

# A CFD Based Performance Analysis of Heat Transfer Enhancement in Solar Air Heater Provided with Transverse Semi-Circular Ribs

Kanhaiya Jhariya<sup>1</sup>, Rajeev Ranjan<sup>2</sup>, M.K.Paswan<sup>3</sup>

M.Tech Student, Department of Mechanical Engineering, NIT Jamshedpur, Jamshedpur, India<sup>1</sup>

Research Scholar, Department of Mechanical Engineering, NIT Jamshedpur, Jamshedpur, India<sup>2</sup>

Professor, Department of Mechanical Engineering, NIT Jamshedpur, Jamshedpur, India<sup>3</sup>

**ABSTRACT:** This article is present for a detailed investigation of the design of solar air heater having semi-circular transverse rib roughness on the absorber plate by using the application of computational fluid dynamics (CFD). In this Solar air heater an absorber plate is made of 'Al' and roughened with semi-circular sectioned transverse ribs due which creates the turbulence in the flow of fluid (air) and increase the heat transfer from the absorber plate to the fluid. Quantity of heat transfer is depending on Nusselt number. Nusselt number depends on surface heat transfer coefficient 'h'. Heat transfer coefficient is strongly depends on the relative roughness height (e). Here ANSYS FLUENT v 14.5 is used to simulate fluid flow through a conventional solar air heater.

The results predicted by the present CFD investigation are much closer to previous experimental results. It can, therefore, be concluded that the present numerical results have demonstrated the validity of the proposed system. Thus, it is possible to establish a validated model for the prediction of heat transfer and fluid flow phenomena in artificially roughened solar air heater roughened with square-sectioned transverse rib roughness on the absorber plate.

**KEYWORDS:** Solar air heater, Solar energy, CFD, THPP, Nu, Re.

## I.INTRODUCTION

The Solar air heater is one of the basic equipment through which solar energy is converted into thermal energy. A solar air heater is a type of heat exchanger which transfers solar radiation into heat energy. Solar air heaters, because of their simple designing, are cheap and most widely used as a collection devices of solar energy. A solar air heater requires little maintenance. Solar air heater is a type of solar thermal system where air is heated in a collector and either transferred directly to the interior space or to a storage medium. 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. The air gets heated up while the absorber plate absorbs the heat. The hot air is drawn through the plates with a blower which is operated electrically.

The main applications of solar air heater 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. The other applications of solar air heater are drying of agro and allied products, food items such as fruits, vegetables, chilies, tea-leaves, fish, salt, etc. The solar air heater can be used in many industrial activities (drying/heating) such as chemical, pharmaceutical, limited areas of textiles and hoseries, tannery, edible oil, etc.

The thermal performance of conventional solar air heater has been found to be poor because of the low convective heat transfer coefficient from the absorber plate to the air (Duffie and Beckman 1980) [1]. Conventional solar air heaters also have poor thermal efficiency primarily due to high heat losses and low convective heat transfer coefficient between the absorber plate and flowing air stream, leading to higher absorber plate temperature and greater thermal losses. The duct

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

formed by the absorber and the bottom plate is the air flow duct, where the air is heated by the absorbed solar radiation on the absorber. Among the many effective ways to augment the convective heat transfer rate in channel flows, those that increase the heat transfer surface area and those that increase turbulence inside the channel with fins or corrugated surfaces and surface roughness have been the most popular. For this reason, the surfaces are sometime roughened in the air flow passage. The use of artificial roughness on a surface is an effective technique to enhance the rate of heat transfer to fluid flowing in a duct. Artificial roughness in the form of repeated ribs has been found to be a convenient method to enhance the rate of heat transfer. Ribs of various shapes and orientations have been employed and the performance of such system has been investigated. Artificial roughness have been used to enhance the heat transfer coefficient by creating turbulence in the flow. However, it would also result in an increase in friction losses and hence greater pumping power requirements for air through the duct. In order to keep the friction losses at a low level, the turbulence must be created only in the region very close to the duct surface, i.e. in the laminar sub-layer. The application of artificial roughness in the form of fine wires or ribs of different geometries on the heat transfer surface has been recommended to enhance the heat transfer coefficient by several investigators using experimental and computational fluid dynamics (CFD) approaches.

