

# Homogeneous and Inhomogeneous Model for Flow and Heat Transfer in Porous Materials as High Temperature Solar Air Receivers

Olena Smirnova<sup>1\*</sup>, Thomas Fend<sup>1</sup>, Schwarzbözl Peter<sup>1</sup>, Daniel Schöllgen<sup>1</sup>

<sup>1</sup>German Aerospace Center, Institute of Technical Thermodynamics, DLR, Koeln, Germany

\*Linder Höhe, 51147 Köln; E-Mail <mailto:olena.smirnova@dlr.de>;  
Telephone +49 2203 601-2901

**Abstracts:** Results of calculations on flow and heat transfer in a porous Silicon Carbide honeycomb structure applied as a solar air receiver are presented. In this application porous materials are put in concentrated solar radiation. Flux densities of up to 1000 MW/m<sup>2</sup> are reached. Simultaneously, ambient air flows through the material to be heated up to temperatures of app. 800°C. This hot air is then used to feed the steam generator of a steam turbine to generate solar electricity (solar tower technology).

The results are describing the resulting temperature field in the receiver. The main problem of the solar receiver is connected with the overheating and destruction of parts of the working surface. For the simulation material properties such as permeability, thermal conductivity and volumetric convective heat transfer are needed. These have been determined experimentally.

The study has been carried out in 2 phases:

1. Simulation of the velocity distributions and temperature fields in a single channel of the honeycomb structure
2. Combined model with the honeycomb structure as a homogenized structure and the support structure taken with its real geometry.

The obtained results of the numerical calculation for the mentioned cases are presented and discussed. The working temperatures, velocities and heat flux distributions which correspond to the used inlet parameters were found from the simulations. The results can be used as a base for the prediction of the working regimes. This approach can help to perform a safe operation and avoid overheating and damage of the receiver material.

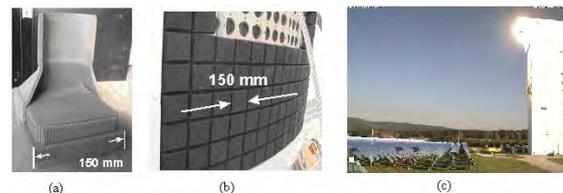
**Keywords:** Volumetric receiver; Solar Tower technology; Coefficient of the Term Specific Heat Transfer

## Introduction

During the last years a lot of theoretical and experimental research has been connected with the properties of the open solar volumetric receiver, because of the increasing importance of solar power technology [1-2]. A solar volumetric receiver is the central element of the so-called *solar tower*

*technology* which converts concentrated solar radiation into high temperature heat. In case of a volumetric absorber a cellular material is employed to absorb concentrated solar radiation and to transfer the energy to a fluid flowing through its open cells. The numerical calculation of this process is of increasing interest, since practical problems like the overheating of the receiver can be assessed and possible solutions can be found.

A general view of the receiver can be seen in Fig. 1.



**Figure 1:** a) photograph of a single absorber module, which has been cut for a better view, b) photograph of the SOLAIR 3 MW receiver from the front during installation showing many installed absorber modules (see a) next to each other. In the upper part of the picture there are still some openings waiting for modules. c) The experimental solar tower power station in Juelich (Germany)

The task of the numerical calculation of the flow and the heat transfer in the volumetric solar receiver was divided in two independent subtasks:

1. Single channel model. This gives the possibility to select the optimum geometry of the receiver by comparison of the efficiency of the receiver with a different relation between channel size and wall thickness.
  2. Homogeneous model. This considers the receiver as a solid porous continuum with effective permeability, heat conductivity and heat transfer properties.
- For these subtasks the flow and heat transfer processes have been considered as stationary and the following COMSOL application modes have been used:

1. Weakly compressible Navier–Stokes;
2. Convection and heat conduction in air;
3. Heat conduction in the solid body.

The use of two heat transfer modes made it possible to find the temperature distribution both of the solid body and the air independent from each other.

