Revue des Energies Renouvelables Vol. 20 N°4 (2017) 713 - 720
DOI: https://doi.org/10.54966/jreen.v20i4.662

Comparison between two solar tower receivers of different geometry

Messaoud Hazmoune [1,2,*], Benaoumeur Aour [2], Mohamed Moundji Hadjiat [1] and Ali Malek [1]

[1] Centre de Développement des Energies Renouvelables, CDER
B.P. 62 Route de l'observatoire, 16340 Bouzaréah, Alger, Algeria

[2] Laboratoire de Biomécanique Appliqué et Biomatériaux, LABAB
Ecole Nationale Polytechnique d'Oran, Oran, Algeria

(reçu le 10 Décembre 2017 - accepté le 20 Décembre 2017)

Abstract - *This study presents a comparison between two solar power tower receivers. Selecting the optimum location of thousands of heliostats, the most profitable tower height, receiver size and geometric shape remains a challenge. In our case, we are interested in the geometric shape of the receiver. Two solar receivers connected in series and the other in parallel with different cross-sections and identical internal surface areas were studied. We performed a comparison to find the right pipe connection that gives us the good thermal capacity and solar tower efficiency. The heat transfer fluids used are water vapor and molten salts ($NaNO_3$-$NaNO_2$ - KNO_3/$NaNO_3$ - KNO_3/LiF - NaF - BeF_2), the software used for this study is ANSYS CFX with an unstructured grid with 825 300 cells and standard (k-ε) turbulent model and the flow of heat imposed on the receiver tubes 0.5 MW. We have been assumed that heat transfer occurs only by conduction and convection. The simulation results show the influence of parameters on the temperature field between the input and output of the receiver. The numerical simulation was done by the ANSYS CFX calculation software and using a Z620 Workstation computing machine. The objective of this work is to define the performance geometry for receivers in solar towers.*

Résumé - *Cette étude présente une comparaison entre deux récepteurs de tours d'énergie solaire. En choisissant l'emplacement optimal de milliers d'héliostats, la hauteur de la tour la plus rentable, la taille du récepteur et la forme géométrique restent un défi. Dans notre cas, nous nous intéressons à la forme géométrique du récepteur. Deux récepteurs solaires connectés en série et l'autre en parallèle avec des sections transversales différentes et des surfaces internes identiques ont été étudiés. Nous avons effectué une comparaison pour trouver la bonne connexion de tuyau qui nous donne la bonne capacité thermique et l'efficacité de la tour solaire. Les fluides caloporteurs utilisés sont la vapeur d'eau et les sels fondus ($NaNO_3$-$NaNO_2$ - KNO_3 / $NaNO_3$ - KNO_3 / LiF - NaF - BeF_2), le logiciel utilisé dans cette étude est l'Ansys CFX avec une grille non structurée de 825 300 cellules et modèle turbulent standard (k-ε) et le flux de chaleur imposé aux tubes récepteurs 0.5 MW. Nous avons supposé que le transfert de chaleur se produit uniquement par conduction et convection. Les résultats de la simulation montrent l'influence des paramètres sur le champ de température entre l'entrée et la sortie du récepteur. La simulation numérique est réalisée par le logiciel de calcul Ansys CFX et à l'aide de la machine informatique Z620 Workstation. L'objectif de ce travail est de définir une géométrie performante pour les récepteurs dans les tours solaires.*

Keywords: 3D-simulation - Solar power tower - Solar receivers - Flow - Heat transfer.

1. INTRODUCTION

Recently, the uses of solar concentration technologies has registered a big development. In 2013, registered 2.136 GW as an exploited power amount, 2.447 GW as next realization program and 10.135 GW as announced solar energy program in USA. Where the crescent dunes solar energy project and central of Ivanpah represent one of big

* m.hazmoune@cder.dz

investment in solar energy in 2013. In China, big CSP projects of 17 GW of production capacity are in realization.

The generation of energy by using the technology solar power tower has a considered advantages compared by the other technologies. This technology uses consider number of mobile mirrors (heliostats) for reaches the high temperatures under high concentration of the solar radiation on receiver at top of tower as presented in figure 1. These mirrors have the same focal point at the receiver. Where the energy of solar rays converted to the heat energy that used to feed the turbine related by electrical generator to produce the electricity. The efficiency of this system related by various design parameters and constraints.

