Iranian (Iranica) Journal of Energy & Environment
Journal Homepage: wwwijeenet
IJEE an official peer review journal of Babol Noshirvani University of Technology, ISSN:2079-2115

Research Note

Heat Exchange Optimization by Adding Baffles to Streaming Duct of Solar Air Collector

Z. Aouissi*[1,2], F. Chabane[1,2], M. S. Teguia[1,2], N. Belghar[1,2], N. Moummi[1,2], A. Brima[1,2]

[1] *Department of Mechanical Engineering, University of Biskra, Biskra, Algeria*
[2] *Laboratoire de Génie Mécanique (LGM), Faculty of Technology, University of Biskra 07000, Algeria*

P A P E R I N F O

Paper history:
Received 10 May 2022
Accepted in revised form 08 June 2022

Keywords:
ANSYS Fluent
Baffles
Computational fluid dynamics
Heat transfer
Solar air collector

A B S T R A C T

This numerical and experimental work aims to improve the heat transfer inside a solar thermal collector. By incorporating rectangular baffles in the middle of the distributed air passing channel at different angles of inclination (β= 90°, β= 180°, β= 180° and β= 90°). That is called the model H. These experiments were carried out in the Biskra region of Algeria in good natural conditions with an average solar radiation approximately constant I= 869 W/m^2 varying from 11:30 to 14:00. After the completion of the experimental investigation, a computational fluid dynamics (CFD) model was created that matches this experimental model with the same experimental boundary conditions. In the numerical study, ANSYS Fluent 18.1 was used to conduct simulations and compare the results of the thermal and hydraulic performance of the collector. It was concluded that the effectiveness of the CFD model, meaning that the theoretical and numerical data were very close to each other for all mass flow rates. As the mass flow increased the heat transfer process increased, while the absorber plate temperature inside the collector for experimental and numerical studies decreased. Addition of baffles increased heat transfer, due to the creation of turbulent flow that leads to crack the dead thermal layers near the absorber plate, which leads to an increase in heat transfer from the absorber plate to the air.

doi: *10.5829/ijee.2022.13.04.04*

NOMENCLATURE

Φ	Heat flux *(W)*	*abs*	Absorber
m	Mass flow rate (kg/s)	*In*	Inlet
D_h	Haydraulic diameter (m)	*Out*	Outlet
T	Temperature (K)	**Greek Symbols**	
C_p	Specific heat (J/kg.K)	*d*	Width of duct (m)
Subscripts		*s*	Distance between two baffles (m)
air	Air	*e*	Width of baffles (m)

INTRODUCTION

Solar energy is one of the renewable, environmentally friendly and easy-to-use energies that ensures future development. One of the most important means used to benefit from this energy, as it transfers energy from solar radiation to the air passing through its channel through the absorber plate, which is easy to install and simple to use. In the literature, there are many researchers who tried to improve these transformers, some of them conducted experimental investigations and others used numerically by adding a new gemetrical shape of baffles [1-4] and changed the form of the duct [5-8]. Among these works, we mention the study of Hu et al. [9] where prepared numerical and experimental work of the heat tranfer inside a channel divided into serval compartments, the pricipal characteristics where changed in this study is the geometrical dimension of the compartments. In the same

*Corresponding Author Email: *zouhair.aouissi@univ-biskra.dz*
(Z. Aouissi)

Please cite this article as: Z. Aouissi, F. Chabane, M. S. Teguia, N. Belghar, N. Moummi, A. Brima, 2022. Heat Exchange Optimization by Adding Baffles to Streaming Duct of Solar Air Collector, Iranian (Iranica) Journal of Energy and Environment, 13(4), pp. 349-353. Doi: 10.5829/ijee.2022.13.04.04

contex Akpinar and Koçyiğit [10] conducted an experimental investigation of the heat exchange inside solar air collectoe with baffles, then compared the results with smooth plate collector. This study showed the the thermal and hydrolic perfermence are related with the geometrical shape of baffles. Also, Wang et al. [11] made an experimental study where added the S shepe of baffles inside their collector, from the results the the effect of the baffles on the thermal perfermances is confirmed. On the other side double-pass solar air collector was studied by Wijeysundera and Tjioe [12].

In this study, many of the operating conditions were modified and they were compared with a single-channel solar thermal collector. Through its results, it confirmed the superiority of the double-pass solar collector compared to the single-channel solar collector. Based on the mentioned works, in this study, rectangular barriers were added perpendicular to the air inside the collector channel, and the heat transfer was studied by numerical and experimental methods.

