International Journal of Conceptions on Computing and Information Technology
Vol. 6, Issue. 1, August' 2018; ISSN: 2345 - 9808

Computational Heat transfer analysis of a Solar Air Heater Having Square Rib Roughness on The Absorber Plate

Amit Kumar and Apurba Layek
Department of Mechanical Engineering
National Institute of Technology, Durgapur
Durgapur, India
amit4310@rediffmail.com and apurba_layek@yahoo.co.in

Abstract— **In this paper CFD based computational analysis of heat transfer enhancement in a rectangular channel duct of a solar air heater equipped with the square rib roughness on the absorber plate has been investigated. The effects of Reynolds number on Nusselt number for square shaped transverse wire rib roughness was examined and their relative evaluation has been plotted. The 2-D computational based analysis is performed using ANSYS FLUENT 16.2 code with the SIMPLE algorithm have been done for the air flow to visualize the characteristics and distribution of the flowing fluid across the duct considering the Reynolds numbers ranging from 3800-18000.The turbulence model which are considered for the evaluation of the smooth duct with square rib roughness were found to be in good agreement with experimental results and with the standard theoretical approaches. The result obtained by the present investigation shows that the Nusselt number strongly dependent on the Reynolds number and it increases as the Reynolds number increases for the range of parameter investigated.**

Keywords-Solar Air Heater; Artificial roughness; Heat transfer; CFD

I. INTRODUCTION

Solar energy is one of the most important non-conventional source of energy. To utilize this energy requires solar collector which absorbs the incident solar radiations and converted it into thermal energy for many applications. The conversion of solar energy into thermal energy is poor due to its low heat transfer coefficient between the absorber plate and the flowing fluid. To make solar air heater more economical by advancing the heat transfer coefficient between the absorber and the flowing fluid which greatly increase the thermal performance of solar air heater. The use of artificial roughness on the absorber plate is an efficient technique to enhance the rate of heat transfer to the flowing fluid. To some extent number of experimental work [3-5] on solar air heater has been done to increase the rate of heat transfer coefficient between the flowing fluid and the absorber plate which increases the overall thermal performance of solar air heater system. The critical analysis on solar air heater has been done by using CFD approach resulted by advanced mathematical models can put together to get large volume of results added no expense and considered to be very cheap to execute parametric analysis to get the most effective equipment performance. In the same conditions some parameters are very difficult to test experimentally which requires both time and cost, such problem can be overcome by using computational technique. The quality of results obtained by CFD approach are found to be in acceptable range.

Bhushan et al.[1] investigated Thermo-hydraulic performance parameter of solar air heater system consist of protruded absorber plate using numerical simulation approach to get the optimum value of the required parameter on the basis of its thermal efficiency. **Chaube et al. [2]** presents a 2-D numerical analysis of a solar air heater having different roughness geometry using ANSYS FLUENT 6.1.and the enhancement of heat transfer was based with minimum pressure penalty. Different rib roughness shapes viz. Rectangular, Square, Chamfered, Triangular, Semicircle etc., were investigated at the Reynolds number range from about 3000–20000. **Prasad and Mullick [3]** studied the effect of heat transfer coefficient using transverse ribs roughness on the absorber plate with experimental studies in order to determine an optimum roughness parameter for attaining maximum heat transfer between the flowing fluid and the absorber plate. **Kumar and Saini** [4] examined CFD based numerical analysis of solar air heater to get the fluid flow and heat transfer characteristics provided with artificial roughness technique for the arc shaped geometry using 3-D models which shows that Nusselt number found to increase with increase in Reynolds number where friction factor decreases with increase in Reynolds number for all combinations of relative roughness height (e/D) and relative arc angle (α/90). **Gupta and Garg [5]** studies with modified absorber plate configurations with heat transfer and their performance of different solar air heaters. **Yadav and Bhagoria [6]** investigated a detailed analysis on heat transfer and fluid flow enquiry on solar air heaters system using artificial roughness on the absorber plate considering 2-D CFD simulation of solar air heater shows that the Renormalization-group (RNG) k-ε turbulence model found to be the best results compared to other turbulence models. Actually, CFD approach predicts very close to physical phenomenon of fluid flow happening in a duct with great accuracy. **Yadav and Bhagoria [7]** investigated the numerical analysis of the artificially roughened solar air heater system which is strong function of the Reynolds number, relative roughness pitch and relative roughness height. **Singh et al. [8]** CFD (computational fluid dynamics) investigation has been carried out to study the heat transfer and friction characteristics of solar air heater duct roughened with periodic transverse rib.

