18èmes Journées Internationales de Thermique (JITH 2017)
Monastir (Tunisie), 25 - 27 Octobre 2017

Numerical study of heat transfer diluted particulate fluid for concentrating solar systems

Mohamed GHARAFI, AbdelIlah HAKIM, Naji ABDENOURI
Department of Mathematics, Laboratory of Applied Mathematics and Computer Science (LAMAI), FSTG
Cadi Ayaad University, Abdelkarim Elkhattabi, Marrakech, 40000, Morrocco.
n.abdenouri@uca.ac.ma

Abstract: This paper presents a numerical study of the temperature field and the heat transfer properties of a suspension of solid particles inside a tubular solar receiver used as a heat transfer fluid (HTF). This process is getting more attention due to its wide operating temperature range and its high heat transfer efficiency. Despite recent works on the same scheme, exploring its full potential and modeling the process are still challenging. In this study, we used an upward (DPS) of fine Silicon Carbide (SiC) particles (of a mean Sauter diameter of 63.9µm), with air as the carrying fluid inside a vertical solar receiver. The receiver is exposed to a concentrated solar radiation ranging from 200 to 250 kW/m² in the front face of the tube on a length of 0.5m The simulation was performed using Euler-Lagrange equation to model the fluid-particle suspension one-way interaction[1]. The results were significantly close to what have been obtained by using a Dense Discrete Phase Model (DDPM)[2]which requires far more resources and data. The particles reach a temperature of 780 K and the overall heat transfer coefficient of the outer-wall to particle suspension obtained was about 580 *W/m².K*. We observed also a pattern on how fast, or in which length the particles will attain a temperature depending on their initial escape position. The results will be next used to study deeply the reactive particles in the air flow at high temperature.

Keywords: Solar power, dense particle suspension, Heat transfer fluid, CFD.

1. Introduction:

The chemical industry and the energy conversion uses processes that are extremely endothermic, either to perform chemical reactions or for energy conversion[3].The concerned processes imply much conventional energy and still subject to optimization. On the other hand, the resulting CO_2 emissions are far too high regarding the international standards especially the amounts related to combustion processes[2]. Our aim in this paper is to study a potentially cost effective solution that makes use of the solar energy in such processes. The proposed method, known as an upward dense particle suspension (DPS), is a tubular solar receiver, exposed partially to a concentrated solar power. The receiver is in a vertical position, inside of it a fluidized bed of solid particles (SiC) is circulating in an upward stream. Some variations of this solution has been studied, and they suggest that the technology has more potential and presents a lot of opportunities of improvements[3]. In part, this is because of the models implicated in such study are still challenging and in active research, on the other hand, the computational resources required are still expensive. The temperature field in the mixture air-particle flow and the heat transfer coefficient are the determinant parameters for using such facility in chemical reactions or for the thermal energy storage.

The main advantages of this configuration are the wide range of operating temperature and the heat transfer efficiency [4]. The temperature is limited only by the melting temperature of the receiver's material, and the heat transfer coefficient is significantly high in comparison with other heat conversion or transport processes. The use of Silicon Carbide (SiC) particles is also essential to this configuration due to its specific chemical and thermal properties and its availability. In the next paragraph, we will present the detail of problem in hand.

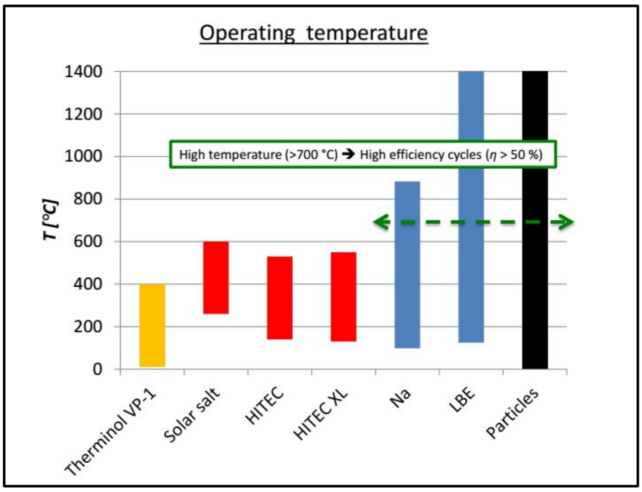


Figure 1: Operating temperature of different HTF[3]

2.Physical problem:

2.1. The solar receiver:

The principal component of the facility is the solar receiver, its function is to contain the fluidized bed and to receive, absorb and transfer the heat flux coming from the solar concentrator. It is a tubular pipe in a vertical position. In our model, we adjusted the position of the pipe so that the heat flux comes at the center of its height. It is made of a mild steel material. As for the geometry of the receiver, it is constrained by the exposed area. In fact, to have a viable data and results in general, the air-particle flow must be fully developed in terms of velocities and temperature. The length of the tube before and after the exposed area must be sufficient to avoid the disturbances and the refluxes of the flow while exiting the pipe.

