Performance enhancement of a baffle-type solar heat collector through CFD simulation study

A S T Tan¹, J Janaun¹, H J Tham¹, N J Siambun¹ and A Abdullah²

¹ Industrial Dryer and Palm Oil Recovery Research Group, Universiti Malaysia Sabah, Jalan UMS, 88400 Kota Kinabalu, Sabah, Malaysia
² Chancellery Office, Universiti Malaysia Kelantan, 16300 Bachok, Kelantan, Malaysia

Email: jidon@ums.edu.my; arham.a@umk.edu.my

Abstract. The application of solar energy conversion has been extensively utilized as an alternative energy source to generate heat. This approach would be a step towards sustainable energy development particularly in the manufacturing industry with energy-intensive process. In this paper, thermal enhancement on the key component of a solar energy device – solar heat collector (SHC), has been evaluated by proposing a baffle-type SHC with various geometric configuration in the air passage namely longitudinal baffle and transversal baffle. The performance of SHC is evaluated in term of efficiency, temperature distribution, airflow pattern and pressure drop across the collector outlet through Computational Fluid Dynamic (CFD) investigation. It was observed that maximum collector efficiency was achieved in the Longitudinal-SHC (L-SHC), with a value of 46.2 % followed by Transversal-SHC (T-SHC) and without baffles. Maximum drying temperature at the collector outlet was 332.43 K for L-SHC, showing temperature rise of 0.35 % and 4.21 % from T-SHC and without baffles, respectively. The velocity vector indicated that turbulence flow was created in the T-SHC which consequently improved the heat transfer. Whereas in L-SHC, enhancement was achieved through the prolonged heating time in the passage. Considering the thermo-hydraulic performance factor evaluated, these enhancement features had diminished the effect of pressure drop.

1. Introduction

In recent years, the application of solar energy has been extensively utilized in the industry for drying, space heating, water heating and other industrial application [1–3]. Solar energy is a clean, sustainable and universally accessible energy source. Hence, numerous research have been performed to study the potential of applying solar energy in rural areas. According to Borhanazad et al. [4], electricity coverage of rural areas in Malaysia is about 79 % compared to Peninsular Malaysia with 99.62 %. Since the major source of income in these areas is agricultural production, implementation of least expensive energy solution is essential. Utilization of solar energy not only offers a cheap alternative energy source, but also resolves the issues of limited accessibility to electricity in rural areas. Ultimately, this could boost the local economic opportunities.

As a key component to harvest solar energy and convert into useful heat, many researchers have introduced several thermal enhancement techniques for solar heat collector (SHC). Wide research has been performed to improve the convective heat transfer in the SHC by means of modifying the flow passage as reported by Arunkumar et al. [5]. The two major enhancement mechanism in the air passage
that are widely discussed in the literature are number of flow pass and attachment of tabulator. Satcunanathan et al. [6] first introduced the concept of double pass. It was reported that the efficiency was improved 10 to 15% higher than that of single-pass SHC. Ho et al. [7] confirmed that the enhancement was achieved due to the increase heat transfer area and time hence, reduce the heat loss from the absorber plate. Jia et al. [8] proposed a flow path modification where air is forced to flow through in a spiral path. An average efficiency ranged from 58% to 62% was recorded under irradiance above 700 W/m² and wind velocity of 0 – 3 m/s.

Introduction of baffles into the air passage improve the heat transfer in solar heat collector by increasing the heating time between the fluid and absorber (residence time), to create turbulence for intense mixing and to avoid ‘hot spot’ which consequently reduces the heat loss [9]. These enhancement were reported by Romdhabe [10]. However, findings indicate that the addition of baffles in the air passage increases the pressure drop. Hu et al. [11] studied the pressure drop in a solar heat collector with baffles positioned perpendicularly towards the air direction. It was found that the variation of pressure drop across the passage is low with increasing baffle number. In Hu et al. [12] later study, it was reported that reducing the width at the first chamber air duct shows significant improvement on the thermal efficiency but less influence on the pressure drop. Mummi et al. [13] developed a solar heat collector with baffles attached perpendicularly to the flow to create high turbulence flow. The thermal improvement was remarkable compared to that without baffles. Bensaci et al. [14] performed experimental testing to study the effect of transversal baffle arrangement on the heat transfer coefficient and thermal efficiency of solar heat collector. Highest performance was found in solar heat collector with baffles attached all over the air passage. Although the enhancement on thermal performance was significant with the presence of baffles, several researchers analysed the overall performance through the evaluation of thermo-hydraulic performance, which consider the heat transfer enhancement with respect to the pressure drop established throughout the passage [15–16].

