International Journal of Thermal Technologies
ISSN 2277 - 4114

Available at http://inpressco.com/category/ijtt

Research Article

Computational Fluid Dynamics Based Analysis of Angled Rib Roughened Solar Air Heater Duct

Anil Kumar[a], Muneesh Sethi[a], Khushmeet Kumar[a*], Sourabh Khurana[a] and Abhilash Pathania[a]

[a] Mechanical Engineering Department, Shoolini University, Solan, (India)-173212

Accepted 24 April 2013, Available online 15June 2013, **Vol.3, No.2 (June 2013)**

Abstract

An artificial roughness on the heat transfer surface in the form of projections mainly creates turbulence near the wall or breaks the laminar sub-layer and thus enhances the heat transfer coefficient. In the present work the performance of a solar air heater duct provided with artificial roughness in the form of thin circular wire in discrete angled rib geometries has been analyzed using Computational Fluid Dynamics (CFD). The effect of this geometry on heat transfer and friction factor and performance enhancement was investigated covering the range of roughness parameters, P/e=8, e/D=0.043, d/W=0.25, g/e=1.0, α=60° and working parameters (Reynolds number, Re from 2,000 to 20,000). Different turbulent models have been used for the analysis heat transfer and friction factor and their results are compared with Dittus-Boelter Empirical relationship for smooth surface. Renormalization k-epsilon model based results have been found in good agreement and accordingly this model is used to predict heat transfer and friction factor in the duct.

Keywords: *CFD, Solar Air Heater Duct*

1. Introduction

Thermal efficiency of solar air heater is significantly low because of low convective heat transfer coefficient between the absorber plate and air, leading to high absorber plate temperature and greater amount of heat losses to the ambient. It has been found that the main thermal resistance to the convective heat transfer is due to the formation of boundary layer on the heat transferring surface. Efforts for enhancing heat transfer have been directed towards artificially destroying or disturbing this boundary layer. The application of artificial roughness in the form of repeated ribs on the heat transfer surface has been recommended to enhance the heat transfer coefficient by several investigators.

Early studies on transverse rib roughness by Prasad and Saini discussed the effect of relative roughness height (e/D) and relative roughness pitch (P/e) on Nusselt number and friction factor. The maximum enhancement in Nusselt number and friction factor were found to be 2.38 and 4.25 times respectively in comparison to that for smooth duct. Gupta et al investigated the effect of relative roughness height, angle of attack and Reynolds number on Nusselt number and friction factor in rectangular duct having inclined circular wire ribs on the absorber plate. Cho et al investigated the characteristics of square duct roughened on two opposite sides with inclined rib having small gap equal to rib height. The heat transfer coefficient is reported to be higher due to gap in the inclined rib which is thought to accelerate and energize the retarded boundary layer flow. The gap provided on the downstream side of inclined rib resulted in higher enhancement in heat transfer than that on the upstream side of the inclined rib. Aharwal et al experimentally studied the effect of width and position of gap in an inclined ribs having square cross-section on heat transfer and friction characteristics of a rectangular duct and reported that the inclined rib with gap arrangement shows higher enhancement in heat transfer compared to that of the continuous inclined rib arrangement.

The literature review shows that the use of artificial roughness in different forms and shapes is an effective means of improving the performance of rectangular ducts. Numbers of experimental investigations involving roughness elements of different shapes, sizes and orientations with respect to flow direction have been carried out in order to obtain an optimum arrangement of roughness element geometry. It was found that a transverse rib roughness enhances the heat transfer coefficient by flow separation and generation of vortices on the upstream and downstream of rib and reattachment of flow in the inter-rib spaces. Angling of transverse rib further enhances the heat transfer on account of the movement of vortices along the rib and formation of a secondary flow cell which results in high heat flow region near the leading end.