MATERIAL AND METHODS

Experimental investigation

The experimental work was carried out in the Biskra region, specifically at the University of Biskra, near the Technological Hall with inclination angle β=38°.

The studied collector receiver consists of a single channel, to which were added rectangular baffles made of galvanic located in the middle of the channel at angles of inclination of β=90°, β=180°, β=180°and β=90° as depicted in Figures 1 and 2. Also, this collector is well thermally insulated with layers of polystyrene to ensure the correctness of the results reported in the literature [13]. The purpose of adding baffles is to create turbulent flow and increase heat transfer within the transformer; this was the target focused by Menasria et al. [14]. Geometric dimensions of the building is tabulated in Table 1.

The experiment was done on 01/12/2021, from 11:30 to 14:00 where the mass flow rate changed from 0.011 to 0.068 kg/s. Every quarter of an hour we measured absorber plate temperature, bottom plate temperature, solar radiation, inlet and outlet temperatures. All measuring devices were available in the laboratory, where five measuring points were placed on the absorbent plate and like them in parallel on the bottom plate of Figure 1.

CFD set-up

The same experimental studied model was created for numerical studies using ANSYS Fluent 18.1, the generated model is a two-dimensional model with the same dimensions as the experimental model and the baffles situated in it. The CFD model consists of three basic steps, which are:

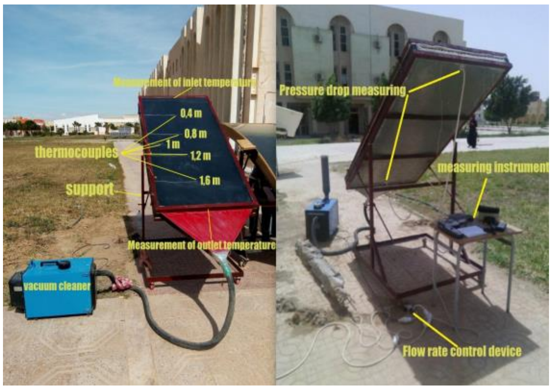

Figure 1. Experimental set-up

Figure 2. Position of baffles inside the channel

Table 1. Geometric dimensions

Building elements	Length (m)	Width (m)	Thickness (mm)
Transparent cover	1.94	0.94	3
Absorber	1.94	0.94	0.8
Wood frame	2	1	30×30
Wood sticks	1.94	0.03	30
Insulator	2	1	40
Baffles	0.88	0.02	0.8

- Design the model using Design Modeler Figure 3, where S= 10 cm, d= 4 cm, and e= 2 cm
- Meshing the model using the same software, the division is carried out into several nodes, which are controlled by the stability of the extracted results. The most accurate division was adopted at the walls to observe the effect of the walls on the thermal properties, where did the number of nodes reached to 22775 nodes as illustrated in Figure 4.
- After the division we pass to the introduction of the boundary conditions extracted from the experiment, the determination of the turbulence model (RNG k-ε), and the introduction of the physical properties of the materials. Through the studie conducted by Bensaci et al. [15] and Lee et al. [16] the boundary conditions were constant temperature on the absorber plate and bottom plate, velocity, turbulence intensity, temperature at the inlet, and hydraulic diameter.

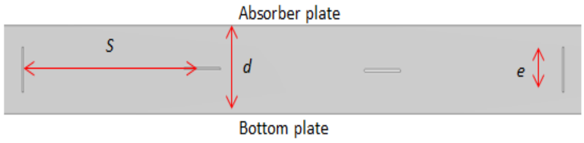

Figure 3. Numerical model design

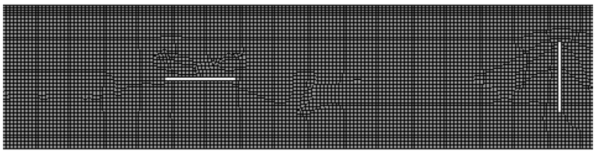

Figure 4. Meshing the domain

Modeling

In this study, many relations that have to do with hydraulic or thermal properties in both dimensional and non-dimensional forms were used.