According to the literature review of research in the field of thermal enhancement of solar heat collector through the modification of flow configuration, it is evident that limited study on solar heat collector with baffles positioned parallel to the air direction was reported. It was confirmed by Potgieter et al. [17] that counter-/parallel flow created by this baffle arrangement could improves the turbulence and convection heat transfer in the collector. In addition, less pressure drop is expected as the baffles arranged in parallel does not provokes high pressure drop along the passage. In this present study, a comparative study of the Longitudinal Solar Heat Collector and Transversal Solar Heat Collector is performed through Computational Fluid Dynamics (CFD) simulation to identify the enhancement mechanism occurs in the air passage with different baffle arrangement.

2. Methodology
This study aims to evaluate the thermal performance enhancement on an air-based solar heat collector with different baffles arrangement, namely Longitudinal Solar Heat Collector (L-SHC) and Transversal Solar Heat Collector (T-SHC), using Computational Fluid Dynamics (CFD) simulation. Computational modelling adopted in this study was developed based on the assumptions as follow:

- Three-dimensional fluid flow and heat transfer.
- Steady-state and incompressible flow condition.
- Negligible heat loss from the bottom and side wall of the solar heat collector frame.
- Negligible heat loss from the inlet and outlet of solar heat collector.
- Thermo-physical properties of fluid (air) and solid (aluminium frame and baffle plate) remain constant throughout the process.

Firstly, the computational model adopted in this present study was extracted from previous work that studied the performance of solar heat collector integrated to UMS Eco-Solar Dryer and validated with the experimental results [18]. Experimental testing performed previously was based on solar heat collector with absorber made from recycled aluminium cans, enclosed with plywood frame and acrylic cover at the top. In this present study, recycled aluminium cans absorber was replaced with baffle plate arranged in longitudinal and transversal with regard to the collector. The computational model was
developed based on the geometry and dimension as of the previous work. Quality of mesh generated was then analysed through grid independence study. After the validation of CFD model developed, it was adopted to study the influence of baffles arrangement on the performance of solar heat collector. The enhancement mechanism in solar heat collector was evaluated in respect with the temperature rise of fluid across the collector air passage, velocity vector and pressure drop established approaching to the outlet air vent. In order to analyse the overall performance of solar heat collector, parameters such as Nusselt number, local heat transfer coefficient and friction factor across the air passage was calculated to identify the thermo-hydraulic performance of solar heat collector.

Two different cases were considered in this study depending on the baffle arrangement in the air passage – L-SHC and T-SHC. A solar heat collector without baffle in the air passage was simulated for comparison purpose. For each case, simulation was performed according to the boundary condition specified under natural convection with air velocity of 0.12 m/s. The results were then compared in the aspect of heat transfer characteristics, fluid flow condition, thermal efficiency and thermo-hydraulic performance.

2.1. Description of solar heat collectors

The 3D geometry of solar heat collectors covered in this study are as illustrated in figure 1. The solar heat collectors consist of five main parts: transparent cover, absorber plate, baffles, collector frame and air passage between the absorber plate and bottom plate where heated air circulates. The dimensions of solar heat collector were 2112 mm length, 910 mm width, and 100 mm height according to the prototype – UMS Eco-Solar Dryer, developed in Faculty of Engineering, Universiti Malaysia Sabah (UMS). The cover was a transparent acrylic with 5 mm thickness. Aluminium absorber with thickness of 3 mm was fixed below the cover to absorb solar radiation pass through it. An air gap with 25 mm thickness was left in between the cover and the absorber to ensure greenhouse heating and to prevent heat loss through the top cover. In addition, collector frame made from aluminium was well-insulated to ensure minimal heat loss to the surrounding. Baffles were attached on the bottom plate of the collector in a way to cover 80% of the total height of the air passage to achieve maximum heat transfer between the drying air and absorber wall [19].

