves the time-consuming job of manually modifying the BC sub-elements whenever other radiation conditions are required. Moreover, discretization can lead to other limitations such as the maximum number of rays that can be traced in the optical software.

Other approaches and methods that have modeled the directionality of concentrated solar radiation have also considered the coupling of MCRT with the Finite Volume Method (FVM). Qiu et al., for example, studied the optical and thermal performance of linear Fresnel concentrators [17]. The MCRT model, developed in FORTRAN, provided a non-uniform solar flux distribution that was assigned to the absorber outer wall as an irradiance heat source. A similar approach was adopted but for a different application by Du et al. [18], where the outcome of the MCRT code is coupled as a source term to the porous absorber

domain within a pressurized volumetric receiver. Moreover, the FVM model is simultaneously coupled to a finite element (FEM) model of the quartz window which is used to determine the stress characteristics of the quartz structure. Another study used the coupled MCRT-FVM scheme for conveying the radiative flux information onto absorber tubes inside a solar receiver with specularly reflective cavity walls [19]. A solar flux map obtained from SolTrace is passed onto the cavity domain through the cavity window. The MCRT method is then used to determine the stochastic paths of each ray impinging on the window and trace them through a sequence of surface interactions until fully absorbed. The experimentally validated results of this study showed an acceptable temperature profile match of 4% with temperatures reaching up to 1820 K.

Similar to the above MCRT-FVM coupled models, this paper presents a CFD optimization on the design of a 3 kW solar cavity-receiver that is irradiated by a 10 kW_e High Flux Solar Simulator (HFSS). A systematic approach is provided to develop a model to manufacture a new receiver with optimized geometry for more uniform temperature distribution. Similar to outdoor solar concentrators, optics of indoor solar simulators result in a non-collimated beam of radiative power at the window of the cavity. Therefore, variability on the direction of the incident radiation needs to be accounted for as accurately as possible. In this CFD model, which is solved using the FVM method, directionality of the ray paths incident at the aperture of the receiver is obtained through the MCRT method. This information is then passed on to the fluid domain without limitations on discretization and independent of the mesh refinement. The three-dimensional flow field is solved for steady-state conditions. Prior to being used as a design optimization tool, the CFD model is validated against experimental data. The validated receiver design is altered towards enhanced flow and heat transfer and the results are presented in terms of a parametric study per systematic design optimization.

2. SYSTEM DESCRIPTION

The system studied consists of a cavity receiver irradiated by a solar simulator as depicted in Figure 1. The radiation source in the experimental setup consists of an ellipsoidal reflector that houses a 10 kW_e xenon arc discharge bulb, as shown in the schematic in Figure 1a.

Figure 1 (a) Schematic of the experimental setup, and (b) Photo of the cavity receiver irradiated by the solar simulator.

The reflector concentrates radiation generated by the xenon arc, which is placed at the first focal point (f_1 = 75 mm from the apex), into the secondary focal point f_2 at a distance of 900 mm from the reflector apex. The receiver is coincidentally aligned with the optical axis of the concentrator. The aperture of the receiver is aligned with focal point f_2 . Power generated by the xenon bulb is adjusted by controlling the current input within the range from 80-200 A, where 160 A was considered as the maximum power level to prevent

irreversible damage of the receiver. The maximum HFSS power level can then be correlated to the maximum irradiation during day time, i.e. noon, for experiments mimicking the transient nature of the sun.

For this study, a 3 kW solar cavity-receiver was designed and manufactured which is depicted in Figure 1b, and in more detail in Figure 2. A cylindrical volumetric cavity was chosen in the present study for its' ability to maximize the net absorption of concentrated radiation, which is essential in high temperature solar thermochemical processes [20]. The length was determined based on a prior study investigating the effect of the D/L ratio on the temperature distribution inside the cavity domain [13]. The receiver body is made of 316L stainless steel with a melting point of around 1650 K and has a thickness of 1.5 cm in the present setup. Irradiation of the receiver occurs at the window, which is made of fused quartz and has a thickness of 6.35 mm. The aperture has a radius of $R_{ap} = 4.4$ cm (#2 in Figure 2). The window is clamped between two flange plates with ceramic fiber seals to seal the inner chamber from the atmosphere and to prevent leakages. The receiver body is wrapped with 12.2 cm of ceramic fiber insulation and held in place with an aluminum sheet cover [21] (#5 in Figure 2). The receiver chamber has an inner diameter of 10.2 cm and length of 29.3 cm. The receiver is fed with air flow through 4 inlet ports positioned at the back of the chamber and which are positioned behind the receiver, and thus not visible, in Figure 2. Each inlet is tilted at an angle of 40 degrees with the tangent of the cylindrical body. The exhaust of the receiver is positioned at the front bottom of the chamber, slightly shifted from the vertical centerline of the receiver. A secondary flow stream configuration is used at the front of the receiver which primarily serves as a window screening flow (#4 in Figure 2). Four inlet streams are pointed towards the window at 30-degree angle of attack. A set of ten K-type thermocouples (TC in Figure 2) are placed at 3 axial locations, two at the front and four in the middle and rear end, and are evenly distributed over the outer wall circumference to monitor the wall temperatures during experiments. Two additional thermocouples monitor the backplate temperature at two radial locations, i.e. center and 3 cm above center. For all the thermocouples mounted around the cavity, holes were drilled on the outer shell leaving a shell thickness of approximately 2 mm of stainless steel between the chamber and the hole. The probes are fixed in such a way that the measuring tip of the probe touches the wall. Further information on the experimental setup can be found elsewhere [22].

Figure 2 Photo of the cavity receiver with its components: (1) High Flux Solar Simulator, (2) Aperture window, (3) Exit port, (4) Window screening inlet, (5) Insulation and shielding, and (TC) Thermocouples

3. NUMERICAL MODEL

3.1. Determination of radiative flux via MCRT

Prior to conducting the experiments with the cavity receiver, meticulous characterization of the HFSS was obtained via indirect heat flux mapping method. Elaboration of the flux characterization for that particular setup is explained in [23]. Flux map of the HFSS provided information on the spatial distribution of the flux density magnitude at the focal plane. In order to determine directionality and the respective spatial-dependent angular distribution of ray directions, an optical analysis was carried out via Monte Carlo ray tracing method. The optical setup modeled via MCRT code and the respective occurrences along a ray path are depicted in Figure 3.

Figure 3 (a) Schematic illustration of the optical setup with (b) Generation of the ray origin P_0 and (c) Surface reflection treatment in the MCRT method.

The ray tracing procedure initiates with generation of a random ray origin. For this, an origin point P_0 is computed within the xenon arc geometric bounds:

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

where R_{cyl} is the radius of the arc bounds and equals to 0.285 mm, L_{cyl} is the length of the arc and equals to 11.7 mm. Probability of a ray originating at a particular y-coordinate, i.e. along the optical axis, is directly correlated to the brilliance distribution obtained from technical reports of the xenon lamp [24]. Figure 3b shows the probability density function (PDF) that was fitted to the brilliance data and is a function of the y-coordinate of the ray.

The PDF is then used to generate a pseudo-random number $\mathcal{R}_y$ which computes the y-coordinate of P_θ in Equation (1). In the radial direction of the xenon arc P_θ is assumed to be randomly generated. Hence, the pseudo-random number $\mathcal{R}_c$ is uniformly distributed and results in a value between 0 and 1. Once P_θ is computed, ray direction is generated using a different set of pseudo-random numbers to determine the polar and azimuthal angles with respect to the shell surface element in P_θ . In a similar way as the PDF used for determining the y-coordinate of P_θ , the pseudo-random number that determines the azimuthal direction of the initiated ray is bound by a PDF. This PDF was fitted to the luminance polar plot of the xenon arc, that was provided by the manufacturer [25].

