Effect of Baffles Position on Thermo-Hydraulic Efficiency of a Solar Air Heater

Rahul Kumar Bhattar1,*, Ali Abbas2 and Anant Kumar Rai3

1,3Department of Mechanical Engineering, National Institute of Technology, Warangal, India
2Department of Mechanical Engineering, Madanapalle Institute of Technology & Science, Madanapalle, A.P, India
*Email: rahul_193713@student.nitw.ac.in

Abstract. Providing artificial irregularity in the form of baffles on the bottom plate of a solar air heater improves the thermal performance of the heater, whereas it reduces the hydraulic efficiency due to friction losses. To optimize the thermo-hydraulic performance of a rectangular duct solar air heater, a thermo-hydraulic performance factor (THPF), dependent on shape, position and size of baffles, is considered in this study. The performance of the heater in various configurations related to the position of the baffles was evaluated considering Renormalization group k-ε model to simulate the fully turbulent flow for a forced convection condition. At a constant value of heat flux of 1000 W/m² on the absorber plate and a fixed value of Reynolds number as 2350, pitch between the baffles was varied from 15mm to 75mm. Further, the distance of 1st row of baffles from the inlet was varied from 200mm to 500mm and the optimum configuration was figured out. For pitch value of 60mm between the baffles and the distance of 1st row of baffles as 200mm from the inlet, the highest value of THPF was obtained as 0.9339. The fluid behavior in presence of baffles was analyzed together with its influence on the thermo-hydraulics parameters.

Keywords: Solar air heaters, Baffles, Nusselt number, Skin friction coefficient, Reynolds number

1. Introduction

Twenty first century is looking towards the renewable sources of energy for fulfilling its energy demands sustainably. The day-to-day rising prices of energy obtained using fossil fuels, depletion in supply of fossil fuel and rising environmental health impact concern are changing the global energy scenario. Establishing renewable energy as a means to supply the growing demand of the rising population of the world is one of the greatest challenge. It is estimated that the world’s population will be about 10 billion by the end of twenty-first century [1]. The increasing demand for electric vehicles in the future plays a crucial role to explore the charging of the electric vehicle batteries using renewable sources of energy such as solar energy. Solar energy is perennial, abundantly and freely available [2, 3, 4]. Harnessing the renewable solar
energy is important in supplying both thermal and electricity demand as solar radiation can be converted to heat or electricity. The technology to convert solar energy into either heat or electrical energy is available and improving rapidly [5, 6]. Due to the advantages like ease of fabrication, low cost, collection of beam and diffuse radiation as well as less maintenance, the flat plate solar air collectors are one of the most commonly used technology for capturing solar radiation [2, 4, 6]. However, the major challenge with the use of flat plate solar collectors is its considerably low efficiency due to the poor heat transfer characteristics in the system.

To increase the efficiency of the collector, researchers provided several ways like use of double or multiple pass ducts instead of using single pass ducts, incorporating baffles in the duct etc. [6, 7]. In case of turbulent flow over a smooth heated duct, the rate at which heat is transferred to the surrounding air is significantly low due to viscous sub-layer present near wall of the duct. The heat transfer through this sub-layer is dominated by conduction mode whereas; convective mode of heat transfer is reduced. The viscous sub-layer is disturbed by roughening the surface using baffles, surface modifications etc. [7, 8]. In such attempt, Bensaci et al. [9] provided the numerical results of thermal and hydraulic performance of a solar air heater with different positions of the baffle in the entire length of the duct. In their study, the positions of the baffles were changed and found that the thermal as well as hydraulic performance depends significantly on the position of baffles in addition to the shape, size or other geometric parameters of the baffles. However, they only considered four configurations with respect to the baffle positions and no change in inter space between baffles were considered. Thus, some more study is required considering the position of baffles. In present research, the baffles placement is varied from the inlet and the inter baffle spacing is also varied to obtain an optimum value of thermo-hydraulic performance factor.

