Air Behavior Inside Duct of Air Solar Collector with Three Models of Baffles

Henaoui Mustapha[*], Aliane Khaled.

Departement of Mechanical ingineering, Laboratory MECACOMP, Tlemcen University
Telmcen, Algeria
[*]Corresponding author Email: mus196510@live.fr

Article Info

Article history:

Received 21 April 2020
Revised 25 May 2020
Accepted 07 June 2020

Keywords:

Solar collector
Finite volume method
K-Epsilon turbulent model
Baffles

ABSTRACT

The objective of this work is to study numerically the effect of the geometry of the baffles on the heat exchange in a solar air collector. Three models of collector were used in this study, fitted with simple baffles and perforated baffles. Fluid dynamics calculation (CFD) tool has been used to simulate the geometries of the solar collectors. Its three models involving air intake, are modeled by the FLUENT6.3 software and the grids were created with the Gambit software. The shape of the perforations is in the forms strips perforated in the baffles. The numerical resolution uses the finite volume method and the turbulence modeling K-Epsilon. The results have been validated by previous work and the simulation results are in terms of the evolution of the axial velocity and temperature distribution for the three models.

I. Introduction

Flow in channels with baffles occurs in many industrial applications such as heat exchangers, chemical reactors, filtration and desalination.

The introduction of obstacles into the flow path of solar air collectors was the subject of several experimental, theorical and numerical works which showed an improvement in the air outlet temperature of solar collectors and consequently of their performance. In [1], a numerical three-dimensional study of solar air collector with rectangular baffles and a comparison of the results with experimental data have been done. [2], study numerical and experimental investigations on turbulent flow inside a rectangular channel containing two rectangular baffles. They found that numerical results were in good agreement with those obtained by experiment. [3], examined the effect of the geometric parameters on the steady turbulent flow passing through a pipe with baffles. The effect of the orientation and the distance between nine baffles on the improvement of heat transfer was highlighted in this work Ackermann *et al.* [4], presented a computational study to examine the fully developed laminar flow and heat transfer characteristics in solar collector panels with internal, longitudinal, corrugated fins, the single pass with front duct, rear duct, double duct and double pass K. Aliane and al [5], study the numeric analysis of the turbulent flow inside a channel of rectangular section, with two types of obstacles in the two-dimensional case. The existence of the obstacle in a flow certainly causes a recirculation zone located upstream and downstream of the obstacle. These zones may represent pollution areas, where the pollution remains retained. [6], the impact of the inclined baffles on the flow structure of the near wall for different Reynolds numbers.

The objective of this study is to perform the flat plate collector with using computational fluid dynamics simulation to perform detailed fluid flow and heat transfer analysis. Hocine Moungar et al [7], presents a theoretical and experimental study of a double slope still with and without immersed fins compared to the single solar still monthly production. The influences of the distance between fins, fin heights, fin numbers and water layer thickness on the solar still production have been widely researched. Hocine Guellil et al [8], presents an experimental study of thermal performances of a latent heat thermal storage device made in laboratory. The obtained results showes that the storage unit with three exchangers stores 73 and 32% more thermal energy than a storage unit with one and two exchangers. At the end of the discharging duration, the first exchanger releases its total heat in Conf. 1, 2 and 3 respectively, after 126, 149 and 160 min. P. Dutta and al [9-10], have investigated the local heat transfer characteristics and the friction head loss in a rectangular channel with inclined solid and perforated baffles. Experimental results show that the local Nusselt number distribution is strongly depended on the position, orientation, and geometry of the baffles. The friction factor ratio goes up with an increase in the Reynolds number, but its value depends on the arrangement of baffles. A numerical validation of this work has been done by R. Saim and al [11]. The numerical results have showed that the inclined baffles improve the friction factor and the heat transfer.

This study focuses on the orientation of the flow by the perforation of strips in the baffles of the solar air collector. The objective consists in a two-dimensional numerical study on the hydrodynamic and thermal behavior of three models of solar collectors equipped with solid baffle and perforated baffle in the first case and perforated baffles in the second and the third case (strips perforated in the baffles).

II. Analysis Model

II.1. Geometric Modelling

The geometries of the two models of solar collector are illustrated in Fig.1 solar collector with solid baffles and perforated baffle, Fig.2. solar collector with perforated baffles (strips perforated in the baffles) and Fig. 3 solar collector with perforated baffles. The purpose of this work is to allow an orientation adequate of the coolant in order to maximize the heat transfer between the coolant (air) and the absorber.