Once directionality of the ray is known, the next event along the ray path can be tracked and computed. Either a ray exits the ellipsoidal reflector through one of both truncated holes, or it intersects with the specular surface of the reflector. In the former case, the ray can eventually reach the focal plane but outside the area of interest (i.e. aperture size of the receiver) while in the latter case surface reflection treatment is considered. Another pseudo-random number is used to determine whether a ray is absorbed by the reflector. Based on an absorptivity of 0.9 [26], if the random number that is computed between 0 and 1 is higher than 0.9, the ray is absorbed, and the path of the ray is terminated. Otherwise direction of the reflected ray is computed, as shown in Figure 3c. For the surface treatment a non-ideal reflection is assumed by considering a normal vector $\hat{n}_{re}$ of the reflector surface element on which the ray is incident which is distributed by a Rayleigh distribution with respect to the ideal normal vector $\hat{n}_{id}$. Hence, $\hat{n}_{re}$ can be expressed as

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

where $\hat{t}_1$ and $\hat{t}_2$ are the tangential components of surface element dS . The Rayleigh distribution yields

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

where θ_m is the Rayleigh mode and $\mathcal{R}_{err}$ is a pseudo-random number that is distributed uniformly between 0 and 1. In the current optical model a Rayleigh mode of 1.5 mrad was used which is within acceptable range compared to values used in similar models elsewhere [27], [28]. The polar dispersion error then follows from Eq. 3 and varies per ray-iteration as it depends on the randomly generated number $\mathcal{R}_{err}$. The azimuthal dispersion error ϕ_{err} , on the other hand, is presumed to vary uniformly between 0 and 2π rad.

Hence, the real normal vector $\hat{n}_{re}$ of surface element dS (Figure 3c) will account for possible surface irregularities. The reflected ray direction $\hat{s}_{ref,id}$ in Figure 3c can then be related to $\hat{n}_{re}$ by the following equation

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

The ellipsoidal reflecting surface is defined by the reflector geometry. Therefore, the mathematical function describing an ellipsoid is used to determine the point of intersection:

$$\frac{x^2}{b^2} + \frac{(y-a)^2}{a^2} + \frac{z^2}{b^2} = 1 \quad (5)$$

where $a = 440$ mm and $b = 213$ mm are the lengths of the semi-major and semi-minor axes, respectively. Point of reflection is bounded by y which is constrained by the length of the reflector.

A target is placed at the focal plane of the optical setup to determine the flux data of the incident rays. Figure 4 shows one half of the flux distribution cross-section for different numbers of rays traced. The plot shows clear fluctuations in the tail of the distribution when 200k or 500k rays are used in the model. These fluctuations dampen out as the number of rays increases. In order to assess reliability of the numerical results, effect of the number of rays traced in the model is examined by evaluating convergence of the flux results. Table 1 summarizes the flux data in terms of average and total fluxes on specified target sizes, as well as the peak flux density at the focal plane, the computational time and an average relative difference (ARD). The peak flux is significantly higher for the computation with 200k and 1M rays.

Figure 4 Numerical flux distribution ($x+$ axis) for different numbers of rays traced

However, for the other ray numbers not much difference is observed. This is mainly due to the fact that the radiative energy from the arc source is evenly distributed over each ray and the ray paths are independent of the number of rays used in the model. Since smoothness of the distribution decreases with decreasing number of rays, as seen in Figure 4, the ARD is used to quantify this smoothness. It computes the average difference between values of a given flux profile and the average value (i.e. average value of all six cases in Table 1) at the respective spatial locations. Starting around 10M rays the values of ARD start to stabilize. Table 1 also includes the computational time required to run the code. Going beyond 20M rays, the computational time increase significantly without significant changes or improvement of the ARD. Considering the ARD and computational time to be the determining factors, a value of 20M rays was chosen to provide sufficiently reliable data.

Table 1: Effect of traced rays number on resulting flux

		Number of rays traced						
		200k	500k	1M	10M	20M	50M	100M
Peak flux	$[kW/m^2]$	6842	5310	5310	5043	5030	5091	5044
Avg. flux – 10x10 mm ²	$[kW/m^2]$	2723	2723	2753	2756	2749	2752	2752
Avg. flux – 50x50 mm ²	$[kW/m^2]$	725	724	724	725	725	725	725
Avg. flux – 160x160 mm ²	$[kW/m^2]$	70	70	70	70	70	70	70
Tot. power – 10x10 mm ²	$[W]$	329	329	333	333	333	333	333
Tot. power – 50x50 mm ²	$[W]$	1887	1882	1883	1886	1886	1886	1886
Tot. power – 160x160 mm ²	$[W]$	4069	4072	4074	4078	4078	4077	4078
ARD	$[%]$	36.8	19.9	16.6	9.3	7.5	7.2	5.8
Computational time	$[s]$	52	130	265	2,896	5,709	14,367	28,112

3.2. Computational fluid domain

The fluid domain was modeled according to the aforementioned dimensions of the receiver using Solidworks® as the solid modeler. The resulting unstructured computational grid with its respective inlets, outlet and irradiated boundary is shown in Figure 5. In order to prove accuracy of the numerical results for the given grid size, four mesh sizes are compared in a grid independence study. Results of this study are shown in Table 2. For each grid the maximum skewness, average velocity at the exhaust $V_{avg,out}$, y^+ at the walls and volumetric average velocity of the fluid domain $V_{avg,vol}$ are quantified. Refinements of the grids were obtained by changing the maximum face size from 3 mm to 2 mm, 1.8 mm and 1.5 mm, respectively.

Figure 5 Computational fluid domain (858,794 tetrahedral cells)

All four grid sizes have an acceptable maximum skewness that does not exceed 0.8. Since a smaller y^+ value is beneficial to capturing the viscous sublayer at the wall, smallest y^+ value is targeted [29]. The average y^+ values at the wall for the different grid sizes are listed in Table 2. All values are below 1 and, thus, yield several wall-adjacent cells within the viscous sublayer which have a beneficial effect on obtaining accurate wall thermal results. Given that all y^+ values are below 1, the variable is not considered the determining factor in the grid independence study. The average outlet velocity $V_{avg,out}$ decreases when the grid is refined with a factor of approximately 2 from 367,583 tetrahedral cells to 858,794 cells. The outlet velocity is much less affected when the latter grid undergoes further refinement, i.e., grid sizes 1,101,011 and 1,699,136 cells. This indicates that refining the grid beyond 858,794 cells does not affect the monitored variable. A similar trend occurs for the volumetric average velocity $V_{avg,vol}$. Hence, an unstructured grid of 858,794 cells was chosen for this CFD study.

Table 2: Results of the grid independence study

Grid size [number of cells]	Face size [m]	Maximum skewness [-]	$V_{avg, out}$ [m/s]	y^+ [-]	$V_{avg,vol}$ [m/s]
367,583	0.0030	0.7966	0.5684	0.9958	0.0789
858,794	0.0020	0.7783	0.5639	0.7015	0.0803
1,101,011	0.0018	0.7907	0.5632	0.6392	0.0806
1,699,136	0.0015	0.7996	0.5634	0.5482	0.0812

The FLUENT solver is used for the computation of the CFD model. FLUENT uses the finite volume method to discretize the Navier-Stokes equations, where the governing equations are integrated over every control volume. The computation is performed assuming steady-state conditions and incompressible flow. The SIMPLE algorithm is used for coupling velocity and pressure and PRESTO scheme was used for the interpolation of face pressures. Second order upwind schemes were applied to the remaining flow variables. Turbulence is assumed inside the computational domain for which the standard k- ϵ model will be used given the presumed rotational and recirculation nature of the flow [30].

