CENTRAL SOLAR RECEIVER CFD MODELLING UTILISING GENERATED HELIOSTAT FIELD HEAT FLUX MAPS

Marsberg J., Craig K.J.* and Meyer J.P.
*Author for correspondence
Department of Mechanical and Aeronautical Engineering,
University of Pretoria, Pretoria, 0002, South Africa,
E-mail: ken.craig@up.ac.za

ABSTRACT

A numerical method to couple Monte Carlo ray tracing data to a Finite Volume (FV) semi-transparent surface to allow for the determination of thermal efficiency due to an input heat flux profile and corresponding ray directions within a central cavity receiver is presented. A sample Biomass cavity receiver[1, 2] is used as a 2-D validation case to demonstrate that a CFD FV approach can be used as an accurate solution to the Radiative Transfer Equation (RTE). A 3-D representation of this cavity allows for the approximation of cavity thermal efficiency to be compared between various input heat flux profiles due to the addition of conjugate heat transfer. Results allow for deductions to be made on the benefits of more accurate representations of heat flux maps due to the point concentration of solar energy from a heliostat field. These representations of heat profiles can be used in future applications such as cavity and heliostat field optimization by creating the critical link between ray tracing and conjugate heat transfer solution methods to evaluate central tower cavity receiver designs.

INTRODUCTION

The purpose of this study is to develop a method to improve the accuracy of optimization for real world central solar receivers. It is necessary to be able to accurately predict the heat flux distribution which enters the cavity. Various methods to predict these heat flux distributions have been developed including the ray-tracing software, SolTrace [3]. These generated ray data including the magnitude and direction of the rays can then be mapped to a surface as a boundary condition (BC) in a Computational Fluid Dynamics (CFD) environment. Since the thermal efficiency of the cavity is dependent on the optical efficiency of the heliostat field, this boundary condition can be used for accurate thermal/optical optimization of a cavity receiver or the heliostat field.

NOMENCLATURE

a	[m^{-1}]	Absorption coefficient
I	[W/sr]	Radiative intensity
n	[-]	Refractive index
N	[-]	Number
q	[W/m^2]	Radiative flux in non-gray medium

Special characters

β	[-]	Extinction coefficient
σ_s	[m^{-1}]	Scattering coefficient
φ	[-]	Phase function
ω′	[sr]	Solid angle
∇.	[m^{-1}]	Divergence operator
$\vec{s}'$	[-]	Scattering direction vector
$\vec{r}$	[m]	Position vector
$\vec{s}$	[m]	Direction vector
ε	[-]	Emissivity

Subscripts

θ		Angular direction
φ		Angular direction
λ		Spectral

TOWER RECEIVER MODELLING

An accurate estimation of the heat flux magnitude and its directional components intersecting the inlet of a cavity's aperture is essential in the optimization process to allow for the objective function(s) to converge to a truly optimal result. The effect which different heat flux distributions have on the efficiency of a cavity, by replicating different times of day, locations and time of year, with specific consideration given to amount of energy absorbed into the heat transfer fluid, and hot spots on surfaces on the inside of the cavity, can be determined. Minimization of convective and re-radiation losses will be

essential in the improving of the thermal efficiency of the cavity receiver.

A method has been developed which allows for the coupling between ray-tracing and CFD software. This method will allow for an inexpensive simulation of a system, which includes conjugate as well as radiative heat transfer, while not compromising on solution accuracy. Figure 1 shows the proposed coupling of the software systems for the PS-10 heliostat field in Spain. A Monte Carlo ray tracing method is used to solve the system up until the virtual surface as shown, after which these data are patched onto a surface within a Finite Volume (FV) Method Solver, in this case ANSYS Fluent, where the conjugate heat transfer solution within the cavity receiver is obtained.
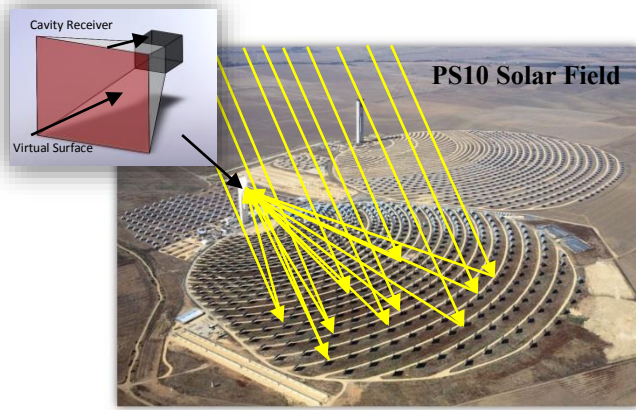