Serval works have been done in the aim to determine the best selected design parameters under constraints to reach the maximum solar collected energy. Xu *et al.* [3] applied the steady heat and mass transfer models of the porous media on solar receivers, also they analyze the typical influences of the porosity, average particle diameter, air inlet velocity, and thickness on the temperature distribution.

Different designs of receiver are exist such us cavity receiver's types. This designs developed to reduce the losses in reflected radiation and increase the efficiency. In technical designs, radiative losses are eventually reduced to the same order of magnitude as the convective losses [4]. The complexity of analyses and the important investment in this field push us to use the efficient packages and methods to study and to predict the losses in applied technologies. In this context, CFD analysis methods represent one of the best method used actually.

In this case, in aims to minimize the thermal losses by increases the attack surface, Garbrecht *et al.* [7] investigated a new design solar central receiver by introducing of hexagonal pyramid shapes, which are surround the heliostat field the results gave a decreases of 1.3 % of the reflection radiation losses with thermal efficiency of 91.2%.

Crocker *et al.* [6] found that countercurrent flow seemed to lead to the most efficient receptor. but these earlier studies lacked some physics, such as the effects of the window and true solar irradiation from the heliostat field, and were. Limited to straight cylindrical geometry.

In this work, we test a receiver of a solar tower with two different geometries with different parameters such as the flow of heat, heat transfer fluids, and the speed of the entry of the calorific fluids.

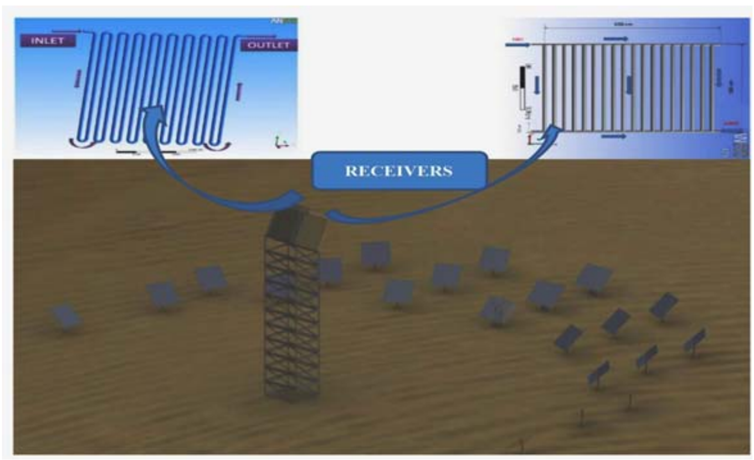


Fig. 1: Solar power tower

2. GEOMETRY DETAILS AND MESHING

Ansys CFX software (Version 13) was used in this study. The geometry of receiver is obtained by Ansys Workbench. It have $D = 2.2\ \text{cm}$ of the receiver diameter selected. With length of $L = 100\ \text{cm}$ and height of $H = 100\ \text{cm}$, (figure 2 and 3). A non-uniform grid system is used to discretize the governing equations, as shown in figure 4 and 5.

In addition, different tests were carried out to study the mesh sensitivity as shown in **Table 1**. The best results registered by meshing of 825 300 tetrahedral cells for a receiver connected in series. Thus the meshing by 921 320 tetrahedral cells for a receiver connected in parallel.

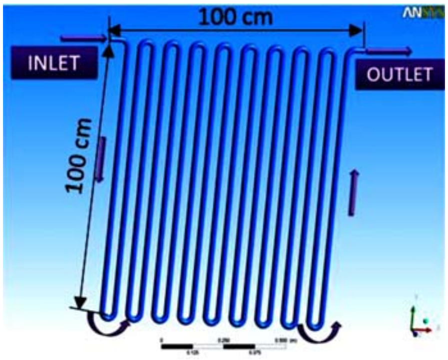


Fig. 2: Fluid flow receiver connected in series

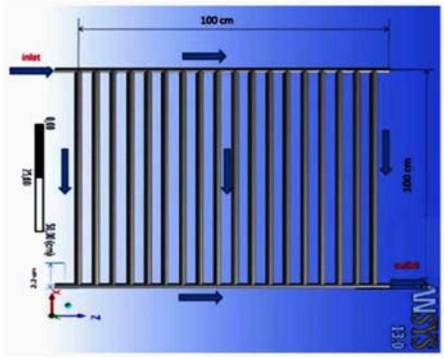


Fig. 3: Fluid flow receiver connected in parallel

Table 1: Meshing results for both receivers

Case	Number of nodes (in series)	Number of nodes (in parallel)
1	102 900	130 809
2	250 456	270 389
3	562 320	669 323
4	825 300	921 320
5	1 032 256	1 049 236