Table 1 : geometry of the receiver

Feature	Symbol	Value (m)
Inner diameter	D_{in}	0.0424
Outer diameter	D_{out}	0.0488
Height	H	2.6300
Exposed length	$L_{exposed}$	0.5000
Exposed Area	$A_{exposed}$	$4.6759\ 10^{-4}$

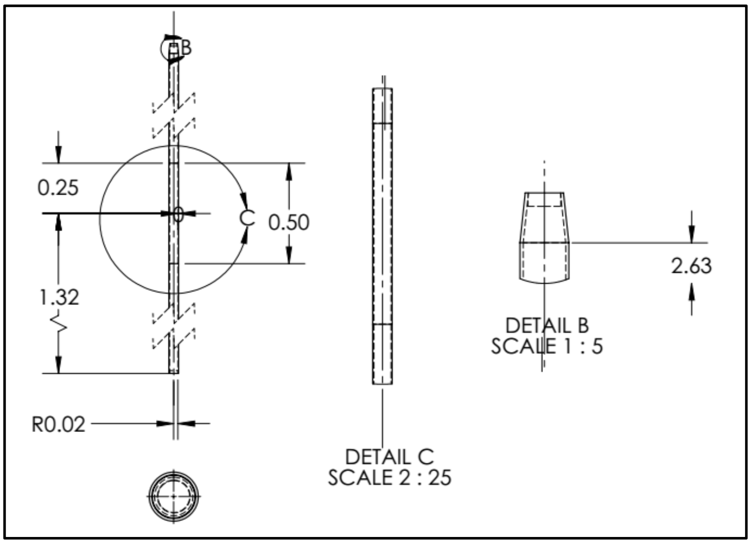


Figure 2 : the tubular receiver schema

The table [Table 2] below lists the main properties of the solar receiver's material.

Table 2 : the solar receiver's material properties

Property	Symbol	Value	Unit
Density	ρ_r	7870	kg/m^3
Specific heat	$C_{p,r}$	472	*J/(kg.k)*
Thermal conductivity	k	51.9	*W/(m.k)*
Melting temperature	$T_{melting}$	1673.5	*K*
Emissivity coefficient	ε_r	0.14 – 0.32	_

2.2 The suspension of solid particles:

The suspension of solid particles is acting in this design as heat transfer fluid HTF. The solid particles circulation the suspension is a fine Silicon carbide particles SiC of a mean Sauter diameter of $63.9\mu m$to allow a better heat exchanges. Their thermal properties [table 3] are appropriate for this use. For instants, they have a high sintering temperature, a high heat capacity. Moreover, they are available and have low-cost. The table [Table 3] below lists the physical properties of the SiC particles[4].

Table 3 : Silicon carbide properties

Name	ρ_p *(kg/m³)*	$C_{p,p}$ *(J/kg.K)*	k *(W/m.K)*	T_{sinter} *(°K)*	d_{sauter} *(μm)*
Silicon Carbide	3210	0.97 to 1.26 $0.284+1.614.10^{-3}$ T -7.10^{-7} T^2	180 to 40 $289.35-0.4262T+0.1810^{-3}T^2$	1620	63.9