Heat flux

$$d\Phi = m \times C_p \times dT_f \quad (1)$$

This relation becomes:

$$\Phi = m \times C_p \times (T_{out} - T_{in}) \quad (2)$$

The phenomenon that occurs inside the channel is a forced convection heat transfer, so that the heat transfer coefficient becomes one of the most important characteristics, which is given as follows:

$$h = \frac{\Phi}{(T_{abs} - T_{air})} \quad (3)$$

Hydraulic diameter

The hydraulic diameter depended on the cross-section and section perimeter

$$D_h = \frac{4 \times A}{P} \quad (4)$$

RESULTS AND DISCUSSION

In this manuscript, the focus was on the thermal properties of the transformer (heat transfer from the absorber plate to the passing air) [13], where the results were a detailed study of the heat of the absorber plate in its various parts. In addition to the temperature of the bottom plate, and the coefficient of heat transfer by convection and to take a closer look the heat fields were extracted and the velocity fields, to understand the phenomena inside the studied collector.

Figure 5 shows the temperature of the absorber plate in terms of its length at different mass flow rates. The first observation we notice is that all curves have the same changes, which is the rise in the temperature of the plate from the inlet to the outlet, and all the maximum values were recorded at the last point of the measurement at the length x= 1.50 m. The second observation we noticed was that as the mass flow increased, the average temperature of the absorber plate decreased; which indicates an increase in heat transfer from the device to the air. The effect of the baffels on the heat transfer inside the collector was evident on the temperature of the absorber plate, as in the middle region there is a decrease in the temperature of the absorber plate in the region between x= 0.752 m to x= 1.128 m, which is the area of the positioning of the four baffels, and this is due to the creation of turbulent flow in that area and the cracking of layers dead thermal near to the walls.

As for the temperature of the bottom plate Figure 6, it has the same trend of change curve with the absorber plate due to its gaining thermal resistance from the air by convection. The highest temperature was recorded at the position x= 1.5, and in the flows, the highest temperature was recorded in the lowest flow rate m= 0.011 kg/s.

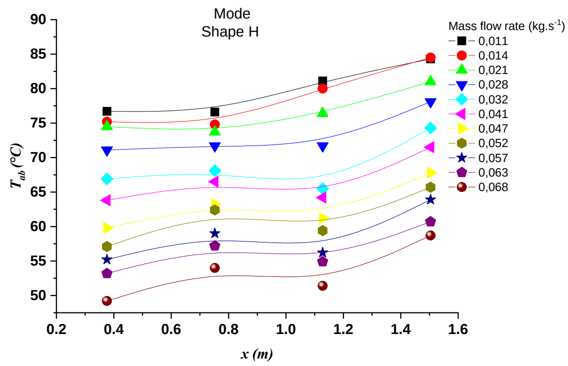

Figure 5. Temperature of the absorber plate in terms of its length at different mass flow rates

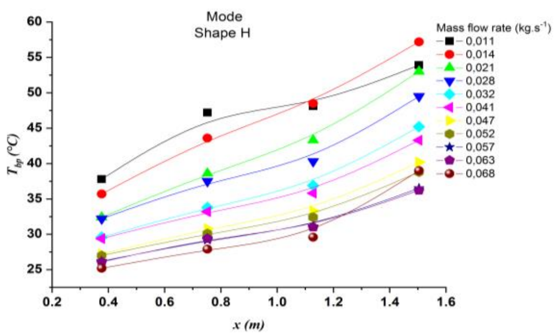

Figure 6. Temperature of the bottom plate in terms of its length at different mass flow rates

Figure 7 represents the change of the heat transfer coefficient in terms of mass flow in both the experimental and numerical investigations. Through this picture, we notice that there is a great convergence between the experimental results and the results of CFD, which leads to the validity of the designed model, as the greater the mass flow rate increases the heat transfer coefficient, it means, an increase in the heat transfer, and this confirms the heat absorber plate results. The highest value reached by the heat transfer coefficient was h= 32.14 W/m^2.K at mass flow rate of 0.068 kg/s, and that is in the experimental investigation. As for the CFD, at the same flow rate the heat transfer coefficient was h= 30.62 W/m^2.K .

We now pass to another result that is no less important than the above, which is the temperature difference between the inlet and outlet in terms of the mass flow by the two methods CFD analysis and experimental investigation as illustrated in Figure 8. This curve confirms the validity of the numerical model based on the convergence between the experimental and numerical values. Due to the decrease in the temperature of the

