COMPUTATIONAL STUDY ON PERFORMANCE OF SOLAR AIR HEATER USING ‘S’ SHAPE RIBS

Khushmeet Kumar
Mechanical Engineering Department Baddi University of Emerging Sciences and Technology, Baddi-173205, INDIA

Sourabh Khurana
Mechanical Engineering Department OM Institute of Technology and Management, Juglan, Haryana-125001, INDIA

Abstract—Solar air heater is used for direct conversion of solar radiation to heat energy. In solar air heaters air is used as a working fluid. Due to low thermal conductivity of air efficiency of solar air heater is less. To overcome this limitation the artificial roughness is provided on the heat transferring surface. The artificial roughness leads to the turbulence of air and hence increases the heat transfer rate. This paper presents the Computational Fluid Dynamics (CFD) analysis of a solar air heater having one of its broad wall artificially roughened with arc shape wire arranged in ‘S’ shape. The effect of geometry on heat transfer, friction factor and performance enhancement was investigated covering the range of roughness parameters \(p/e=8\), \(e/D_h=0.043\), \(\alpha=60^\circ\), \(W/w=3\) and Reynolds number (Re) ranges from 2000-21000. Renormalized k-\(\varepsilon\) model had been used to analyze the problem in ANSYS fluent. It was observed from the study that heat transfer increases with increase in Reynolds number while the friction factor decreases with increase in Reynolds number. Thermal enhancement factor has also been evaluated and a maximum value of thermal enhancement factor has been found to be 3.6 for the investigated range of parameters.

Keywords—Energy, CFD, Fluent, flux, Roughness, Heat transfer, friction.

I. Introduction

Solar air heaters are used for direct conversion of solar energy to thermal energy (heating air) which is used in various industrial and agriculture applications as processing heat. Air heating by using solar energy is one of its simplest applications[1]. A conventional solar air heater generally has its three walls well insulated with fourth wall as the absorber plate which absorbs the solar energy and transfer it to the air flowing underneath the wall (plate). The value of heat transfer coefficient between the plate and air is low due to poor thermal conductivity of air and hence leads to lower efficiency of the solar air heaters[2]. To increase the efficiency the flow of air is made turbulence which leads to increase in the heat transfer rate due to mixing of air. But it also increases the obstruction to air flow and hence the pumping power requirements which is not desirable for efficiency to be maximum [3].

It was found from the literature very less work has been found on CFD investigations as there number of experimental studies has been carried out on solar air heaters. This is due to complexity of flow pattern and computational limitations. With the aid of new development in the field of computers and also the development of new simulations software attracts the researchers for analytical investigations. Increased computational capabilities of computers can produce the results in conformity with the experimental results, so researchers are now diverting towards numerical investigations for solar air heaters.

In the present study a 3-D model of the solar air heater duct was developed and solved by using the standard k-\(\varepsilon\), Renormalized k-\(\varepsilon\) and realizable k-\(\varepsilon\) model for the smooth duct. Renormalized (RNG) k-\(\varepsilon\) model shows that the results are in good agreement with the Dittus-Boelter and Blasius equations. Thus the developed 3-D model of roughened solar air heaters duct having roughness created by arc shaped ribs arranged in ‘S’ pattern were solved by using the Renormalized k-\(\varepsilon\) solution model.

II. Computational Fluid Dynamics Simulation

To investigate the heat transfer and fluid flow characteristics a 3-Dimensional model of the solar air heater duct had been developed and was solved by using ANSYS FLUENT 14.5. The details of developed model for the simulation and solution method are presented in the following subsections.

A. Solution Domain

A rectangular duct having height (H) of 25 mm and width (W) of 300 mm with an aspect ratio of 12 was prepared as shown in Figure 1(a) and Figure 1(b). One broad (Top wall) wall of the duct has been provided with artificial roughness by fixing arc of wires in ‘S’ shape. A uniform
heat flux \((q)\) of 1200 W/m² was added at the roughened wall. The simulation analysis of the model had been carried out by solving the model using ANSYS 14.5.

Figure 1(a) Schematic diagram of absorber plate; (b) Solution Domain used in ANSYS for analysis.

The values of the geometrical and operating parameters used for the simulation are shown in Table 1.

<table>
<thead>
<tr>
<th>S. N.</th>
<th>Parameters</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Reynolds number (Re)</td>
<td>2000 to 21000</td>
</tr>
<tr>
<td>2.</td>
<td>Duct aspect ratio (W/H)</td>
<td>12</td>
</tr>
<tr>
<td>3.</td>
<td>Relative roughness pitch ((p/e))</td>
<td>8</td>
</tr>
<tr>
<td>4.</td>
<td>Relative roughness width ((W/w))</td>
<td>3</td>
</tr>
<tr>
<td>5.</td>
<td>Relative roughness height ((e/D_{th}))</td>
<td>0.043</td>
</tr>
<tr>
<td>6.</td>
<td>Arc angle ((\alpha))</td>
<td>60°</td>
</tr>
<tr>
<td>7.</td>
<td>Heat Flux ((q))</td>
<td>1200 W/m²</td>
</tr>
</tbody>
</table>

\(\text{Table 1 : Range of geometrical and operating parameters}\)

**B. Assumptions for simulation**

The assumptions made for the 3-Dimensional fluid flow are:

1. Fluid flow is fully developed turbulent flow.
2. Flow is steady and incompressible as density variation along the width and height is very less.
3. The duct wall, absorber plate and rib material are homogenous and isotropic.
4. The walls in contact with the fluid are assigned with no-slip boundary condition.