In the present work, design and simulation of a 2-D duct of solar air heater having square rib roughness on the absorber plate is investigated by means of CFD approach to know the best possible outcome of turbulent forced convection. The top side of the wall is subjected to a uniform heat flux while the bottom and other two side walls, except inlet and outlet are insulated.

II. Thermal Performance Analysis

The thermal efficiency of solar air heater is defined as the ratio of useful heat gain by the incident solar radiation to the incident solar radiation intensity on the collector plane per unit area **[9]**. Thermal performance regulates the energy performance and it refers to how well the heat transfer process gains within the collector. Figure 1. shows a conventional type solar air heater where design and details regarding construction are described by **Garg and Prakash [10]**.

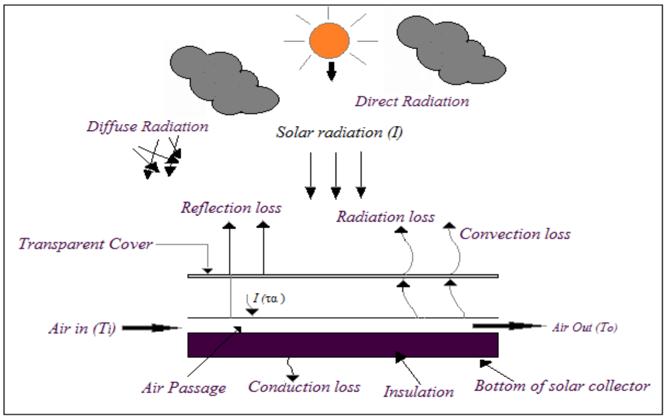

Figure 1. Conventional type solar air heater [8]

The following equation have been taken from Hottel–Whillier–Bliss equation reported by **Duffie and Beckman [11]** for the representation of thermal efficiency.

$$Q_u=A_CF_R[I(\tau\alpha)–UL(T_o\text{-}T_a)] \qquad (1)$$

Where,F_R is Collector heat removal factor, A_c is a collector area,$m^2$and U_L,Overall heat loss coefficient,W/ m^2K

The rate of energy gain by air in the duct of a solar air heater given by the equation (2),

$$Q_u=mC_p(T_o–T_i) \qquad (2)$$

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).

$$Nu=hD/K \qquad (3)$$

Further, thermal efficiency of a solar air heater can be expressed by the equation (4),

$$\eta_{th}=F_R(\tau\alpha)–F_RU_L(T_o\text{-}T_a)/I \qquad (4)$$

III. CFD Approach

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses the applied mathematics, physics and data structure software that analyze to visualize the behavior of the flowing fluid i.e. gas or liquid.CFD is based on the Navier-Stokes equations which describe how the velocity, pressure, temperature and density of a flowing fluid are associated to each other. Dynamics of fluids which are governed by some non-linear partial differential equations generally derived from the laws of conservation of mass, momentum, and energy equation. The equation which are used for the Analytical solutions are only possible for a very simple type of flow domains considering the certain assumptions to be made about the properties of the fluids. In addition to this there are many intrinsic problems encountered with these conventional design processes. In the designing of solar air heater system, the computational technique is simply the helpful method to know the best optimum condition by some design modifications. It works on a very simple principle to solve the complete system of design in a small cells or grids formation by applying governing equations on the discrete elements to find the numerical solutions on velocity, pressure, temperature gradients and density of a flowing fluid which requires minimum time and more economical due to reduction in experimental work **[12-13]**.CFD based analysis generally build a computational model which represents a system that we want to study on it then apply the fluid flow physics to this virtual prototype model and the computational tool software provides a prediction of the fluid flow pattern and other physical phenomena. CFD analysis basically not only an appendage testing for experimental work, but it also leads to a substantial saving of time as a large number of options can be tested.