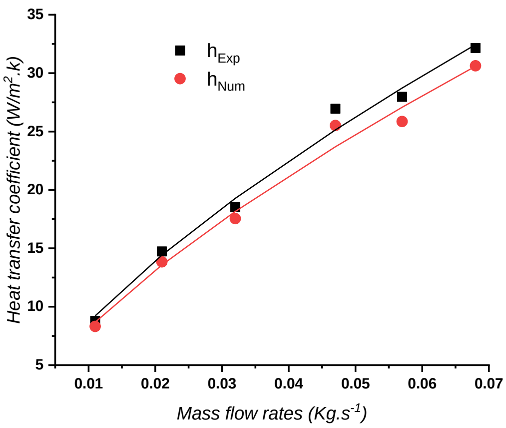

Figure 7. Heat transfer coefficient in terms of mass flow rates

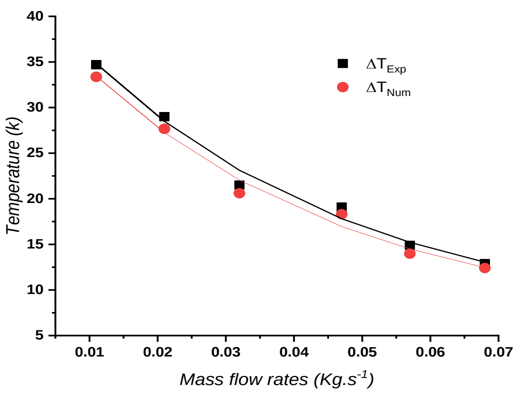

Figure 8. Temperature difference as a function of mass flow rates

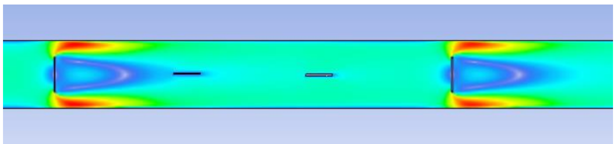

Figure 9. Velocity change field

absorber plate, the highest value of the temperature difference was ΔT= 34.7 °C at a mass flow rate of 0.011 kg/s in the experimental, and ΔT= 33.36 °C in the CFD method.

Figure 9 represents the velocity change field inside the channel and in the baffles area, where we see the effect of these baffles on the air velocity (the effect of baffles is an evident in the speed field change) [15, 16], especially the first baffle, as it is considered a stage of change in the air temperature due to the turbulence caused by these baffles.

CONCLUSION

The problem that was addressed in this study is the effect of baffles inside the channel of the solar thermal collector on the absorber plate and the heat transfer in general. Through the experimental and numerical results, it was found that the baffles have a great effect on the heat of the absorber plate and on the heat transfer, as their presence ensures an increase in the heat transfer and a decrease in the heat of the absorber plate due to the turbulence that it causes to the air, and the results also showed on the mass flow rate. It is inversely proportional to the heat of the absorber plate and directly to the heat transfer. The last conclusion of this study is to prove the effectiveness of the CFD model, through the convergence of its results with the experimental results, where we can rely on it to study other situations of baffles without resorting to the experimental investigation.

REFERENCES

1. Aouissi, Z., Chabane, F., Teguia, M.-S., Bensahal, D., Moummi, N. and Brima, A., 2021. Determination of the heat transfer coefficient by convection, according to shape of the baffles (solar air collector). Available at: www.easychair.org
2. Chabane, F., Moummi, N. and Benramache, S., 2013. Experimental analysis on thermal performance of a solar air collector with longitudinal fins in a region of Biskra, Algeria, *Journal of Power Technologies,* 93(1).
3. Moummi, N., Youcef-Ali, S., Moummi, A. and Desmons, J., 2004. Energy analysis of a solar air collector with rows of fins, *Renewable Energy,* 29(13), pp. 2053-2064. Doi:10.1016/j.renene.2003.11.006
4. Ozgen, F., Esen, M. and Esen, H., 2009. Experimental investigation of thermal performance of a double-flow solar air heater having aluminium cans, *Renewable Energy,* 34(11), pp. 2391-2398. Doi:10.1016/j.renene.2009.03.029

