

Analysis of Heat Transfer Augmentation and Flow Characteristics due to Rib Roughness of a Solar Air Heater

Manoj Kumar, Ashish Agarwal

Mechanical Engineering Department

Technocrats Institute of Technology, Bhopal, MP, INDIA.

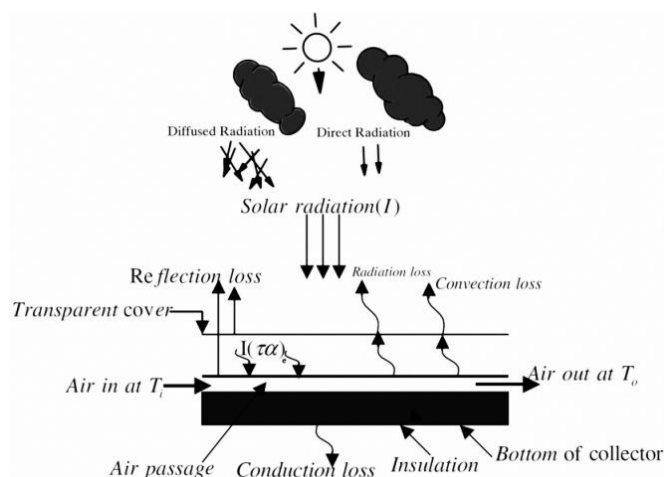
Abstract: This paper presents the study of fluid flow and heat transfer in a plain rectangular duct of a solar air heater by using Computational Fluid Dynamics (CFD). The effect of Reynolds number on heat transfer coefficient and friction factor was investigated. A commercial finite volume package ANSYS FLUENT 12.1 is used to analyze and visualize the nature of the flow across the duct of a solar air heater. CFD simulation results were found to be in good agreement with experimental results and with the standard theoretical approaches. It has been found that the Nusselt number increases with increase in Reynolds number and friction factor decreases with increase in Reynolds number.

Keywords: Energy, Solar Air Heater, Heat transfer, Pressure Drop, CFD

I. Introduction

A conventional solar air heater generally consists of an absorber plate, a rear plate, insulation below the rear plate, transparent cover on the exposed side, and the air flows between the absorbing plate and rear plate. A solar air heater is simple in design and requires little maintenance. However, the value of the heat transfer coefficient between the absorber plate and air is low and this results in a lower efficiency. For this reason, the surfaces are sometimes roughened in the air flow passage. The main application of solar air heaters are space heating, seasoning of timber, curing of industrial products and these can also be effectively used for curing/drying of concrete/clay building components [1].

Figure 1: Solar air heater



A conventional solar air heater generally consists of an absorber plate with a parallel plate below forming a small passage through which the air is to be heated and flows as shown in Fig. 1. A solar air heater is simple in design and requires little maintenance [2]. Performance of any system represents the degree of utilization of input to the system. It is required to analyze thermal and hydraulic performance of a solar air heater for making an efficient design of such type of a system. Thermal performance concerns with heat transfer process within the collector and hydraulic performance concerns with pressure drop in the duct. A conventional solar air heater (Fig. 1) is considered for brief analysis of thermal and hydraulic performance in the following sub-sections [3-6].

Thermal performance of solar air heater can be expressed in terms of the fraction of incident solar radiation utilized to increase the temperature of air. In other words, Thermal efficiency is a measure of thermal

performance of a solar air heater. Thermal performance of solar air heater can be computed with the help of Hottel–Whillier–Bliss equation reported by Duffie and Beckman.

$$Q_u = A_c F_R [I(\tau\alpha)_e - U_L(T_i - T_a)]$$

or

$$q_u = \frac{Q_u}{A_c} = F_R [I(\tau\alpha)_e - U_L(T_i - T_a)]$$

The rate of valuable energy gain by flowing air in the course of duct of a solar air heater can be intended as follows equation:

$$Q_u = \dot{m} c_p (T_o - T_i) = h A_c (T_{pm} - T_{am})$$

The value of heat transfer coefficient (h) can be increased by various active and passive augmentation techniques. It can be represented in non-dimensional form of Nusselt number (Nu).