Figure 1: Illustration of coupled ray-tracing with CFD system for PS-10 heliostat field

Since the heliostat field is large, the most efficient method of simulating the field is by using ray-tracing software to determine energy and ray direction profiles on a virtual surface. These data are processed into a User Defined Profile (UDP) which can be patched to a surface in the Fluent solver. This surface would then contain the accumulated information of all the ray hits on the virtual surface and serve as a Boundary Condition to the CFD model. This model can then be solved while considering the conjugate heat transfer within the system due to the radiation and other fluid dynamic effects.

A brief description of the two solution methods is described in the proceeding subsections.

Monte Carlo ray-tracing

By using this ray-tracing method it is possible to predict the radiation heat transfer to a surface by means of algorithms which are able to track randomly released rays from an emitting surface, and tracing them along their path until fully absorbed or lost [4]. Interaction with surfaces can be either reflective or refractive and is therefore ideal for the prediction of surface incident radiation. By keeping track of the amount of energy lost due to surface-ray interactions within the system it is possible to quantify the heat flux at the surface of interest. Since this method is not dependent on a mesh size, if enough emissions from a source are used, an accurate prediction of the radiation heat transfer within the system is possible. If conjugate heat transfer is not present this method is extremely effective due to its high accuracy and low cost of the simulation, especially when dealing with extremely large systems. More detailed reviews of this method are available [5, 6].

Finite Volume Method

The finite volume method implemented for the solving of the radiative transfer equation (RTE) in ANSYS Fluent v15.0 is the Discrete Ordinates (DO) method. This method mathematically describes the balance of energy through the scattering, absorption and emission due to the interaction with participating mediums in a domain. A beam with a radiative intensity of $I_\lambda(\vec{r}, \vec{s})$ which is a function of the spectral variable (λ), the position ($\vec{r}$) and direction ($\vec{s}$) that travels in an absorbing, scattering, and emitting medium in a defined direction. Beam energy decreases due to absorption and its scattering from the initial trajectory to other directions, while energy increases due to medium volume thermal radiation emission and scattering from other trajectories towards its own trajectory. This is known as out-scattering and in-scattering respectively and is expressed mathematically with the following partial differential equation:

$$\nabla \cdot (I_\lambda(\vec{r} \cdot \vec{s})\vec{s}) + \beta_\lambda I_\lambda(\vec{r} \cdot \vec{s}) = a_\lambda n^2 I_{b\lambda} + \frac{\sigma_{s\lambda}}{4\pi} \int_0^{4\pi} I_\lambda(\vec{r} \cdot \vec{s}')\Phi(\vec{s} \cdot \vec{s}') \, d\omega' \quad (1)$$

Furthermore the radiative heat flux by definition is given as

$$q(r) = \int_0^\infty \int_0^{4\pi} I_\lambda(\vec{r} \cdot \vec{s}) \vec{s} d\omega' d\lambda \quad (2)$$

By double integration of the RTE equation over all solid angles over all wavelengths, the divergence of heat flux can be calculated as

$$\nabla \cdot q = \int_0^\infty a_\lambda \left(4\pi I_{b\lambda} - \int_0^{4\pi} I_\lambda(\vec{s}')d\omega' \right) d\lambda \quad (3)$$

The divergence of the radiative heat flux is determined in ANSYS Fluent with the S_2 method [7], a subset of the Discrete Ordinates (DO) using the S_N approach, where N is number of ordinate directions. The angular space is subdivided into $N_\theta \times N_\phi$ control angles, each of which is further subdivided by pixels. In the 1-D case, $2 \times N_\theta \times N_\phi$ directions of the RTE equations are solved, for 2-D, $4 \times N_\theta \times N_\phi$ directions, and 3-D, $8 \times N_\theta \times N_\phi$ directions are computed as illustrated in Figure 2. This implies that the computational cost and memory requirements increase linearly with each angular discretization division and that for each spatial dimension that is added, the overhead doubles.