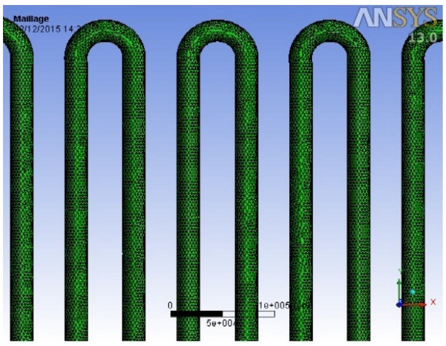


Fig. 4: Mesh of the receiver connected in series

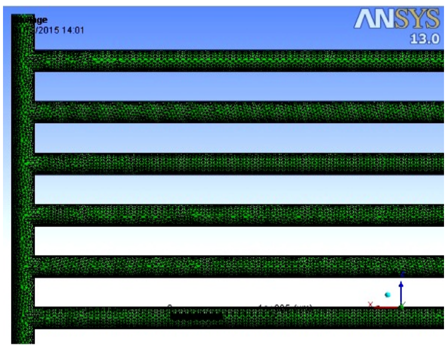


Fig. 5: Mesh of the receiver connected in parallel

3. MATHEMATICAL MODEL

Based on the fundamental equations of mass conservation, momentum, and energy. The steady state is expressed by:

Continuity

$$\frac{\partial \overline{u_l}}{\partial x_i} = 0 \tag{1}$$

Momentum

$$\rho \overline{u_j} \frac{\partial \overline{u_l}}{\partial x_i} = \frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u_l}}{\partial x_j} - \rho \overline{u_i^{'} u_j^{'}} \right) + \rho g_i \tag{2}$$

Energy

$$\rho \overline{u_j} \frac{\partial \overline{T}}{\partial x_j} = \frac{1}{C_p} \frac{\partial}{\partial x_j}\left(k \frac{\partial \overline{T}}{\partial x_j} - \rho C_p \overline{T^{'}} \overline{u_j^{'}} \right) \tag{3}$$

4. TURBULENCE

After the outlet of the pipe, the change in flow direction created the turbulence flow that have the significantly influence on heat transfer. As known, the turbulence modeling is so necessary in simulation works. In the case, the $k-\varepsilon$ model is used with introducing the wall function in aim to analysis the flow near wall. The boundary layer is not resolved by the mesh.

The implementation method of the $k-\varepsilon$ model in Ansys CFX 13 was done based on Launder *et al.* [80] works and Ansys documentation. In aim to analyze near the outled, the turbulent kinetic energy (k_t) and the dissipation of the turbulent kinetic energy (ε_t) are obtained from their transport equations.

Turbulent kinetic energy (k_t)

$$\rho \overline{u_j} \frac{\partial \overline{k_t}}{\partial x_t} = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_{kt}} \right) \frac{\partial \overline{k_t}}{\partial x_j} \right) + P_{kt} + G_{kt} + \rho \varepsilon_t \tag{4}$$

Dissipation of the turbulent kinetic energy (ε_t)

$$\begin{aligned} \rho u_j \frac{\partial \varepsilon_t}{\partial x_j} &= \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_{\varepsilon t}} \right) \mu \frac{\partial \overline{\varepsilon_t}}{\partial x_j} \right) \\ &\quad + c_{1\varepsilon_t} \frac{\varepsilon_t}{k}\left(P_{kt} + c_{3\varepsilon_t} G_{kt} \right) - c_{2\varepsilon t} \rho \frac{\varepsilon_t^2}{k_t} \end{aligned} \tag{5}$$

5. RESULTS AND DISCUSSION

In this study, the flux distribution is estimated by assuming that the solar rays have a normal incidence from punctual source. An energetic modeling have been established to determine the temperature distribution in all transversal pipe sections of the receivers.