The geometry of both the single channel model and the homogeneous model are completely symmetric. Therefore - for simplicity - only one quarter of the total volume of interest has been defined. The 3D-geometry of the single channel model was done in COMSOL. For the drawing of the 3D-geometry of the homogenous model *Autodesk Inventor* was used.

The volumetric receiver was considered as a porous continuum with determined macro properties such as porosity and permeability. The model includes the heat transfer from the hot surface of the solid body to the air flow by taking into account the *volumetric heat transfer coefficient*, a quantity to be determined experimentally. The absorbed concentrated solar radiation was considered in this model as a volumetric heat source.



**Figure2:** 3D-geometry for the calculation of the homogenous model

As an example, the homogeneous model of the receiver is described in detail in the following section.

### Theoretical model and use of COMSOL Multiphysics

#### Area Conditions

1. For the numerical calculation of the velocity and pressure fields the *Weakly Compressible Navier – Stokes* application mode was used in this model. This application mode describes the connection of the fluid velocity  $u$  and the pressure  $p$  according to the equation:

$$\eta \cdot \nabla (\nabla u + (\nabla u)^T) = \rho \cdot u \cdot \nabla u \quad (1)$$

Since it should be treated as a porous medium, the area of the receiver was described by a slight modification of equation 1, the *Brinkman–equation*, which simplifies the consequence of the boundary conditions between porous structure and air:

$$\left( \frac{1}{\varepsilon_p} \right) \eta \cdot \nabla (\nabla u + (\nabla u)^T) = \left( \frac{\eta}{K} \right) \cdot u \quad (2)$$

Here,  $\eta$  is the viscosity [kg/m s],  $K$  the permeability [m<sup>2</sup>],  $\rho$  the density [kg/m<sup>3</sup>] and  $\varepsilon_p$  the porosity.

2. The mode *Convection and Heat Conduction* allows simulating the heat transfer in the air, according to the next equation:

$$\nabla (-k \cdot \nabla T + \rho \cdot c_p \cdot u \cdot T) = Q \quad (3)$$

Here,  $k$  [W/m K],  $c_p$  [J/kg K] and  $Q$  [W/m<sup>3</sup>] denote the coefficient of the thermal conductivity, the specific isobaric heat and the heat source term (heating power per unit volume) respectively.

The heating power per unit volume for the porous structure was determined through the so called volumetric heat transfer coefficient  $\alpha A_V$  [W/m<sup>3</sup> K] according to the equation:

$$Q = \alpha \cdot A_V \cdot (T_2 - T) \quad (4)$$

Here,  $T_2$  and  $T$  denote the solid body and the air temperature respectively. The volumetric heat transfer coefficient  $\alpha A_V$  was calculated according to the following equation:

$$N\bar{u} \cdot A_V = c_{ht} \cdot \text{Re}^m \quad (5)$$

Here  $c_{ht}$  and  $m$  are constants which have been determined in a former study [3].  $A_V$  is the specific surface [m<sup>2</sup>/m<sup>3</sup>].

The volumetric heat transfer coefficient was then calculated from the Nusselt-numbers with the equation:

$$\alpha \cdot A_V = \frac{N\bar{u} \cdot A_V \cdot k}{\delta} \quad (6)$$

Here  $\delta$  denotes the characteristic length of the pore structure, which is in this case the channel diameter.

3. The mode *Heat Conduction* simulates the heat transfer in the solid body, according to the equation:

$$-\nabla (k_{por} \cdot \nabla T) = q_0 \quad (7)$$

Here,  $q_0$  is the heating power per unit volume (heat source term), which takes into account the absorption of the solar radiation of the receiver as well as the heat transfer to the fluid.  $k_{por}$  [W/m K] is the heat conductivity in the porous structure, which was determined experimentally as a function of temperature:

$$k_{por} = (1 - \varepsilon_p) \cdot 280,7 \cdot \exp(-0,0021 \cdot T_2) \quad (8)$$

The complete equation for the heat source term is:

$$q_0 = I_0 \cdot \varepsilon_p \cdot \xi \cdot \exp(-\xi \cdot z) + \alpha \cdot A_v \cdot (T - T_2) \quad (9)$$

Here,  $I_0$  [W/m<sup>2</sup>],  $\xi$  [1/m] and  $z$  [m] denote the intensity of the radiation, the measured value of the “extinction coefficient”, and the coordinate in flow direction.