3.3. Governing equations

Since the computational domain is solved for steady-state conditions and the fluid flow is turbulent and incompressible, the governing equations are expressed as follows:

The continuity equation:

$$\nabla \cdot (\rho \vec{v}) = 0 \quad (6)$$

The momentum equation:

$$\nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \mu [(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I] \quad (7)$$

k-equation:

$$\frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon \quad (8)$$

ϵ -equation:

$$\frac{\partial}{\partial x_j} (\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S \epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}} + C_{1\epsilon} C_{3\epsilon} G_b \frac{\epsilon}{k} \quad (9)$$

Energy equation:

$$\nabla \cdot (\vec{v} (\rho E + p)) = \nabla \cdot (k_{eff} \nabla T + \vec{v} \mu [(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I]) + Q_{rad} \quad (10)$$

The radiative heat transfer equation (RTE) for the computational domain is solved using the discrete ordinates radiation model (DORM) and denotes as

$$\nabla \cdot (I(\vec{r}, \vec{s})\vec{s}) + (a + \sigma_s)I(\vec{r}, \vec{s}) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s}') \Phi(\vec{s} \cdot \vec{s}') d\Omega \quad (11)$$

Detailed description of the governing equations above can be found in Ref. [30]. An overview of the model constants and parameters are listed in the Annex A of this paper. Material properties of the solids and fluids are defined in Table 3. Since air is used as the feedstock, the flow medium can be presumed quasi-transparent to incident radiation. Therefore, absorption coefficient a_g and scattering coefficient σ_g are both equal to $1e^{-9} \text{ m}^{-1}$, which are the default values for air in the Fluent material database. The scattering phase function is presumed isotropic [30]. Furthermore, wavelength independence of the radiative properties is considered in the computational model.

Table 3: Thermo-physical properties of the solids and fluid used in the CFD model.

Property	Unit	Correlation (temperature-dependent)	Reference
Density	kg/m^3	$\rho_g(T) = \frac{PM}{RT} \mid \rho_{316L} = 7970 \mid \rho_{ins} = 128 \mid \rho_{quartz} = 2214$	[30], [31],[*]
		$C_{p,g}(T) = 947.51 + 0.26667T - 5e^{-5}T^2$	[32]
Specific heat	J/kg.K	$C_{p,316L}(T) = 349.145 + 0.469T - 4.007e^{-4}T^2 + 1.34e^{-7}T^3$	[31]
		$C_{p,ins} = 1130 \mid C_{p,quartz} = 700$	[*]
Thermal conductivity	W/m.K	$k_g(T) = 0.0146 + 5e^{-5}T$	[32]
		$k_{316L}(T) = 11.133 + 0.013T$	[31]
		$k_{ins}(T) = 0.0504 - 1e^{-4}T + 3e^{-7}T^2$	[*]
		$k_{quartz} = 1.4$	[*]
Viscosity	kg/m.s	$\mu_g(T) = 4e^{-5} + 5e^{-7}T - 10^{-10}T^2$	[33]

[*] provided by the manufacturer

3.4. Boundary conditions

Referring to Figure 5 for visual interpretation, distinction of five boundary conditions is made for the fluid domain: (1) four main flow inlets which are tangentially configured to generate a vortex flow pattern are modeled as Dirichlet boundary conditions with an inlet velocity of 0.6086 m/s corresponding to a total volume flow rate of 10 L/min of air; (2) the window screening flow is not considered at this time (i.e. $\dot{V}_{screen} = 0 \text{ m/s}$); (3) cavity walls are modeled as no-slip boundaries with multi-layer shell conduction and outer shell convection ($h = 20 \text{ W/m}^2.\text{K}$) and radiation heat transfer to the ambient air. In terms of internal radiation, the stainless steel walls are considered opaque to the incoming radiative flux. An internal emissivity of 0.8 is assigned to the wall [26] and 10% of the reflected radiation incident on the walls is considered to be diffusely reflected. (4) the receiver exhaust is defined as an atmospheric pressure outlet.

The most crucial boundary condition (5) of the computational domain is the aperture of the cavity. Although this boundary is dominated by the incident radiation originating at the solar concentrator, it is considered as a semi-transparent wall with a thickness of 6.35 mm. A transmissivity of 0.9 is accounted for as well, which is in accordance with common transmissivity values for solar cavity windows [34]–[36]. The window boundary accounts both for direct radiative losses as well as convective losses to the environment outside the fluid domain. Both the outer free stream temperature and external radiation temperature were set to 293 K and a free convection heat transfer coefficient of 20 W/m².K were assigned to the outer surface. With respect to the incident radiation, both directionality and intensity are accounted for. As aforementioned, the spatially variable flux density at the focal plane (i.e. receiver window) of the optical setup was obtained via the indirect heat flux mapping method. Elaboration of this method for the optical setup used in this study is described in Ref. [23]. Considering axi-symmetry of the obtained Gaussian-like distribution, the experimental data was curve-fitted using the least square method as shown in Figure 6. A curve described by a superposition of three Gaussian distributions provided an accurate fit where the fitting parameters were found to be $R^2 = 0.9998$ and $RMSE = 17,515$ W/m². A user-defined function (UDF) was then assigned to the boundary depicting the axi-symmetrically revolved Gaussian superposition curve.

Figure 6 Horizontal flux density distribution of the experimental flux map, the fitted Gaussian superposition curve and the numerical results of the MCRT model

Ray directionality was accounted for by using the ray-tracing data obtained from the MCRT model. Incorporating the directional information of radiation yields more accurate flux distributions on the cavity walls inside the fluid domain. Similar to the three-dimensional flux profile at the window boundary plane, ray directionality was assigned as a multi-variable UDF to the window as well. The shape of the curve used

in the latter UDF was obtained from the results of the MCRT simulations. The resulting flux density distribution at the focal plane of the optical setup obtained through ray-tracing is shown by the blue dashed line in Figure 6. Comparison with the experimentally obtained flux data shows equal peak flux with discrepancies in the steeper portion of the distribution and the tail of the distribution. In terms of agreement between numerical and experimental data, an R^2 of 0.9123 and a 0.6% under-estimation of the total incident flux on a 240 mm by 240 mm target was determined. The MCRT model is used to extract directional information on the flux and therefore the results of the numerical model are considered to be within acceptable limits. Knowing the point of reflection on the ellipsoidal reflector and direction of the ray $\hat{s}_{ref}$ (Eq. 4) incident at the window of the receiver, a distribution for the angle θ of the ray and the surface normal of the window can be obtained. Figure 7 shows how the angle θ changes in the radial direction. A maximum angle of 7.5° is observed at the theoretical focal point and decreases in a non-linear manner towards the edges of the window. FLUENT requires the unit vector components of the incident beam or rays. Therefore, from the available ray data, i.e. $\hat{s}_{ref}$, a spatial distribution can be obtained for all three unit vectors $\hat{i}$, $\hat{j}$ and $\hat{k}$ at the receiver's window. With the use of the surface fitting toolbox a bivariate distribution is obtained and converted into C code. The latter is used in the UDF that is assigned to the window boundary in FLUENT. Hence, the user-defined function (UDF) which is a function of the spatial coordinates at the window boundary plane depicts the ray direction in terms of unit vector components. This approach saves a considerable amount of time and can be more accurate when compared to alternative approaches where the window boundary would manually be discretized into cells with constant ray directions [15], [16]. Limitation of this UDF approach is the unavailability of an analytical relation that describes the spatially dependent variable.

Figure 7 Incident ray angle distribution at the receiver's window

4. RESULTS AND DISCUSSION

4.1. DORM discretization

The DORM that is used throughout the computational domain solves the radiative transfer equation for a finite number of discretized solid angles. For a three-dimensional fluid domain, the angular space 4π is divided into $8 \times N_\theta \times N_\phi$ solid angles for the polar and azimuthal directions respectively. In addition, in order to account for overhanging control angles due to unaligned angular discretization of the DORM with the computational mesh a sub-discretization is assigned to the solid angle in the form of pixelation, i.e. $N_{\theta p} \times N_{\phi p}$ pixels are assigned to each solid angle [16], [30]. Dordevich developed a computational model for a small particle heat exchange receiver and performed an extensive study on the effect of the DORM discretization [16]. He observed a linear trend in terms of computational time as a function of angular discretization and concluded that directional dependent radiation requires sufficient levels of discretization. In order to find a compromise in terms of computational time and accurate assessment of numerical versus experimental results of the solar receiver, a preliminary study on the DORM discretization of the fluid domain was performed.