![Figure 1. Three-dimensional geometry of solar heat collectors used for CFD model – (a) Longitudinal Solar Heat Collector (L-SHC) and (b) Transversal Solar Heat Collector (T-SHC).](image)

In L-SHC, baffles were arranged parallel with regard to the air direction. In this design, fluid is forced to flow through three time the length of collector while approaching to the outlet air vent. As shown in figure 1 (a), the longitudinal baffle arrangement adopted in this study creates three sets of this particular air channel. In an attempt to minimize the pressure drop across the collector, the width of air duct between two baffles was increased from 50, 100 to 150 mm from the first to third air duct in a single air channel. In T-SHC, baffles were arranged perpendicularly occupying 80% of the total width of solar
heat collector to create a zig-zag air passage across the collector. The width of air duct between baffles was fixed at 182 mm in the collector, which is equivalent to the width of the inlet air vent.

3. Computational model

The CFD model of solar heat collector used in the present study was made up with a rectangular solid geometry comprises seven major domain namely top wall (cover), side wall (collector frame), absorber wall, baffle plates, fluid domain flow through air channel between the top and absorber wall, fluid domain flow through air channel between absorber and bottom wall, air inlet and air outlet. The L-SHC consists of three sets of air channel, each with an air inlet vent and a corresponding outlet vent, whereas the T-SHC consists only one inlet and outlet air vent respectively at each end of the solar heat collector. The simulation was done using ANSYS 2021 R1 Fluent package. The flow in solar heat collector is considered steady-state, three dimensional and incompressible. In addition, since this study aimed to investigate the performance of solar heat collector under natural convection, gravity is considered. All fluid and solid material properties remain constant throughout the process at average air temperature of 300.15 K.

3.1. Boundary condition

The radiation model utilized in this study was Rosseland model incorporated with solar ray tracing algorithm. Realizable k-ε turbulent model with enhanced wall treatment was selected for the calculation of turbulence flow created in the air passage due to the presence of baffles. All wall interfaces were defined as no-slip condition. The top wall (cover) was defined as semi-transparent acrylic with 0.8 transmissivity and participated in solar ray tracing. The side wall (collector frame) was aluminium material defined as adiabatic wall (heat flux = 0) which does not participate in solar ray tracing. Absorber plate positioned below the cover was assigned as aluminium and subject to solar radiation. A mass flow rate condition was assigned at the inlet section whereas the outlet was modelled as pressure outlet.

Coupled algorithm was selected to use with the pressure-based solver to solve the momentum equation and pressure-based continuity equations together [20]. Residual value of $10^{-6}$ and $10^{-3}$ was selected to monitor the convergence of solution for energy and velocity equation respectively.

3.2. Modelling

Performance parameters used to evaluate the enhancement mechanism in the solar heat collector are the outlet temperature, velocity and pressure drop across the air passage, which contour plot was extracted from CFD post-processing software. Dimensionless parameters such as Nusselt number, $Nu$ and friction factor, $f$ that define the thermo-hydraulic performance of solar heat collector can be estimated from the following equations [21]:

$$Nu = \frac{hD}{k}$$  \hspace{1cm} (1)

where $h$ is the heat transfer coefficient (W/m$^2$-K) obtained directly from Fluent simulation and $k$ is the thermal conductivity (W/m-K) of fluid.

$$f = \frac{\Delta PD}{2\rho LV^2}$$  \hspace{1cm} (2)

where $\Delta P$ is the pressure drop (Pa), $\rho$ is the fluid density (kg/m$^3$), $L$ is the total length of air passage (m) and $V$ is the velocity (m/s). $D$ is the hydraulic diameter of the air duct (m), calculated by multiplying $A$, the cross-sectional area (m$^2$), of the air duct by 2 and dividing by the sum of width, $W$ (m) and height, $H$ (m) of the air duct as follow:
\[ D = \frac{2A}{W + H} \]  