### Boundary Conditions

For the simulation the hydrodynamic process, the following boundary conditions were set:

Inlet: velocity  $u=u_0$ .

Outlet: pressure without viscous stress:  $p=p_0$

$$\left( \frac{1}{\varepsilon_p} \right) \cdot \eta \cdot (\nabla u + (\nabla u)^T) = 0$$

Other boundaries: wall “no - slip”  $u=0$ .

For the simulation of the temperature in the air:

Inlet: entrance temperature  $T=T_0$ .

Outlet: convective flow:

$$n \cdot (-k \cdot \nabla T) = 0.$$

On the boundary Cup – Air:  $T=T_2$ .

Other boundaries: thermal isolation

$$n \cdot (-k \cdot \nabla T + \rho \cdot c_p \cdot u \cdot T) = 0$$

For the simulation of the temperature in the solid body:

Inlet into the receiver: radiation heat flow:

$$n \cdot (k \cdot \nabla T_2) = q_0 + C \cdot (T_{amb}^4 - T_2^4).$$

Here

$$q_0 = I_0 \cdot \cos \alpha \cdot (1 - \varepsilon_p) - (1 - \varepsilon_p) \cdot F \cdot (T_2 - T_0)$$

describes the absorbed radiation on the front boundary receiver – air and also the losses due to thermal radiation from the receiver to the ambient ( $C=\varepsilon \cdot 5.67 \cdot 10^{-8}$  [W/m<sup>2</sup> K<sup>4</sup>] is the emissive constant with  $\varepsilon$  denotes the emissivity of the receiver material and  $T_{amb}$  is the environment temperature).  $F$  is the convective heat transfer coefficient describing the convective losses at the front of the receiver.

At the outlet of the receiver the temperature  $T_2$  was set to  $T_2=T$ . The boundary conditions at the lateral

boundaries of the receiver were set to thermal isolation:

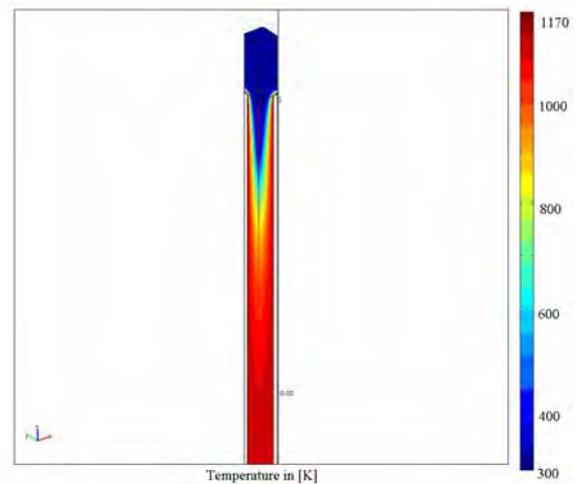
$$n \cdot (-k \cdot \nabla T + \rho \cdot c_p \cdot u \cdot T) = 0$$

For the model, adaptive meshing with different meshing steps for each area was used.

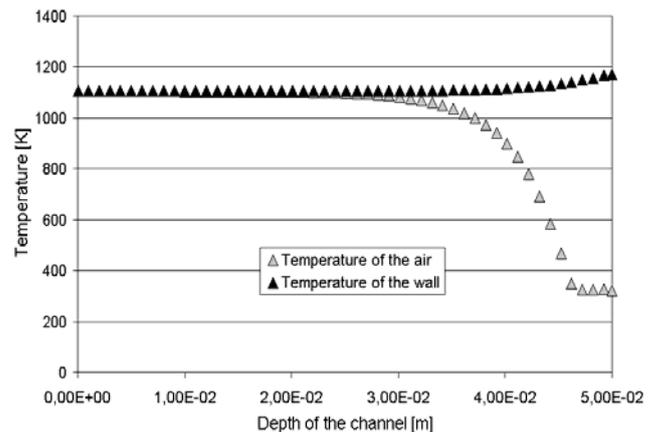
### Results and discussion

The results of the temperature distribution and velocity fields for the both models are presented here to compare the two numerical approaches.