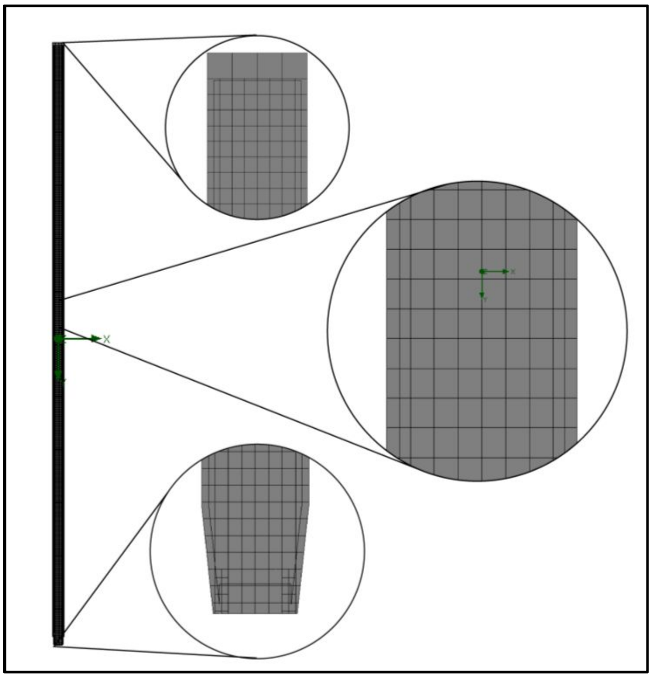
3. Simulation setup:

In this simulation, we used a 3D model, to solve a steady stat problem, taking into consideration the heat conductivity in part, gravity and radiation. Although the solar radiation was neglected except for the concentrated heat flux was modeled as a surface heat flux applied to a boundary face. In the next paragraphs, we'll describe the simulation setup: geometry, mesh, models and material, boundary conditions and the solution. It is also worth mentioning that we used the Euler-Lagrange equations to model the fluid-particle interaction. Thus, we run the simulation in two steps: first we calculated the flow field parameters, then, we proceeded to a particle study. It is also important to keep in mind that the interaction is a one-way interaction – fluid to particle and not the way around. In fact, the particulate phase in this model is considered sufficiently diluted to neglect its effects on the fluid[1].

3.1 The geometry and the mesh:

The tubular geometry of the solar receiver was obtained by creating a planar sketch containing the curve profile to be revolved. We used also split line feature to underline the face receiving the heat flux. The inner volume of the tube is preserved to the fluid domain. The end results were mentioned previously in the figure 2.

As for the mesh, we used a cut cell type of mesh for both cell zones (Solid and fluid). The following figures represent the different views of the mesh used in the simulation.



3.2. Models and material:

The materials we used in this simulation are mild steel for the tubular receiver, air as the carrying fluid and silicon carbide as solid particle in suspension.

The models that were used to perform the simulation are the energy model, turbulent and laminar model for the fluid flow and the discrete ordinates for radiation model. As for the particle study, after the flow simulation we introduced a uniform particle injection on the inlet face. Afterward, the coupling of the gaseous and the particulate phase was made using the Euler Lagrange model.

This model predicts the trajectory of the discrete phase by integrating the force balance on the particle written in Lagrangian frame of reference. This force is written as follow[5]:

$$\frac{du_p}{dt} = F_D\left(u - u_p\right) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x \quad (1)$$

With:

$F_D(u_p - u)$: is the drag force per unit particle mass;
F_x: the additional acceleration term;
u: the fluid phase velocity;
u_p: the particle velocity;

As for the drag coefficient, it is written as follows:

$$F_D = \frac{18\mu}{\rho_p d_p^2}\frac{C_D R_e}{24} \quad (2)$$

With:

μ: is the molecular viscosity of the fluid
ρ: the fluid density
ρ_p: the particle density
d_p: *the particle diameter*

Re is the Reynolds number defined as:

$$R_e \equiv \frac{\rho d_p |u - u_p|}{\mu} \quad (3)$$

The viscous model makes use of the following equations[6]:

$$\frac{\bar{D}k}{\bar{D}t} = -\nabla.\left(\frac{\nu_T}{\sigma_k}\nabla k\right) + P - \varepsilon \qquad (4)$$

Whereσ_k is a model constant, T' is the flux of Reynolds stress and P is the rate of production of turbulent kinetic energy, given by:

$$P = -\langle u_i u_j \rangle \langle \frac{\partial U}{\partial x_j} \rangle \qquad (5)$$

And viscosity can be computed by:

$$\nu_T = C_\mu \frac{k^2}{\varepsilon} \qquad (6)$$

Where k is the kinetic energy and ε is the dissipation rate and we have:

$$\frac{\bar{D}\varepsilon}{\bar{D}t} = -\nabla.\left(\frac{\nu_T}{\sigma_\varepsilon}\nabla \varepsilon\right) + C_{\varepsilon 1}\frac{P_\varepsilon}{k} - C_{\varepsilon 2}\frac{\varepsilon^2}{k} \qquad (7)$$