5. Khanlari, A., Güler, H. Ö., Tuncer, A. D., Şirin, C., Bilge, Y. C., Yılmaz, Y. and Güngör, A., 2020. Experimental and numerical study of the effect of integrating plus-shaped perforated baffles to solar air collector in drying application, *Renewable Energy*, 145, pp. 1677-1692. Doi:10.1016/j.renene.2019.07.076
6. Naphon, P. and Kongtragool, B., 2003. Theoretical study on heat transfer characteristics and performance of the flat-plate solar air heaters, *International Communications in Heat and Mass Transfer*, 30(8), pp. 1125-1136. Doi:10.1016/S0735-1933(03)00178-7
7. Sopian, K., Daud, W. R. W., Othman, M. Y. and Yatim, B., 1999. Thermal performance of the double-pass solar collector with and without porous media, *Renewable Energy*, 18(4), pp. 557-564. Doi:10.1016/S0960-1481(99)00007-5
8. Yeh, H.-M. and Ho, C.-D., 2013. Collector efficiency in downward-type internal-recycle solar air heaters with attached fins, *Energies*, 6(10), pp. 5130-5144. Doi:10.3390/en6105130
9. Hu, J., Liu, K., Guo, M., Zhang, G., Chu, Z. and Wang, M., 2019. Performance improvement of baffle-type solar air collector based on first chamber narrowing, *Renewable Energy*, 135, pp. 701-710. Doi:10.1016/j.renene.2018.12.049
10. Akpinar, E. K. and Koçyiğit, F., 2010. Energy and exergy analysis of a new flat-plate solar air heater having different obstacles on absorber plates, *Applied Energy*, 87(11), pp. 3438-3450. Doi:10.1016/j.apenergy.2010.05.017
11. Wang, D., Liu, J., Liu, Y., Wang, Y., Li, B. and Liu, J., 2020. Evaluation of the performance of an improved solar air heater with "S" shaped ribs with gap, *Solar Energy*, 195, pp. 89-101. Doi:10.1016/j.solener.2019.11.034
12. Wijeysundera, N., Ah, L. L. and Tjioe, L. E., 1982. Thermal performance study of two-pass solar air heaters, *Solar Energy*, 28(5), pp. 363-370. Doi:10.1016/0038-092X(82)90253-5
13. Chabane, F., Kherroubi, D., Arif, A., Moummi, N. and Brima, A., 2020. Influence of the rectangular baffle on heat transfer and pressure drop in the solar collector, *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, pp. 1-17. Doi:10.1080/15567036.2020.1767727
14. Menasria, F., Zedairia, M. and Moummi, A., 2017. Numerical study of thermohydraulic performance of solar air heater duct equipped with novel continuous rectangular baffles with high aspect ratio, *Energy*, 133, pp. 593-608. Doi:10.1016/j.energy.2017.05.002
15. Bensaci, C.-E., Moummi, A., de la Flor, F. J. S., Jara, E. A. R., Rincon-Casado, A. and Ruiz-Pardo, A., 2020. Numerical and experimental study of the heat transfer and hydraulic performance of solar air heaters with different baffle positions, *Renewable Energy*, 155, pp. 1231-1244. Doi:10.1016/j.renene.2020.04.017
16. Lee, C. and Abdel-Moneim, S., 2001. Computational analysis of heat transfer in turbulent flow past a horizontal surface with two-dimensional ribs, *International Communications in Heat and Mass Transfer*, 28(2), pp. 161-170. Doi:10.1016/S0735-1933(01)00223-8

Persian Abstract

چکیده

هدف از این پژوهش عددی و تجربی، بهبود انتقال حرارت در درون یک کلکتور گرمایی خورشیدی است. با ترکیب بافل‌های مستطیل شکل در وسط کانال، عبور هوای در زوایای شیب مختلف (°β=90، °β=180 ، °β=180 و °β =90) توزیع شده که این مدل H نامیده شده است. این آزمایش‌ها در منطقه بیسکرا واقع در کشور الجزایر و در شرایط طبیعی مناسب با میانگین تابش خورشیدی حدودا ثابت ۸۶۹ وات بر مترمربع، که طی ساعات ۱۱:۳۰ الی ۱۴:۰۰ متغیر است، انجام شد. پس از تکمیل تحقیقات تجربی، یک مدل دینامیک سیالات محاسباتی (CFD) ایجاد شد که این مدل تجربی را با همان شرایط مرزی تجربی مطابقت می‌دهد. در مطالعه عددی از نرم‌افزار ANSYS Fluent 18.1 برای انجام شبیه‌سازی و مقایسه نتایج عملکرد حرارتی و هیدرولیکی کلکتور استفاده شد. بر اساس نتایج حاصله نتیجه‌گیری شد که اثربخشی مدل CFD - به این معنی که داده‌های نظری و عددی برای همه دبی جریان جرمی به یکدیگر بسیار نزدیک هستند- با افزایش جریان جرمی، فرآیند انتقال حرارت افزایش یافت. در حالی که دمای صفحه جاذب در داخل کلکتور برای مطالعات تجربی و عددی کاهش یافت. افزودن بافل باعث افزایش انتقال حرارت می‌شود که به دلیل ایجاد جریان آشفته‌ای است که منجر به برهم‌خوردن لایه‌های حرارتی مرده در نزدیکی صفحه جاذب می‌شود. این امر منجر به افزایش انتقال حرارت از صفحه جاذب به هوا می‌شود.