The FV implementation of the RTE equation leads to both false scattering (or numerical diffusion) and a ray effect [8]. Numerical diffusion causes smearing of the propagated radiation while the ray effect causes an incorrect direction of the wave front. In ANSYS Fluent, these can be reduced by three methods: refining the mesh, increasing the number of angular

discretizations, and increasing the order of the spatial discretization of the DO method.
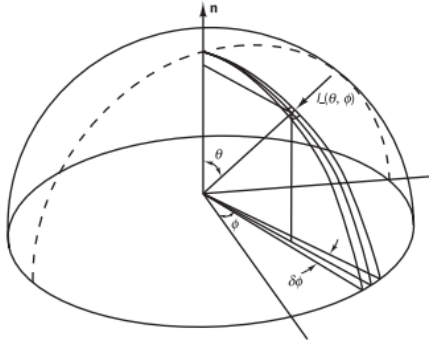

Figure 2: Angular Discretization of Discrete Ordinates Method [4]

APERTURE BOUNDARY CONDITION

The prediction of the heat flux map and ray directions which intersect the aperture of a cavity receiver is necessary to accurately model the real radiation heat transfer due to a real source, in this case a field of heliostats.

The ray properties at the aperture are linked to the sun shape and the position for a specific heliostat field and receiver layout. From the developed method it is possible to predict what the rays' magnitude and direction are for various field layouts and solar conditions. The method uses Soltrace's scripting abilities such that numerous inputs are taken and results are written to a file in the form of a User Defined Profile (UDP), which can be mapped to a geometry's surface in ANSYS Fluent.

The proposed heliostat field layout and properties are based on the PS-10 field in Spain [9] as listed in Table 1.

Table 1 – PS-10 heliostat field properties

Location	
Latitude	37°26′N
Longitude	6°15′W
Heliostats	
Number	624
Width	12.84m
Height	9.45m
Reflectivity	0.88
Receiver	
Tower Height	115m
Normal Direction	(0, 0.97, -0.217)
Aperture Width	13.78m
Aperture Height	12m

Various profiles can be generated through variation of parameters such as time of day, geographical location and time of year, allowing for a large scope of research. Figure 3 shows a sample SolTrace ray trace using 2E06 rays for the PS-10 field. The corresponding heat flux map is displayed in Figure 4. The results of varying the time of day can be seen in Figure 5.

As previously explained this method requires a number of statistically determined rays to be released from a surface towards a target. To increase the accuracy of the simulation, the number of released rays should be increased such that surfaces which interact with these rays become saturated with hits. This will ensure that data at the surface of interest will accurately represent the solution of the system.
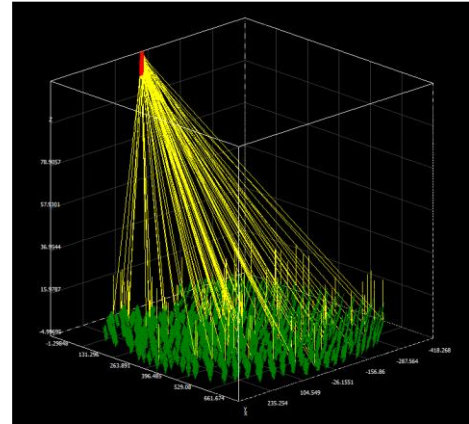

