Numerical investigations of fluid flow and heat transfer characteristics in solar air collector with curved perforated baffles

Emad M.S. El-Said

Faculty of Engineering, Mechanical Engineering Department, Fayoum University, Fayoum, Egypt

Correspondence
Emad M.S. El-Said, Faculty of Engineering, Mechanical Engineering Department, Fayoum University, Fayoum, Egypt.
Email: emadsaad@fayoum.edu.eg

Abstract
In this article, an improved double-pass solar air heater with multiple curved perforated baffles is proposed; the solar air heater can enhance the heat transfer between air and heat absorber plate sufficiently by using a perforated baffles. At the same time, to reduce the flow resistance of air, a curved shape was created on the baffles. This research is aimed at evaluating a novel solar air heater design in terms of its thermal efficiency and temperature distributions within the solar air heater in forced convection. This investigation is performed based on numerical simulations in two-dimensional, steady, incompressible Navier-Stokes and energy equations utilizing CFDRC software with the shear stress transport turbulence model. The main factors including the hole diameter, baffle inclination angle, and air mass flow rate were evaluated. The effects of these factors on the thermal efficiency of solar air heater and the temperature and pressure difference between the inlet and outlet were analyzed. The results obtained for the different cases show that the maximum thermal efficiency of the air heater is about 77% at air mass flow rate = 0.03 kg/s, hole diameter = 3 mm, and baffle angle = 7°. However, the work section pressure drop across this new solar air heater is varied between about 15.05 and 1.7 N/m². These numerical results are important for the application of this improved solar air heater and show that the curved perforated baffles in the air channel is a crucial factor for improving of collector performance.

KEYWORDS
curved baffle, heat transfer, performance improvement, solar air heater

1 INTRODUCTION
A solar air heater (SAH) is a straightforward heating system that converts solar energy into thermal energy. Thanks to their sustainability, portability, and ease of use and installation, SAHs have been applied to a variety of purposes, such as space heating, preheating for industrial applications, and drying of various materials.¹ SAHs are also commonly employed...
for preheating in saline water desalination in small systems.2–9 The radiation energy of the sun is absorbed by an absorber plate and then transferred to the air. This heated air can be used further, according to our requirement. The design augments and operating conditions optimization of SAH continues to concern many researchers.10–15 Passive enhancement technique such as turbulators, baffles, and fins had been widely used for heat transfer augmentation in different types of heat exchange systems.16 The passive technique objective was to generate the swirling flow, vortex flow, and impinging flow in the heating area which disturb the thermal boundary layer on the heat transfer zone that helps to increase both the efficiency and heat transfer rate. Turbulator is one of the passive heat transfer augmentation technique, which is used widely in the heat exchangers design.17 Hence, many studies presented valuable attempts utilizing these enhancement methods.18–28 Henaoui et al.29 presented a numerical study to optimize the performance of a solar air flat plate collector. In this study, the authors used a simple baffle and perforated baffle on the absorber and the insulation with dynamic and thermal stationary air flow. Menasria et al.30 studied numerically a turbulent flow and convective heat transfer of air inside channel of rectangular cross-section. This channel containing rectangular baffles with inclined upper part planted on the opposite surface of the absorber plate under SAH boundary conditions. They found that the baffle geometry and locations in addition to Reynolds number are the main controller of thermohydraulic flow fields. Sahel et al.31 used the numerical simulation approach to enhance the heat transfer, baffle design, and performance of a SAH channel. They concerned a perforated baffle having a row of four holes placed in three unique locations. Their results demonstrated that the pore axis ratio of 0.190 was the best design with increment in the heat transfer rate from 2% to 65% compared with the simple baffle. Alam and Kim32 studied the heat transfer and friction characteristics in a rectangular SAH flow duct with semieliptical shape barriers utilizing numerical simulation. They found that the angle of attack and barriers arrangement essentially affects the Nusselt number and friction factor. For all values of angle of attack, they showed that the staggered arrangement had an effect on the SAH performance more than the inline arrangement. Hamza and Gari33 presented a numerical simulation of the baffles with different height influence on heat transfer and pressure drop inside flow channel. They showed that the supporting tall baffles augment the Nusselt number by 3% by comparing with same average baffle heights. Basra et al.34 presented a theoretical model for a SAH with V-groove absorber and single- and double-pass flow thermal to investigate the efficiency and pressure loss through the flow channels. They studied the effect of the different air mass flow rate, flow channel height, and length on the SAH performance. They found that the porous media utilizing in double-pass flow increments the thermal efficiency to be 7% higher than in a single pass, and 2% to 3% higher than in a double-pass flow without porous media. Liu et al.35 studied a perforated rib with inclined holes arrangements in a cooling duct with rectangular cross-section. They found that the enhancement in overall averaged Nusselt number ratio was about 1.85% to 4.94%. Ibrahim et al.36 studied the performance of fins with and without perforated geometry under forced convection heat transfer. They found that the thermal performance of the perforated fins was superior over the nonperforated ones with a reduction in fin temperature up to 8.5°C. Also, their results showed that the perforated fins enhanced the hydraulic performance by reducing the friction factor and the pumping power. Chang et al.37 studied the development of heat transfer enhancement element utilizing the plate insert with periodical inclined baffles and perforated slots for augmenting the thermohydraulic performance of a heat exchanger. They found that the inclined baffles and perforated slots augmented the thermohydraulic efficiencies.