(3)

Thermal efficiency, \( \eta \) of the collector is calculated by the equation as follows [22]:

\[ \eta = \frac{Q_u}{G \times A} \]  

(4)

where \( G \) is the global solar radiation (W/m\(^2\)) and \( A \) is the area of the collector (m\(^2\)). The useful heat gain, \( Q_u \) (W) in the solar heat collector is expressed as:

\[ Q_u = m \times C_p \times (T_{out} - T_{in}) \]  

(5)

where \( m \) is the mass flow rate (kg/s), \( C_p \) is the specific heat (J/kg-K) of the fluid, \( T_{out} \) is the outlet temperature (K) and \( T_{in} \) is the inlet temperature (K).

4. Validation of model

The validation of computational model proposed in Section 3 was performed in the aspect of grid independency analysis and by experimental results. The latter approach was reported in previous work, which identified the computational model with a percentage deviation of 1.76 % and 8.60 % compared to the measured value for single-pass solar heat collector and multiple-pass solar heat collector respectively [18]. The basic structure of L-SHC and T-SHC discussed in this study is very similar to the model reported in previous work, which used recycled aluminium cans to modify the air flow configuration in the air passage. Hence, it is safe to assume that the experimental validation reported is applicable to the present study.

4.1. Grid independence test

The computational grid for the domains was generated using unstructured mesh with uniform growth rate of 1.2. During meshing, it is crucial to identify the number of nodes generated in the domain is sufficient prior to obtain an accurate numerical result [23]. Hence, it is necessary to perform grid independence test to evaluate the mesh quality at different grid size. Five different mesh size was used in this study – 0.014, 0.015, 0.018, 0.020, and 0.025 mm. The number of cells were increased from 101,991 to 443,928 using the built-in meshing software in ANSYS Fluent. Each case was carried out under turbulent flow with Re = 7580 (velocity = 2 m/s). The quality of mesh was evaluated based on the temperature of drying air at the outlet vent as this performance parameter is well established in the literature to study the performance of solar heat collector. Results of the mesh independence test for L-SHC and T-SHC are shown in table 1 and table 2 respectively. Based on the results presented, coarse mesh method provides less accurate solution for both model with floating error indicated in L-SHC and large percentage error in T-SHC. Hence, very fine element size was selected for both case due to less percentage deviation (less than 1%) was recorded.

| Trial | Mesh size       | Cell number | Outlet temperature (K) | Percentage deviation (%) |
|-------|-----------------|-------------|-------------------------|--------------------------|
| 1     | Extremely coarse| 111,164     | Floating error          | -                        |
| 2     | Less coarse     | 183,968     | 298.870                 | -                        |
| 3     | Normal          | 260,788     | 301.625                 | 0.920                    |
| 4     | Fine            | 397,750     | 301.648                 | 0.008                    |
| 5     | Very fine       | 457,284     | 301.660                 | 0.004                    |
Table 2. Grid independence test results for Transversal Solar Heat Collector (T-SHC).

| Trial | Mesh size       | Cell number | Outlet temperature (K) | Percentage deviation (%) |
|-------|-----------------|-------------|-------------------------|--------------------------|
| 1     | Extremely coarse| 101,991     | 80.798                  | -                        |
| 2     | Less coarse     | 177,498     | 284.217                 | 251.8                    |
| 3     | Normal          | 241,879     | 292.441                 | 2.894                    |
| 4     | Fine            | 382,380     | 301.581                 | 3.125                    |
| 5     | Very fine       | 443,928     | 301.872                 | 0.096                    |

5. Results and discussion
In this section, the predicted heat transfer and fluid flow characteristics across the solar heat collector are presented in term of contour plot extracted from CFD. Results presented were simulated under natural convection with Reynold number of 1303. The effect of different baffle arrangement on the performance of solar heat collector is discussed in the aspect of thermal efficiency, Nusselt number, heat transfer coefficient and friction factor. Lastly, the thermo-hydraulic performance of each solar heat collector is studied by also taking account of the pressure drop together with the heat transfer enhancement achieved in the air passage.