IV. CFD Simulation

A. Computational Domain

The guidelines for 2-D computational domain CFD analysis has been taken from ASHRAE standard 93-2003 **[14]**. The 2-D domain is a simple rectangle duct having three sections, namely Inlet section, Test section and Exit section as shown in Figure 2. In the present work, roughness element in the form of 2-D square shaped transverse rib has been used. The inlet section length as per the ASHRAE standard **[14]** chosen to develop the thermally fully developed flow and the exit section is used just after the test section in order to reduce the end effect in the test section. Figure 3. shows the roughness geometry of the absorber plate in the square rib shaped.

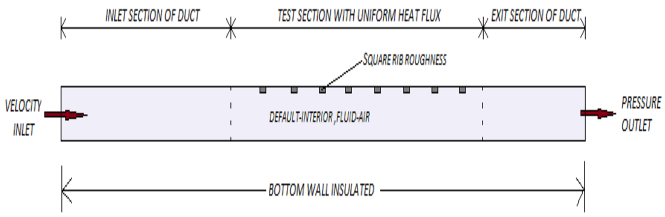

Figure 2.2-D Computational domain

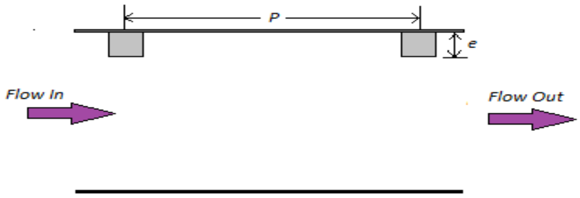

Figure 3. Square shaped rib roughness

Table 1 shows the operating and geometrical parameters of square sectioned transverse wire rib roughness on the absorber plate of solar air heater system that have been used in the present CFD investigation. The top of the computational domain is heated at a constant heat flux of 1000 W/m2 for computational analysis. The flow is assumed to be steady, incompressible and turbulent two-dimensional. The duct wall, absorber plate and roughness material are homogeneous and isotropic.

Table 1: Geometrical and operating parameters of the square rib roughness for CFD investigation.

1	Relative roughness pitch, P/e	10,14.29
2	Heat flux, I	1000 W/m^2
3	Reynolds number range, Re	3800-18000
4	Duct depth, H	20
5	Duct width, W	100
6	Hydraulic diameter, D_h	33.33
7	Relative roughness Height, e/D_h	0.021,0.030
8	Rib height, e	0.7, 1
9	Plate length, L	280 mm
10	Pitch, mm	10 mm

In the present study, a 2-D computational domain rectangular solar air heater duct roughened with square wire transverse ribs on the absorber plate is used for numerical simulation to analyze the flow characteristics of the flowing fluid. ANSYS FLUENT 16.2, is used for the fluid dynamics and heat transfer analysis for the artificially roughened solar air heater system.

The assumptions to be considered for the computational analysis are as follows,

1) The flow is steady, fully developed, turbulent and two dimensional.
2) The thermal conductivity of the duct wall, absorber plate and roughness material are independent of temperature.
3) The duct wall, absorber plate and roughness material are homogeneous and isotropic.
4) The working fluid, air is assumed to be incompressible for the operating range of solar air heaters since variation in density is very less.
5) No-slip boundary condition is assigned to the walls in contact with the fluid in the model.
6) Negligible radiation heat transfer and other heat losses.