From the previous literature review in addition to the large numbers of works that have been conducted on augmenting the performance of the SAH, still works are required to be presented to improve the SAH performance especially augmenting the SAH designs. Enhancing the SAH design could be one of the best techniques that improves the SAH performance, especially if it is possible to apply also the other improvement techniques applying for the conventional SAHs. Hence, in the present study, a new design of SAH of novel absorber plate and its impact on the SAH performance are investigated numerically utilizing a three-dimensional Navier-Stokes software (CFDRC38) with modeling the turbulence effect, which was not presented before. The new designed SAH absorber consists of multiple curved perforated baffles with various hole diameter, which can enhance the heat transfer between air and heat absorber plate sufficiently. Also, a curved shape baffle with different inclination angles was created to reduce the pressure drop. The performance (thermal efficiency, outlet air temperature, and pressure drop) of the new SAH is studied. This study is carried out at different inlet air mass flow rates to the SAHs.

2 | NUMERICAL METHODOLOGY

The numerical study comprises the steps illustrated in the flowchart in Figure 1.


Figure 2 shows the schematic of the SAH computational domain studied in the present work. The SAH consists of a glass cover, thermal absorber, and back plate. The air is flowing in the lower channel between the absorber plate and the back plate and then goes to the upper channel between the absorber plate and the glass cover in the opposite direction. The absorber has been fitted with six curved perforated baffles of 1 mm thicknesses at an angle $\theta$ with the horizontal. The angle $\theta$ is adjusted to different values between 7° and 27°. The baffle is perforated, in a perpendicular direction with the baffle surface, with five holes with diameter varied from 1 to 3 mm. The holes diameter and angle $\theta$ are set according to the test case. Table 1 lists the values of the dimensional variables. The baffle width is unity for 2D simulation.

2.2 | Numerical modeling approaches and grid generation

The flow field in the SAH channels is numerically modeled utilizing the three-dimensional CFD software CFDRC-2008.38 The governing equations are discretized on an unstructured grid utilizing an upwind difference method. For different conditions: temperature, velocity, and pressure variation are computed.

2.3 | Governing equation

According to the main equations of the mass, momentum, and energy conservations can be expressed as follows39: Mass conservation

$$\nabla \cdot (\rho \vec{V}) = 0. \quad (1)$$
Momentum conservation
\[ \nabla [\bar{V} \cdot (\rho \bar{V})] = -\nabla p - \frac{2}{3} \nabla [\mu (\nabla \cdot \bar{V})] + \nabla \cdot [\mu (\nabla \cdot \bar{V})^T] + \nabla \cdot [\mu (\nabla \cdot \bar{V})]. \] (2)

Energy conservation.
\[ \rho c_p \bar{V} \cdot \nabla T = \nabla \cdot [k (\nabla T)] + \left[ \frac{\partial p}{\partial t} + \bar{V} \cdot \nabla p \right] + \phi. \] (3)