In this CFD study, four different discretizations have been compared with $N_\theta \times N_\phi \times N_{\theta p} \times N_{\phi p}$ indicating the angular subdivision and pixelation. Since the fluid domain contains air which is quasi transparent to the incoming radiation, only energy equation is solved in conjunction with the RTE to save computational time. The results of this analysis are shown in Figure 8.

Figure 8 depicts radiation throughout the fluid domain by plotting the incident radiation at eight radial cross-sections equally spaced at 4 cm interval. The incident radiation accounts for all radiative fluxes from neighboring cells. Hence, both the propagation of radiative flux originating from the solar simulator as well as the radiative emission from the hot walls are accounted for. However, irradiation through the window dominates the thermal emission of the walls. Therefore, the incident radiation is a good measure for evaluating the performance of the DORM discretization on the concentrated radiation entering the fluid domain. In Figure 8, one can clearly observe the dispersion of incident radiation as it reaches the backplate of the receiver. The DORM discretization of $5 \times 5 \times 3 \times 3$ shows formation of four lobes which does not agree with the axisymmetric behavior that is expected. When the angular discretization is increased by a factor of 2, i.e. DORM $10 \times 10 \times 3 \times 3$, the lobes disappear but axis-symmetry is not preserved at the back part of the cavity receiver. Another notable observation is the effect of pixelation. Mutual comparison of the $10 \times 10 \times 3 \times 3$ and the $10 \times 10 \times 15 \times 15$ DORM model indicates that increasing the number of pixels per solid angle does not affect the propagation of radiation in the computational domain. Increasing the subdivision of solid angles from 10×10 to 15×15 solid angles resulted in improved axial symmetry. Nevertheless, the increased discretization had a significant impact on the computational time and thus, the $15 \times 15 \times 5 \times 5$ discretization was considered in the remainder of this paper.

4.2. Model validation with experimental data

Validation of the numerical model was obtained by comparison of the computed wall temperatures with the wall temperatures measured at different positions along the receiver body as shown in the photo of the experimental setup seen in Figure 9a. A total of twelve K-type thermocouples are used to monitor the spatial temperature distribution of the cavity receiver. Exact position of the thermocouple with respect to the aperture window is shown in Figure 9b. Temperature is measured at three axial positions, i.e., front (T_f), middle (T_m) and back (T_b). At each axial position temperature is measured at two (front) or four (middle and back) radially distributed locations with 90° angles. Two additional thermocouples (T_{bp}) are fixed on the backplate to determine the radial temperature distribution but have not been considered in the present study since temperatures observed at the backplate were at the limit of the operating temperature of the stainless steel. Therefore, uncertainty associated with these measurements might be high.

Figure 9 (a) Photo of the cavity receiver with annotated thermocouples and (b) configuration of all twelve thermocouples

The experimentally measured wall temperatures at the respective thermocouple location are plotted in Figure 10. A gradual increase in temperature towards the back of the receiver can be observed. The uncertainty within the experimental measurements can be determined following the method defined by the American National Standard Institute in [37]. 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 ± 1.6 K and ± 3.9 K, respectively, at the maximum temperature of 1250 K. This gives an overall maximum uncertainty of the experimental results of ± 8.5 K with a 95% confidence level. The uncertainty intervals for other temperature measurements are shown in Figure 10, where the absolute uncertainty values decrease with decreasing temperatures and vary within the range of ± 7.2 K up to ± 9.5 K. These also account for the uncertainty of the data acquisition equipment which were determined to be ± 0.6 K.

The numerical model, shown in Figure 10 as well, depicts a similar trend with a more pronounced temperature gradient in the longitudinal direction of the cavity. An average receiver wall temperature of 1087 K was obtained from the experimental campaign and compares to 1081 K based on the computational results. Taking into consideration all ten thermocouples, an average absolute deviation (AAD) between experimental and numerical results was computed and equals to 47 K, which corresponds to an average relative deviation (ARD) of 4.2%. The relative temperature errors between the experimental measurements and the numerical results in Figure 10 are -0.85% as average for the front two thermocouples T_{f1} and T_{f2} . This error increases to an average of -5.7% for T_{m1} to T_{m4} and reduces to an average error of 4.3% for T_{b1}

to T_{b4} . Nevertheless, the maximum relative error is around -6.4% (T_{m4}) and corresponds to a maximum absolute error of 71K. The error falls within an acceptable range of accuracy. In addition, the agreements of the respective T_f , T_m and T_b temperatures in Figure 10 indicate the radial temperature homogeneity within the cavity receiver.

Assuming that the radiative flux incident on the walls inherently affects the wall temperature distribution, the semi-constant temperature differences for $T_{m,1-4}$ and $T_{b,1-4}$ possibly indicates that the angle distribution for the ray directional information assigned at the window boundary of the numerical domain was underestimated. In other words, the numerically computed incident ray angle, as seen in Figure 7, should be relatively higher. This underestimation of the ray angle leads to a numerical overestimation of the incident radiation intensity at the back of the cavity receiver. Therefore, extracting information on the directionality of the radiative flux experimentally would result in more accurate input data and will be taken into consideration in future work [38].

Figure 10 Comparison of the experimentally measured wall temperatures and the numerical results

Flow behavior has been visualized by the velocity streamlines in Figure 11a. The thermocouple planes T_f , T_m and T_b in Figure 10 respectively correspond to the dimensionless distances $y/L = 0.13$, $y/L = 0.54$ and $y/L = 0.95$. The plane at $y/L = 0.66$ represents the midplane between the inlets and the exhaust. The tangential configuration of the four main flow inlet ports at the back of the receiver demonstrates their effect, based on the vortex that is generated. This enhances mixing of the flow and increases flow

turbulence. From a heat transfer point, the convective heat transfer from the heated walls to the gas benefits from this and causes the gas to absorb more heat and lower the wall temperature. Swirl strength gradually decreases as the flow approaches the exit port. The same applies to the velocity magnitude of the flow as higher velocities are observed at the rear of the cavity. Axial and tangential velocity were determined at four different positions along the optical axis of the receiver and are plotted in Figure 11b-c. Since no window screening flow is present, the front part between the screening inlets and the outlet can be considered as being a quasi-stagnant zone. This is confirmed by the relatively flat profiles at $y/L = 0.13$ in Figure 11b-c. Hence, the vortex runs between the main flow inlets and outlet and yields a maximum axial velocity at $y/L = 0.66$ which is midway through the vortex. At the same time, the negative axial velocity causes reversed flow in the core region which allows to maintain the vortex structure [39]. The tangential velocity profiles in Figure 11c confirm the gradual decrease of swirl strength from back to front. Moreover, these profiles indicate the vortex characteristic of a Rankine vortex: a forced vortex in the central region is surrounded by a free vortex which is free of vorticity.

Figure 11 Velocities inside the cavity receiver: (a) streamlines, (b) axial velocity profile and (c) tangential velocity profile

Spatial temperature distribution inside the receiver obtained via the CFD model is shown in Figure 12. The temperature contours for both the XY- and ZY-plane are quasi-identical and confirm the axi-symmetry of the temperature profile in the fluid domain. Also, the induced strong swirl in the main flow fed at an inlet temperature of 293 K is observed in the temperature contours. The cyclone flow at the back of the cavity strongly entrains heat away from the wall, weakens as it progresses towards the front of the receiver and, thus, yields a step-shaped profile near the backplate of the receiver. This profile can be explained by the

strong peaks in tangential velocity at $y/L = 0.93$ as was observed in Figure 11c. The stronger swirl entrains more heat towards the outside of the vortex while the axial and tangential velocities in the core region are closer to 0 and thus creating a stagnant flow region. Average gas temperature for both planes were computed as 1310 K and 1306 K for the XY- and ZY-plane, respectively. A distinct gradient of temperature decrease from the back of the receiver to the front is visible with a clear hot spot in the proximity of the backplate, reaching a maximum temperature of approximately 1475 K. This hot spot is caused by the propagation of the Gaussian-like flux density distribution incident on the receiver window.