B. *Grid Generation*

Uniform grid generation are used for fluid flow distribution and thermal analysis. After defining the computational domain, uniform meshing is done by triangular elements and is created by using ANSYS ICEM CFD V16.2 software. During mesh creation, it is very important to have more no of cells near the heating plate section because we want to resolve the turbulent boundary layer, which is very thin compared to the height of the flowing field. A Grid containing Triangular meshing as shown in Figure 4.

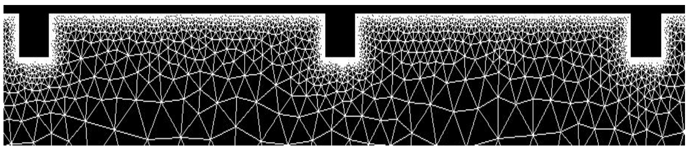

Figure 4. Grid containing Triangular meshing

The CFD modeling involves governing equations for solar air heater duct with artificial roughness is governed by the steady 2D form of continuity, the time independent incompressible Navier stokes equations and the energy equation. These equations can be written as:

Continuity equation is as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0. \qquad (1)$$

Momentum equation is as follows:

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \upsilon\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \qquad (2)$$

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \upsilon\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right). \qquad (3)$$

Energy equation is as follows:

$$u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right), \qquad (4)$$

where, v is the kinematic viscosity and α is the thermal diffusivity.

C. Solver

In this work, the CFD (ANSYS FLUENT v16.2) is used to predict the heat transfer. The top wall is subjected to a uniform heat flux while the other three walls are insulated. The results of the present CFD simulation analysis have been correlate with available experimental results. The RNG k-ε and Realizable k- ε turbulence model are selected for the analysis from the different turbulence model after comparing the results of these models with empirical correlation as (RNG) k-ε model results was found in good agreement.

V. Result and Discussion

The CFD based analysis has been done for the square rib roughness on the underside of the absorber plate of solar air heater systems and the results which is obtained has been compared with the case of smooth duct operating under the same conditions generally used to evaluate the enhancement in heat transfer. Generally governing equation of continuity, energy and momentum equation are used for present simulation process using finite volume method and the equation related to this governing equation is solved by using ANSYS Fluent 16.2. The outcome of fluid flow and its design parameters on the heat transfer is simulated by CFD approach. To consider the best turbulence model for validation such as Renormalization group (RNG) k-ε and Realizable k- ε turbulence it is found that the results of Nusselt Number obtained from RNG k-ε turbulence model is in good agreement with the literature results of Yadav and Bhagoria [6]. The uniform air velocity is considered at the inlet condition while the pressure outlet condition is considered at the outlet of the rectangular duct. The constant heat flux condition is applied to the upper duct wall of test section. The contour plot of temperature distribution along the length of the duct is shown in Figure-5 at Re=18000. The patterns of temperature contour at regions behind and ahead of the rib illustrate the overall temperature field and the degree of heat transfer.

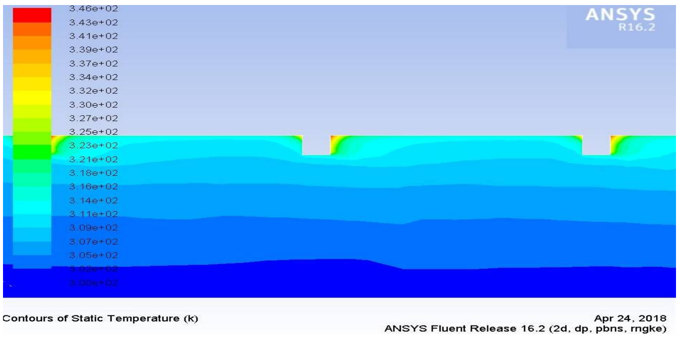

Figure 5. Temperature Contour

Figure 6. and 7. shows the contour of velocity vector and velocity magnitude for the square shape of ribs embedded on the bottom side of absorber plate.