### 2.4 Turbulence model

In the present work, the turbulence model utilized in all the cases is one of the variants of the well-known turbulence model available within CFDRC, known as the shear stress transport (SST) model, which employ averaged governing equations to solve for the mean flow quantities and provide the distribution of the turbulent viscosity that is needed in the approximations of the Equation (2). The predictions utilizing this model were validated and compared with the experimental data by many researchers\(^{40-42}\) and found that the SST model provided the most appropriate prediction of behavior of similar cases such as film cooling. The SST model is a two equation model presented by Menter.\(^{43}\) It is a modification of Wilcox\(^{44}\) \(k-\omega\) model. This model utilizes a blending function that allows switching between the \(k-\omega\) model in the sub and log layer and the \(k-\epsilon\) model in the outer region of the boundary layer and in free-shear flows.\(^{35}\) The complete formulation of the SST model is given as follows\(^{40}\):

\[ \frac{\partial }{\partial t} (\rho k) + \frac{\partial }{\partial x_i} (\rho u_i k) = \bar{P}_k - \beta^* \rho k \omega + \frac{\partial }{\partial x_i} \left[ (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right]. \] (4)

\[ \frac{\partial }{\partial t} (\rho \omega) + \frac{\partial }{\partial x_i} (\rho u_i \omega) = \alpha \frac{1}{\nu_i} \bar{P}_k - \beta \rho \omega^2 + \frac{\partial }{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] + 2(1 - F_1) \sigma_\omega \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}. \] (5)

\[ \nu_i = \frac{\alpha_1 k}{\max(\alpha_1 \omega, \Omega F_2)}; \quad S = \sqrt{2S_i S_i}. \]
where

\[ P_k = \mu_t \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]

\[ \tilde{P}_k = \min(P_k, 20 \cdot \beta^* \rho k \omega), \]

\[
F_1 = \tanh \left\{ \min \left( \frac{\max \left( \frac{\sqrt{k}}{\beta^* \omega y}, \frac{500}{\omega y^2} \right)}{4 \sigma_{K2} k} \right) \right\}^4,
\]

\[
F_2 = \tanh \left[ \max \left( \frac{2 \sqrt{k}}{\beta^* \omega y}, \frac{500}{\omega y^2} \right) \right]^2,
\]

\[
C D_{k\omega} = \max \left( 2 \rho \sigma_{o2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right),
\]

where \( k \) is the turbulence kinetic energy, \( \omega \) is the turbulence frequency, \( y \) is the distance to the nearest wall, \( \Omega \) is the absolute value of the vorticity, \( \rho \) is the density, and \( u_i \) is the flow velocity. \( F_1 \) and \( F_2 \) are blending functions that are equal to zero away from the surface (k-\( \epsilon \) model) and switch over to one inside the boundary layer (k-\( \omega \) model). It should be noted that a production limiter is used in the SST model to prevent the buildup of turbulence in stagnation regions as an essential part of the SST model.

All constants are computed by a blend from the corresponding constants of the k-\( \epsilon \) and the k-\( \omega \) model via \( \alpha = \alpha_1 F_1 + \alpha_2 (1 - F_1) \), and so on. The constants for this model are:

\[ \beta^* = 0.09, \quad \alpha_1 = 5/9, \quad \beta_1 = 3/40, \]

\[ \sigma_{k1} = 0.85, \]

\[ \sigma_{o1} = 0.5, \quad \alpha_2 = 0.44, \quad \beta_2 = 0.0828, \quad \sigma_{k2} = 1, \quad \sigma_{o2} = 0.856. \]

Additional details of the SST model and the parameters can be found in CFDRC-2008 users’ manual and Menter.

### 2.5 Boundary, initial, and volume conditions

A fully developed air flow at constant temperature \( T_i = 300 \text{ K} \) enters the lower channel from the right with constant mass flow rate (0.07, 0.05 and 0.03 kg/s) at the inlet for each simulation case. Constant heat flux \( q_w = 1000 \text{ W/m}^2 \) equivalent to net solar radiation is applied to the top surface of the collector, and all other sides are assumed to be fully isolated (adiabatic). Set fixed pressure boundary conditions at the outlet. All numerical model boundaries have an emissivity, \( \varepsilon = 1 \) aside from the top glass, face, and bottom. The air properties are varied values; density, thermal conductivity, W/m·K, and dynamic viscosity, which is a function of temperature is used for air as:

\[
\rho_a = 3.9147 - 0.016082 T + 2.9013 \times 10^{-5} T^2 - 1.9407 \times 10^{-8} T^3,
\]

\[
k_a = (0.0015215 + 0.097459 T - 3.3322 \times 10^{-5} T^2) \times 10^{-3},
\]

\[
\mu_a = (1.6157 + 0.06523 T - 3.0297 \times 10^{-5} T^2) \times 10^{-6}.
\]

The air is assumed to be transparent to radiation; transmissivity = 1. Also, the specific heat of air is constant and equal 1007 J/kg·K. The same was done for the absorber plate and baffles according to Table 2, which shows all these properties.