In the present work, CFD based study on the performance of rectangular duct, having the absorber plate

*Corresponding author: Khushmeet Kumar

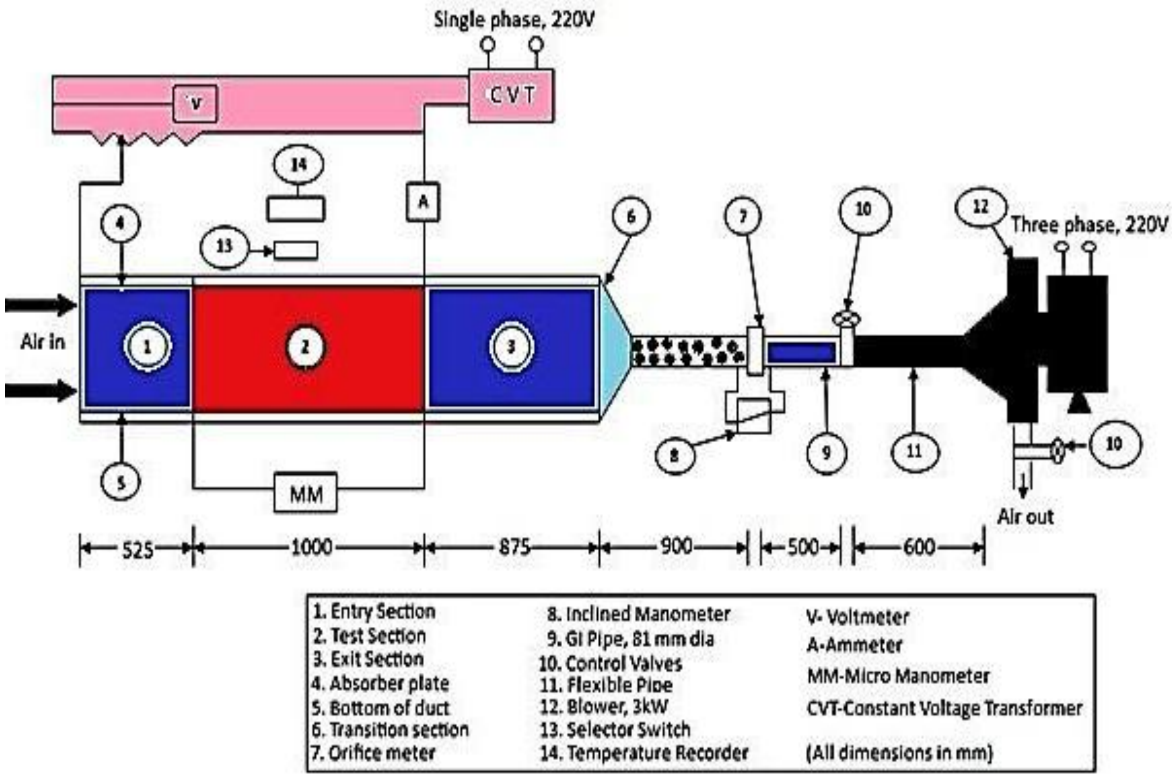

Figure.1 Schematic diagram of experimental set up

with artificial roughness in the form of discrete angled ribs has been carried out.

2. Details of Experimental set-up

A schematic diagram of experimental setup is shown in Fig.1. It consists of an entry section, test section, exit section, transition section, a flow measuring orifice-meter and a centrifugal blower with a control valve. The wooden rectangular duct having a size of 2400 mm×300mm×25mm and is provided with an entrance section, a test section and an exit section of lengths 525mm, 1000mm and 875mm, respectively, including the length of plenum on the exit side in order to minimize the end effects on the test section as per the recommendations of ASHRAE Standard 93-97.

3. Details of roughness parameters for CFD analysis

The relative gap position (d/W) and relative gap width (g/e) values are selected as 0.25 and 1.0 based on the optimum value of these parameter reported in the literature. Selected other parameters such as, relative roughness pitch (P/e) values is selected as 8, relative roughness height (e/D) values is selected as 0.043 and angle of attack (α) value is selected as 60°, based on the optimum values of theses parameters reported in the literature. Table.1 shows in the values of flow and roughness parameters for this study.

Table.1: Range of parameters

Sr. No.	Parameters	Values
1	Relative roughness gap postion (d/W)	6
2	Relative roughness pitch(P/e)	8
3	Relative roughness height (e/D)	0.043
4	Angle of attack(α)	60°
5	Duct aspect ratio (W/H)	12
6	Reynolds number(Re)	2000-20000
7	Relative gap width (g/e)	1

4. Computational fluid dynamics (CFD)

Computational fluid dynamics is analysis of the system involving fluid flow heat transfer and associated phenomena such as chemical reaction by means computer based simulation.