The purpose of this work is to investigate heat transfer characteristics of a 2D rectangular duct of a solar air heater provided with semi-circular sectioned transverse rib roughness by adopting CFD approach. In this work, the CFD code, ANSYS FLUENT v14.5 is used to predict the heat transfer. The upper wall is subjected to a uniform heat flux condition while the lower wall is insulated. The ribs are provided on the underside of the absorber plate whereas other sides of the duct are kept smooth. The results of the present CFD analysis have been compared with available experimental results. The main aim of the present CFD analysis is:

- To investigate the effect of roughness height, roughness pitch, relative roughness pitch and relative roughness height on heat transfer.

## II.RELATED WORK

The concept of artificial roughness was first applied by Joule (1861) [2] to enhance heat transfer coefficients for in-tube condensation of steam and since then many experimental investigations were carried out on the application of artificial roughness in the areas of cooling of gas turbine, electronic equipments, nuclear reactors, and compact heat exchangers etc. Nunner (1956) [3] was the first who developed a flow model and likened this model to the temperature profile in smooth tube flow at increased Prandtl number. The proposed flow model predicts that roughness reduces the thermal resistance of the turbulence dominated wall region without significantly affecting the viscous region. A friction correlation for flow over sand-grain roughness was developed by Nikuradse (1950) [4]. Based on law of the wall similarity, Nikuradse presented the pressure drop results in terms of roughness function  $R$  and roughness Reynolds number  $e^+$ . Dipprey and Sabersky (1963) [5] developed a heat-momentum transfer analogy relation for flow in a sand-grain roughened tube and achieved excellent correlation of their data. The concept proposed by Dipprey and Sabersky was so common and it can be applied to any roughness for which law of the wall similarity holds. Prasad and Mullick (1983) [6] were the first who introduced the application of artificial roughness in the form of small diameter wire attached on the underside of absorber plate to improve the thermal performance of solar air heater for drying purposes. Karwa et al. (2001) [7] conducted an experimental investigation of the performance of solar air heaters with chamfered repeated rib-roughness on the air flow side of the absorber plates. Authors found substantial enhancement in thermal efficiency (10-40%) over solar air heaters with smooth absorber plates. Saini and Verma (2008) [8] conducted an experimental investigation on fluid flow and heat transfer characteristics of solar air heater duct having dimple-shaped artificial roughness. Authors found maximum value of Nusselt number corresponds to relative roughness height ( $e/D$ ) of 0.0379 and relative roughness pitch ( $P/e$ ) of 10. Authors also found minimum value of friction factor correspond to relative roughness height ( $e/D$ ) of 0.0289 and relative pitch ( $P/e$ ) of 10. Singh et al. (2011) [9] experimentally investigated the heat transfer characteristics of rectangular duct having its one broad wall heated and roughened with periodic 'discrete V-down rib'. Authors found maximum value of Nusselt number and friction factor corresponds to relative roughness pitch ( $P/e$ ) of 8. Bhagoria et al. (2002) [10] performed experiments to determine the effect of relative roughness pitch, relative roughness height and wedge angle on the heat transfer and friction factor in a solar air heater roughened duct having wedge shaped rib roughness. Authors found maximum enhancement of Nusselt number up to

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

2.4 times while the friction factor up to 5.3 times for the range of parameters investigated. Gandhi and Singh (2010) [11] conducted a CFD study to investigate the effect of artificial surface roughness on flow through a rectangular duct having bottom wall roughened with repeated transverse ribs of wedge shaped cross-section. Two dimensional numerical modeling of the duct flow using FLUENT showed reasonably good agreement with the experimental observations except for the friction factor. Numerical results obtained by commercial computational fluid dynamics (CFD) code FLUENT were compared with the experimental results. Yadav and Bhagoria (2013) [12] conducted CFD investigations of an artificially roughened solar air heater to examine the effect of circular transverse ribs roughness on the absorber plate. 2D CFD simulation of the duct was based on CFD code FLUENT v12.1. The simulation results revealed that RNG  $k-\varepsilon$  turbulence models predicted flow distribution and heat transfer very well. A CFD simulation was conducted with different turbulence models in order to find out the best appropriate turbulence model for simulation of artificially roughened solar air heater. The results obtained by the RNG  $k-\varepsilon$  model were in good agreement with the Dittus-Boelter and Blasius empirical correlations. Yadav and Bhagoria (2013) [13] conducted a numerical analysis of the heat transfer and flow friction characteristics in an artificially roughened solar air heater having square sectioned transverse rib roughness considered at the underside of the top heated wall. The thermohydraulic performance parameter under the same pumping power constraint was calculated in order to examine the overall effect of the relative roughness pitch. The maximum value of the thermohydraulic performance parameter was found to be 1.82 corresponding to a relative roughness pitch of 10.71.