The radiation effects on the domain side and bottom walls were neglected; emissivity equals zero.
| Parameter                      | Working fluid | Absorber plate and baffles |
|-------------------------------|---------------|---------------------------|
| Type                          | Air           | Copper                    |
| Density, $\rho$ (kg/m$^3$)    | See Equation (1) | 8960                     |
| Specific heat, $c_p$ (J/kg-K) | 1007          | 385                       |
| Thermal conductivity (W/m-K)  | See Equation (2) | 385                     |
| Viscosity, $\mu$ (kg/m-s)     | See Equation (3) | —                        |
| Transmissivity                | 1             | 0                         |
| Emissivity                    | —             | 1                         |

**TABLE 2** Volume condition setting of properties of blocks

Initial and boundary values for the turbulence quantities are critical for the accurate turbulent flow numerical modeling. The values of $k$ and $\omega$ at the inlet are determined utilizing the inlet conditions from the following equations:\textsuperscript{46}

$$k = \frac{1}{2}(u'^2 + v'^2 + w'^2),$$

(11)

where $u'$ is the turbulent fluctuation velocity, assuming $u'$, $v'$, and $w'$ are all equal to the turbulence intensity ($I$) multiplied by the main flow inlet velocity, we can calculate the inlet turbulent kinetic energy as follow:

$$k = \frac{3}{2}(u_m \cdot I),$$

(12)

$$\omega = \frac{k^{0.5}}{\beta L},$$

(13)

where $L$ is the turbulent characteristic length scale. For current study, we will choose $L$ as the channel with ($W$).

The turbulent intensity ($I$) for each case can be calculated as follows:\textsuperscript{47}

$$I = 0.16(Re)^{-0.125}.$$  

(14)

The assumptions considered for the simulation are: Air is used as a working fluid; it is compressible fluid. All surfaces considered in geometry are smooth air flow over it is frictionless. Ambient temperature is considered constant. Thermophysical properties of absorber plate and baffle metal were constant.

In spite of the fact that the physical domain is a three-dimensional model, in the current work it can be assumed to a two-dimensional one and the impact of side walls can be dismissed as the channel width to height ratio equal 7.77. Chaube et al\textsuperscript{48} presented a two-dimensional simulation of flow in duct with ribs and contrasted the results with the experimental work of Karwa,\textsuperscript{49} where the channel width to height ratio was 7.5. Also, they studied two- and three-dimensional numerical models of the experimental work of Tanda,\textsuperscript{50} where the channel width to height ratio was 5. Contrasting the two- and three-dimensional numerical models, they showed that the two-dimensional results were agreed with the experimental ones.

### 2.6 Grid generation

Commercial software (CFDRC) was utilized to generate the computational domain mesh as presented in Figure 3 with an unstructured mesh. Mesh size is not uniform, being clustered near the holes due to the high-velocity and pressure variations in that region.

### 2.7 Grid independence test

A grid independence test was performed for the computational model to ensure that the grid size has no effect on the results accuracy. Computations were performed utilizing four different grid sizes: 65 321 to 87 957 grid nodes. Figure 4 shows the outlet temperature with different grid nodes for hole diameter ($D_h$) = 1 mm, inclination angle ($\theta$) = 17°, inlet
mass flow rate ($\dot{m}_a = 0.07 \text{ kg/s}$) and heat flux ($q_w = 1000 \text{ W/m}^2$). As can be seen the outlet temperature variation in results obtained from utilizing 83631 to 87957 node grid size is little value. Hence, in this work, a grid size of 83631 nodes is utilized without accuracy compromising or/and taking more time consuming of the computation.

### 2.8 Run time for a simulation

The run time for a simulation to calculate the development of fluid motion and heat exchanging during 1 hour on DELL OPTIPLEX 780 with Intel (R) core (TM) 2 Due CPU: 2.93 GHZ, RAM: 4 GB. The number of iterations was 1000.
3 | PERFORMANCE PARAMETERS

3.1 | Solar collector efficiency

Variations of solar water heater efficiency ($\eta$) are calculated according to the following equation:

$$\eta = \frac{\dot{m}_a C_p(T_o - T_i)}{SA}.$$  (15)
FIGURE 9  Thermal efficiency various baffle angle (θ) with different hole diameter (Dh) for \( \dot{m}_a = 0.03 \) kg/s

FIGURE 10  Thermal efficiency various baffle angle (θ) with different hole diameter (Dh) for \( \dot{m}_a = 0.05 \) kg/s

FIGURE 11  Thermal efficiency various baffle angle (θ) with different hole diameter (Dh) for \( \dot{m}_a = 0.07 \) kg/s

4 | RESULTS AND DISCUSSIONS

4.1 | Model results validation

To verify our numerical simulation, a comparison between the present numerical methodology with those reported by Demartini et al\(^5\) was shown. Figure 5 shows the velocity variation that presented by Demartini et al\(^5\) and the calculated values of flow through a channel with baffle plates. As observed, a good agreement has been obtained with calculated
values from numerical simulation at the same conditions. Also, an experimental investigation of the current model now in progress approves the assumptions and predictions.