In all cases, the heat flow equation is used for different temperature values between the inlet and outlet section, it given by:

$$Q = \frac{T^4 - T_{min}^4}{T_{max}^4 - T_{min}^4} \tag{6}$$

In addition, the obtained results of both solar receivers geometries (serial and parallel connected) was compared. The initial and limit conditions was token the in both study case.

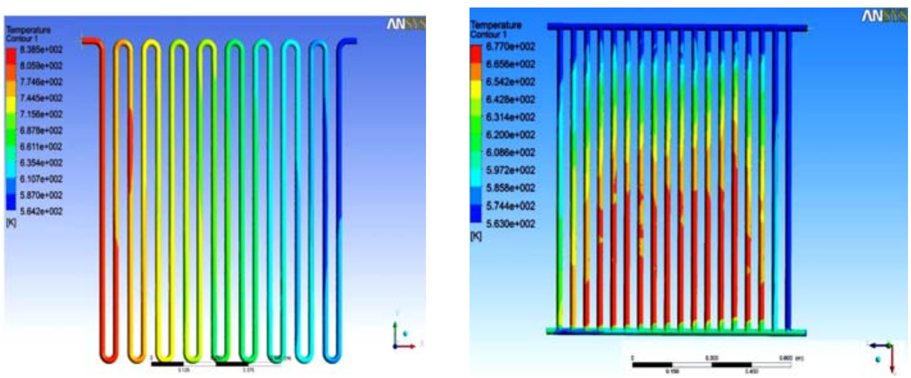
5.1 Distribution of temperature

In order to investigate the effect of flux on the solar receivers, numerical simulations were performed using the same configurations and the same fluid. The same initial and boundary conditions have been applied for both studied cases. An inlet temperature of 503 K and molten salts (HTF) with flow speeds of 0.5 m/s have investigated.

The obtained result are presented in figure 6 and 7. For these particular receivers design, it can be seen that the surface temperatures obtained for the case of the receiver connected in series are very high compared to that of the parallel connected receiver. In addition, the difference in temperature between the two receivers in 161 °C.

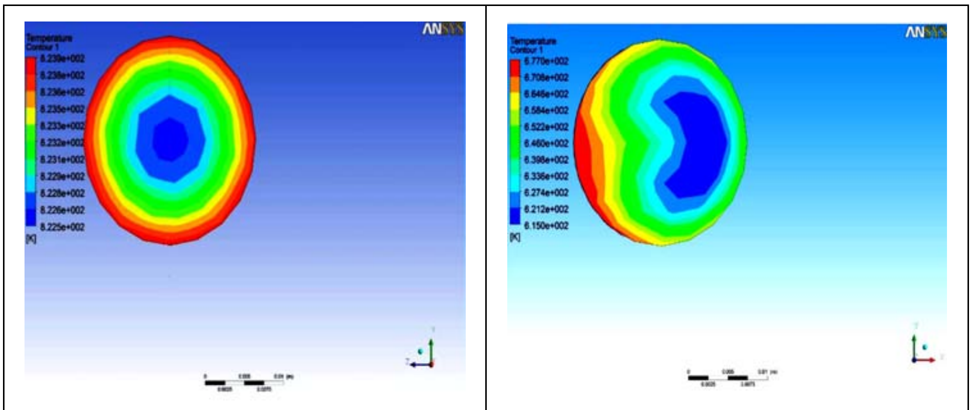
This difference can be justified by the fact that the circulation of the heat transfer fluid in the case of a receiver connected in series is very easy compared to that of the receiver connected in parallel because the flow does not find geometric obstacles created by the connection in parallel.

Furthermore, the figures (6) and (7) also shows that the temperature of solar radiation can reach 565°C as maximum value. Which represents the maximum temperature operation of the solar radiation incident on the receiver.



(**a**) solar receiver in series (**b**) solar receiver in parallel

Fig. 6: Temperature distribution on the tube surfaces of the two types receivers



(**a**) Solar receiver connected in series	(**b**) Solar receiver connected in parallel

Fig. 7: The temperature distribution at the output of the receiver

5.2 Influence of the speed of the heat transfer fluid on the thermal transfer

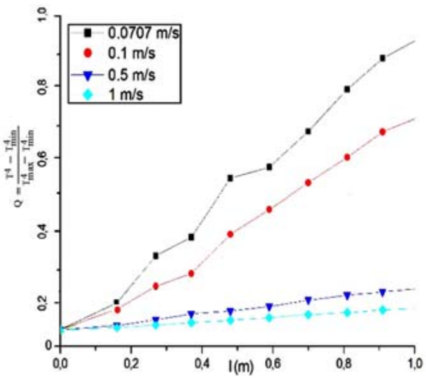


Fig. 8: Influence of the speed of the heat transfer fluid on the output temperature {Receiver connected in series}