$$Nu = hl/k$$

Further, thermal efficiency of a solar air heater can be expressed by the following equation;

$$\eta_{th} = \frac{Q_u}{I} = F_R [(\tau\alpha)_e - U_L(T_i - T_a)/I]$$

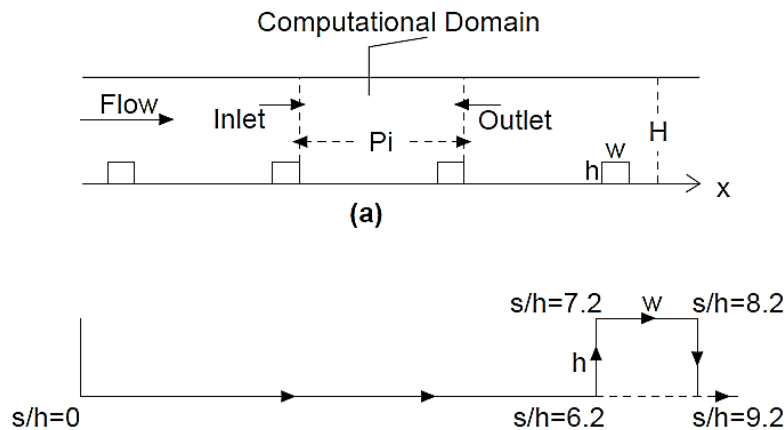
Hydraulic performance of a solar air heater concerns with pressure drop (ΔP) in the duct. Pressure drop accounts for energy consumption by blower to propel air through the duct. The pressure drop for fully developed turbulent flow through duct with $Re < 50,000$ is given as

$$f = \frac{\left(\frac{\Delta P}{l}\right) D_H}{2\rho v^2}$$

II. CFD Modeling and Analysis

Computational Fluid Dynamics (CFD) is the science of determining numerical solution of governing equation for the fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest. The equation can represent steady or unsteady, Compressible or Incompressible, and in viscid or viscous flows, including non-ideal and reacting fluid behavior. The particular form chosen depends on intended application. The state of the art is characterized by the complexity of the geometry, the flow physics, and the computing time required obtaining a solution. The 2-D computational domain used for CFD analysis having the height (H) of 20 mm and width (W) 100 mm as shown in Fig. 2. In the present analysis, a 2-dimensional computational domain of artificially roughened solar air heater has been adopted which is similar to computational domain of Chaube *et al.* [7].

Figure 2: 2-D computational domain



After defining the computational domain, non-uniform mesh is generated. In creating this mesh, it is desirable to have more cells near the plate because we want to resolve the turbulent boundary layer, which is very thin compared to the height of the flow field. After generating mesh, boundary conditions have been specified. We will first specify that the left edge is the duct inlet and right edge is the duct outlet. Top edge is top surface and

bottom edges are inlet length, outlet length and solar plate. All internal edges of rectangle 2D duct are defined as turbulator wall. Meshing of the domain is done using ANSYS ICEM CFD V12.1 software. Since low-Reynolds-number turbulence models are employed, the grids are generated so as to be very fine. Present non-uniform quadrilateral mesh contained 161,568 quad cells with non-uniform quad grid of 0.22 mm cell size. This size is suitable to resolve the laminar sub-layer. For grid independence test, the number of cells is varied from 103,231 to 197,977 in five steps. It is found that after 161,568 cells, further increase in cells has less than 1% variation in Nusselt number and friction factor value which is taken as criterion for grid independence. To select the turbulence model, the previous experimental study is simulated using different low Reynolds number models such as Standard $k-\omega$ model, Renormalization-group $k\epsilon$ model, Realizable $k\epsilon$ model and Shear stress transport $k-\omega$ model. The results of different models are compared with experimental results. The RNG $k-\epsilon$ model is selected on the basis of its closer results to the experimental results. The working fluid, air is assumed to be incompressible for the operating range of duct since variation is very less. The mean inlet velocity of the flow was calculated using Reynolds number. Velocity boundary condition has been considered as inlet boundary condition and outflow at outlet. Second order upwind and SIMPLE algorithm were used to discretize the governing equations. The FLUENT software solves the following mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem [9-10].