5.1. Temperature distribution and thermal efficiency
The predicted temperature across the length of the smooth and baffled solar heat collectors (L-SHC and T-SHC) are shown in figure 2. The results were simulated under the condition with inlet velocity of 0.12 m/s and solar radiation of 438.578 W/m². Based on the plotted graph, results simulated in CFD indicate an increase in fluid temperature while approaching to the outlet air vent of the solar heat collector. Both baffled solar heat collector offers more uniform distribution along the length, indicating the reduction of dead zone due to the presence of baffles in the air passage. In addition, the performance of baffled solar heat collector is significantly improved in term of outlet temperature as compared to the smooth solar heat collector, which indicates a maximum fluid temperature of 319.00 K at the collector outlet. For L-SHC and T-SHC, the highest predicted outlet temperature was 332.43 K and 331.26 K respectively. From the plotted graph, it is noticeable that in the L-SHC, a sudden drop of drying temperature is observed at length of 1.2 m. This can be inferred as the reattachment of fluid flow at the turning edge results higher heat transfer rate between the absorber and the fluid hence, higher fluid temperature was observed in the turning region compared to the middle zone (1.2 m).

Temperature uniformity for L-SHC and T-SHC is apparent in figure 3. The temperature contour indicates larger temperature gradient in the T-SHC compared to L-SHC, inferring vortex was formed in the air passage of T-SHC, particularly at the high temperature zone observed near the outlet vent (figure 3b). In this region, heat transfer between the fluid and the absorber is improved due to the separation and reattachment of airflow that would occurs at the downstream of baffle, allowing the air to be constantly heated by the absorber hence, resulting higher temperature at this region. Hu et al. [11] reported similar trend in a four-baffles solar heat collector. The study reveals that in a serpentine flow channel, the fluid temperature attained across the air channel is exactly correspond to the flow pattern. For L-SHC, high temperature zone was observed in the turning region (figure 3a). At each turn, an increment in temperature was observed, followed by a slight reduction until the next turning edge. Similarly, it can be inferred that reattachment of flow occurs in the turning edge resulting high heat transfer rate at this region. In addition, a more uniform temperature distribution was established in L-SHC compared to T-SHC. Potgieter et al. [17] revealed that the presence of fins that allow fluid to flow in parallel and counter pattern exhibits uniform temperature at each consecutive pass.

The predicted thermal efficiency for smooth, L-SHC and T-SHC is 26.6, 46.2, and 43.4%, respectively. It can be concluded that the installation of baffles in solar heat collector significantly improve the efficiency. Despite the fact that vortex zone formed in the air passage enhance the heat transfer between the fluid and absorber, the flow configuration in baffled solar heat collectors increases...
the residence time of fluid in the air passage. Consequently, heat on the absorber plate is fully utilized by the fluid due to prolonged heating time thus, improve the efficiency of collector. A slightly higher thermal enhancement was observed in the L-SHC, which can be explained due to the parallel and counter flow pattern produced in the air passage.

![Figure 2](image2.png)

**Figure 2.** The predicted fluid temperature at the middle region \((y = 0.05\, \text{m})\) of the solar heat collector across the length of air passage \((m = 0.012\, \text{kg/s})\).

![Figure 3](image3.png)

**Figure 3.** Temperature distribution of fluid at the middle region \((y = 0.05\, \text{m})\) of air passage in (a) L-SHC and (b) T-SHC.

5.2. Heat transfer characteristics

In this section, heat transfer characteristics between the fluid and the absorber within the air passage is studied in terms of the local convection heat transfer coefficient \((h_x)\) obtained directly from Fluent and Nusselt number \((Nu)\) calculated as in equation (6). The variation of predicted local convective heat
transfer coefficient along the length of air passage for each solar heat collector is presented in figure 4. As illustrated in the graph plotted, both baffled solar heat collector exhibits higher $h_x$ compared to the smooth one, with maximum value of 4.15 W/m$^2$-K recorded along the air passage. As a comparison, highest heat transfer coefficient recorded in L-SHC and T-SHC was 16.68 W/m$^2$-K and 18.81 W/m$^2$-K respectively. This finding indicates that the heat transfer in solar heat collector can be significantly improved by using baffles. Mahanand and Senapati [24] reported in their previous study that the convective heat transfer in solar heat collector attached with ribs is improved due to the breaking of boundary layer in the air passage, promoting rapid heat transfer between the fluid and absorber plate.