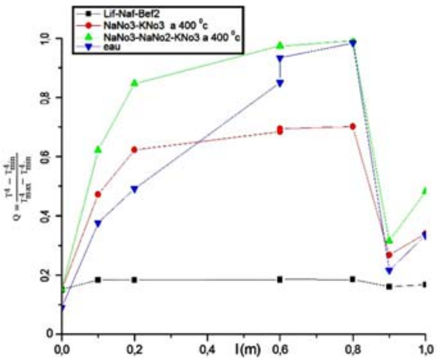


Fig. 9: Influence of the speed of the heat transfer fluid on the Temperature {receiver connected in parallel}

It should be noted that, in the solar tower, we are interested to ensure a higher temperature at the output of the receiver to allow the turbine rotation which gives mechanical energy to the generator. In this comparative study between the both receivers {connected in series and in parallel}, we were found that the output temperatures of the receiver connected in series are very high then that of the receiver connected in parallel, despite the change in speed of the heat transfer fluid between 0.0707 m/s and 1 m/s.

Figures 8 and 9 show respectively the Q evolution along the length of the both receivers connected in series and in parallel. It can be observed that the velocity of the heat transfer fluid clearly influences the efficiency of the receiver. Indeed, the temperature curve always reaches an elevation at the output of the receiver connected in series while, in the case of the receiver connected in parallel the temperature reaches its maximum values at the middle region and a low temperature at the output of the receiver.

5.3 Influence of the heat transfer fluid on the thermal transfer

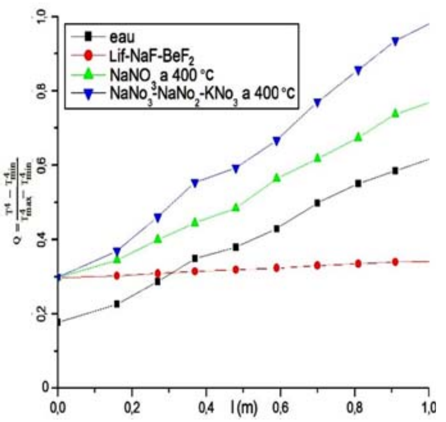


Fig. 10: Influence of the heat transfer fluid on the output temperature {receiver connected in series}

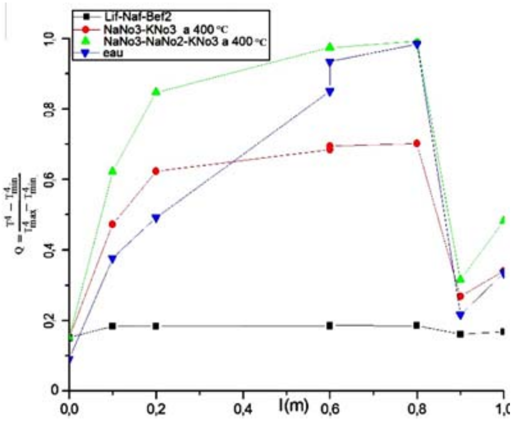


Fig. 11: Influence of the heat transfer fluid on the output temperature {receiver connected in parallel}

The used heat fluid influences directly on the efficiency of solar tower. That related by the thermal and chemical proprieties and his resistance at high temperatures values. The salts is produced by using the heat exchanger. The steam produced feeds a turbine coupled by an electric generator.

The uses of molten salts is related by their higher temperature as the major advantage. That allows to produce the steam under pressure and improve the global performance.

The consideration of the coolant effect on liquid, four types of fluids have been tested such us {$NaNO_3$-$NaNO_2$-KNO_3 / $NaNO_3$-KNO_3 / LiF-NaF-BeF_2 and water}. The figure (10) and (11) illustrates the obtained numerical results of serial and parallel connected receiver respectively. The best results is the results registred by using the heat fluid ($NaNO_3$-$NaNO_2$-KNO_3).