The constants introduced are evaluated as follows:

$$C_\mu = 0.09, C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

3.3. Boundary conditions:

3.3.1. Inlet boundary condition:

The inlet boundary condition was applied normal to the bottom face of the tube, as an inlet fluid velocity of a magnitude of 2m/s. In previous attempts, lower velocities though particle study was used but wasn't successful as none the particles trajectories fate was "escaped". In previous attempts of air-particle flow study, the lower velocities lead to the falling of particles which can't be driven by air flow. We also made sure that the fluid velocity was fully developed.

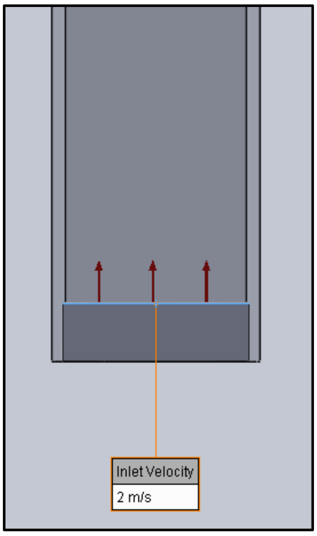


Figure 6: Inlet boundary condition

3.3.2. Outlet boundary condition:

As for the outlet boundary condition, we applied a pressure opening using environment pressure. And to make sure that the fraction of the flow that refluxes is as low as possible, we reduced the outlet opening radius. This allowed us to have the solution converges as fast as possible.

3.3.3. Wall condition:

There are two types of wall conditions used in the simulation. The first one is a mixed thermal wall condition, modeling the external convection with a heat transfer coefficient of 15$W.m^{-1}K^{-1}$as[7] well as the external radiation in form an emissivity coefficient. This condition was applied to all the external face of the tubular pipe, except the one receiving solar heat flux. The second one is surface heat generation rate that represents the concentrated solar radiation coming from the solar plant.

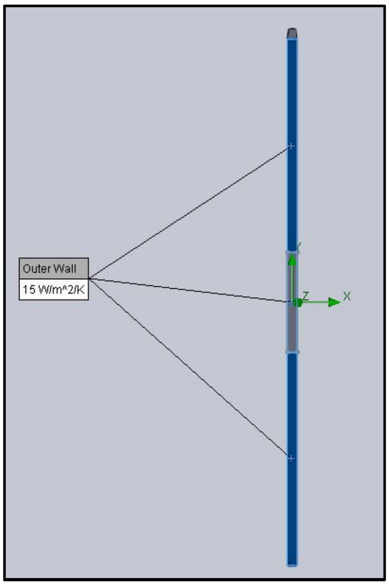


Figure 8: the first wall condition type

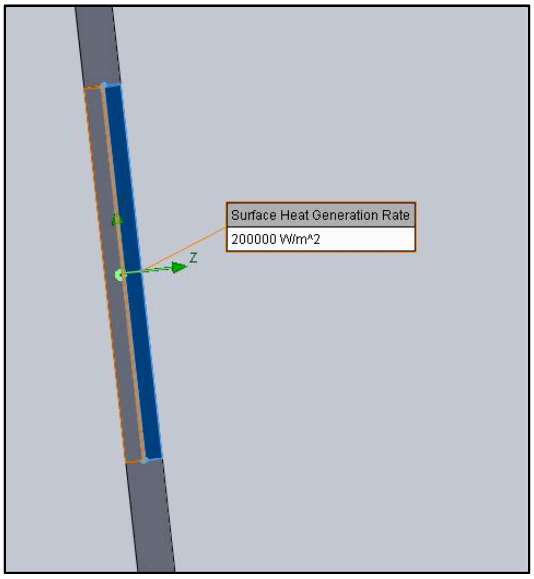


Figure 9: the second wall condition type

3.4. Computationalsetting:

The convergence of the solution was monitored by the residual values of the continuity, the component of the fluid velocity. The number of iteration was set to 1000. The solid cell count is 24 660 and the fluid cell count is 18 365, the total cell count is 43 016. The simulation was run on a 2.67 Hz computer for 15minutes.

