NUMERICAL ANALYSIS FOR NUSSELT NUMBER AND HEAT TRANSFER AUGMENTATION ON SOLAR AIR HEATER ROUGHENED WITH SQUARE RIB ROUGHNESS ON THE ABSORBER PLATE

Amit Kumar[1], Apurba Layek

Abstract

Heat transfer rate on solar air heater system is generally achieved by suitable modification of absorber plate surface with acceptable geometry of rib roughness. In this paper a numerical study is performed using CFD based computational analysis of heat transfer augmentation equipped with the square rib roughness created artificially on the absorber plate. The 2D analysis is performed using ANSYS 16.2 Code with RNG k-ε turbulence model to investigate the heat transfer and fluid flow characteristics. The augmentation of heat transfer on increase of Reynolds number enhance the Nusselt number which is referred as energy gain ratio with the use of rib roughness was examined and their relative evaluation has been plotted. The parameter which is considered for the range of analysis is taken as relative roughness pitch (P/e), relative roughness height (e/D) and the Reynolds numbers (Re) ranging from 3800-18000. The enhancement of heat transfers due to rib roughness on the absorber plate have been compared with those for smooth ducts for the same flow and thermal boundary condition to determine the energy gain ratio of solar air heater. The result obtained by the present investigation shows that the maximum value of energy gain ratio is found to be about 1.93 times the smooth duct for the studied range of parameters.

Key words: *Solar Air Heater, Artificial roughness, Heat transfer, CFD.*

INTRODUCTION

The most important non-conventional source of energy is the Solar energy which may be extensively used energy source. A solar collector generally used to utilize this energy which absorbs the incident solar radiations and converted it into thermal energy for many applications. Solar collectors fond to be low thermal efficiency due to low convective heat transfer coefficient between absorber plate and the flowing fluid causes absorber plate to higher temperature and maximum losses to environment **[1]**. Figure 1. shows a conventional type solar air heater system. A conventional type solar air Heater consist of an absorber plate and a transparent cover on the top side of the heated plate. 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 [5-6] 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. Beside with the experimental investigations, researchers also go through for numerical analysis of solar air heater system with rib roughness on the absorber plate to get the predicted value for the range of parameters investigated before actual experimentation done. 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.

Sethi et al. [2] did experimental investigation using dimple shaped as a roughness on the absorber plate to see the effect on heat transfer and flow characteristics of solar air heater duct. **Chaube et al. [3]** investigated a 2-D numerical analysis of a solar air heater for different roughness geometry 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.**Bhushan et al. [4]** 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. **Prasad and Mullick [5]** 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. **Gupta and Garg [6]** studies with modified absorber plate configurations with heat transfer and their performance of different solar air heaters. **Kumar and Saini [7]** 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. **Yadav and Bhagoria [8]** investigated heat transfer analysis on solar air heaters system using rib roughness on the absorber plate considering 2-D CFD simulation of solar air heater found to be the good results compared to data available. Actually, CFD approach predicts very close to physical phenomenon of fluid flow happening in a duct with great accuracy.

Present work Corresponds to simulation study of a 2-D rectangular duct of solar air heater for square rib roughness on the absorber plate is investigated by means of CFD approach to get the best possible outcome of turbulent forced convection for maximum heat transfer enhancement. The uniform heat flux applied on top side of the wall while the bottom and other two side walls, except inlet and outlet are insulated.

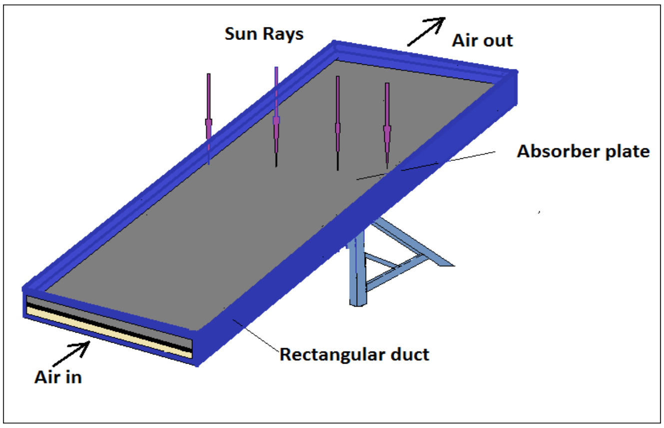

Figure 1. Conventional type solar air heater

I. CFD ANALYSIS

Computational fluid dynamics (CFD) is a branch of fluid science 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. Fluid dynamics 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 generally used for the solutions analytically used 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 design process of solar air heater duct, 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 **[11-12]**.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.

II. CFD SIMULATION

This section generally explains the procedure adopted for solving the 2D numerical simulation of solar air heater system with square rib roughness on the surface.