Figure 3: Ray tracing results for PS-10 Heliostat field generated by Soltrace using 2E06 rays
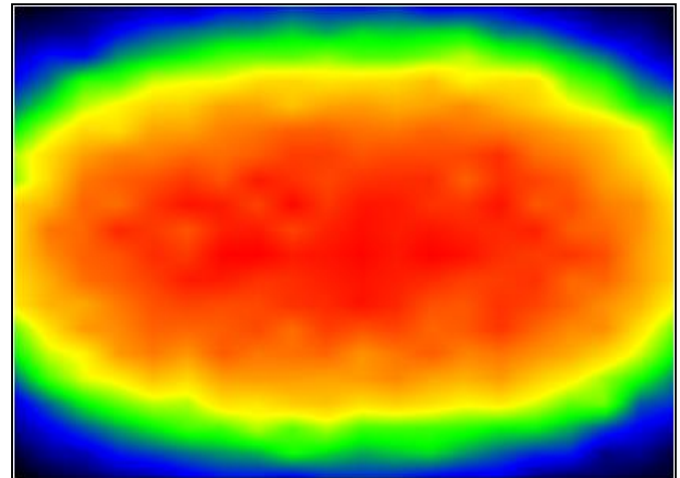

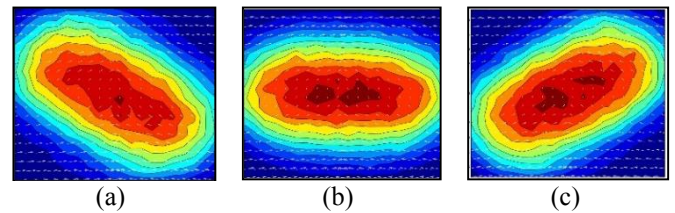**Figure 4:** Heat Flux Map generated by Soltrace using 2E06 rays

| (a) | (b) | (c) |

Figure 5: Heat flux map and corresponding ray directions for (a) morning, (b) midday and (c) afternoon sun

NUMERICAL APPROACH AND DISCUSSION

The numerical simulations which were performed are based on a Rapid High Temperature Solar Thermal Biomass Gasification Prototype Cavity illustrated in Figure 6 [1]. This cavity was selected as a test case for both a 2-D numerical validation [2], as well as comparison between the effects of different energy input profiles at the aperture for the 3-D case.
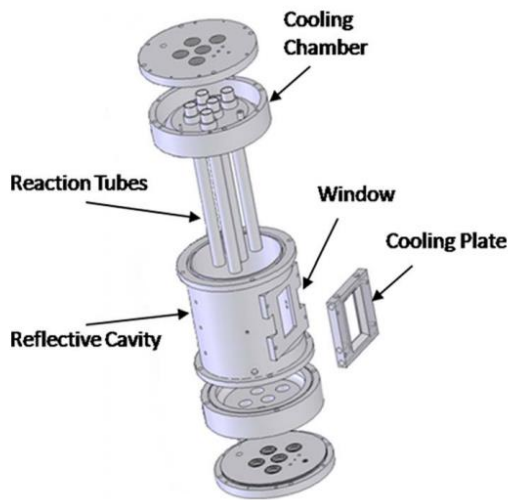
Figure 6: Reflective Cavity Multi-tube Solar Reactor [1]

As previously discussed, ANSYS Fluent is used to perform numerical simulations on the Biomass Cavity Receiver using the Discrete Ordinates model to predict radiative heat transfer.

2-D Validation of Numerical Model

Since the Monte Carlo ray-tracing method is widely considered to produce an accurate solution to the RTE, it is compared with the FV method [2] as obtained in ANSYS Fluent.

Figure 7 shows the cross-section of the biomass cavity which is modelled in the ANSYS environment used in the validation cases. Various boundary conditions are applied to the surfaces within the domain to match those used in the comparison test cases. These boundary conditions are as follows:

- It is always assumed that there is a constant radiative heat flux (semi-transparent wall) at the aperture with ray-directions normal to the aperture surface.
- All surfaces within the cavity are kept at 1K to ensure re-radiation effects are negligible.
- Pipe walls are assumed to be perfectly absorbing surfaces by setting the emissivity BC as $\varepsilon = 1$.
- Cavity walls are assumed to be either perfectly absorbing or perfectly reflecting with internal emissivity BC's set as $\varepsilon = 1$, $\varepsilon = 0$ respectively and diffuse fraction $= 0$.