4.Results and discussion:

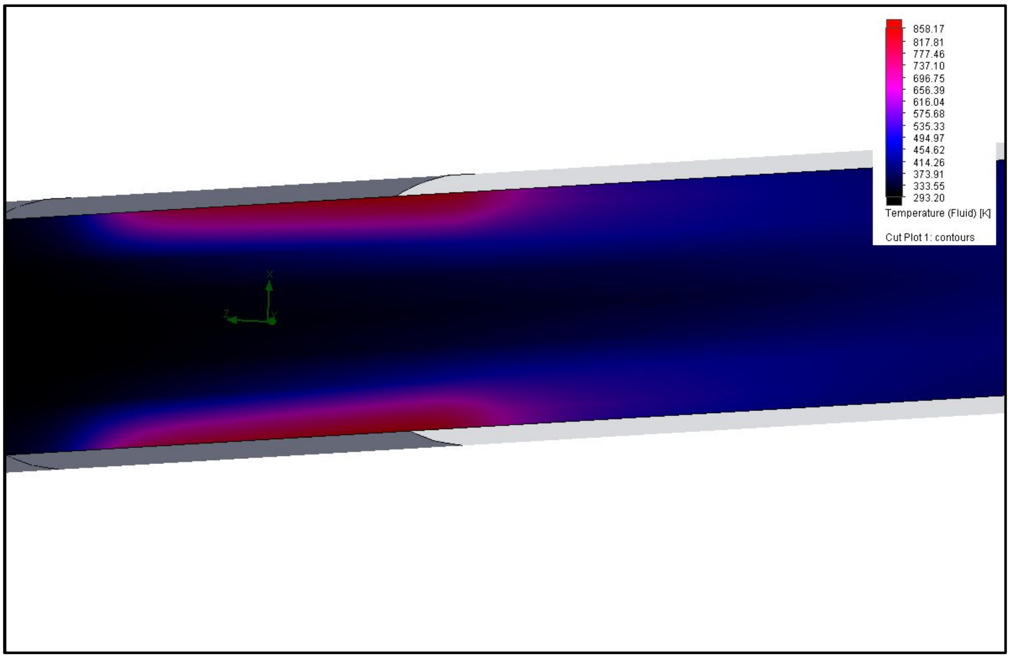


Figure 10 - a: the fluid temperature plot near the heating area the XY view

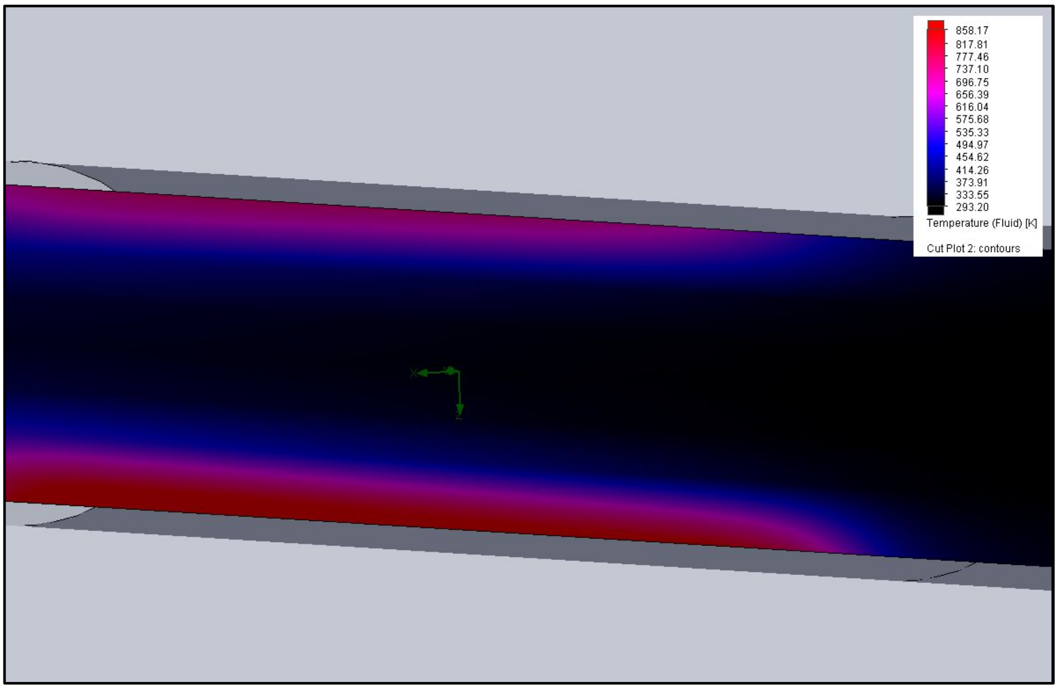


Figure 10 - b: the fluid temperature field near the heating area the ZY view

The results were coherent with what we would have expected from the previous work [4]. The temperature of the fluid reached 953*K*, while the solid wall attains a temperature of 1080 *K.