Table 1. Dimensions of the models of solar collector

Distance (m)	*AB*	*CE*	*AC*	*CB*	*EF*	*Perforations diameter d*
Collector with solid baffles	*0.218*	*0.37*	*0.146*	*0.01*	*0.174*	*0.005*

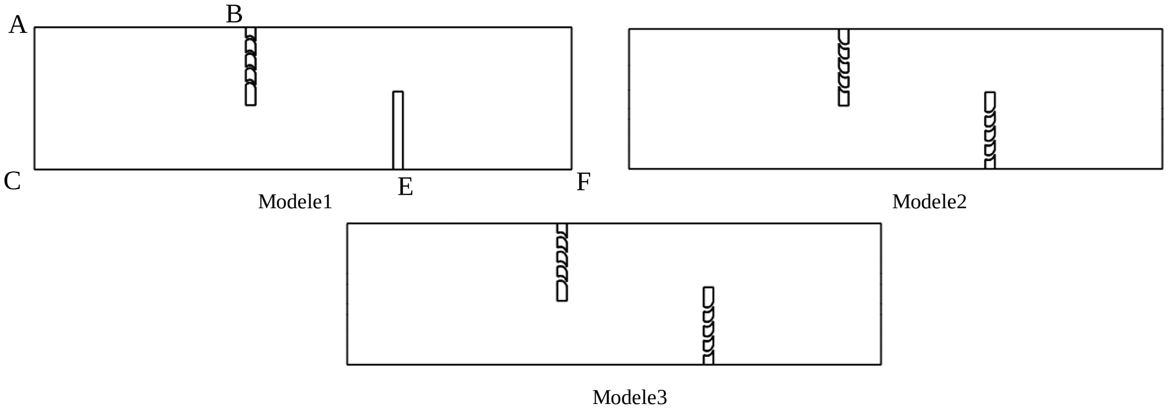

Figure 1. Physical domain for solar collector

II.2. Numerical Model

Fluid dynamics is described by partial differential equations, known as Navier-Stokes equations. For this reason, through the CFD (computational fluid dynamics) the problem is traced back to a numerical problem. In the context of CFD, the set of Navier-Stokes equations is transformed into a system of algebraic equations, to which is added the creation of the spatial domain calculation grid.

II.2.1. Mathematical Model

Airflow is supposed to be turbulent under steady-state condition, two-dimensional, Newtonian and incompressible.

The most general mathematical model of fluid dynamics consists of the principle of conservation of mass, the principle of conservation of momentum and the principle of conservation of energy. The equations obtained from these, in differential form:

1) **Equations**

Continuity Equation

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0 \tag{1}$$

Momentum Equation in x-direction

$$\frac{\partial}{\partial x}(\rho u^2)+\frac{\partial}{\partial y}(\rho uv)=-\frac{\partial P}{\partial x}+\frac{\partial}{\partial x}\left[(\mu+\mu_t)(\frac{\partial u}{\partial x})\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)(\frac{\partial u}{\partial y})\right] \tag{2}$$

Momentum Equation in y-direction

$$\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho v^2)=-\frac{\partial P}{\partial y}+\frac{\partial}{\partial x}\left[(\mu+\mu_t)(\frac{\partial v}{\partial x})\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)(\frac{\partial v}{\partial y})\right] \tag{3}$$

Energy equation

$$\frac{\partial}{\partial x}(\rho uT)+\frac{\partial}{\partial y}(\rho vT)=\frac{\partial}{\partial x}\left[(\mu+\frac{\mu_t}{\sigma_T})\frac{\partial T}{\partial x}\right]+\frac{\partial}{\partial y}\left[(\mu+\frac{\mu_t}{\sigma_T})\frac{\partial T}{\partial y}\right] \tag{4}$$

2) **Turbulence Model**

The k-ε model implies two-transport equations turbulent kinetic energy and the dissipation of turbulent kinetic [1, 8].

$$\frac{\partial}{\partial x}(\rho uk)+\frac{\partial}{\partial y}(\rho vk)=\frac{\partial}{\partial x}\left[(\mu_l+\frac{\mu_t}{\sigma_K})\frac{\partial k}{\partial x}\right]+\frac{\partial}{\partial y}\left[(\mu_l+\frac{\mu_t}{\sigma_K})\frac{\partial k}{\partial y}\right]-\rho\varepsilon+G_k \tag{5}$$

$$\frac{\partial}{\partial x}(\rho u\varepsilon)+\frac{\partial}{\partial y}(\rho v\varepsilon)=\frac{\partial}{\partial x}\left[(\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x}\right]+\frac{\partial}{\partial y}\left[(\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial y}\right]+C_1G_k\frac{\varepsilon}{k}+C_2\rho\frac{\varepsilon^2}{k} \tag{6}$$

$$G_k=\mu_t\left[2(\frac{\partial u}{\partial x})^2+2(\frac{\partial v}{\partial y})^2+(\frac{\partial v}{\partial y}+\frac{\partial u}{\partial x})^2\right] \qquad \mu_t=C_\mu\frac{\rho k^2}{\varepsilon}$$