In this investigation a three dimensional numerical simulation of the conjugate heat transfer and fluid flow was conducted using the CFD code FLUENT 6.3.26. The modeling was carried out in order to predict and explain the experimental observations.

4.1 Modeling in GAMBIT

4.1.1 Geometry and grid arrangement

The solution domains used for CFD analysis has been generated as shown Fig.2. The arrangement of domain solution in the form of discrete angled rib. The duct used for CFD analysis having the height (H) of 25 mm and width (W) of 300 mm. A uniform heat flux of 1000 W/m^2 was considered for analysis. Roughness was considered at the underside of the top of the duct to have roughened surface while other three sides were considered as smooth surface.

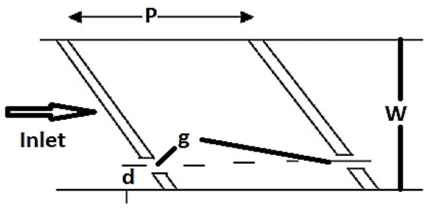

Figure.2 Discrete angled rib roughness geometry.

The 3D solution domain and grid were selected as shown in Fig.3. For the present work, meshing has been done using commercially available software GAMBIT 2.3.16. The size and number of control volumes (mesh density) are user determined and will strongly influence the accuracy of the solutions. After boundary conditions have been implemented the flow and energy balances are solved by an iteration process the decreases the error in the solution until a satisfactory results has been reached.

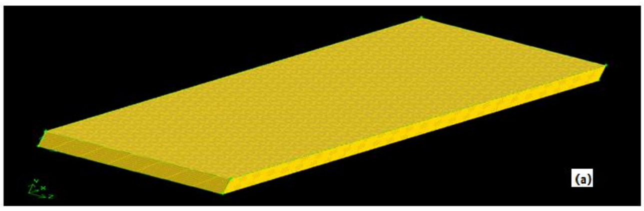

Figure.3 Meshing of the solar air heater duct

4.2 Analysis in FLUENT

4.2.1 Boundary conditions

The boundary conditions as shown in Fig.4 as inlet, outlet, heat flux (absorber plate), insulated wall and the fluid zones are defined as per experimental set up.

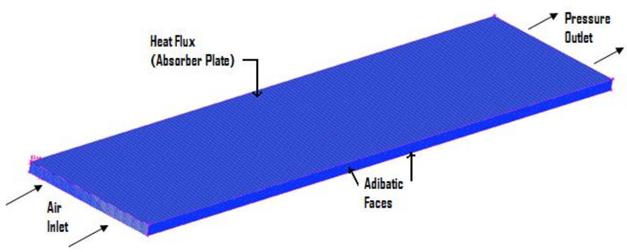

Figure.4 Boundary conditions

4.2.2 Selection and validation of the model

The selection of model is carried out by comparing the predictions by different turbulent models with experimental results available in the literature. The different models namely

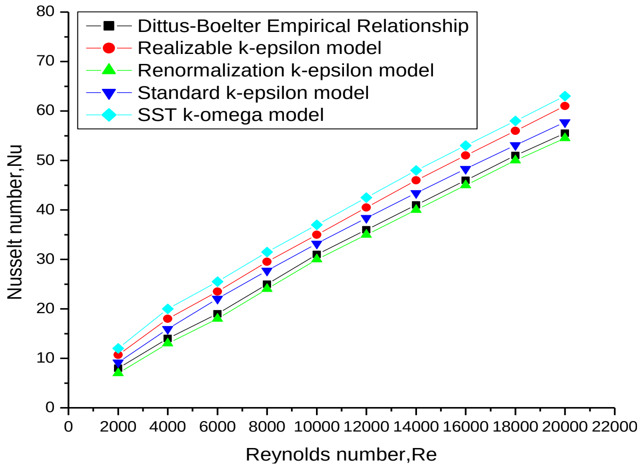

Figure.5 Comparison between Nusselt number with different models with Dittus-Boelter empirical relationship for smooth surface

Renormalization (RNG) group k-epsilon model, Standard k-epsilon model, Realizable k-epsilon model, and Shear Stress Transport (SST) k-omega model have been tested for smooth duct to find out the validity of the models.

$$Nu=0.024Re^{0.8}Pr^{0.4} \qquad (1)$$

Fig.5 shows the variation of Nusselt number with Reynolds number for different models and the results are compared with results computed from the Dittus-Boelter empirical relationship for a smooth duct. It has been observed that the results obtained by Renormalization (RNG) group k-epsilon model are in good agreement with Dittus-Boelter empirical results. It is therefore, for the present numerical study Renormalization (RNG) group k-epsilon model has been employed to simulate the flow and heat transfer.