* The particles though have an interesting pattern. In fact, the more their initial position at the inlet face is far away from the center, the quicker they heat up. Or in terms of trajectory length required to increase their temperature by 200 *k*, we found that it is significantly shorter.

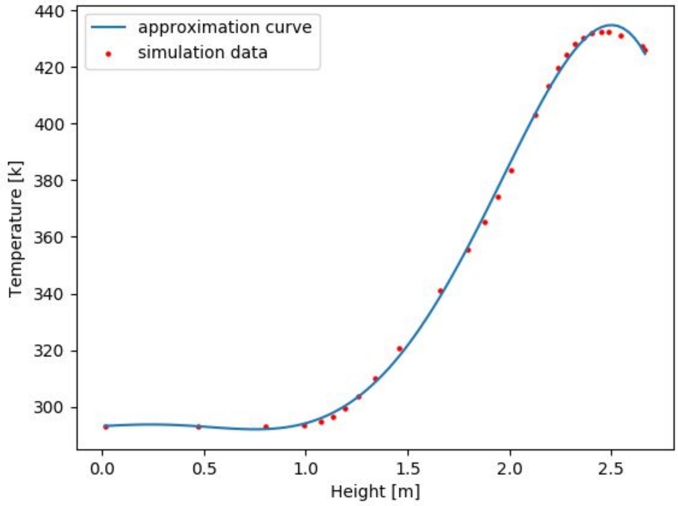


Figure 11: The temperature curve in the core of the tube

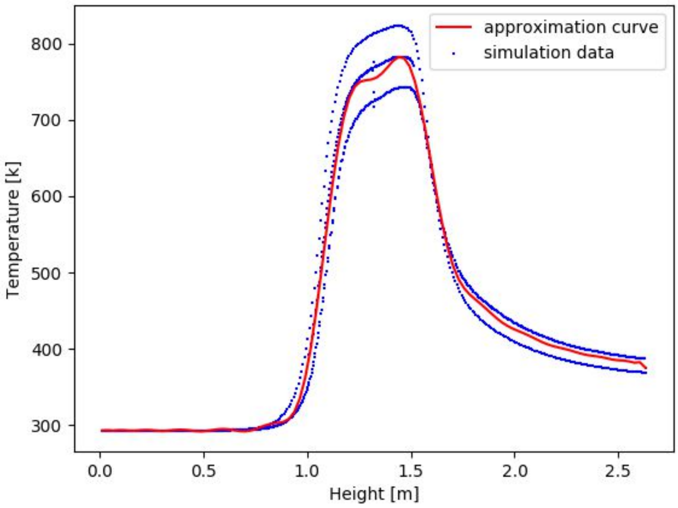
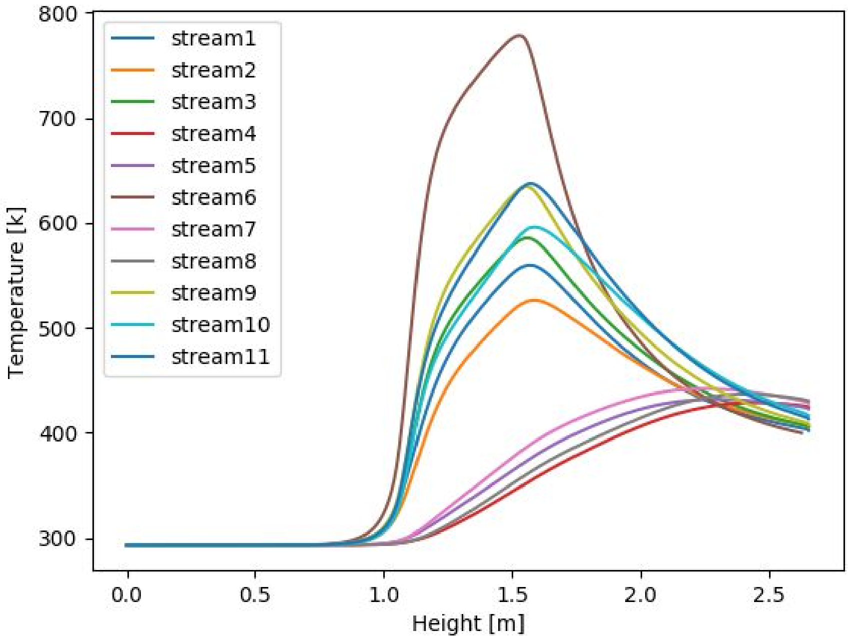