Menasria et al. [10] performed a 2-dimensional numerical study on solar air heater with bottom plate containing rectangular shaped baffles having upper part in an inclination position. Keeping the heat flux value constant at 1000 W/m², they studied the combination of four different non-dimensional values of baffle blockage ratio (Bk) between 0.7 to 0.98 and four different non-dimensional values related to baffle pitch (Pk) between 2 to 8. The Reynolds number range between 4000 and 18000 was studied and found that the arrangement with Bk = 0.7, Pk = 2 at Re = 5000 provides the highest value of thermohydraulic performance factor (THPF) – a factor representing the combined effect of thermal and fluid dynamic performance as 0.857. This optimum configuration leads to improvement in both heat transfer and friction factor by 2.16 and 15.95 times, respectively compared to a smooth duct configuration. They concluded that THPF depends on pitch, height, shape and position of the baffles. Prasad and Saini [11] observed consequences of the non-dimensional height (e/D) and non-dimensional pitch (P/e) on the frictional as well as the heat transfer characteristics of flow in the duct of a solar air heater by providing the irregularities on the bottom surface of the absorber plate. They obtained design curves pertaining to the flow parameters and the roughness for obtaining the optimum value of THPF.

Karwa et al. [12] carried out experiments on a rectangular solar air duct having rectangular cross section artificial ribs arranged in both V-shaped patterns. They observed that the thermo-hydraulic performance of V-down ribs were better than the V-up ribs considering equal amount of pumping power. Similarly, the effects of discrete W-shaped ribs for improving the performance of an absorber plate was also explored [13]. An increase in the thermal performance by a factor of 1.2-1.8 was observed for this arrangement of the duct compared to a smooth configuration of the duct for the range of parameters considered. Lanjewar et al. [14] performed experiments to study the thermal and hydrodynamic performance of a rectangular shape duct fitted with W-shaped ribs roughness on its underside on a wall, which is kept leaning to the flow path. For an angle of attack of 60°, the highest improvement in the performance parameters due to W-shaped roughness was observed compared to smooth duct. Researchers considered different shapes of the fins like helicoidal spring for improving the THPF [15]. The
characteristics of the spring fin like wire and spring diameter, and helicoidal pitch were varied and the heat transfer was found to enhance without considerable resistance to flow.

As the thermal efficiency of solar air heaters is significantly low, to increase its efficiency, roughness in form of baffles are introduced taking into consideration that thermal efficiency increases with minimum power loss because of friction. A lot of experimental study is done in this field whereas, a very few numerical studies are done. The main motive of this work is to check the variation of THPF with respect to the position of baffles and to compare the heat transfer and hydraulic parameters for smooth case and baffled case of the duct.

2. Methodology
2.1. Geometry of computational domain
The experimental data and the geometry for present study are considered from Bensaci et al. [9] as shown in Figure 1. In case of experimental arrangement, the baffles were placed on the 2nd half of the fluid domain, i.e. between the mid and outlet section of the duct. A numerical model was developed based on the experimental set up. The numerical results obtained for temperature variation from inlet to the outlet were validated with the experimental results.

A 3D geometrical model was developed in Design Modeler with the assumptions that fluid is incompressible and having turbulent flow behavior, fluid sustains single phase, negligible radiation and natural convective heat transfer between different parts, steady state situation is achieved, thermo physical properties of air and absorber plate are constant.

Further, two cases were analyzed to compare the results of the present study:
Case-1: solar air duct without baffles throughout the fluid domain.
Case-2: solar air duct with baffles on the second half of the fluid domain.

Figure 2 and 3 shows a 3D computational model of the solar air duct for Case-1 and Case-2, respectively. The geometry is created using ANSYS Design Modeler. Table 1 shows various geometrical parameters of the computational domain. The materials of absorber plate and the bottom plate are aluminum and wood (an insulating material), respectively. Air is considered as the working fluid medium which enters through inlet section and exits through outlet section. To match the simulation conditions with the experiments, the properties of air are taken from Bensaci et al. [9] as shown in Table 2.
Figure 2. Computational scheme for Case-1.

Figure 3. Computational scheme for Case-2.