5. Results and discussion

The major objectives of this investigation is to see the effects of discrete angled rib roughened solar air heater duct with the help of Computational fluid dynamics (CFD) software on heat transfer and friction factor.

5.1 Temperature Profile (Heat Transfer)

Most of the flow which occurs in practical applications is in general turbulent in nature. In the turbulent region, the velocity of the particles very near to surface becomes almost zero. In this region the particle have very low kinetic energy. This region is called laminar sub-layer. These laminar sub-layer acts as a barrier of heat transfer from heated surface to fluid medium. As shown in the

Fig.6 discrete angled ribs are attached underside of the absorber plate these ribs breaking and disturbing the laminar sub layer and flow is wavy flow.

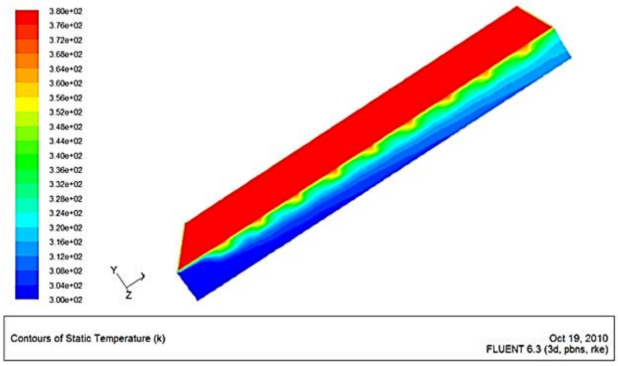

Figure.6. Temperature profile in the roughened solar air heater duct

5.2 Velocity Profile

As shown in the Fig.7 velocity profile on the roughened surface and increase the fluid temperature along the length of the duct and outlet of the duct a fluid temperature is maximum. As shown in Fig.7 in the present Computational fluid dynamics (CFD) based analysis angled rib with gap considered underside of the absorber plate. The atmospheric air enters at the rectangular duct and flows under absorber plate which is uniform distribution of heat flux. Creating gap in the inclined ribs allow release of the secondary flow along the rib joins the main flow to accelerate it, which energizes the retarded boundary layer flow along surface resulting in enhancement of heat transfer as shown in Fig.7 the air temperature increase (indicate red color is maximum temperature) along the length of the duct.

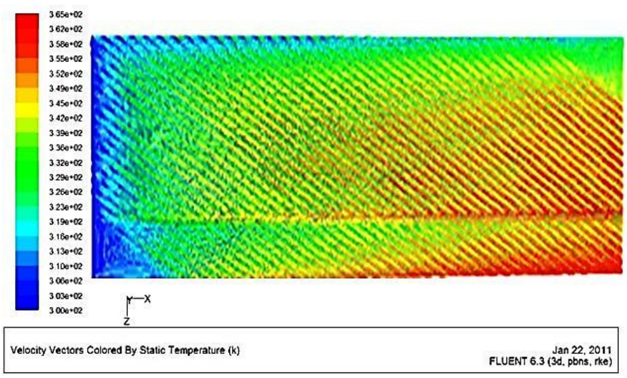

Figure 7. Velocity profile on the smooth surface solar air heater duct

The experimental results and CFD results of the Nusselt number ratios (Nu/Nu_s) as a function of discrete angled rib roughness geometry for different values of Reynolds number as shown in the Fig.8. It is observer that the CFD values and experimental values of friction factor (pressure drop) very close to each other. It has been observed from fig.8 that Nusselt number ratio (Nu/Nus) increases monotonically with increase in Reynolds number snd the maximum value of Nusselt number ratio corresponding to Reynolds number value of 12000.