III. Results And Discussion

Study of the heat transfer and flow friction characteristics of the artificially roughened ducts shows that an enhancement in heat transfer is accompanied with friction power penalty due to a corresponding increase in the friction factor. The present CFD investigation shows that the roughened duct with relative roughness height (e/d) of 0.06 yields the maximum value of average Nusselt number in the order of 2.78 times that of the smooth duct at higher Reynolds number (18,000) whereas similar roughened duct with similar relative roughness height (e/d) and roughness pitch results in the maximum value of friction factor in the order of 4.24 times that of the smooth duct at lower Reynolds number (3800). Therefore, it is essential to determine the optimal rib dimension and arrangement that will result in maximum enhancement in heat transfer with minimum friction power penalty. A value of thermal enhancement factor higher than unity ensures the effectiveness of using an enhancement device and can be used to compare the performance of number of arrangements to decide the best among these

Figure 3: Thermal enhancement factor vs Reynolds number

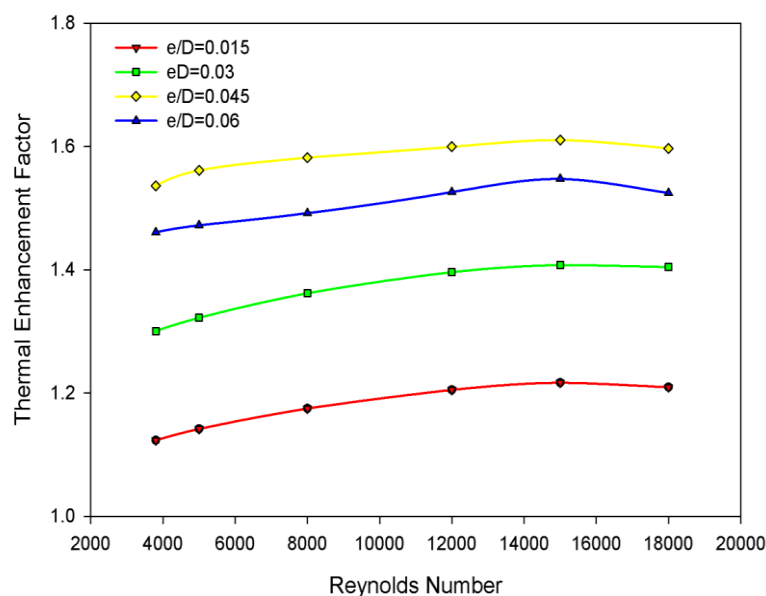
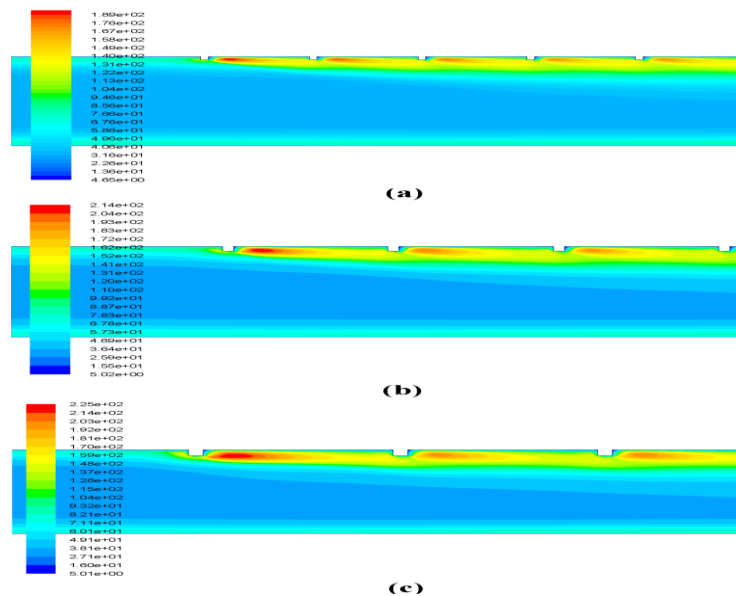


Fig. 3 shows the variation of the thermal enhancement factor with Reynolds number for all cases. It is found that the thermal enhancement factor values vary between 1.12 and 1.61 for the range of parameters investigated. It is observed that roughened duct having square transverse wire rib with $e = 1.5$ mm and $P = 10$ mm (i.e. $e/D = 0.045$) gives better thermal enhancement factor (TEF=1.61) at a Reynolds number of 15,000.