6. CONCLUSION

CFD simulation was carried out to investigate heat loss of central tower solar receiver. Eight (08) different cases covering two different shapes of receiver from fully external to full cavity type, four different fluid velocities (0.0707 m/s; 0.1 m/s; 0.5 m/s; and 1 m/s), were considered for the simulation and four types of fluid ($NaNO_3$ - $NaNO_2$ - KNO_3 / $NaNO_3$ - KNO_3 / LiF - NaF - BeF_2 and water).

The outcome of the simulation was then correlated to find a simplified model for predicting the heat loss. According to our bibliographic study on the work of researchers and academics on the field of the solar recipient, and according to the numerical results obtained from the calculation software Ansys CFX.

We note that the heat transfer fluid adequate to the transfer of heat within the receiver for the power plants is the molten salts of type ($NaNO_3$ - $NaNO_2$ - KNO_3) and the geometry which gives us a good yield and a high temperature at the output of the receiver is connected in series.

NOMENCLATURE

3D, Three-Dimensional	HTF, Heat transfer fluid
CFD, Computational Fluid Dynamics	SPT, Solar power tower
$k-\varepsilon$, Turbulence model	$\overline{T}$, The mean temperature, K
u_i', Instant fluctuation of x-velocity direction, m/s	G_{kt}, Generation of turbulent kinetic energy due to the buoyancy force
T', Fluctuation of temperature, K	$\sigma_{\varepsilon t}$ and σ_{kt}, Turbulent Prandtl number
$c_{3\varepsilon t}$ and $c_{2\varepsilon t}$, are coefficients	c_p, Specific heat, J/kg.K
P_{kt}, Generation of the turbulent kinetic energy	ε_t, Dissipation of the turbulent kinetic energy
ε, Dissipation of the kinetic energy	i and j, i_{th} and j_{th} elements
D, Height, cm	H, Diameter, cm
L, Length, cm	T, Temperature of fluid, K
u, Inlet velocity, m/s	g, Gravity, m/s
k, Kinetic energy	p, Pressure, Pa
μ, Viscosity, kg/m.s	t, Total
x, System coordinate	$(i=x,y,z - j=x,y,z)$

REFERENCES

[1] O. Behar, K. Khellaf, and K. Mohammedi, '*A Review of Studies on Central Receiver Solar Thermal Power Plants*', Renewable and Sustainable Energy Reviews, Vol. 23, pp. 12 - 39, 2013.

[2] Report IRENA, '*Concentrating Solar Power. Cost Analysis Series*', In: Renewable Energy Technologies, Power Sector, Vol. 1, N°2/5, 2012.

[3] C. Xu, Zh. Song, Le. Chen, Y. Zhen, '*Numerical Investigation on Porous Media Heat Transfer in a Solar Tower Receiver*', Renewable Energy, Vol. 36, N°3, pp. 1138 – 44, 2011.

[4] R.K. McMordie, '*Convection Heat-Loss From a Cavity Receiver*', Journal of Solar Energy Engineering-Transactions of the ASME, Vol. 106, N°1, pp. 98 - 100, 1984.

[5] J.B. Fang, J.J. Wei, X.W. Dong and Y.S. Wang, '*Thermal Performance Simulation of a Solar Cavity Receiver Under Windy Conditions*', Solar Energy, Vol. 85, N°1, pp. 126 – 138, 2011.

[6] A. Crocker and F.J. Miller, '*Coupled Fluid Flow and Radiation Modeling of a Cylindrical Small Particle Solar Receiver*', In: Proceedings of the ASME, 6th International Conference on Energy Sustainability, ES Fuel Cell 2012-91235, San Diego, CA, 2012.
<http://proceedings.asmedigitalcollection.asme.org/proceeding.aspx?articleid=1719575>.

[7] O. Garbrecht, F. Al-Sibai, R. Kneer and K. Wieghardt, '*CFD-Simulation of a New Receiver Design for a Molten Salt Solar Power Tower*', Solar Energy. Vol. 90, pp. 94 - 106, 2013.

[8] Scientific Report, '*ANSYS*', ANSYS, Inc. FLUENT 12.0 Theory Guide, 2009.

[9] N.Z. Ince and B.E. Launder, '*On the Computation of Buoyancy Driven Turbulent Flows in Rectangular Enclosures*', International Journal of Heat and Fluid Flow, Vol. 10, pp. 110 - 117, 1989.