The validation is performed in two sections. To determine whether convergence has been reached, and numerical diffusion (false scattering) effects [8] have been minimized, convergence tests were run on both mesh size as well as the discretization of the radiation model.

Verification of Convergence

Tests on the 2-D case of the specularly reflective 5 pipe biogas receiver were performed to determine the convergence of the solution by calculating the sum of the absorbed radiation flux on the surfaces of the tubes.
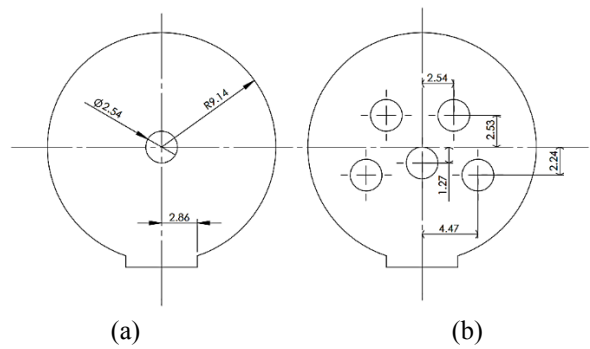

(a)	(b)

Figure 7: Biogas Receiver Configuration for (a) 1 pipe and (b) 5 pipes [2]

As expected, the convergence of the solution is both dependent on mesh size as well as the DO discretization as is illustrated in Figure 8. An acceptable error of less than 0.1% is achieved by using a DO setting of 5×100 with a mesh size of approximately 4.5E05 elements. By increasing the discretization level to 2× and 4× this level only improved the solution accuracy by 0.094% and 0.096%, respectively.

The mesh was determined to reach a converged state by approximately 3E05 number of elements. By increasing the mesh size to a value greater than this did not result in a significant increase in the accuracy of the solution.
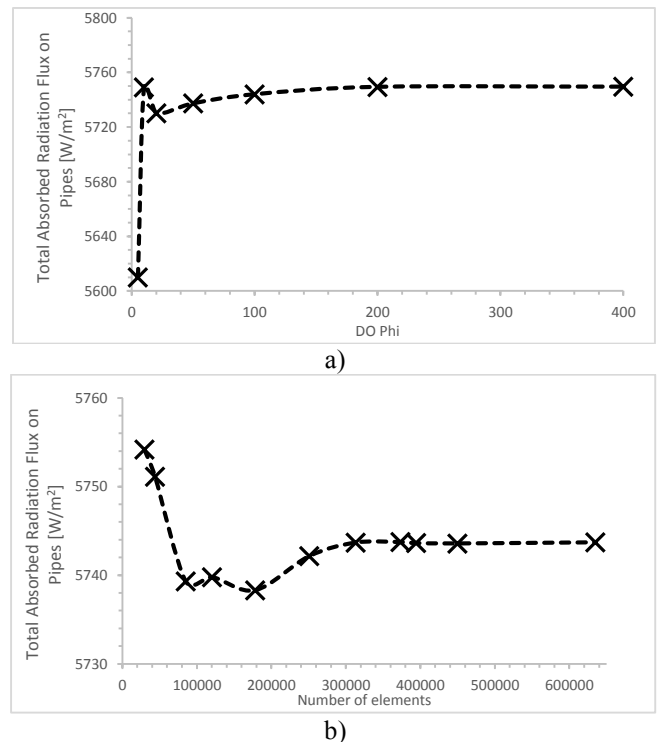

a)

b)

Figure 8: Convergence of (a) DO discretisation and (b) Mesh for a 2-D 5 pipe reflecting Biomass Cavity

The 2-D validation geometry is used in comparison with the results published by Martinek et al. [2] for both the single tube as well as a 5 pipe geometry. In these test cases the absorbed heat flux predictions for the Monte Carlo as well as Finite Volume

methods are used as comparison. Since the Monte Carlo method is considered to be the accepted accurate answer results will ideally be closely matched to these data.