The secondary flow exerts a measurable effect in disturbing the axial flow profile, which increases the friction coefficient in non-circular duct .

The experimental results and CFD results of the friction factor ratio (f/f_s) as a function of discrete angled rib roughness geometry for different values of Reynolds number as shown in the Fig.9. It is observer that the CFD values and experimental values of friction factor (pressure drop) very close to each other.

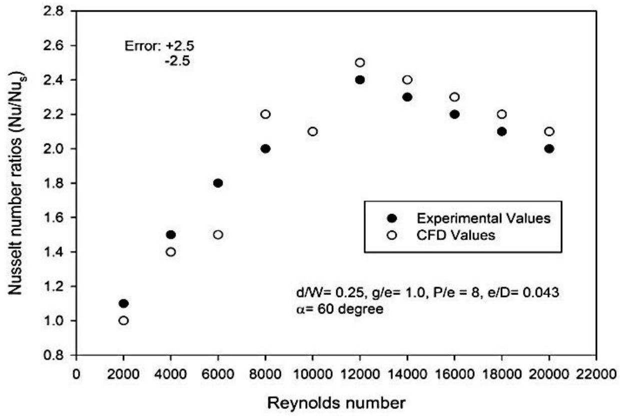

Figure.8 Variation of Nusselt number ratio with Reynolds number for discrete angled rib geometry

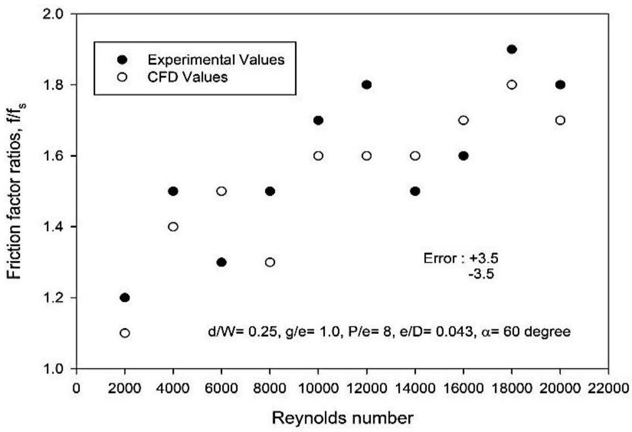

Figure.9 Variation of friction factor ratio with Reynolds number Reynolds number for discrete angled rib geometry

Conclusions

On the basis Computational fluid dynamics (CFD) analysis of heat transfer and friction factor characteristics of the solar air heater duct with discrete angled rib roughness geometry on the absorber plate. The enhancement in heat transfer is found to be increased to 1.45 times that of the heat transfer with that of the heat transfer of smooth surface for discrete angled rib roughened solar air heater. CFD results have been also validated the smooth duct and different CFD model results were compared with Dittus-Boelter empirical relationship for smooth surface. Among all the models used, Renormalization k-epsilon model results have been found to have good agreement.

References

Prasad, B.N., Saini, J.S., 1988. Effect of artificial roughness on heat transfer and friction factor in a solar air heater. *Solar Energy* 41, 555–560.

Gupta, D., Solanki, S.C., Saini, J.S., 1993. Heat and fluid flow in rectangular solar air heater ducts having transverse rib roughness on absorber plates. *Solar Energy* 51, 31–37

Cho, H.H., Kim, Y.Y., Rhee, D.H., Lee, S.Y., Wu, S.J., 2003. The effect of gap position in discrete ribs on local heat/mass transfer in a square duct. *Journal Enhanced Heat Transfer*

Aharwal, K.R., Gandhi, B.K.,Saini, J.S., 2008. Experimental investigation on heat-transfer enhancement due to a gap in an inclined continuous rib arrangement in a rectangular duct of solar air heater. *Renewable Energy* 33,585–96.

ASHRAE, Standard, 93–97, 1977. Method of Testing to determine the thermal performance of solar air heater. American Society for Heating, *Refrigeration and Air Conditioning Engineering,* New York, 1-34.

Kumar S, Saini RP. 2009, CFD based performance analysis of a solar air heater duct provided with artificial roughness. *Renewable energy* ; 34:1285-1291.