3.1 Computational Domain

As per ASHRAE standard 93-2003 **[13]**, the guidelines have been taken for 2-D computational domain analysis. 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 **[13]** 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.

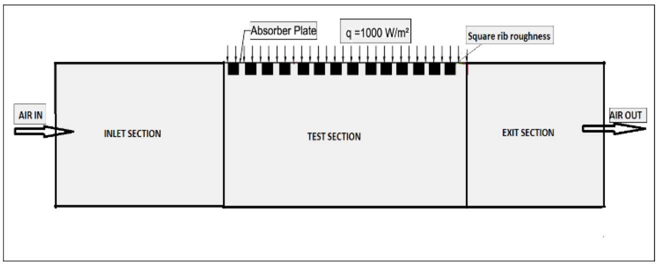

Figure 2. 2D Computational domain

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.

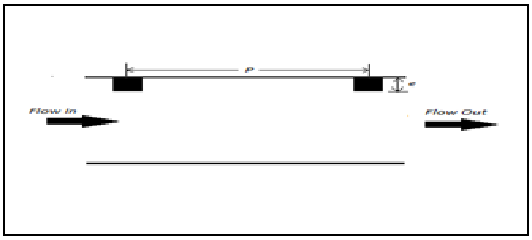

Figure 3. Square shaped rib roughness

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.

3.2 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.

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, υ is the kinematic viscosity and α is the thermal diffusivity.

3.3 Solver

In the present work ANSYS FLUENT v16.2 is used to predict the heat transfer between the heated plate and flowing fluid. 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.

III. DATA REDUCTION FOR EXPERIMENTAL AND CFD ANALYSIS

The parameters which are considered during experiments measured at quasi state condition. The Nusselt number and friction factor are measured from the above parameters considered. Thermal performance regulates the energy performance and it refers to how well the heat transfer process gains within the collector.

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

$$Q_u=mC_p(T_o-T_i) \qquad (5)$$

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).

$$Nu=hD/K \qquad (6)$$

Friction factor from the above parameter for the analysis were calculated by using the following relations,

$$fr=\frac{(\Delta P/l)D}{2\rho V^2} \qquad (7)$$

where D is the hydraulic diameter.

Further, the optimum value for square shaped rib roughness was used using a factor called Thermo hydraulic performance parameter (THPP) given by **Webb and Eckert [9]** expressed by the equation (8),

$$THPP=\frac{(Nu_r/Nu_s)}{(\frac{fr}{fs})^{1/3}} \qquad (8)$$

Where, (N_{ur}/N_{us}) generally known as Nusselt number enhancement ratio.

The general modified equation generally used to find out the Nusselt number for the smooth duct has been compared in Figure. 4 with corresponding values obtained from correlation given by the Dittus–Bolter equation given by **McAdams [10],**

$$N_{us} = 0.023Re^{0.8}Pr^{0.4} \qquad (9)$$

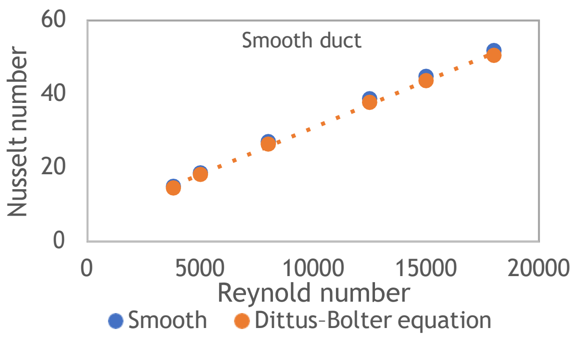

Figure 4. Nu Vs Re for smooth duct.

V. RESULT AND DISCUSSION

In the present investigation the roughness parameter effect on heat transfer using CFD based analysis has been discussed 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 [8]**. 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**.** Figure 5. shows the contour of velocity magnitude for the square shape of ribs embedded on the bottom side of absorber plate.