The heat transfer phenomenon can be observed and described by the contour plot of turbulence kinetic energy. The contour plot of turbulence intensity is shown in Fig. 4. The intensities of turbulence are reduced at

the flow field near the rib and wall and a high turbulence intensity region is found between the adjacent ribs close to the main flow which yields the strong influence of turbulence intensity on heat transfer enhancement.

Figure 4: Contour plot of turbulence intensity



IV. Conclusion

A 2-dimensional CFD analysis has been carried out to study heat transfer and fluid flow behavior in a rectangular duct of a solar air heater with one roughened wall having square transverse wire rib roughness. The effect of Reynolds number and relative roughness pitch on the heat transfer coefficient and friction factor have been studied. In order to validate the present numerical model, results have been compared with available experimental results under similar flow conditions. CFD Investigation has been carried out in medium Reynolds number flow ($Re = 3800-18,000$). The following conclusions are drawn from present analysis:

1. The Renormalization-group (RNG) $k-\epsilon$ turbulence model predicted very close results to the experimental results, which yields confidence in the predictions done by CFD analysis in the present study. RNG $k-\epsilon$ turbulence model has been validated for smooth duct and grid independence test has also been conducted to check the variation with increasing number of cells.
2. It is found that the square transverse wire rib roughness with rib height $e=1.7\text{mm}$, pitch $P=10\text{mm}$ and $e/D=0.051$ provides the better thermal enhancement factor ($TEF=1.66$) for the studied range of Reynolds number and hence can be employed for heat transfer augmentation.
3. The discrepancy between available experimental data and present computational results is less than $\pm 10\%$. It can therefore be concluded that the present computational results are reasonably satisfactory.

V. References

- [1] J.L. Bhagoria, J.S. Saini, S.C. Solanki. Heat transfer coefficient and friction factor correlations for rectangular solar air heater duct having transverse wedge shaped rib roughness on the absorber plate. *Renewable Energy* 25 (2002) 341–369.
- [2] R.P. Saini, Jitendra Verma. Heat transfer and friction factor correlations for a duct having dimple-shape artificial roughness for solar air heaters. *Energy* 33 (2008) 1277–1287.
- [3] Gupta Dhananjay, Solanki SC, Saini JS. Heat and fluid flow in rectangular solar Air heater ducts having transverse rib roughness on absorber plate. *Solar Energy* 1993; 51:31–7.
- [4] J.C. Han, J.S. Park, C.K. Lei. Heat transfer enhancement in channels with turbulence promoters. *J. Eng. Gas Turb.Power* 107 (1985) 628–635.
- [5] Y.M. Zhang, W.Z. Gu, J.C. Han, Heat transfer and friction in Rectangular channel with ribbed or ribbed-grooved walls. *ASME/J.Heat Transfer* 116 (1994) 58–65.
- [6] Liou TM, Hwang JJ. Effect of ridge shapes on turbulent heat transfer and Friction in a rectangular channel. *International Journal of Heat and Mass Transfer* 1993; 36:931–40.
- [7] Alok Chaube, P.K. Sahoo, S.C. Solanki. Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater. *Renewable Energy* 31 (2006) 317–331.
- [8] R. Kamali, A.R. Binesh. The importance of rib shape effects on the local heat transfer and flow friction characteristics of square ducts with ribbed internal Surfaces. *International Communications in Heat and Mass Transfer* 35 (2008) 1032–1040.
- [9] Han, J.C., Chandra, P.R., Lau, S.C., 1988. Local heat/mass transfer distributions around sharp 180 deg turns in two-pass smooth and rib roughened channels. *J. Heat Transfer* 110 (February), 91–98.
- [10] Taslim, M.E., Li, T., Spring, S.D. Measurements of heat transfer coefficients and friction factors in passages rib-roughened on all walls. *ASME J. Turbomach.* 1998; 120, 564–570.