Figure 12 Static temperature contours for the XY-plane (top) and ZY-plane (bottom)

A one-dimensional distribution of both the gas and wall temperature of the computational domain are plotted in Figure 13. The gas temperature is evaluated along the centerline of the fluid domain, whereas the plotted wall temperature is representing the temperature distribution at the very bottom of the receiver cavity. Wall temperature was also computed along the top part of the cavity and showed quasi the same trend as the bottom wall temperature, with a maximum difference of no more than 3.33 K close to the window. Hence, this indicates axi-symmetry of the wall temperature distribution throughout the receiver's length, as was earlier observed in the experimental temperature readings in Figure 10. The wall temperature agrees well with the temperatures observed close to the wall in the YZ temperature contours in Figure 12.

Furthermore, comparison of both temperature distributions in Figure 13 shows relatively low discrepancies for $0.1 \leq y/L \leq 0.9$. The lower gas temperature at the window ($y/L = 0$) is due to the convective losses. This, opposed to the increase of the gas temperature at the backplate ($y/L = 1$), which can be prescribed to the fluid motion obtaining heat through convection along the hotspot (i.e. backplate) of the receiver.

Figure 13 Temperature of the gas along the receiver's centerline versus wall temperature at the bottom of the receiver

The above observed temperature gradient throughout the cavity is caused by the small rim angle of the ellipsoidal reflector with respect to the focal plane. Unlike parabolic solar concentrators which reflect solar radiation as a wide-angle conical beam into its focal plane, such as solar furnaces or parabolic dishes, the solar simulator's ellipsoidal reflector used in this study yields a narrow-angle conical beam of artificial sunlight at the receiver's aperture. This results in a relatively small incident ray angle as was previously shown in Figure 7. The penetration of incident radiation within the fluid domain as seen in Figure 8 results in an incident radiation distribution on the cavity walls as depicted in Figure 14. The aforementioned hot spot is clearly visible as an axisymmetric distribution on the backplate. The initial flux density distribution of the radiation at the window with a peak flux of approximately $5,000 \text{ kW/m}^2$ is diminished. However, this still yields a peak flux of approximately $1,600 \text{ kW/m}^2$ at the backplate and explains the presence of the hot fluid region observed in Figure 12.

Figure 14 Incident radiation on inner cavity walls

4.3. Window screening flow

In the current receiver configuration, four inlet ports are located at the front of the receiver pointing towards the window at a 30 degree angle with the surface normal. The flow stream generated by these four inlets acts as a window screening flow to protect the quartz window from non-volatile reaction products such as particles. For the solar thermochemical dissociation of methane for example, the produced nano-scale carbon particles tend to deposit on the receiver walls as well as the window. This has been investigated experimentally [40] while other studies have been performed to numerically evaluate the effectiveness of screening flows to prevent carbon deposition [41], [42]. At present, window screening of the current receiver design has not been extensively investigated. Nonetheless, few experimental campaigns were conducted with an active window screening flow and were also numerically investigated. Velocity streamlines have been obtained from the CFD results, as shown in Figure 15. A window screening flow (i.e. air) of 2 L/min is injected at the front side of the receiver while the main flow rate remains 10 L/min. The streamlines visually indicate the unmixed state of both flows as well as the aperture region being undisturbed by the main flow. Furthermore, in terms of temperature validation the results showed similar agreement as the case where no window screening flow was considered. Overall, the temperature differences between both cases did not exceed more than 15 K at the respective thermocouple locations.

Hence, the presence of the window screening flow, which enters the cavity at room temperature, causes an increase of flow rate by 20% but a negligible temperature difference. This is possibly due to the fact that the window screening flow is positioned close to the exhaust and tends to reach for the exit port without getting a chance to mix with the hot main flow. And so one can state that window screening did not significantly affect the thermal behavior of the system and, thus, can be considered an improvement that does not affect the flow regime.

Figure 15 Velocity streamlines colored per flow type: main flow (red) and window screening flow (yellow)

5. DESIGN OPTIMIZATION

Although considering the overall temperature distribution in Figure 12 to be relatively uniform, presence of a hot spot indicates the possibility of design improvement of the receiver. The parametric study in the present paper comprises the effect of flow configuration of the current receiver design on the thermal hydraulics of the system. Considering the validated numerical model presented above as being the base case, several parameters have been varied and their effect have been quantitatively observed. An overview of this parametric study and the respective flow rates are shown in Table 4. Validation of case 1 and case 2 has been described in prior sections and are compared with cases 3-6 in this section.

Table 4 Overview of the parametric study cases.

Case	Description	Main Flow Rate [L/min]	Window Screening Flow Rate [L/min]
1	Base case	10	0
2	Window screening enabled	10	2
3	Inlet ports pivoted 45°	10	2
4	16 main inlet ports	10	2
5	Main flow rate increased to 20 L/min	20	0
6	Main flow rate increased to 40 L/min	40	0

5.1. Flow inlet configuration

First parameter that was altered was the angle of the main flow inlet ports (case 3 in Table 4). In the original receiver design the inlet ports are tangentially configured in a plane parallel to the window. Based on the observation of the hot spot, by pivoting the inlets and pointing them towards the backplate the convective heat transfer between the hot wall and the gas flow will presumably be increased. Hence, heat is effectively carried away from the highly irradiated backplate. The inlet ports were tilted at a 45° angle with the XZ-plane while keeping the tangential configuration for vortex formation. The second parameter that was changed, was the amount of main inlet ports. Sixteen inlet ports were used while the cross-sectional area was decreased proportionally to keep the inlet velocity and total flow rate constant. Comparison was made between cases 1-4 with respect to the flow field and temperature distribution inside the fluid domain. The velocity vector field and temperature contours of the enclosed gas are shown in Figure 16 for the four different cases. The velocity vectors are normalized, and magnitude is visualized in greyscale gradient value.

The absence of a window screening flow in case 1 (Figure 16a) results in a less significant lower temperature region at the window, compared to the other three cases. The main flow propagates through the receiver towards the window where the upper flow entrains heat to the front of the receiver. As the flow reaches the window, the fluid cools down and momentum decreases. A recirculation zone is formed at the front of the receiver due to the receiver exhaust position and the collision of two flow streams. Hence, this yields a lobe in the bottom part of the lower temperature region. The other three cases, as depicted in Figure 16b-d, demonstrate the impact of window screening flow on this lower temperature region at the window. The inlets at the window assume a gas inlet temperature at room temperature. Given the position of the

exhaust near the window screening inlet ports, which also are pointing towards the window, this results in a secondary flow stream being re-directed into the cavity. Analogous to case 1, the position of the receiver exhaust yields a low temperature region that shifts towards the exit port.