Table 1. Geometrical parameters.

| Parameter                              | Value  |
|----------------------------------------|--------|
| Duct length (mm)                       | 1500   |
| Duct width (mm)                        | 750    |
| Height of air column (mm)              | 25     |
| Width of baffles (mm)                  | 60     |
| Height of baffles (mm)                 | 20     |
| Thickness of baffles (mm)              | 2      |
| Distance between two adjacent baffles in a row (mm) | 30    |
| Distance between two adjacent rows of baffles (mm) | 50    |

Table 2. Details of the properties of air and absorber plate.

| Property                  | Air(Fluid) | Aluminum(Absorber plate) |
|---------------------------|------------|--------------------------|
| Viscosity (kg/m.s)        | 1.85 e-05  | -                        |
| Density (kg/m³)           | 1.167      | 2719                     |
| Specific heat (J/kg.k)    | 1006       | 871                      |
| Thermal conductivity (W/m.k) | 0.0262   | 202.4                    |
2.1.1. Grid generation
For Case-1, uniform structured grid consisting of only hexahedral elements were used with enough elements near the wall by keeping the y plus value less than one. The mesh metric parameters such as aspect ratio, orthogonal quality and skewness were kept within the permissible range for convergence of the solution. For Case-2, hexahedral elements were used in the first half of fluid domain, whereas tetragonal elements were used for the second half of the fluid domain where baffles were placed. The mesh metric parameters are shown in Table 3. A flow chart of the methodology used for the study is shown in Figure 4.

| Mesh parameters            | Case-1     | Case-2     |
|----------------------------|------------|------------|
| First layer thickness (mm) | 0.209      | 2          |
| Maximum aspect ratio       | 38.5       | 9.8        |
| Minimum orthogonal quality | 1          | 0.23       |
| Maximum skewness           | 0.008      | 0.84       |
| Number of elements         | 206448     | 709148     |

The Reynolds number is defined as equation (1).

\[ Re = \frac{UD_h}{v} \]  

(1)

where, \( U \), \( v \) and \( D_h \) are velocity, kinematic viscosity of air and hydraulic diameter, respectively. The hydraulic diameter is calculated using equation (2).

\[ D_h = \frac{4A}{P} \]  

(2)

where, \( A \) and \( P \) are the cross sectional area and the perimeter of the duct, respectively.

The local heat transfer coefficient is obtained using equation (3).

\[ h_x = \frac{q}{T_{w,x} - T_{b,x}} \]  

(3)

where, \( T_{w,x} \), \( T_{b,x} \) and \( q \) are the local wall temperature, bulk fluid temperature and heat flux, respectively.

The local Nusselt number, calculated near the absorber plate wall, is computed directly using equation (4).

\[ Nu_x = \frac{h_x D_h}{K} \]  

(4)

where, \( K \) is defined as the thermal conductivity.

The hydraulic performance is obtained by calculating the friction factor defined as equation (5).

\[ f = \frac{(\Delta P/L)D_h}{2\rho U^2} \]  

(5)

where, \( \Delta P \) is the pressure difference observed between the inlet and the outlet sections.
THPF is defined as a ratio of thermal and hydraulic outputs which is expressed as equation (6).

\[ \frac{THPF}{(Nu/Nu_s)/(ffs)^{1/3}} \]

where, \( Nu \) and \( Nu_s \) are Nusselt number for baffled and smooth duct, respectively. Similarly, \( f \) and \( f_s \) are friction factors for baffled and smooth duct, respectively.

2.1.2. Boundary conditions
As shown in Table 4, the boundary conditions used for the validation are obtained from the experimental values from Bensaci et al. [9].

| Boundary condition 1 (BC1) | Boundary condition 2 (BC2) |
|---------------------------|---------------------------|
| Mass flow rate            | 0.017 kg/s                |
| Absorber temperature      | 351.4 K                   |
| Bottom temperature        | 332.28 K                  |
| Side heat flux            | 0 W (adiabatic)           |

Table 4. Boundary conditions used for validation.
The inlet air temperature were 305K and 308K for boundary condition 1 (BC1) and 2 (BC2), respectively. The outlet pressure condition was considered as 0 Pascal (atmospheric pressure). A double precision pressure based solver was used to obtain the solution. For Case-1, the residual for convergence was set to the value of $10^{-6}$ for energy equation. Such criterion for other parameters like momentum, continuity, k and epsilon equations was set to a value of $10^{-5}$. For Case-2, the residuals values were taken as $10^{-5}$ and $10^{-3}$ for respective convergence conditions. The SIMPLE numerical algorithm was used to solve the solution using RNG k-epsilon turbulence model. The turbulence intensity values were related to the Reynolds number using equation (7).

$$I = 0.16 \times (Re)^{-1/8}$$  \hspace{1cm} (7)

The mesh independency was carried out for four different mesh densities for Case-1 condition and the effect of different mesh densities on outlet temperature of air has been plotted as Figure 4.