For the single tube absorbing cavity as seen in Figure 9a even with a fairly low DO discretization of 5×20 results matched the Monte Carlo method with negligible error. This result is due to the reduction of the false scattering effect since there is no reflection within the cavity. The results for a specularly reflecting cavity (Figure 9b) were more expensive to achieve an accurate result. In the single pipe case 5×20 DO discretization achieved an acceptable result with an approximate accuracy of 85% when compared with the MC method.

The results for the specularly reflecting 5 pipe cavity achieved similar accuracy to that which was achieved in the comparison case when using the FV method. These results were considered too inaccurate so the DO refinement of 5×100 was used which achieved a much more acceptable accuracy when compared with the MC method data, with an approximate average error of 5% on the front pipe and 8% on the back pipe as can be seen in Figure 10a and 10b, respectively.

Due to the accurate solution values achieved, these results validate the FV method as a useful ray-tracing tool, and simulations can therefore be considered valid for the 3-D case.
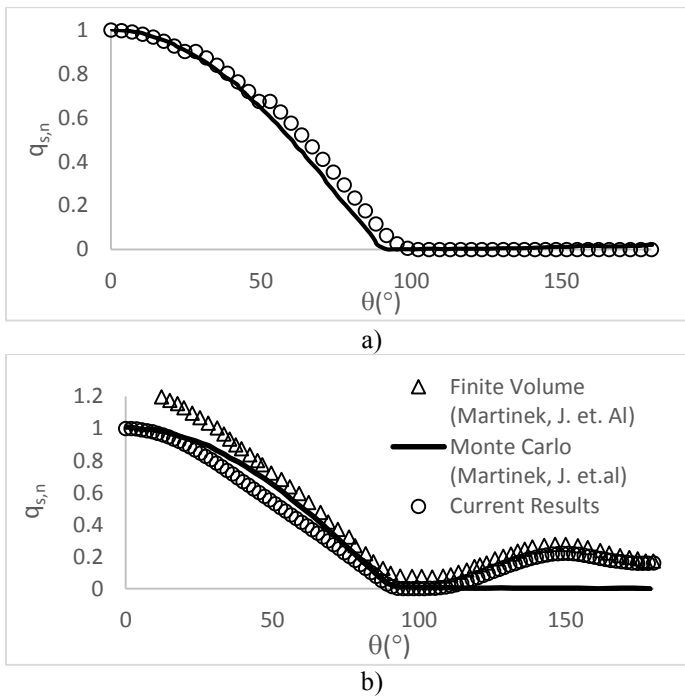

a)

b)

Figure 9: Normalized Heat Flux on pipe surface for an Single Pipe (a) Absorbing Cavity and (b) Reflecting Cavity