The step-shape profile of the hot gas near the backplate of the receiver, which was previously observed in Figure 12, can be explained based on the results in Figure 16. Cases 1, 2 and 4 show a similar axisymmetrical velocity field in the rear area of the fluid domain. With the given flow configuration and with the main inlet ports parallel to the XZ-plane, formation of vortices near the corner region occurs. As it is close to the inlet ports, the concentrated flow momentum and high swirl in this region enhances convective heat transfer. Velocity of the flow that is recirculated from the cavity center towards the backplate decreases and becomes zero at the wall. Hence, convective heat transfer at the center of the backplate is much less. Because of the directionality of the incoming radiation, most of this radiative heat flux is incident on the backplate. And as such, this locally concentrated heat flux results in the hot spot at the respective location. By directing the inlet ports towards the backplate, this effect is significantly reduced, as shown in Figure 16c. The alteration of the inlet ports in case 3 has a diminishing effect on the hot spot observed in the other three cases. Moreover, there are no recirculation zones in the corner region of the cavity for the flow configuration in case 3. Figure 16d shows the altered cavity receiver with 16 inlet ports equally distributed around the periphery of the cavity. Other than slightly increased recirculation zones at the back of the fluid domain, increase of the amount of inlet ports yielded no significant improvement on the flow field or the temperature distribution.

Figure 16 Temperature contours and velocity vector field in the XY-plane for (a) case 1, (b) case 2, (c) case 3, and (d) case 4

5.2. Flow rate

In a subsequent step the effect of the main flow rate was investigated. As indicated in Table 4 the main flow rate was increased by a factor of 2 and 4 for case 5 and case 6, respectively. Window screening flow was disabled. Increase of the flow rate by a factor 2 (i.e. case 5) resulted in a volume averaged temperature decrease of 51 K with respect to reference case 1, yielding $T_{gas,avg} = 950$ K. Increasing the flow rate further to 40 L/min (i.e. case 6) resulted in $T_{gas,avg} = 860$ K. Resulting static temperature contours are shown in Figure 17. Increased inlet velocity at the back of the receiver increases the fluid momentum and, thus, has an enhanced effect on the forced convection along the backplate of the receiver. As a result, the initial hot spot in the fluid domain (Figure 17a) is decreased with increase of flow rate. Furthermore, temperature uniformity seems to increase with higher flow rates, which is very evident in Figure 17c.

Figure 17 Static temperature contours in the XY-plane for (a) case 1, (b) case 5 and (c) case 6

5.3. Velocity profiles and temperature uniformity

The parametric changes to the flow rate and flow configuration have altered the flow regime in all the cases. To have a general idea of how the flow field has been affected, axial and tangential velocity profiles have been plotted in Figure 18. The plots are obtained along the x-axis at the location $y/L = 0.66$. As seen in Figure 11a, $y/L = 0.66$ is halfway between the inlets and the exhaust and, thus, can be considered a good representation of the bulk flow in the fluid domain. The first main observation is the increased momentum for the increased flow rates cases 5 and 6. The higher tangential velocity peaks (Figure 18b) also indicate stronger swirl strength. In both plots axi-symmetry, i.e. a shift towards $r/R = -1$, is also observed when flow rate is increased. This can visually be interpreted as the vortex core, earlier observed in Figure 11, bending towards $r/R = -1$ and is due to the outlet being positioned out of plane, i.e. not aligned with the YZ-plane.

The bended vortex funnel yields stronger momentum and swirl at the opposite side of the YZ-plane with respect to where the outlet is located. Enabling window screening (case 2) nor increasing the number of inlet ports in case 4 does significantly affect the flow regime. Pivoting the inlet angles towards the backplate in case 3, on the other hand, shows a weakening effect on the flow momentum and swirl strength.

Figure 18 Velocity profiles for the different cases at $y/L = 0.66$: (a) axial velocity and (b) tangential velocity

Closer comparison of all six cases of the parametric study in terms of temperature results was obtained by computation of the temperature distribution at the centerline of the fluid domain which is shown in Figure 19. Overall, one can see that the main discrepancy between the distributions occurs at the front side of the receiver, i.e. where $0 < y/L < 0.3$. When comparing the presence of a window screening flow (case 2) with no window screening flow (case 1), it is seen in Figure 19 that the presence of a window screening flow diminished the gas temperature between $y/L = 0$ and $y/L = 0.4$, while the temperature distributions for both case 1 and 2 coincide for $y/L > 0.4$. This is due to the window screening flow entering the fluid domain at room temperature, preventing the hotter main flow to reach the window, and reaching the exhaust rapidly.

While the altered 45-degree flow configuration of the main inlet ports (case 3) showed effective reduction of the hot spot in the back of the receiver when compared to case 2 in Figure 16, it does not significantly affect the temperature distribution towards the front of the cavity, nor does the wall temperature reduce (Figure 19). Moreover, both the gas temperature distributions along the centerline for case 3 and case 4 showed no substantial difference with case 2. Nevertheless, towards the back of the receiver larger discrepancies are observed, especially at $y/L = 0.8$ and $y/L > 0.85$ for cases 4 and 3, respectively. These temperature differences are due to the increased convective heat transfer by the gas stream along the backplate for case 3, and the weaker momentum at the back of the receiver for case 4. In the region closer to the window the temperature distribution of case 4 coincides with the distribution of case 2 for $y/L < 0.4$, and further coincides with the distribution of case 3 when $y/L < 0.2$. On the other hand, increasing the flow rates in case 5 and case 6 proportionally decreased the gas temperature along the centerline yielding a slightly more uniform temperature distribution. While this mainly applies to the middle part of the receiver, in the front part the gas temperature experiences a drop around $y/L = 0.15$ (case 6) and $y/L = 0.2$ (case 5) in Figure 19. This trend is confirmed by the temperature contours in Figure 17b-c. The increased momentum of the flow allows the hot gas circulation to carry its absorbed heat towards the window region. Once reaching the solid-fluid interface at the window heat is transferred from the gas through the window to the environment by conduction and convection and hence, resulting in the second more significant temperature drop at $y/L = 0$.

Figure 19 Gas temperature distribution along centerline of the fluid domain

To further examine the effect of the altered flow configurations and respective flow rates on the thermal behavior and heat transfer, the temperature non-uniformity (TNU) is quantified. The latter serves as an indicator for the uniformity of the temperature distribution and is expressed as

$$\text{TNU} = \frac{\sum \frac{|T - \bar{T}|}{\bar{T}}}{n} \cdot 100 \quad (12)$$

where T is the observed temperature for every position y along the centerline of the receiver, $\bar{T}$ is the average temperature over the full longitudinal range $0 \leq y \leq L$, with $y/L = 0$ at the receiver window, and n is the total amount of temperature data points computed. A TNU of 0% corresponds to a completely uniform temperature distribution. To evaluate temperature uniformity in the complete three-dimensional fluid domain and account for non-axisymmetric phenomena the TNU was also computed over all cells of the computational domain. Both T and $\bar{T}$ in Equation 12 were then taken as the cell temperatures and volume average gas temperature of the domain, respectively.

The average temperatures and TNU for all the cases are depicted in Table 5. Comparison between the one-dimensional and three-dimensional averaged temperature shows similar values except for case 1 and case 5. This possibly indicates non-axisymmetric distribution of the gas temperature in the fluid domain. Furthermore, the TNU computed over the whole computational domain is found to be consistently lower than the TNU based on the centerline values for all cases. Reason for this is that the temperature distribution along the centerline experiences larger gradients which results in larger temperature differences with the average centerline gas temperature. Computation of the TNU over the full computational domain is a more reliable measure for temperature uniformity. Hence, accounting for all regions of the fluid domain shows that overall temperature difference with the average gas temperature is smaller.