The effect of baffles arrangement on the local convective heat transfer coefficient can also be identified in figure 4. As observed, T-SHC exhibits higher $h_x$ along the air passage compared to L-SHC, inferring larger turbulence was created in T-SHC. As baffles were arranged transversely across the passage, fluid flow will be constantly interrupted by the baffles while approaching towards the outlet vent. This disturbance created in the air passage then results rapid fluid mixing, leading to the improvement of convective heat transfer. Conversely in L-SHC, flow modification occurs only at the turning edge. This explained the lower $h_x$ observed across the air duct.

The Nusselt number calculated using the average convective heat transfer coefficient across the smooth passage, L-SHC and T-SHC is 27.63, 110.93, and 125.12, respectively. As expected, T-SHC has a higher Nusselt number value, inferring fewer dead zones among the tested solar heat collector. Ozgen et al. [25] and Abdullah et al. [26] confirmed the reduction of dead zone corresponding to the turbulence regime. In the L-SHC, less turbulence region is expected hence, yields slightly lower Nu value. Significantly, the heat transfer enhancement in a transversal baffle type solar heat collector is a result of convective heat transfer improvement correspond to the strong vortex flow produced in the air passage. Whereas the major enhancement mechanism occurs in a longitudinal baffle type solar heat collector is the result of prolonged heating time between the fluid and absorber plate.

![Figure 4. Variation of predicted local convective heat transfer coefficient at the middle region (y = 0.05 m) of air passage in L-SHC and T-SHC (m = 0.012 kg/s).](image)

5.3. Fluid flow characteristics
This section presents the internal flow of both L-SHC and T-SHC at the middle region (y = 0.05 m) of the air passage. The velocity vectors are displayed in figure 5. It was found that the flow pattern in L-SHC is less turbulence than that of the T-SHC with significant vortex indicated along the passage. This finding revealed that the thermal enhancement in solar heat collector through the separation and reattachment of flow in longitudinal baffles arrangement is less significant than the transverse baffles which are attached perpendicular to the airflow direction. It was reported that the flow pattern created by the presence of transversal baffles could increase the air velocity, resulting turbulence flow across
the passage [27]. In L-SHC, no significant increment in the variation of air velocity was observed, indicating the heat transfer in this collector occurs completely under laminar flow regime. Hence, explained T-SHC exhibits more efficient convective heat transfer.

Despite to ensure maximum heat transfer between the fluid and absorber occurs in the passage, it is equally important to reduce heat loss from the absorber to the external surrounding through radiation heat transfer. Therefore, it is vital to ensure that convective heat transfer dominates the heat transfer process in the solar heat collector. Temperature distribution across the absorber for L-SHC and T-SHC is as illustrated in figure 6. As expected, L-SHC exhibits higher temperature zone on the absorber, indicating more convective heat loss. Abo-Elfadl et al. [28] confirmed that the heat loss from solar heat collector depends on the absorber temperature. In spite of the fact that the convective heat transfer in L-SHC is less efficient, this flaw is compensated by the increasing residence time of fluid over the absorber. Consequently, the efficiency can be improved through the prolonged heating time in the air passage. Therefore, it may be argued that the drawback of heat loss in L-SHC can be diminished.