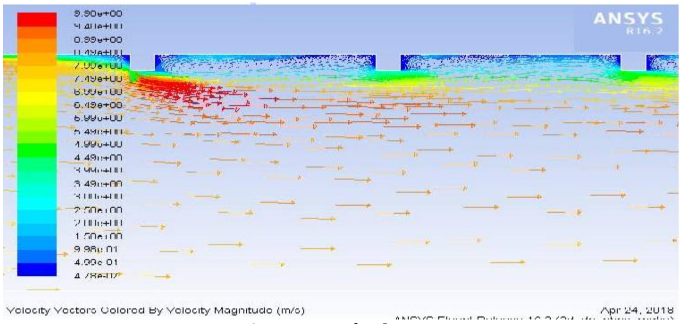

Figure 6. contour of velocity vector

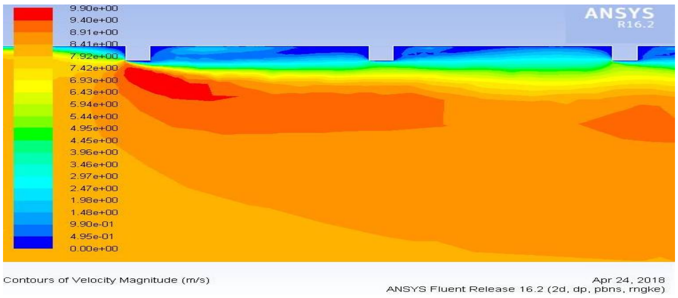

Figure 7. contour of velocity magnitude

The data has been plotted to get the effect of relative roughness height (e/D) on heat transfer of square shaped ribs roughness through CFD approach which is signified through the Fig.8. It can be seen from the Fig 8 that there is a substantial enhancement caused by placing the artificial rib roughness in the form of square shaped embedded below the absorber plate. The average Nusselt number increases as it is observed to increase with increase in Reynolds number mainly due to the increase in turbulent intensity caused by increase in turbulent kinetic energy and turbulent dissipation rate. The enhancement in Nusselt number attain through simulation process varies from 1.7 to 1.93 times as compared to that of the smooth surface. Fig. 8 shows that the enhancement in heat transfer coefficient between the flowing fluid and the absorber plate of the roughened duct with respect to the smooth duct also increases with an increase in Reynolds number. It can also be seen that Nusselt number values increases with the increase in relative roughness height (e/D) for fixed value of roughness pitch, (P) mainly due to the fact that that the laminar sub-la

yer just adjacent to the plate would be of the same order as of roughness height and by increasing the relative roughness height (e/D) the reattachment points over the absorber plate increases which greatly enhance the turbulence effect between the flowing fluid and the absorber plate. The values of rib pitch and height used in the present analysis have been taken from the experimental work of **Yadav and Bhagoria [6]**.

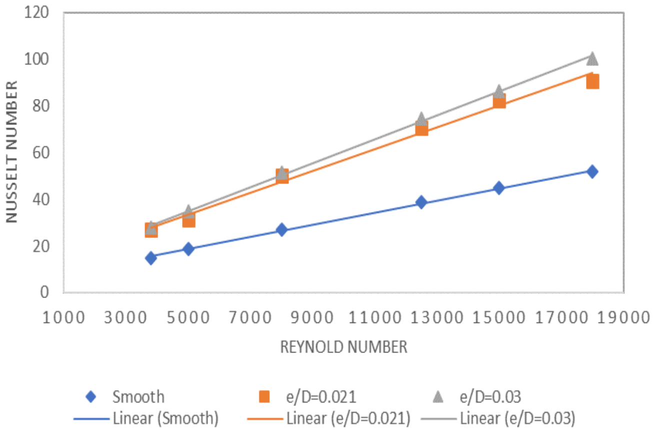

Figure 8. Variation of Nusselt number Vs Reynold number

VI. CONCLUSION

In the present CFD investigation, a 2-D numerical analysis has been governed to study heat transfer and fluid flow behaviors of a rectangular duct of a solar air heater having square rib roughness on the absorber plate. The following conclusions are taken from the present investigation:

- The major focus of CFD based numerical analysis of solar air heater is done to enhance the design process that deals with the heat transfer and fluid flow behavior which does not require a prototype structure to build for testing and experimentation.
- The quality of the results that obtained from CFD simulations technique are mainly within the acceptable range assign that CFD approach is an efficient tool for analyzing the behavior and implementation of a solar air heater system.
- It can be seen from the Fig.8 that Nusselt number increases with the increase of Reynolds number due to increase in heat transfer coefficient between the flowing fluid and absorber plate.
- Solar air heater system having square rib roughness on the absorber plate attain 1.7 to 1.93 times enhancement in Nusselt number as compared to smooth surface.
- It is found that the square rib roughness with relative roughness height (e/D) of 0.03 having constant rib pitch, P of 10 provides the maximum value of Nusselt number for the studied range of Reynolds number and can be employed for better heat transfer augmentation.

REFERENCES

[1] Brij Bhushan, Ranjit Singh," Thermal and thermo hydraulic performance of roughened solar air heater having protruded absorber plate "Solar energy Vol.86, Issue 11, pp.3388-3396,2012.

[2] Chaube A., Sahoo P. K. and Solanki S. C., Effect of Roughness Shape on Heat Transfer and Flow Friction Characteristics of Solar Air Heater with Roughened Absorber Plate, WIT Transactions on Engineering Sciences,2006, 53,43-51.

[3] Prasad. K and S.C. Mullick. Heat Transfer Characteristics of a Solar Air Heater used for drying purposes, Applied Energy. 1983 13: 83-93.

[4] Kumar S. and Saini, R. P., CFD based Performance Analysis of a Solar Air Heater Duct Provided with Artificial Roughness, Renewable Energy, 2009, 34 (5), 1285-1291.

[5] C.L. Gupta and H.P. Garg, Performance Studies on Solar Air Heaters. Solar Energy. 1967,11(1): 25-31.

[6] A.S. Yadav, J.L. Bhagoria, Numerical investigation of flow through an artificially roughened solar air heater, Int.J. Ambient Energy (2013).

[7] Yadav AS, Bhagoria JL. A CFD based thermo-hydraulic performance analysis of an artificially roughened solar air heater having equilateral triangular sectioned rib roughness on the absorber plate. Int J Heat Mass Transf;70: 1016-39, 2014.

[8] Singh S, Hans V.S, Gill R.S. CFD (computational fluid dynamics) investigation on Nusselt number and friction factor of solar air heater duct roughened with non-uniform cross-section transverse rib "Energy" 84,509-517,2015.

[9] R. Karwa and K. Chauhan, "Performance evaluation of solar air heaters having V-down discrete rib roughness on the absorber plate, "Energy, 2010, vol.35, no.1, pp.398-409.

[10] Garg H. P. and Prakash J., Solar Energy Fundamentals and Applications, 1st ed., Tata McGraw-Hill, New Delhi,2000.

[11] Duffie J. A.and Beckman W. A., Solar Engineering of Thermal Processes. 2nd ed., Wiley, New York, 1980.

[12] Date AW. Introduction to computational fluid dynamics.1st ed. New York: Cambridge university Press;2005.

[13] Andwerson Jr JD. Computational fluid Dynamics-the basics with applications.1st ed. New York: McGraw- Hill;1995.

[14] ASHRAE Standard 93, Method of Testing to Determine the Thermal Performance of Solar Collectors, American Society of Heating, Refrigeration and Air Conditioning Engineers, Atlanta, GA 30329, 2003.