**C. Data reduction**

The main non dimensional factors which are of interest in this computational analysis are Reynolds number, Nusselt number, friction factor and thermo-hydraulic performance parameter. The values of Reynolds number \((Re)\), nusselt number \((Nu)\) and friction factor \((f)\) are given by equation (1-3):

\[Re = \frac{\rho v D_h}{\mu}\]  
\[Nu = \frac{h D_h}{k}\]  
\[f = \frac{\Delta p D_h}{2 p v^2}\]

Providing artificial roughness leads to the enhancement in the heat transfer along with the rise in friction factor. This leads to the increase in the requirement of the pumping power \((P_m)\) which is not desired. So the performance of the solar air heater is estimated by simultaneously considering the heat transfer and friction factor. This is termed as Thermo-hydraulic performance parameter \((\eta)\).

Webb and Eckert [4] proposed a new parameter called thermo-hydraulic performance parameter \((\eta)\) which is used to calculate the enhancement in the heat transfer for the roughened duct as compared to the smooth duct for the same pumping power \((P_m)\) and is given by equation (4):

\[\eta = \left(\frac{Nu_s / Nu_r}{f_s / f_r}\right)^{1/3}\]

**D. Boundary Conditions**

No-slip conditions for the fluid velocity in the solid surfaces were assumed. The two side walls and bottom wall are set as the adiabatic walls and a uniform heat flux \((q)\) of 1200 W/m² was added at the top wall. The temperature of the air inside the duct is assumed at 300K at the initial stage. Reynolds number \((Re)\) is varied from 2000-21000 at the inlet and the mean inlet flow velocity is calculated using the Reynolds number \((Re)\). A pressure outlet condition is applied at exit of duct.

**E. Selection and validation of Model**

The selection of model is carried out by comparing the Nusselt number \((Nu)\) obtained from simulation results by using different turbulence models (standard \(k-\varepsilon\), renormalized (RNG) \(k-\varepsilon\) and realizable \(k-\varepsilon\)) with the Dittus-Boelter equation [5] available for the smooth plate solar air heater i.e. equation (5) and (6).

\[Nu_s = 0.023Re^{0.8}Pr^{0.4}\]  
\[f_s = 0.0791Re^{0.25}\]

The absolute percentage deviation of the Nusselt number \((Nu)\) data by RNG \(k-\varepsilon\) model was found to be \(\pm 5.9\%\) while the prediction by other models shows large deviations from empirical correlation as shown in Figure 2.
III. Result and Discussion

The simulation is performed on ANSYS Fluent14.5. The heat transfer and friction factor were obtained from the simulation results. Based on the results obtained thermal enhancement factor (ε) for solar air heater was also evaluated and presented in the following subsections.

A. Heat Transfer (Nusselt number)

The values of Nusselt number (Nu) for artificially roughened and smooth plate solar air heater were plotted against Reynolds number (Re) as shown in Figure 3.

It is clear from the Figure 3 that Nusselt number increases with the increase in Reynolds Number (Re). Also it is seen that the heat transfer is more for roughened plate as compared to the smooth plate. This increase in the Nusselt number (Nu) for roughened solar air heater; compared to that of the smooth solar air heater is due to the turbulence that is caused by the presence of ribs on heat transferring surface. The turbulence is due to the generation of secondary flow which disturbs the main flow. The profiles of turbulent kinetic energy are shown in Figure 4 for different values of Reynolds number.

B. Friction factor (f)

Figure 5 show the friction factor values for both roughened and smooth plates. The value of friction factor decreases with increase in Reynolds number values. This decrease in the friction factor is due to the suppression of viscous sub layer along the absorber plate and this suppression increases with the increase in the Reynolds number values as seen in the Figure 4.
Variation trend of the Nusselt number and friction factor observed in this simulation study are in broad agreement with the variation reported in experimental results by Kumar et al. [6], Mittal et al. [7], Varun et al. [8] and Varun et al. [9] for different rib arrangements.

C. Effect on Thermal Enhancement Factor

It is seen that for roughened solar air heater duct the heat transfer rate increases; compared to the smooth duct. But it also results the increase in penalty (rise in friction factor). Hence it is desired to evaluate the performance of the solar air heater in terms of a parameter called thermo-hydraulic parameter. Lewis [10] proposed a thermo-hydraulic enhancement parameter (TPP) to evaluate the thermal performance of solar air heater with reference to the performance of the smooth solar air heater. Thus Thermal enhancement factor (ε) for the present study has been calculated by using equation (4). Figure 6 represents the plots of the variation of thermal enhancement factors as function of Reynolds number. It is seen from the Figure 6 that for all values of the Reynolds number thermal enhancement factor is more than 1.

![Figure 6: Variation of thermal enhancement factor with Reynolds number.](image)

IV. Conclusions

In the present study a 3-Dimensional analysis using ANSYS fluent 14.5 has been carried out on artificially roughened solar air heater with arc shaped ribs arranged in ‘S’ shape. Based on the simulation results the following conclusion are may be drawn.

1. The renormalized (RNG) k-ε model predicts the results very close to results predicted by Dittus-Boelter equation for smooth plate. Hence this model can be used to analyze the roughened duct.
2. Nusselt number increase with the increase in Reynolds number values while the friction factor has a reversed trend i.e. friction factor decreases with the increase in Reynolds number values.
3. A significant enhancement in the thermal enhancement factor was reported for roughened duct. The value of thermal enhancement factor varies in the range 1.66 to 3.77 for Reynolds number values from 2000 to 21000.

REFERENCES