**Figure 5.** Velocity vector of fluid at the middle region (y = 0.05 m) of air passage in (a) L-SHC and (b) T-SHC.
5.4. Thermo-hydraulic performance

Thermo-hydraulic performance of a solar heat collector concerns with the enhancement of heat transfer and also the pressure drop established across the collector towards the outlet vent. Pressure contour for both baffled solar heat collector is shown in figure 7. As in the smooth air passage, the predicted pressure drop was 0.038 Pa. From figure 7, it was found that both L-SHC and T-SHC shows higher pressure drop with maximum value of 0.27 and 6.84 Pa respectively recorded in the air passage. This indicates that the position of baffles has significant effect on flow resistance. As calculated using equation (7), the friction factor for L-SHC and T-SHC is 0.58 and 14.66 respectively. The corresponding thermo-hydraulic performance factor (THPF) is 2.1 and 0.8 for L-SHC and T-SHC respectively. Lower THPF was found in the T-SHC, inferring that the flow resistance (friction factor) outweighed the enhancement of heat transfer in the collector. However, the friction factor is expected to reduce at increasing Reynold number [29]. Hence, pressure drop might not have a sensible effect under forced convection. The results predicted by CFD in this study is within the range reported in the literature with Reynolds number ranging from 1200 to 20 000 [30].
Figure 7. Pressure drop at the middle region ($y = 0.05$ m) of air passage in (a) L-SHC and (b) T-SHC.

6. Conclusion

The effect of different baffle arrangement on the heat transfer characteristics, fluid flow characteristics and pressure drop across the solar heat collector has been studied. The main objective of this study is to identify the enhancement mechanism in the proposed arrangement – baffles fixed parallel towards the air direction (L-SHC) and baffles fixed perpendicularly towards the air direction (T-SHC). The major findings from this research are listed as follow:

1. The L-SHC exhibits higher thermal efficiency compared to the T-SHC due to the longer residence time in the air passage. This indicates that fluid pass through the L-SHC fully utilized heat on the absorber through the prolonged heating time thereby attained higher fluid temperature at the collector outlet.

2. Highest local heat transfer coefficient was identified in the T-SHC with a maximum value of 18.81 Wm$^{-2}$, indicating the presence of greater turbulence flow compared to L-SHC. The Nusselt number was found as 125.12. This finding implied that the enhancement in T-SHC was achieved through rapid heat transfer.

3. Velocity contour extracted from CFD shows that fluid in T-SHC has more intense mixing, inferring convective heat transfer dominates the heat transfer process in the collector and that the heat loss can be minimized.

4. Despite the fact that the convective heat transfer in L-SHC is less efficient, the high temperature zone on the absorber plate is insignificant indicating lower potential of heat loss occurs. This can be explained as the prolonged heating time could compensate this drawback.

5. Pressure drop in the T-SHC is significantly higher than that of L-SHC. The corresponding thermo-hydraulic performance factor (THPF) is 2.1 and 0.8 for L-SHC and T-SHC respectively.

References

[1] Chwieduk D A 2016 8—Active solar space heating Advances in Solar Heating and Cooling. (Woodhead Publishing); pp 151–202

[2] Chand T K, Mohanty M K and Mohanty R C 2015 An overview of solar energy and its application
in solar dryers with brief concept of energy and energy analysis \textit{Int. J. Res.} \textbf{2} 870–7

[3] Singh I and Singh S 2018 A review of artificial roughness geometries employed in solar air heaters \textit{Renew. Sustain Energy Rev.} \textbf{92} 405–25

[4] Borhanazad H, Mekhilef S, Saidur R, and Boroumandjazi G 2013 Potential application of renewable energy for rural electrification in Malaysia \textit{Renew. Energy} \textbf{59} 210–9

[5] Arunkumar H S, Vasudeva Karanth K and Kumar S 2020 Review on the design modifications of a solar air heater for improvement in the thermal performance \textit{Sustain. Energy Technol. Assessments} \textbf{39} 100685

[6] Satcunanathan S and Deonarine S 1973 A two-pass solar air heater \textit{Sol. Energy} \textbf{15} 41–9

[7] Ho C D, Yeh H M and Chen T C 2011 Collector efficiency of upward-type double-pass solar air heaters with fins attached \textit{Int. Commun. Heat Mass Transf.} \textbf{38} 49–56

[8] Jia B, Liu F and Wang D 2019 Experimental study on the performance of spiral solar air heater \textit{Sol. Energy} \textbf{182} 16–21

[9] Hu J and Zhang G 2019 Performance improvement of solar air collector based on airflow reorganization: A review \textit{Appl. Therm. Eng.} \textbf{155} 592–611

[10] Romdhane B S 1987 \textit{Contribution to the study and the development of pumps and solar air collectors} (University of Valenciennes, France).