3-D Conjugate Heat Transfer Modelling

A combination of the Monte Carlo method implemented in Soltrace as well as a FV method implemented using ANSYS v15.0 is used to model the energy within the cavity with a DO discretization of 15×15. This geometry is used as a tool to illustrate the effects of various inlet aperture heat flux profiles on the conjugate heat transfer within the cavity.
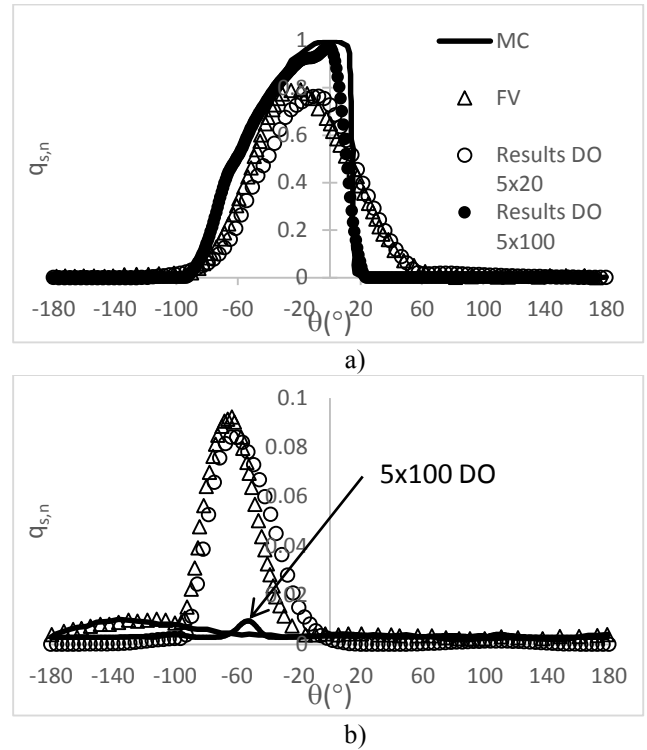

a)

b)

Figure 10: Normalized Heat Flux on (a) Front and (b) Back pipe surface for a 5 Pipe Reflecting Cavity

As shown in Figure 11 the geometry is reproduced in DesignModeler and imported to Fluent with the following Boundary conditions:

- A semi-transparent wall at the inlet aperture is created with flux maps a or b in Figure 12 with their corresponding ray directions.
- The cavity walls are perfectly reflective with no heat transfer at the surface.
- Absorber tubes are perfectly absorbing with copper material properties.
- The HTF is set as water with a $\overline{V_{in}} = 0.005 m/s$.
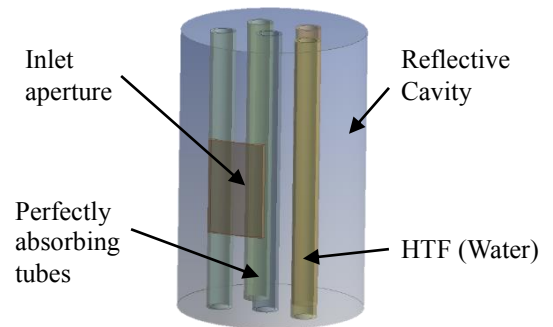

Figure 11: 3-D Geometry representing Biomass Cavity

Figure 13 and Figure 14 highlight the radiation distribution within the cavity due to the flux maps and ray directions and their corresponding effect on the temperature distribution on the surface of the absorber tubes. It is clear that with an ideal heat flux map there is a high concentration of heat flux on the centre

tube creating a hot spot and potential for large re-radiation losses and thermal stresses. Since the heat flux which reaches the surrounding tubes is effectively negligible, the design is subject to large inefficiencies and thermal stresses. These results may drive the optimization towards a geometry which would not efficiently absorb energy from an actual solar field. With the real distribution and ray directions applied, the heat flux distribution is more distributed throughout the cavity, which would reduce thermal stress and increase the thermal efficiency of the cavity.
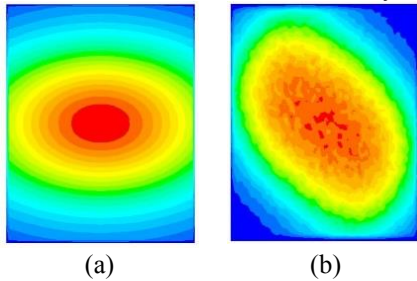

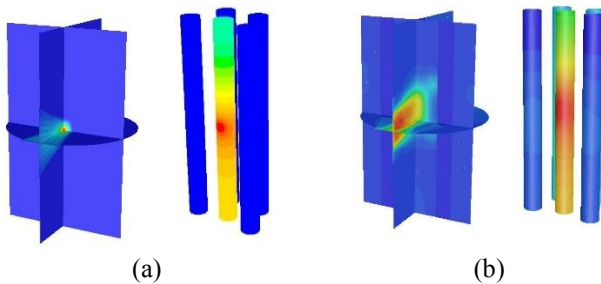**Figure 12:** (a) Ideal and (b) real flux map profiles for mapping onto cavity aperture