From the values in Table 5 one can conclude that maximizing the main flow rate positively affects the overall temperature uniformity. Increasing the flow rate with a factor of 4, i.e. 40 l/min, reduces the TNU by approximately 30% for the full computational domain. However, one should consider that changing the flow rate of a feed stock in a solar thermochemical process can adversely affect reaction kinetics and thus production rate. For example, experimental campaigns on the solar thermal dissociation of methane have shown that a 33% increase of CH_4 flow rate at reactor temperatures of 2133 K brought down the conversion rate by 30% [43]. Therefore, careful consideration of required flow rate is necessary to optimize the thermochemical process housed in the cavity receiver. The results in Table 5 also show that the presence of a window screening flow not only reduces average gas temperature by 1 - 2%, but also negatively affects the temperature uniformity of the fluid domain. Further mutual comparison of the respective cases shows that neither pivoting the main inlet ports, nor increasing the amount of inlet ports contributes to improvement of the temperature uniformity inside the cavity receiver. Pivoting the inlet ports increases the centerline TNU and full computational domain TNU by 0.5% and 1%, respectively, with respect to reference Case 2 (with window screening flow). An increase of 1.1% and 1.2% is observed for both TNU's, respectively, when the number of inlet ports is quadrupled. However, the increased TNU is not supported

by Figure 16 and Figure 19 where temperature homogeneity seems to increase for both cases. Moreover, average temperature of the flowing medium is increased and is slightly higher for Case 4 than for Case 3. Hence, it could be implied that the computation of the TNU is more affected by the colder region at the front of the receiver, i.e. higher temperature difference with T_{avg} , than it is by the hot spot region at the back of the receiver.

Table 5 Average gas temperature and temperature non-uniformity (TNU).

	Case					
	1	2	3	4	5	6
Centerline						
T_{avg} (K)	1044	997	1016	1023	1016	859
TNU_{avg} (%)	10.3	16.6	17.1	17.7	9.1	6.9
Full computational domain						
T_{avg} (K)	1013	995	1016	1025	950	860
TNU_{avg} (%)	8.9	10.9	11.9	12.1	7.2	6.1

6. CONCLUSIONS

CFD simulations play an important role in the design of solar receiver-reactors as they allow to simulate complex processes without the recurrent aid of experimental campaigns when design iterations are implemented. In this study, a 3 kW solar cavity receiver is presented that uses a 10 kW_e high flux solar simulator as a radiation source for indoor experimental testing. Indirect heat flux mapping method provided spatially distributed flux density data incident on the radiative boundary of the fluid domain. A validated Monte Carlo ray tracing (MCRT) model was adopted to obtain information on the directionality of the incoming rays. The MCRT model was coupled to the computational fluid domain.

Observed temperatures of the numerical model at the respective thermocouple locations were compared against experimental results. An average relative deviation of 4.2% confirmed the acceptable accuracy of the computational model. Furthermore, flow pattern and temperature distribution inside the receiver were numerically obtained and visualized. A hot spot was observed at the backplate of the cavity, leaving room for design optimization towards temperature uniformity. The latter was addressed by conducting a parametric study on the flow configuration of the receiver. Outcome of this study is summarized as follows:

- Pivoting the four main inlet ports of the cavity towards the back of the receiver reduces the hot spot within the fluid domain close to the backplate. However, this adjustment of the flow configuration adversely affects the temperature uniformity of the fluid domain.
- Although window screening flow can play a crucial role in protecting the quartz glass and allow continuous operation of solar thermochemical processes, it causes reduction of overall temperature of the fluid and increases temperature non-uniformity. Moreover, it does not significantly affect the bulk flow regime.
- Increasing the main flow rate showed that average gas temperature is reduced, and temperature uniformity is improved. However, flow rate is an important process parameter which affects reaction kinetics and outcome of the chemical process and should therefore not unjustifiably be maximized.

This parametric study pin-pointed areas of optimization for solar cavity receiver-reactors. The outcome obtained from this study can serve as a guideline for the design process of solar reactors.

Acknowledgement

Numerical part of this research was funded by Impulse Fund Project No. IMP/14/049 of KU Leuven and campus De Nayer of KU Leuven. Experimental part of this research was funded by the University of Minnesota Duluth. The corresponding author appreciates the diligent effort and support of Professor Aydin Durgunoglu, Professor Rudy Perrault, Ms. Kathleen Adee, and Mr. Michael Swanson in the materialization of the High Flux Gas Dynamics Laboratory.

References

- [1] G. Levêque, R. Bader, W. Lipiński, and S. Haussener, "High-flux optical systems for solar thermochemistry," *Sol. Energy*, vol. 156, pp. 133–148, 2017.
- [2] D. Yadav and R. Banerjee, "A review of solar thermochemical processes," *Renew. Sustain. Energy Rev.*, vol. 54, pp. 497–532, 2016.
- [3] T. Kodama, "High-temperature solar chemistry for converting solar heat to chemical fuels," *Prog.*

- 827 *Energy Combust. Sci.*, vol. 29, no. 6, pp. 567–597, 2003.
- 828 [4] V. M. Wheeler, R. Bader, P. B. Kreider, M. Hangi, S. Haussener, and W. Lipinski, “Modelling of
829 solar thermochemical reaction systems,” *Sol. Energy*, vol. 156, pp. 149–168, 2017.
- 830 [5] J. Christian and C. Ho, “Alternative designs of a high efficiency, north-facing, solid particle
831 receiver,” *Energy Procedia*, vol. 49, pp. 314–323, 2013.
- 832 [6] J. Costandy, N. El Ghazal, M. T. Mohamed, A. Menon, V. Shilapuram, and N. Ozalp, “Effect of
833 reactor geometry on the temperature distribution of hydrogen producing solar reactors,” *Int. J.*
834 *Hydrogen Energy*, vol. 37, no. 21, pp. 16581–16590, 2012.
- 835 [7] L. O. Schunk, P. Haeberling, S. Wepf, D. Wuillemin, A. Meier, and A. Steinfeld, “A ROTARY
836 RECEIVER-REACTOR FOR THE SOLAR THERMAL DISSOCIATION OF,” in *Proceedings of*
837 *ES2007 Energy Sustainability*, 2018, pp. 1–7.
- 838 [8] Z. D. Cheng, Y. L. He, F. Q. Cui, B. C. Du, Z. J. Zheng, and Y. Xu, “Comparative and sensitive
839 analysis for parabolic trough solar collectors with a detailed Monte Carlo ray-tracing optical model,”
840 *Appl. Energy*, vol. 115, pp. 559–572, 2014.
- 841 [9] Y. Shuai, X. L. Xia, and H. P. Tan, “Radiation performance of dish solar concentrator/cavity receiver
842 systems,” *Sol. Energy*, vol. 82, no. 1, pp. 13–21, 2008.
- 843 [10] C.-A. Asselineau, J. Zapata, and J. Pye, “Integration of Monte-Carlo ray tracing with a stochastic
844 optimisation method: application to the design of solar receiver geometry,” *Opt. Express*, vol. 23,
845 no. 11, p. A437, 2015.
- 846 [11] J. Delatorre *et al.*, “Monte Carlo advances and concentrated solar applications,” *Sol. Energy*, vol.
847 103, pp. 653–681, 2014.
- 848 [12] E. Przenzak, M. Szubel, and M. Filipowicz, “The numerical model of the high temperature receiver
849 for concentrated solar radiation,” *Energy Convers. Manag.*, vol. 125, pp. 97–106, 2016.
- 850 [13] H. Abedini and N. Ozalp, “An advanced modeling and experimental study to improve temperature
851 uniformity of a solar receiver,” *Energy*, vol. 165, pp. 984–998, 2018.
- 852 [14] H. Zhang and J. D. Smith, “Investigating in fl uences of geometric factors on a solar thermochemical
853 reactor for two-step carbon dioxide splitting via CFD models,” *Sol. Energy*, vol. 188, no. January,
854 pp. 935–950, 2019.
- 855 [15] S. S. S. Khalsa and C. K. Ho, “Radiation Boundary Conditions for Computational Fluid Dynamics

856 Models of High-Temperature Cavity Receivers,” *J. Sol. Energy Eng.*, vol. 133, no. 3, p. 031020,
857 2011.

858 [16] M. C. W. Dordevich, “Heat Transfer Analysis of a Lab Scale Solar Receiver Using the Discrete
859 Ordinates Model,” San Diego State University, 2013.