The plot in **Figure 3** shows the distribution of the fluid’s temperature in the single channel model.



**Figure 3:** Temperature distribution of the air in the receiver calculated with the single channel model

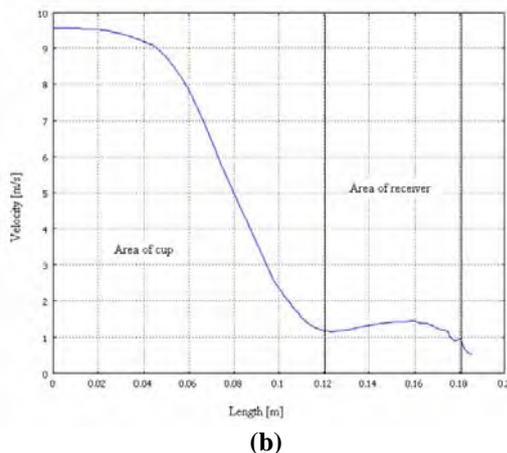
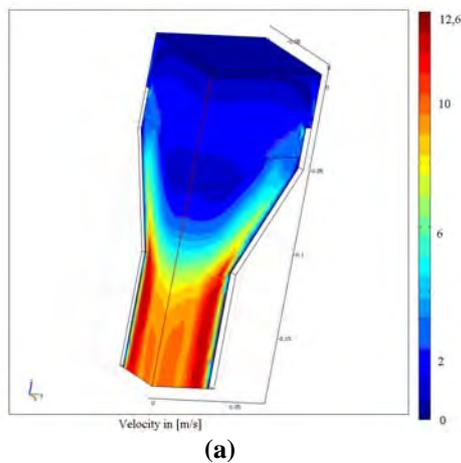


**Figure 4:** Comparison of the temperatures of the air ( in the middle of the channel ) and the wall ( at the middle of the wall ) in flow direction (air inlet is at the right end of the diagram)

**Figure 4** shows the development of the air temperature in the middle of the single channel. This diagram shows that at a depth of  $\approx 25$ mm from the channel entrance the temperature of the solid body and air achieve equal values.

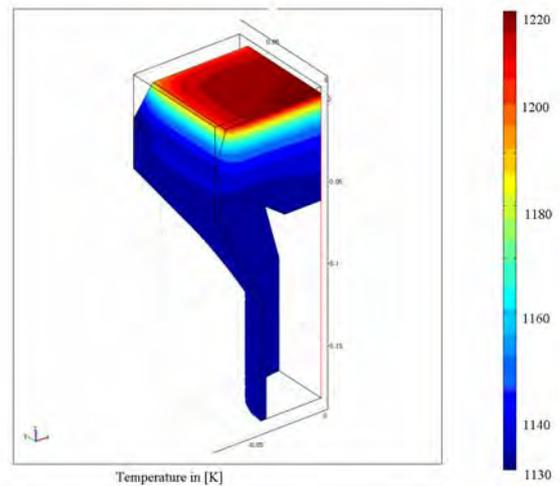
**Figure 5** displays the velocity field for the homogenous model. The plot shows a nearly parabolic distribution of the velocity field in the cup with the areas of the higher velocities along the cup wall in the outlet area. This effect takes place because of the reduction of the entrance cross – section of the cup. It causes an increase of the convective heat flow and of the thermal tension in this area.

The diagram in **Figure 5 (b)** shows the air velocity in the centre of the cup. Three zones can be observed. The receiver area is of special interest because in this zone the velocity firstly increases from the inlet value ( $\approx 1.0$  [m/s]) to a maximum value of  $\approx 1.45$  [m/s]. Then, it decreases again to  $\approx 1.2$  [m/s]. After that it increases to  $9.5$  [m/s] due to the decreasing cross-sectional area of the cup.