### 4.2 Outlet air temperature

Figures 6-8 show the variation of the outlet-air temperature with a baffle inclination angle with different diameter of holes for air mass flow rate equal 0.03, 0.05, and 0.07 kg/s, respectively. It can be clearly seen from the figures that at a constant inlet-air temperature, the increase of holes diameter led to the outlet-air temperature increase, which also decrease with baffle inclination angle increase. These results may be due to the effect of turbulence flow behind the baffles, which affect by holes diameter and angle of baffles. For different air mass flow rate, the outlet-air temperature increases with increase of air mass flow rate.

### 4.3 Collector efficiency

Figures 9-11 show the variation of the thermal efficiency with a baffle inclination angle with different diameter of holes for air mass flow rate equal 0.03, 0.05, and 0.07 kg/s, respectively. The thermal efficiency used to evaluate the performance of the SAH is calculated according to Equation (15). It is found from figures that the thermal efficiency increases with increasing of hole diameter and air mass flow rate and decrease with the increase of baffle angle. This is because the
heat transfer rate is directly proportional to the air mass flow rate. At a given air mass flow rate, the decrease of the baffle angle causing an increase of projected heat transfer surface area results in the increase of heat transfer rate. The maximum thermal efficiency of the air heater about 77% for air mass flow rate $\dot{m}_a = 0.03$ kg/s, hole diameter $D_h = 3$ mm, and baffle angle $\theta = 7^\circ$. From the figures, all the cases have the same trend of efficiency variations with baffle angle and hole diameters. The enhancement ratio with increasing of perforated hole diameter is approximately 12.02% to 20.62% and 14.77% to 18.53% and 17.19% to 31.33% for air mass flow rate 0.03, 0.05, and 0.07 kg/s, respectively. Hence, with the hole diameter becoming large the heat transfer process in the region downstream behind the baffle is further increased due to flow recirculation, which considered the main advantage of the perforated baffles. By contrast, the inclined angle increment usually have negative in heat transfer along the inclined direction.

### 4.4 Pressure drop

Figures 12-14 show the pressure drop variations with a baffle inclination angle with different diameter of holes for air mass flow rate equal 0.03, 0.05, and 0.07 kg/s, respectively. By raising the value of the flow rate through the flow duct, the pressure drop raised as well. Also, pressure drop increase with decrease of hole diameter and increase of baffle angle. The pressure drop is across air heater varied between about 15.05 N/m² for air mass flow rate $\dot{m}_a = 0.07$ kg/s, hole diameter $D_h = 1$ mm, and baffle angle $\theta = 27^\circ$ and 1.7 N/m² for air mass flow rate $\dot{m}_a = 0.03$ kg/s, hole diameter $D_h = 3$ mm, and baffle angle $\theta = 7^\circ$.

### 4.5 Velocity, temperature, and pressure variation

Figure 15 displays the computational contours for velocity, temperature, and pressure variations in the flow ducts. The showed velocity contours presenting a robust change of the values along the fluid flow as shown in Figure 15A. Figure 15B displays the computational contours for temperature variation. The showed temperature contours explain the effect of design and operating conditions on outlet temperature. Figure 15C displays the computational contours for pressure variation. The showed pressure contours are giving the chance for investigation of pressure drop around between air heater inlet and outlet.
5 | CONCLUSIONS

The current study presents a novel development of double passes with curved perforated baffles in forced convection utilizing numerical simulation of two-dimensional, steady, incompressible Navier-Stokes and energy equations utilizing CFDRC software. The present work combines the enhanced effect of perforated baffles and curved baffles aiming to improve thermal performance. For different test conditions the outlet temperature, thermal efficiency, velocity, and pressure variation has been numerically computed. Moreover, we suggest future experiments and three-dimensional simulations for the current proposed SAH model. The main conclusions are summarized:

1. The new development that presented in the current study is suitable for building heating applications.
2. The current numerical model can predict the outlet temperature and pressure drop, so it is considered a good approximation of the experimental data.
3. The pressure drop across air heater was approximately affected by air mass flow rate, hole diameter, and baffle inclination angle.
4. The pressure drop is across air heater varied between about 15.05 N/m² for air mass flow rate = 0.07 kg/s, hole diameter = 1 mm, and baffle angle = 27° and 1.7 N/m² for air mass flow rate = 0.03 kg/s, hole diameter = 3 mm, and baffle angle = 7°.
5. The maximum thermal efficiency of the air heater about 77% for air mass flow rate = 0.03 kg/s, hole diameter = 3 mm, and baffle angle = 7°.

CONFLICT OF INTEREST
The author declares no conflicts of interest.
NOMENCLATURE

LATIN SYMBOLS

\( T \) \hspace{1em} \text{temperature, K}
\( p \) \hspace{1em} \text{pressure, N/m}^2
\( D \) \hspace{1em} \text{diameter, m}
\( u, v, w \) \hspace{1em} \text{velocity in} \ x, y, z, \text{respectively, m/s}
\( W \) \hspace{1em} \text{width, m}
\( L \) \hspace{1em} \text{length, m}
\( m \) \hspace{1em} \text{mass flow rate, kg/s}
\( A \) \hspace{1em} \text{collector surface area, m}^2
\( S \) \hspace{1em} \text{solar heat flux, W/m}^2
\( c_p \) \hspace{1em} \text{specific heat, J/kg-K}
\( g \) \hspace{1em} \text{gravitational constant, m}^2/s

GREEK LETTERS

\( \rho \) \hspace{1em} \text{density, kg/m}^3
\( \Delta \) \hspace{1em} \text{change or difference}
\( \theta \) \hspace{1em} \text{baffle inclination angle, degree}
\( \eta \) \hspace{1em} \text{efficiency, dimensionless}
\( \delta \) \hspace{1em} \text{thickness, mm}
\( \varepsilon \) \hspace{1em} \text{turbulent dissipation rate, m}^2/s^3 \text{ or emissivity}
\( \mu \) \hspace{1em} \text{Dynamic viscosity, kg/m-s}

SUBSCRIPTS

uc \hspace{1em} \text{upper channel}
lc \hspace{1em} \text{lower channel}
ab \hspace{1em} \text{absorber plate}
c \hspace{1em} \text{channel}
i \hspace{1em} \text{inlet}
o \hspace{1em} \text{outlet}
ro \hspace{1em} \text{turn opening}
bf \hspace{1em} \text{baffle}
h \hspace{1em} \text{hole}
a \hspace{1em} \text{air}

ORCID

Emad M.S. El-Said \( \text{https://orcid.org/0000-0002-4247-5763} \)