The model coefficients are as follows:
C_μ=0.09, $C_{\varepsilon 1}$=1.44, $C_{\varepsilon 2}$=1.44, σ_k=1.0, σ_ε=1.3, σ_T=0.9

3) boundary conditions

The boundary conditions are as follows: Constant properties of the air.

- At the inlet of the channel: u_{in}=2.2 m/s, v= 0, k= 0.005u^2, ε= 0.1k,^{2} Te = 300 K,
- At the wall: $u = v$= 0,
- Temperature of absorber Tabs=420 K, Adiabatic condition of the insulator
- At the exit:$p_{out} = p_{atm}$ and all gradients are null

III. Results and Discussions

The results of in the flow simulations in a solar air collector with several models of baffles are presented below:

The axial velocity variation around the baffles is shown in Figures 2 show, with the observation of the influence of the baffle shape. Perforated baffles promote the appearance of secondary flows and reduce the undesirable recirculation zones.

The distribution of the axial velocity field for the third Model is most important while observing reduction in the dead zone downstream of the first baffle and second baffle consequently decrease in recirculation zones. We also notice for this model a good increase in the temperature distribution compared to the other two models Figure 3. The orientation of the perforations according to the third model led to an effective improvement of the heat exchange.

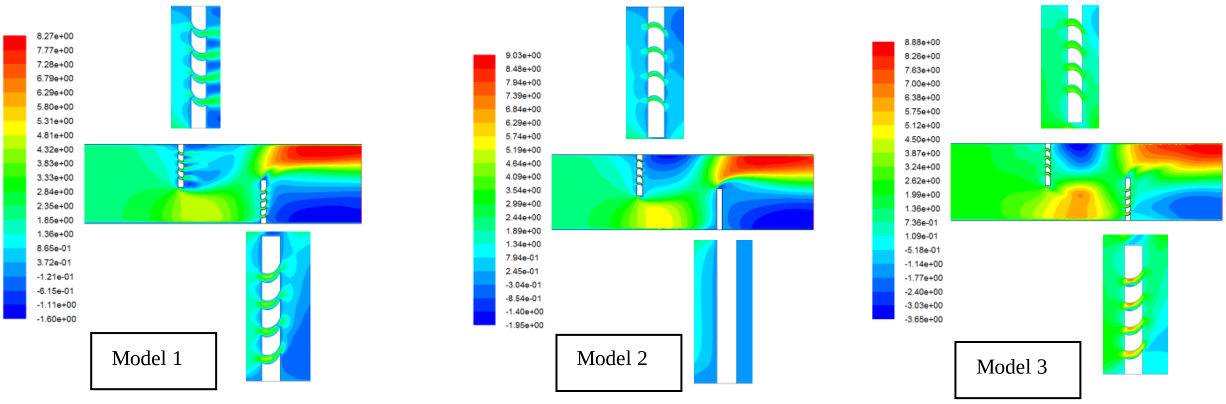

Figure 2. Contour of axial velocity

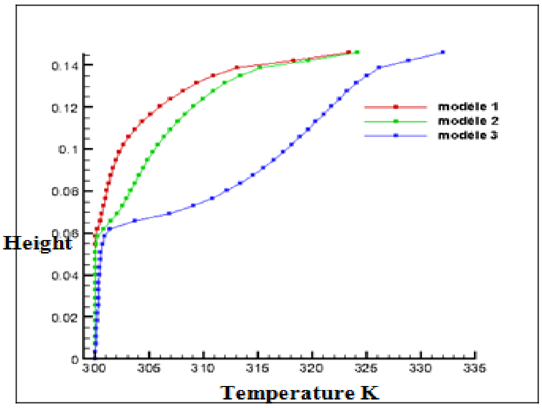

Figure 3. Profils of temperature distribution at the outlet x=0.554m

IV. Conclusion

A two dimensional CFD analysis has been carried out in order to study several geometries of solar air collector with solid baffles and perforated the baffles.