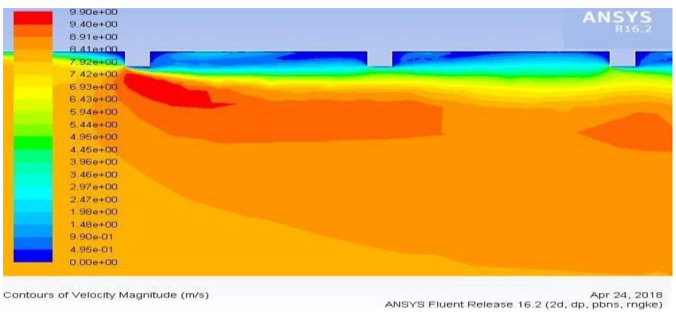

Figure 5. contour of velocity magnitude

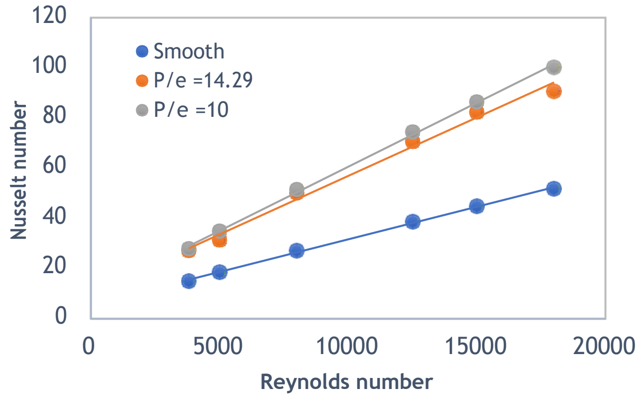

Figure.6 Nusselt number Vs Re

It can be seen from the Fig 6 Nusselt number is an important factor to get the enhancement of heat transfer due to effect the ratio of conductive to the convective resistance of heat flow and s the Reynolds number increases the thickness of boundary layer decreases causes resistance due to convection decreases which leads to increase in Nusselt number. At lower Reynolds number the gain ratio of Nusselt number is generally small due to viscous sub layer just adjacent to the plate retarded by the roughness element which offers resistance to heat flow and hence which lowers the heat transfer coefficient. Fig 8 shows the plot of Nusselt number gain ratio versus Reynolds number for different value of relative roughness pitch(P/e). As 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 maximum gain ratio in Nusselt number is found to be 1.93 times as compared to that of the smooth surface for relative roughness pitch (P/e) of 10 at Reynolds number of 18000 for the range of parameters which is approximately found to be good agreement with experimental data of **Yadav and Bhagoria [8]**.

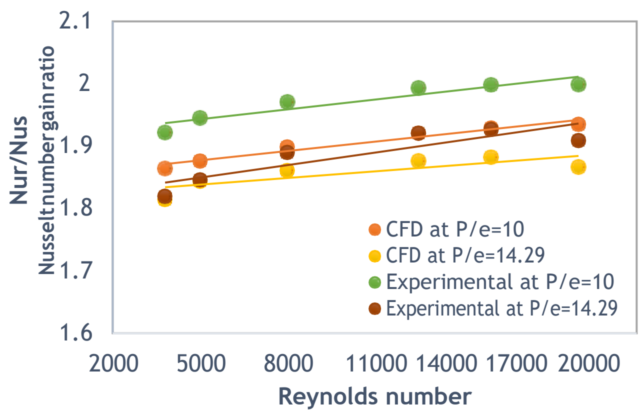

Figure 7. Nusselt number gain ratio Vs Reynold number

VI. CONCLUSION

CFD based numerical analysis is carried out on solar air heater 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 effect of relative roughness pitch on varying Reynolds number are discussed and found that for the entire range of Reynolds number Nusselt number increases attains a maximum at relative roughness pitch (P/e) of 10 and decreases as relative roughness pitch increases. The optimum value of Nusselt number is found to be 100.3 for relative roughness pitch of 10 at 18000 Reynolds number and the maximum enhancement or gain ratio of Nusselt number has been found to be 1.93 times that of smooth duct for relative roughness pitch of 10 at a Reynolds number of 18,000 for the parameters investigated. 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.

REFERENCES

[1] Sukhatme SP., Tata McGraw Hill Publishing Company Limited, Solar Energy,1999, New Delhi Fourth Reprint.
[2] Sethi, M., Varun, Thakur, N.S., Correlations for solar air heater duct with dimpled shape roughness elements on absorber plate. Sol.Energy,2012,86, 2852–2861.
[3] 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.
[4] Brij Bhushan, Ranjit Singh," Thermal and thermo hydraulic performance of roughened solar air heater having protruded absorber plate "Solar energy,2012,Vol.86, Issue 11, 3388-3396.
[5] Prasad. K and S.C. Mullick., Heat Transfer Characteristics of a Solar Air Heater used for drying purposes, Applied Energy. 1983 13: 83-93.
[6] C.L. Gupta and H.P. Garg, Performance Studies on Solar Air Heaters. Solar Energy. 1967,11(1),25-31.
[7] 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.
[8] A.S. Yadav, J.L. Bhagoria, Numerical investigation of flow through an artificially roughened solar air heater, Int.J. Ambient Energy (2013).
[9] Webb, R.L., Eckert, E.R.G., Application of rough surface to heat exchanger design. Int. J. Heat Mass Transfer,1972,15 (9), 1647–1658.
[10] McAdams, W.H., Heat Transmission. Mc Graw-Hill, New York, 1942.
[11] Date AW. Introduction to computational fluid dynamics.1st ed. New York: Cambridge university Press;2005.
[12] Anderson Jr JD. Computational fluid Dynamics-the basics with applications.1st ed. New York: McGraw- Hill;1995.
[13] 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.