REFERENCES

1. Potgieter MSW, Bester CR, Bhamjee M. Experimental and CFD investigation of a hybrid solar air heater. Solar Energy. 2020;195:413-428.
2. Kabeel AE, El-Said EMS. Technological aspects of advancement in low-capacity solar thermal desalination units. Int J Sustain Energy. 2013;32(5):315-332.
3. Kabeel AE, El-Said EMS. Development strategies and solar thermal energy utilization for water desalination systems in remote regions: a review. Desalin Water Treat. 2014;52(22-24):4053-4070.
4. Giwa A, Akther N, Al-Housani A, HarisShadi S, Hasan W. Recent advances in humidification dehumidification (HDH) desalination processes: improved designs and productivity. Renew Sust Energ Rev. 2016;57:929-944.
5. Shalaby SM, Bek MA, Kabeel AE. Design recommendations for humidification-dehumidification solar water desalination systems. Energy Proc. 2017;107:270-274.
6. Kabeel AE, El-Said EMS. A hybrid solar desalination system of air humidification dehumidification and water flashing evaporation: a comparison among different configurations. Desalination. 2013;330:79-89.
7. Poblete R, Salihoglu NK, Salihoglu G. Optimization of the solar brine evaporation process: introduction of a solar air heater. Env Prog Sustain Energy. 2019;38(3):1-9.
8. El-Said EMS, Kabeel AE, Abdulaziz M. Theoretical study on hybrid desalination system coupled with nano-fluid solar heater for arid states. Desalination. 2016;386:84-98.
9. Pfister H, Ralston T, Kim SW. A novel gridded solar air heater and an investigation of its conversion efficiency. Solar Energy. 2016;136:560-570.
10. Singh I, Singh S. A review of artificial roughness geometries employed in solar air heaters. *Renew Sust Energy Rev.* 2018;92:405-425.
11. Bisht VS, Patil AK, Gupta A. Review and performance evaluation of roughened solar air heaters. *Renew Sust Energy Rev.* 2018;81(1):954-977.
12. Alam T, Kim M. A critical review on artificial roughness provided in rectangular solar air heater duct. *Renew Sust Energy Rev.* 2017;69:387-400.
13. Hu J, Zhang G. Performance improvement of solar air collector based on airflow reorganization: a review. *Appl Therm Eng.* 2019;155:592-611.
14. Evangelisti L, Vollaro RD, Asdrubali F. Latest advances on solar thermal collectors: a comprehensive review. *Renew Sust Energy Rev.* 2019;114:109318.
15. Fadholi A, Sopian K. A review of solar air flat plate collector for drying application. *Renew Sust Energy Rev.* 2019;102:333-345.
16. Nanan K, Thianpong M, Pimsarn V, Chuwattanakul S. Eiamsa-ard, flow and thermal mechanisms in a heat exchanger tube inserted with twisted cross-baffle turbulators. *Appl Therm Eng.* 2017;114:130-147.
17. Fawaz HE, Badawy MTS, Abd Rabbo MF. AmrElfeky, numerical investigation of fully developed periodic turbulent flow in a square channel fitted with 45° in-line V-baffle turbulators pointing upstream. *Alex Eng J.* 2018;57(2):633-642.
18. Sheikholeslami M, Jafaryar M, Shafee A, Li Z, Haq R. Heat transfer of nanoparticles employing innovative turbulator considering entropy generation. *Int J Heat Mass Transf.* 2019;136:1233-1240.
19. Sheikholeslami M, Jafaryar M, Hedayat M, et al. Heat transfer and turbulent simulation of nanomaterial due to compound turbulator including irreversibility analysis. *Int J Heat Mass Transf.* 2019;137:1290-1300.
20. Sheikholeslami M, Jafaryar M, Shafee A, Li Z. Simulation of nanoparticles application for expediting melting of PCM inside a finned enclosure. *J Phys A.* 2019;52:544-556.
21. Sheikholeslami M, Haq R, Shafee A, Li Z. Heat transfer behavior of nanoparticle enhanced PCM solidification through an enclosure with V shaped fins. *Int J Heat Mass Transf.* 2019;130:1322-1342.
22. Sheikholeslami M, Haq R, Shafeec A, Lie Z, Elarakig YG, Tlili I. Heat transfer simulation of heat storage unit with nanoparticles and fins through a heat exchanger. *Int J Heat Mass Transf.* 2019;135:470-478.
23. Karima A, Djamel S, Ali N, Houari A. CFD investigations of thermal and dynamic behaviors in a tubular heat exchanger with butterfly baffles. *Front Heat Mass Transf.* 2018;10(27):1-7.
24. Boukhadia K, Ameur H, Sahel D, Bozit M. Effect of the perforation design on the fluid flow and heat transfer characteristics of a plate fin heat heat exchanger. *Int J Therm Sci.* 2018;126:172-180.
25. Sahel D, Ameur H, Kamla Y. A numerical study of fluid flow and heat transfer over a fin and flat tube heat exchangers with complex vortex generators. *Eur Phys J Appl Phys.* 2017;78:34805.
26. Yadav AS, Bhagoria JL. A numerical investigation of turbulent flows through an artificially roughened solar air heater. *Numerical Heat Transf Part A.* 2014;65:679-698.
27. Naphon P. On the performance and entropy generation of the double-pass solar air heater with longitudinal fins. *Renew Energy.* 2005;30(9):1345-1357.