Figure 13: Incident radiation and temperature distributions for (a) ideal and (b) real heat flux profiles

CONCLUSIONS

A method to determine the thermal performance of a cavity receiver which is subject to a heat flux input at its aperture due to a large heliostat field has been proposed, which is able to couple ray-tracing and FV methods. It was determined that FV methods are able to accurately simulate the RTE to a high degree of solution accuracy when compared with the Monte Carlo method equivalent.
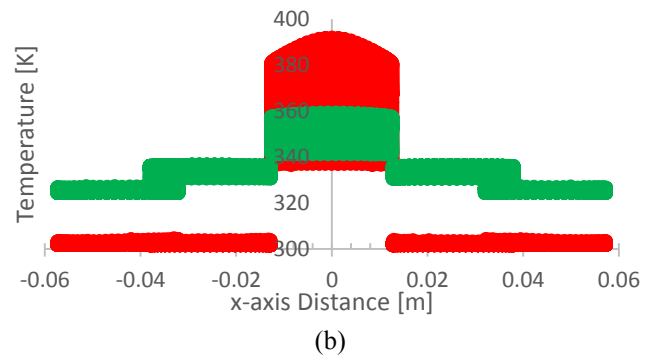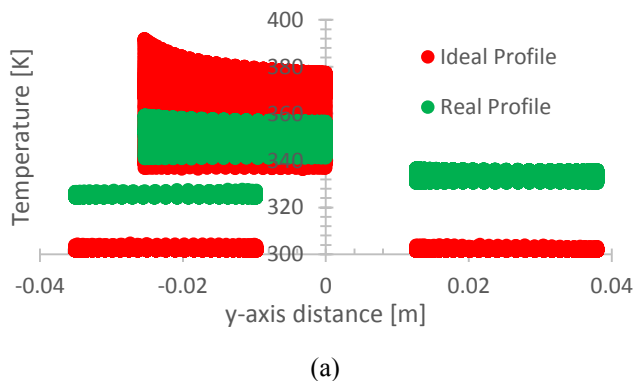

(a)

(b)

Figure 14: Temperature distribution [K] on absorber tubes versus (a) y-axis and (b) x-axis

Differences within the cavity due to various aperture heat flux profiles have been identified. Using a more realistic heat flux profile would benefit the goal of driving towards a design that will optimize heat absorption by a heat transfer fluid through minimizing unwanted effects such as convective and re-radiative losses. In addition, thermal 'hot spots' can be reduced.

ACKNOWLEDGEMENTS

REFERENCES

[1] Lichty, P., et al., *Rapid High Temperature Solar Thermal Biomass Gasification in a Prototype Cavity Reactor.* Journal of Solar Energy Engineering, 2010. **132**(1): p. 011012-011012.

[2] Martinek, J. 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.* International Journal of Heat and Mass Transfer, 2013. **58**(1): p. 585-596.

[3] Wendelin, T., A. Dobos, and A. Lewandowski, *SolTrace: A Ray-Tracing Code for Complex Solar Optical Systems.* NREL - Technical Report, 2013. **303**: p. 275-300.

[4] Versteeg, H.K. and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method.* 2007: Pearson Education.

[5] Binder, K., *Monte Carlo Methods.* Quantum Monte Carlo Methods, 1987: p. 241.

[6] Howell, J., *The Monte Carlo method in radiative heat transfer.* Journal of Heat Transfer, 1998. **120**(3): p. 547-560.

[7] Ansys, A.F., *14.0 Theory Guide.* ANSYS inc, 2011.

[8] Chai, J.C., H.S. Lee, and S.V. Patankar, *Finite volume method for radiation heat transfer.* Journal of thermophysics and heat transfer, 1994. **8**(3): p. 419-425.

[9] Noone, C.J., *Optimization of central receiver concentrated solar thermal: site selection, heliostat layout & canting*, 2011, Massachusetts Institute of Technology.