Figure 12: the fluid temperature near the wall

To demonstrate the pattern of the particles heating per escape position at the inlet boundary, we have the following charts, first we drew the temperature curve per each injection stream to have a main idea about how the temperature of the particle behaves.



After that we tried to trace back each stream to its escape position at the inlet face, and then we drew a reverse length parameter D that characterizes the heating rate of each stream according to its initial coordinate on the inlet face.

With:

$$D = H - L \qquad (1)$$

And:

L is the required length to have a temperature increase by 200 K
H is the length of the tube.

The figures 13 doesn't show the consistency of the pattern, but when combined in one figure using the distance of the particle to the center of the inlet face we observe the relationship between the escape position and the heating rate as shown in the figure 15.

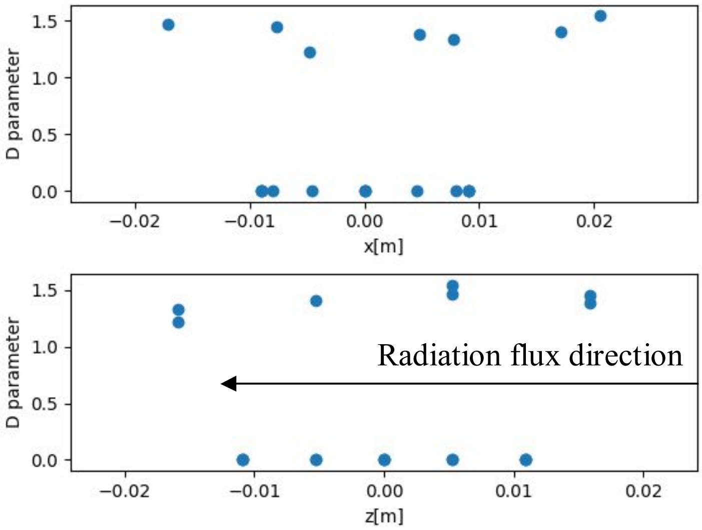


Figure 13: Particle heating depending on its x and z position

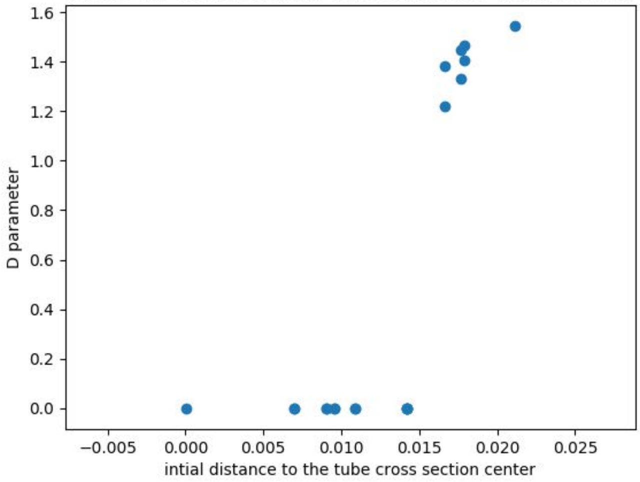


Figure 14: Particle heating depending on its initial position

Though the problem doesn’t have a cylindrical symmetry, hence we did check also the heating depending on the particle’s angular departure position.