28. Alam T, Saini RP, Saini JS. Use of turbulators for heat transfer augmentation in an air duct – A review. *Renewable Energy.* 2014;62:689-715.
29. Henaoui M, Aline K, Imame H. Numerical prediction of turbulent characteristics of fluid flow and heat transfer in solar collector with perforated baffles. Paper presented at: Proceedings of the International Conference on Advances in Science; 2016; ICAS, Istanbul.
30. Menasria F, Zedairia M, Moummi A. Numerical study of thermohydraulic performance of solar air heater duct equipped with novel continuous rectangular baffles with high aspect ratio. *Energy.* 2017;133:593-608.
31. Sahel D, Ameur H, Benzeguir R, Kamla Y. Enhancement of heat transfer in a rectangular channel with perforated baffles. *Appl Therm Eng.* 2016;101:156-164.
32. Alam T, Kim M. Numerical study on thermal hydraulic performance improvement in solar air heater duct with semi ellipse shaped obstacles. *Energy.* 2016;112:588-598.
33. Hamza M, Gari AA. Numerical heat transfer and pressure drop investigation of different height baffles mounted simultaneously in a 2-D channel. *Life Sci J.* 2013;10(2):1711-1712.
34. Basra A, Youssef A, Adam NM, Sopian K, Zaharim A, Alghoul M. Analysis of single and double passes V-grooves solar collector with and without porous media. *Int J Energy Env.* 2007;1(2):109-114.
35. Liu J, Hussain L, Wang W, Xie H, Sundén B. Heat transfer enhancement and turbulent flow in a rectangular channel using perforated ribs with inclined holes. *J Heat Transf.* 2019;141(4):041702.
36. Ibrahim TK, Al-Sammarraie AT, Al-Taha WH, et al. Experimental and numerical investigation of heat transfer augmentation in heat sinks using perforation technique. *Appl Therm Eng.* 2019;160:113974.
37. Chang SW, Chen TW, Chen YW. Detailed heat transfer and friction factor measurements for square channel enhanced by plate insert with inclined baffles and perforated slots. *Appl Therm Eng.* 2019;159:113856.
38. CFDCRC. *CFD-ACE+ Theory and Users’ Manuals Ver.6.4*. Huntsville, AL: CFD Research Corporation; 2008.
39. Hamed MH, Kabeel AE. Emad M.S. El-Said, enhancement of heat and mass transfer performance on humidification tower using injection of different carrier gases into water bed. *Appl Therm Eng.* 2017;111:455-476.
40. Kumar KN, Gowardhan M. Secondary flow loss reduction in a turbine cascade with a linearly varied height streamwise endwall fence. *Int J Rotat Mach.* 2011;2011:1-16.
41. Alameldin AM. Numerical study of film cooling influence on performance of transonic vane Cascade (M.Sc. thesis). The American University in Cairo; 2001.
42. Bradley A. CFD simulations for film cooling- reduced models at engine like conditions (M.Sc. thesis). Linköping University: 2013.
43. Menter FR. Two-equation Eddy-viscosity turbulence models for engineering applications. *AIAA J.* 1994;32:1598-1605.
44. Wicloxy DC. Reassesment of the scale-determining equation for advanced turbulence models. *AIAA J.* 1988;26(11):1299-1310.
45. Weast RC. *Handbook of Tables for Applied Engineering Science.* London: CRC Press; 1970.
46. Maaroofi K, Nobari MRH, Shirani E. A numerical simulation of external heat transfer around turbine blades. *Heat Mass Transf.* 2007;44:61-70.
47. Kurtbas I, Gulecimen F, Kilicarslan A, Kaya M. Effect of swirl generator inserted into a tube on exergy transfer: decaying flow. *Exp Heat Transf.* 2014;27:472-487.
48. Chaube A, Sahoo PK, Solanki SC. Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater. *Renew Energy.* 2006;31(3):317-331.
49. Karwa R. Experimental studies of augmented heat transfer and friction in asymmetrically heated rectangular ducts with ribs on the heated wall in transverse inclined, v continuous and v-discrete pattern. *Int Commun Heat Mass Transf.* 2003;30(2):241-250.
50. Tanda G. Heat transfer in rectangular channels with transverse and v-shaped broken ribs. *Int J Heat Mass Transf.* 2004;47(2):229-243.
51. Kabeel AE, El-Said EMS. Hybrid solar desalination system of air humidification–dehumidification and water flashing evaporation Part I. a numerical investigation. *Desalination.* 2013;320:56-72.
52. Kabeel AE, El-Said EMS. A hybrid solar desalination system of air humidification, dehumidification and water flashing evaporation: Part II Experimental investigation. *Desalination.* 2014;341:50-60.
53. Demartini C, Vielmo HA, Möller SV. Numeric and experimental analysis of the turbulent flow through a channel with baffle plates. *J Braz Soc Mech Sci Eng.* 2004;XXVI(2):153-159.

How to cite this article: El-Said EM. Numerical investigations of fluid flow and heat transfer characteristics in solar air collector with curved perforated baffles. *Engineering Reports.* 2020;2:e12142. 
https://doi.org/10.1002/eng2.12142