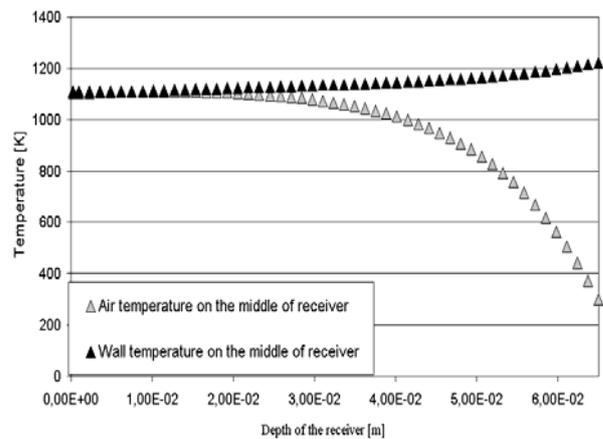


**Figure 5:** Plot and diagram of the development the velocity field in homogenous model

The plot in **Figure 6** displays the temperature field in the solid body. It is interesting to study the temperature distribution in the solid body on the inlet surface of the receiver. The plot shows a symmetrical distribution with a decrease of the temperature from  $1260$  [K] in the middle of the receiver to  $1130$  [K] at the edge. Similar results were shown in experiments [2].



**Figure 6:** The temperature field in the solid body of the homogenous model



**Figure 7:** Comparison of the temperature distribution in the central axis of the receiver both of the wall and of the air (air inlet is at the right end of the diagram)

**Figure 7** shows the temperature distribution in the central axis of the receiver both of the wall and of the air. Both temperatures become equal from a depth of  $\approx 45$ mm. The comparison of two simulation approaches shows that the outlet temperatures are approximately equal. However, the point, from which solid and fluid temperatures become equal, is slightly different. This may be due to the lack of precision in determining the volumetric heat transfer coefficient experimentally.

## Conclusions

Both simulation approaches show good stability of the results with a good correspondence of the outlet temperature. The results of the homogenous model show possible locations of overheating in the center of the inlet surface of the receiver and at the cylindrical surface of the cup in the outlet area. These results correspond to the experimental results. A further work-out of the homogeneous model will give the possibility for a detailed comparison of the numerical and the experimental results.

**Nomenclature:**

<b>u</b>	velocity [m/s]
<b>P</b>	pressure [Pa]
$\eta$	dynamical viscosity [kg/m s]
$\rho$	density [kg/m <sup>3</sup> ]
$\varepsilon_p$	porosity [-]
<b>K</b>	permeability [m <sup>2</sup> ]
$k$	thermal conductivity [W/m K]
$c_p$	specific isobaric heat [J/kg K]
<b>Q</b>	heating power per unit volume for the convection heat transfer [W/m <sup>3</sup> ]
$\alpha$	convective heat transfer coefficient [W/m <sup>2</sup> K]
$T$	air temperature [K]
$T_2$	solid temperature [K]
$A_V$	specific surface [m <sup>2</sup> /m <sup>3</sup> ]
Re	Reynolds number [-]
$N\bar{u}$	Nusselt number [-]
$q_0$	heating power per unit volume for the conductivity heat transfer [W/m <sup>3</sup> ]
$k_{por}$	effective heat conductivity of the porous material [W/m K]
$I_0$	entrance heat radiation per unit surface [W/m <sup>2</sup> ]
$\xi$	extinction coefficient of the radiation [1/m]
$z$	coordinate along the main flow direction [m]

**References:**

- [1] M. Becker, Th. Fend et al.: Theoretical and numerical investigation of flow stability in porous materials applied as volumetric solar receivers, *Solar Energy* 80 (2006) p.1241 - 1248
- [2] Th. Fend, B. Hoffschmidt et al.: Porous materials as open volumetric solar receivers experimental determination of thermophysical and heat transfer properties, *Energy* 29 (2004), (5-6), p.823 - 833
- [3] B. Hoffschmidt: „Comparison and evaluation of different concepts of volumetric radiation receivers“, Doctoral Thesis RWTH Aachen, DLR-Forschungsbericht 97-35, (1997), p.38 - 41