[11] Hu J, Sun X, Xu J and Li Z 2013 Numerical analysis of mechanical ventilation solar air collector with internal baffle \textit{Energy Build} \textbf{62} 230–8

[12] 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 \textit{Renew. Energy} \textbf{135} 701–10

[13] Momeimi N, Youcef-Ali S, Momeimi A and Desmons J Y 2004 Energy analysis of a solar air collector with rows of fins. \textit{Renew. Energy} \textbf{29} 2053–64

[14] Bensaci C E, Momeimi A and Labed A 2020 Experimental investigation on heat transfer coefficient and thermal efficiency of solar air heaters having different baffles \textit{Green Energy and Technology} (Springer International Publishing) pp 309–32

[15] Kumar A and Kim M-H 2016 Thermal hydraulic Performance in a solar air heater channel with Multi V-Type Perforated Baffles \textit{Energies} \textbf{9} 564

[16] Kumar A and Layek A 2018 Thermo-hydraulic performance of solar air heater having twisted rib over the absorber plate \textit{Int. J. Therm. Sci.} \textbf{133} 181–95

[17] Potgieter M S W, Bester C R and Bhamjee M 2020 Experimental and CFD investigation of a hybrid solar air heater \textit{Sol. Energy} \textbf{195} 413–28

[18] Tan A S T, Janaun J, Tham H J, Siambun N J and Abdullah A 2021 Performance analysis of a solar heat collector through experimental and CFD investigation \textit{Renew. Energy} \textbf{81} 139–49

[19] ANSYS 2009 \textit{ANSYS FLUENT 12.0 Theory Guide} \textit{ANSYS Inc.}

[20] Bhushan B and Singh R 2011 Nusselt number and friction factor correlations for solar air heater duct having artificially roughened absorber plate \textit{Sol. Energy} \textbf{85} 1109–18

[21] Chabane F, Girra F, Momeimi N and Brima A 2019 Experimental study of a solar air heater by adding an arrangement of transverse rectangular baffles perpendicular to the air stream. \textit{Int. J. Green Energy} \textbf{16} 1264–77

[22] Lee M, Park G, Park and Kim C 2020 Improvement of grid independence test for computational fluid dynamics model of building based on grid resolution \textit{Adv. Civ. Eng.} \textbf{2020}

[23] Mahanand Y and Senapati J R 2021 Thermo-hydraulic performance analysis of a solar air heater (SAH) with quarter-circular ribs on the absorber plate: A comparative study \textit{Int. J. Therm. Sci.} \textbf{161} 106747

[24] Ozgen F, Esen M and Esen H 2009 Experimental investigation of thermal performance of a double-flow solar air heater having aluminium cans \textit{Renew. Energy} \textbf{34} 2391–8

[25] Abdullah A S, Abou Al-sood M M, Omara Z M, Bek M A and Kabeel A E 2018 Performance evaluation of a new counter flow double pass solar air heater with turbulators \textit{Sol. Energy} \textbf{173}
398–406

[27] El-khawajah M F, Aldabbagh L B Y and Egelioglu F 2011 The effect of using transverse fins on a double pass flow solar air heater using wire mesh as an absorber Sol. Energy 85 1479–87

[28] Abo-Elfadl S, Hassan H and El-Dosoky M F 2020 Study of the performance of double pass solar air heater of a new designed absorber: An experimental work. Sol. Energy 198 479–89

[29] Bensaci C E, Moummi A, Sanchez de la Flor F J, Rodriguez Jara E A, 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 Renew. Energy 155 1231–44

[30] Chamarthi S and Singh S 2021 A comprehensive review of experimental investigation procedures and thermal performance enhancement techniques of solar air heaters Int. J. Energy Res. 45 5098–164

Acknowledgments
This study was supported by Ministry of Finance Malaysia (MOF) Social Enterprise Project (Project Code: R/MOF/B0108/01759A/001/2020/0072), in collaboration between Universiti Malaysia Sabah and Universiti Malaysia Kelantan (UMS Project Code: LPK2001).