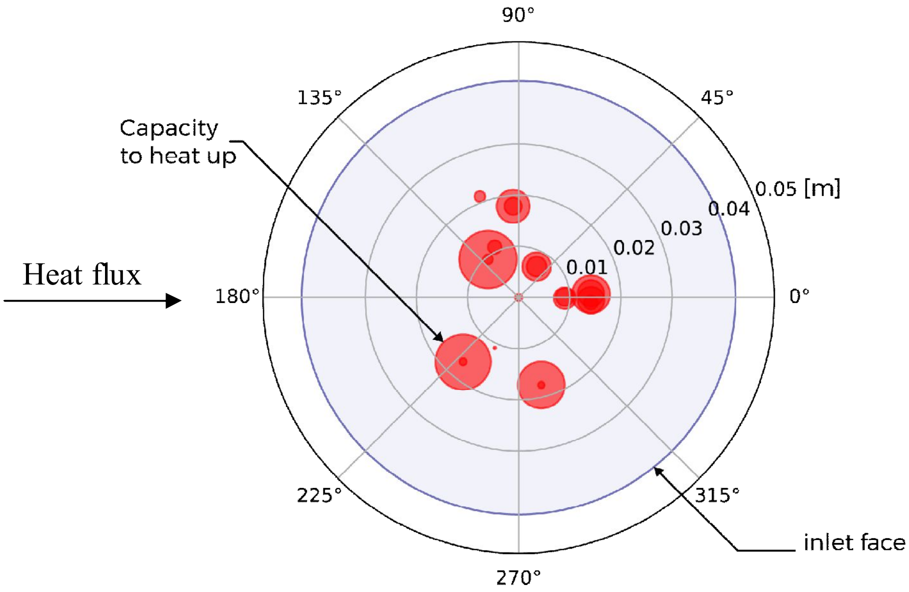


Figure 15: Particle's capacity to heat up per departure position

Conclusion

The numerical study in hand was an attempt to explore the thermal properties of the suspension of solid particles as a heat transfer fluid. This using is a reduced 3D model and reduced computational resources way. The results were viable and can be related to what was conforming to what it found in previous studies. The reduced amount of resources used in this simulation, permit moving on to investigating innovative ways of optimizing this process. For instance, a genetic algorithm and programming based study would yield better results and configurations[8]. Moreover, the pattern of the particles thermal behavior observed, will allow a better particle injection hence an improved thermal efficiency of the (DPS) solar receiver.

Nomenclature

Symbol	Nom, *unite*
k	thermal conductivity, *W/m.K*
T	temperature, *K*
Cp	heat capacity, *J/(kg.k)*
H	height, *m*
$L_{exposed}$	length of the exposed area, *m*

Greek symbols

ρ	density, $kg.m^{-3}$
ε	emissivity coefficient

Exponent, Indices

p	particle
r	receiver

Acknowledgements:

This project was funded from the European Union's European Union Seventh Framework Programme FP7/2007-2013 under grant agreement n° 609837 project acronym STAGE-STE.

References

[1] S. Abanades and G. Flamant, "Experimental study and modeling of a high-temperature solar chemical reactor for hydrogen production from methane cracking," vol. 32, pp. 1508–1515, 2007.

[2] D. K. Fidaros, C. A. Baxevanou, and N. S. Vlachos, "A parametric study of a solar calcinator using computational fluid dynamics," vol. 48, pp. 2784–2791, 2007.

[3] H. Zhang *et al.*, "Particle circulation loops in solar energy capture and storage: Gas-solid flow and heat transfer considerations," *Appl. Energy*, vol. 161, pp. 206–224, 2016.

[4] G. Flamant *et al.*, "Dense suspension of solid particles as a new heat transfer fluid for concentrated solar thermal plants: On-sun proof of concept," *Chem. Eng. Sci.*, vol. 102, pp. 567–576, 2013.

[5] H. H. Hu, N. a. Patankar, and M. Y. Zhu, "Direct Numerical Simulations of Fluid–Solid Systems Using the Arbitrary Lagrangian–Eulerian Technique," *J. Comput. Phys.*, vol. 169, pp. 427–462, 2001.

[6] E. Sciences, "Evaluation of RANS turbulence models for the simulation of channel flow," no. December, 2014.

[7] A. Reyes Urrutia, H. Benoit, M. Zambon, D. Gauthier, G. Flamant, and G. Mazza, "Simulation of the behavior of a dense SiC particle suspension as an energy transporting vector using computational fluid dynamics (CFD)," *Chem. Eng. Res. Des.*, vol. 106, pp. 141–154, 2016.

[8] S. Rajeev and C. S Krishnamoorthy "Discrete Optimization Of Structures Using Genetic Algorithms " vol. 118, no. 5, pp. 1233–1250, 1992.