In our case studied, we used simple baffles and perforated baffles according to three models. The results showed a good increase in the air temperature at the outlet of the solar air collector for the third model of solar collector with perforated baffles. This increase in temperature is the consequence of an adequate distribution and orientation of the air flow towards the absorber. The axial speed distribution field is more developed for the perforated baffle solar collector with the appearance of secondary flows.

A considerable improvement in heat exchange was observed using the third model. The presence of perforated baffles and their geometries for the third model gives a higher heat exchange compared to the other two models.

ACKNOWLEDGEMENTS

My thanks for the director of laboratory MECACOMP Pr. Houmat Abderrahim and may reader Pr. Aliane Khaled for their support.

Nomenclature

C_μ, C_1, C_2	Emperical constant
d	Perforated strips diameter
Gk	Rate of Production
k	Kinetic turbulent energy (m^2/s^2)
P	Pressure (N/m^2)
P_{atm}	Atmospheric pressure (N/m^2)
P_{out}	Outlet pressure (N/m^2)
p’	Fluctuation pressure (N/m^2)
Pr	Prandtl number
T	Temperature (K)
T_e	Entry air temperature (K)
T_{abs}	Absorber temperature (K)
u_i	Velocity in x_i direction (m/s)
u_{in}	Inlet velocity (m/s)
U	Mean velocity (m/s)
u'	Fluctuation velocity (m/s)
σ_k, σ_ε, σ_T	Emperical constant
ε	Turbulent dissipation (m^2/s^3)
μ_t	Turbulent viscosity (kg/m.s)
μ	Dynamic viscosity (kg/m.s)
ρ	Air density (kg/m^3)

References

[1] Amraoui M.A. and al., Three-dimensional Analysis of Air Flow in a Flat Plate Solar Collector Periodica Polytechnica Mechanical Engineering, 62(2), (2018), pp. 126-135.

[2] Ackermann J. A, Ongand, L-E..Lau S. C, Conjugate Heat Transfer in Solar Collector Panels with Internal Longitudinal Corrugated Fins, *Forschung in Ingenieurwesen, 61* , 4, 1995, pp. 84-92.

[3] Demartini L.C., Vielmo H.A., Moller S.V., J. Braz. Soc. Mech. Sci. & Eng, XXVI, No. 2, 2004 pp.153-159.

[4] Ahmet Tandiroglu, Effect of flow geometry parameters on transient heat transfer for turbulent flow in a circular tube with baffle inserts, International Journal of Heat and Mass Transfer, 49, 2006, pp.1559–1567.

[5] Aliane K, Passive control of the turbulent flow over a surface-mounted rectangular block obstacle and a rounded rectangular obstacle, International Review of Mechanical Engineering (I.RE.M.E.), Vol. 5, n. 2,February 2011.

[6] Saim R., Bouchenafa R., Al-salem K., Abboudi S., Benzenine H., Hakan F., A computational work on turbulent flow and heat transfer in a channel fitted with inclined baffles, Journal Heat Mass Transfer Vol. 49, 2011, pp. 761-774.

[7] Moungar H, Azzi A, Sahli Y, Haida A. Comparative Study of Three Solar Desalination Units Based on Theoretical and Experimental Approach. Algerian Journal of Renewable Energy and Sustainable Development, 2019, 1(1),112-118. https://doi.org/10.46657/ajresd.2019.1.1.11

[8] Guellil H, Korti A. Experimental Achievement and Improvement of Latent Heat Energy Storage Unit. Algerian Journal of Renewable Energy and Sustainable Development, 2019, 1(2),182-190. https://doi.org/10.46657/ajresd.2019.1.2.7

[9] Prashanta Dutta, Akram Hossain,. Internal Cooling augmentation in rectangular channel using two inclined baffles. International Journal of Heat and Fluid Flow, 26, pp.223–232. 2005.

[10] P. Dutta, S. Dutta, Effect of Baffle Size, Perforation and Orientation on Internal Heat Transfer Enhancement, Int.J. Heat Mass Transfer, Vol. 41, n. 19, pp. 3005-3013, 1998.

[11] R. Saim, R. Bouchenafa, K. Al-salem, S. Abboudi, H. Benzenine, F. Hakan, A Computational Work on Turbulent Flow and Heat Transfer in a Channel fitted with Inclined Baffles. Journal Heat Mass Transfer, Vol. 49, pp.761-774, 2013.

How to cite this paper:

Henaoui M, Aliane K. Air Behavior Inside Duct of Air Solar Collector with Three Models of Baffles. Algerian Journal of Renewable Energy and Sustainable Development, 2020, 2(1),28-33. https://doi.org/10.46657/ajresd.2020.2.1.4