![Figure 5. Mesh independency test.](image)

From the Figure 5, no significant change of outlet temperature of air was observed from mesh density $2\times10^5$. Thus, for present study, mesh density corresponding to number of mesh value $2.06448\times10^5$ has been taken as an optimal number of mesh counts for further simulation.

3. Results and discussion

For validation of numerical results, air temperature at the outlet section of flow duct obtained from numerical simulation is plotted against the experimental data available from Bensaci et al. [9] for both BC1 and BC2 as shown in Figure 6. For BC2 having mass flow rate 0.025 kg/s, the difference between the experimental and numerical values is 2°, i.e. 0.6% of the experimental value. Further, the difference in outlet temperature reduces to 0.6°, i.e. 0.1% of the experimental value for BC1 having mass flow rate 0.017 kg/s. The minor variation in outlet temperature can be attributed to the fact that the experimental set-up offers minor unaccounted resistance to the flow which leads to higher thermal interaction of fluid within the duct leading to higher outlet temperature. However, the simulations did not account for these minor thermal interactions which lead to slightly less outlet temperature. As the outlet temperature of air obtained numerically is very close to the experimental values for both boundary conditions, the numerical scheme is considered validated and accurate enough for further study.
Figure 6. Temperature of the air at outlet section for various flow conditions.

Figure 8 and 9 shows velocity and pressure contours respectively for the Case-2 having baffled duct. It is evident from the contour plot that the obstruction by the baffles in the flow field led to significant pressure drop from inlet to the outlet which is directly apparent in the form of higher frictional losses. These losses lead to higher power requirement by the motor. A negative pressure is observed near the wall of baffles which is responsible for creating vortices in the flow as shown in Figure 7. The produced vortices create more turbulence in the flow resulting in an increase in air velocity in the baffled region and higher interaction of air with baffles. The increase in velocity means an increase in the Reynolds number which leads to a rise in the Nusselt number (Nu); hence, an increase in heat transfer coefficient.

![Vortices](image)

Figure 7. Velocity vectors showing the vortices generated.
For Case-1 and Case-2, the variation in parameters such as Nusselt number and friction factor along the length of the duct are shown in Figures 10 and 11, respectively. Though the total length of duct remains 1.5 m for both cases, the length scale for Case-1 is shown from 0 to 1.5 m whereas, for Case-2, it is from -0.75 to 0.75 m as the coordinate system for Case-2 is taken at the center of the duct. From Figure 10a showing the surface Nusselt number variation for Case-1 with smooth duct, it is observed that the Nusselt number decreased along the length of duct. However, Figure 10b for Case-2 with baffled duct showed peaks with larger amplitudes in the baffled part region corresponding to high local convective heat transfer coefficient; hence, high local Nusselt number due to high vortices. From Figure 11, showing skin friction coefficient variation for both cases, the skin friction coefficient was found to decrease along the length of flow for smooth duct. However, the introduction of baffles increased the skin friction coefficient in the second half of fluid domain for duct with baffles. Thus, it is observed that roughening the surface of duct improves the heat transfer characteristics; but, it leads to increase in friction, i.e. increase in the hydraulic power requirement for forced circulation. So, an optimum design is required to balance between the heat transfer characteristics and hydraulic characteristics.
The variation of Nusselt number and skin friction coefficient with respect to Reynolds number ranging from 2370 to 8340 is studied as shown in Figure 12a and 12b. The Nusselt number was found to increase with increase in Reynolds number and skin friction coefficient decreases with increase in the value of Reynolds number.