860 [17] Y. Qiu, Y. He, Z. Cheng, and K. Wang, “Study on optical and thermal performance of a linear
861 Fresnel solar reflector using molten salt as HTF with MCRT and FVM methods,” *Appl. Energy*, vol.
862 146, pp. 162–173, 2015.

863 [18] B. Du, Y. Qiu, Y. He, and X. Xue, “Study on heat transfer and stress characteristics of the
864 pressurized volumetric receiver in solar power tower system,” *Appl. Therm. Eng.*, vol. 133, pp. 341–
865 350, 2018.

866 [19] J. Martinek, C. Bingham, and A. W. Weimer, “Computational modeling and on-sun model
867 validation for a multiple tube solar reactor with specularly reflective cavity walls. Part 1: Heat
868 transfer model,” *Chem. Eng. Sci.*, vol. 81, pp. 298–310, 2012.

869 [20] B. J. Hathaway, W. Lipinski, and J. H. Davidson, “Numerical Heat Transfer , Part A : Applications :
870 An International Journal of Computation and Methodology Heat Transfer in a Solar Cavity
871 Receiver : Design Considerations,” vol. 62, no. 5, pp. 445–461.

872 [21] M. T. Ceramics, “Cerablanket AC2,” 2016. .

873 [22] M. Abuseada, “An experimental and numerical study on the heat transfer driven dynamics and
874 control of transient variations in a solar reactor,” University of Minnesota, 2019.

875 [23] M. Abuseada, C. Ophoff, and N. Ozalp, “Characterization of a new 10 kWe high flux solar simulator
876 via indirect radiation mapping technique,” *J. Sol. Energy Eng.*, vol. 141, no. 2, pp. 1–14, 2019.

877 [24] QM Central Laboratory for Light Measurements, “Technical Report: QM CL 111b-08,” Munich,
878 Germany, 2008.

879 [25] Superior Quartz Products Inc, Xenon lamp luminance intensity distribtuion, Air cooled short arc
880 technical information (2018). <https://www.sqpuv.com/acsatechnical.php>.

881 [26] F. Incropera, D. Dewitt, T. Bergman, and A. Lavine, *Fundamentals of heat and mass transfer*, 6th
882 ed. John Wiley & Sons, Inc., 2007.

883 [27] R. Bader, S. Haussener, and W. Lipinski, “Optical design of multisource high-flux solar simulators,”
884 *J. Sol. Energy Eng.*, vol. 137, no. 2, p. 021012, 2014.

- 885 [28] K. R. Krueger, "Design and characterization of a concentrating solar simulator," University of
886 Minnesota, Minneapolis, MN, 2012.
- 887 [29] B. Blocken, T. Defraeye, D. Derome, and J. Carmeliet, "High-resolution CFD simulations for forced
888 convective heat transfer coefficients at the facade of a low-rise building," *Build. Environ.*, vol. 44,
889 no. 12, pp. 2396–2412, 2009.
- 890 [30] ANSYS, "Fluent Theory Guide 17.0." 2016.
- 891 [31] VGB Powertech, "Design Report M218H," 1984.
- 892 [32] M. W. Chase, "NIST-JANAF Thermochemical Tables," *J. Phys. Chem. Ref. Data*, vol. Monograph,
893 1998.
- 894 [33] D. W. Green and R. H. Perry, *Perry's Chemical Engineers' Handbook*, 8th ed. McGraw-Hill, USA,
895 2008.
- 896 [34] C. Ray, T. Method, and F. Miller, "ES2013-18186 OPTICAL ANALYSIS OF A WINDOW FOR
897 SOLAR RECEIVERS USING THE MONTE," pp. 1–13, 2016.
- 898 [35] G. Dai, X. Xia, and G. Hou, "ScienceDirect Transmission performances of solar windows exposed
899 to concentrated sunlight," *Sol. Energy*, vol. 103, pp. 125–133, 2014.
- 900 [36] D. Hirsch and A. Steinfeld, "Solar hydrogen production by thermal decomposition of natural gas
901 using a vortex-flow reactor," *Int. J. Hydrogen Energy*, vol. 29, no. 1, pp. 47–55, 2004.
- 902 [37] ANSI American National Standard for Expressing Uncertainty, "U.S. Guide to the expression of
903 uncertainty in measurement, ANSI/NCSL Z540-2-1997." 1997.
- 904 [38] M. Abuseada and N. Ozalp, "Intensity distribution from a single-bulb solar simulator identification
905 through inverse ray tracing," in *Proceedings of the ASME 2019 13th International Conference on*
906 *Energy Sustainability*, 2019, pp. 1–10.
- 907 [39] N. Ozalp, M.-H. Chien, and G. Morrison, "Computational Fluid Dynamics and Particle Image
908 Velocimetry Characterization of a Solar Cyclone Reactor," *J. Sol. Energy Eng.*, vol. 135, no. 3, p.
909 031003, 2013.
- 910 [40] A. Kogan, M. Kogan, and S. Barak, "Production of hydrogen and carbon by solar thermal methane
911 splitting. II. Room temperature simulation tests of seeded solar reactor," *Int. J. Hydrogen Energy*,
912 vol. 29, no. 12, pp. 1227–1236, 2004.
- 913 [41] D. Jaya Krishna and N. Ozalp, "Numerical investigation of particle deposition inside aero-shielded

914 solar cyclone reactor: A promising solution for reactor clogging,” *Int. J. Heat Fluid Flow*, vol. 40,
915 no. October, pp. 198–209, 2013.

916 [42] N. Ozalp and A. Kanjirakat, “Lagrangian characterization of multi-phase turbulent flow in a solar
917 reactor for particle deposition prediction,” *Int. J. Hydrogen Energy*, vol. 35, no. 10, pp. 4496–4507,
918 2010.

919 [43] J. K. Dahl, K. J. Buechler, A. W. Weimer, A. Lewandowski, and C. Bingham, “Solar-thermal
920 dissociation of methane in a fluid-wall aerosol flow reactor,” *Int. J. Hydrogen Energy*, vol. 29, no.
921 7, pp. 725–736, 2004.

922

923

924

ANNEX A

925

CONSTANTS AND PARAMETERS OF THE GOVERNING EQUATIONS

926

927

928

929

930 **Turbulent viscosity**

931

$$932 \quad \mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

933 with

$$934 \quad C_\mu = \frac{1}{A_0 + A_s \frac{k U^*}{\varepsilon}}$$

935 and

$$936 \quad U^* \equiv \sqrt{S_{ij} S_{ij} + \tilde{\Omega}_{ij} \tilde{\Omega}_{ij}}$$

$$937 \quad \tilde{\Omega}_{ij} = \Omega_{ij} - 2\varepsilon_{ijk} \omega_k$$

$$938 \quad \Omega_{ij} = \overline{\Omega_{ij}} - \varepsilon_{ijk} \omega_k$$

$$939 \quad A_0 = 4.04$$

$$940 \quad A_s = \sqrt{6} \cos \phi$$

$$941 \quad \phi = \frac{1}{3} \cos^{-1}(\sqrt{6}W)$$

$$942 \quad W = \frac{S_{ij} S_{jk} S_{ki}}{\tilde{S}^3}$$

$$943 \quad \tilde{S} = \sqrt{S_{ij} S_{ij}}$$

$$944 \quad S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$$

945

946

947

948

949 **Turbulence dissipation rate constants**

950

$$951 \quad C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right]$$

952 with

$$953 \quad \eta = S \frac{k}{\varepsilon}$$

$$954 \quad S = \sqrt{2 S_{ij} S_{ij}}$$

955 and

$$956 \quad C_{1\varepsilon} = 1.44$$

$$957 \quad C_2 = 1.9$$

$$958 \quad \sigma_k = 1.0$$

$$959 \quad \sigma_\varepsilon = 1.2$$

$$960 \quad C_{3\varepsilon} = \tanh \left| \frac{\nu}{u} \right|$$

961