**Figure 10(a).** Variation of Nu along length of the duct for Case-1.

**Figure 10(b).** Variation of Nu along length of the duct for Case-2.

**Figure 11(a).** Variation of skin friction coefficient along length of the duct for Case-1.

**Figure 11(b).** Variation of skin friction coefficient along length of the duct for Case-2.

**Figure 12(a).** Nu versus Re.

**Figure 12(b).** $f$ versus Re.
The validated model obtained from the previously explained processes is further used for improving the configurations not studied in Bensaci et al. [9]. For the purpose, the boundary conditions were taken from Bensaci et al. [9] as shown in Table 5 and THPF is obtained for various positions of the baffles.

Table 5. Boundary conditions used for the model taken from Bensaci et al. [9].

| BC3 | Mass flow rate | 0.017 kg/s |
|-----|----------------|------------|
|     | Absorber heat flux | 1000 W/m²² |
|     | Bottom and side heat flux | 0 |

Figure 13 shows THPF variation with pitch distance between baffles for 4 different cases of placement of 1st row of baffles from the inlet; namely, i) 1st row at 200mm offset, ii) 1st row at 300mm offset, iii) 1st row at 400mm offset, and iv) 1st row at 500mm offset. In addition to these 4 different cases of placement of 1st row of baffles from the inlet, the inter baffle pitch distance was varied from 15mm to 80mm in steps of 5mm keeping effective baffle area constant, i.e. the total number of baffles were kept fixed for all the cases considered. The highest THPF obtained is 0.9339 for the 1st row at 200mm offset for a pitch distance of 60mm.

\[ \text{Figure 13. THPF versus pitch for different offset distance from inlet.} \]

4. Conclusion
The effect of THPF variation for different position of baffles from inlet has been evaluated. This study also showed the comparison between thermo-hydraulic characteristics of smooth and baffled ducts considering various parameters such as Nusselt number and skin friction coefficient along the duct. The major results obtained from the research are as follows: (1) The maximum THPF was obtained for case with 1st row of baffles at 200mm offset from inlet for a pitch of 60mm between the baffles, the maximum value of THPF is 0.9339 (2) Nusselt number for a baffled duct has three different behaviors along the length of duct i) upstream of the baffled part, peaks with smaller amplitude are seen corresponding to low
Nusselt number because of reversed flow between first row of baffles and flow regime ii) peaks with larger amplitude are observed in baffled part corresponding to high Nusselt number because of more vortices iii) downstream of baffled part, peaks with smaller amplitude are found than upstream because of the flow regime (3) Nusselt number increases along the length of duct with the increase in Reynolds number and skin friction coefficient decreases along the length of duct as the Reynolds number increases.

References

[1] Yadav A S and Bhagoria J L 2013 Renewable and Sustainable Energy Reviews 23 60-79
[2] Foster R, Ghassemi M and Cota A 2010 Solar energy: renewable energy and the environment (New York: CRC Press Taylor & Francis group)
[3] Li X 2011 Green energy: basic concepts and fundamentals (New York: Springer)
[4] Maczulak A 2011 Renewable energy: sources and methods (New York: Infobase Publishing)
[5] Twidell J and Weir T 2006 Renewable energy (New York: Taylor & Francis)
[6] Sukhatme S P and Nayak JP 2011 Solar energy (New Delhi: Tata McGraw Hill)
[7] Ravi R K and Saini RP 2016 Energy 116 507-516
[8] Hans VS, Saini RP and Saini JS 2009 Renewable and Sustainable Energy Reviews 13 1854-1869
[9] Bensaci C E, Moumni A, Flor F J S de la, Jara E A R, Casado A R and Pardo A R 2020 Renewable Energy 155 1231-1244
[10] Menasria F, Zedairia M and Moumni A 2017 Energy 133 593-608
[11] Prasad BN and Saini JS 1991 Solar Energy 47(2) 91–6
[12] Karwa R, Bairwa RD, Jain B P and Karwa N 2005 Journal of Enhanced Heat Transfer 12(4) 343–55
[13] Kumar A, Bhagoria JL and Sarviya RM 2008 Proc. 19th National & 8th ISHMT-ASME heat and mass transfer Int. Conf.
[14] Lanjewar A, Bhagoria JL and Sarviya RM 2011 Energy 36 4531–41
[15] Kumar HSA, Kumar Sand Karanth KV 2020 Renewable Energy 16 0297-311