### III. CFD SIMULATION

The 2D heat transfer and fluid flow phenomenon through an artificially roughened solar air heater is investigated by means of a numerical model. The 2D CFD simulation of artificially roughened solar air heater is carried out using commercial CFD simulation code ANSYS FLUENT (version 14.5) to solve the conservation equations for mass, momentum and energy. The numerical model for fluid flow and heat transfer through an artificially roughened solar air heater is developed under the following assumptions:

- (1) Steady 2D fluid flow and heat transfer.
- (2) Both thermally and hydraulically fully developed flow (steady-state conditions).
- (3) The thermal conductivity of the duct wall, absorber plate and roughness material are independent of temperature.
- (4) The duct wall, absorber plate and roughness material are homogeneous and isotropic.

### IV. COMPUTATIONAL DOMAIN

Artificial roughness has been used to enhance heat transfer by increasing the level of turbulence mixing in the flow. In numerically simulating such flows, it is common to simulate a 3D rib-roughened duct with a 2D domain in order to reduce the computational power and time. Hence, in this work, the 2D flow is therefore carried out for saving computer memory and computational time. The computational domain of an artificially roughened solar air heater is represented in 2D form by a semi-circle and displayed in Figure 1. According to ASHRAE Standard (2003), This investigation is taken for relative roughness height ( $e/D$ ) of 0.06, relative roughness pitch ( $P/e$ ) in the range of  $2.5 \leq P/e \leq 20$  and Reynolds numbers ( $Re$ ) in the range of  $3800 \leq Re \leq 18,000$  (relevant in solar air heater). Artificial roughness in the form of semi-circular sectioned transverse ribs is considered at the underside of the top of the duct, i.e. on the absorber plate while other sides are considered as smooth surface. A uniform heat flux of  $1000\text{W/m}^2$  is considered on the top of the absorber plate for numerical analysis. The geometrical and operating parameters employed in this CFD investigation are listed in Table 1, Table 2, and Table 3.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

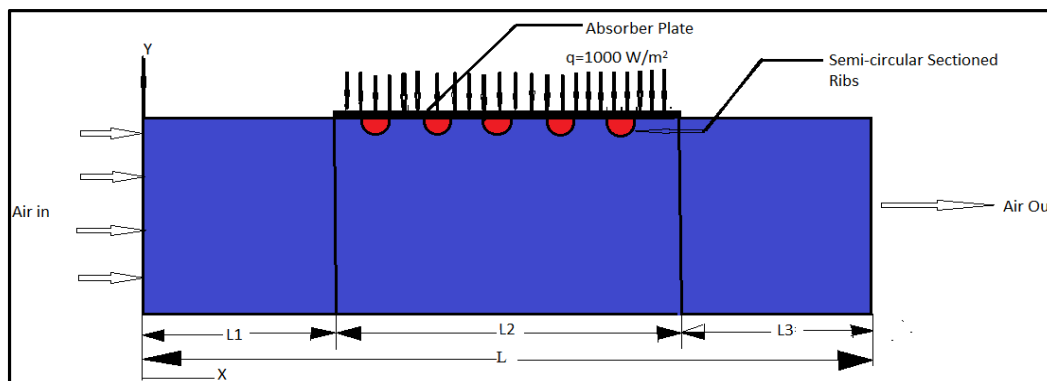


Figure 1: Schematic of 2D computational domain.

the domain consisted of three sections, namely entrance section (L1), test section (L2), and exit section (L3). The internal duct cross section is  $100 \times 20\text{mm}^2$ .

Table 1: Geometric parameters of the artificially roughened solar air heater duct.

L <sub>1</sub> (mm)	L <sub>2</sub> (mm)	L <sub>3</sub> (mm)	W (mm)	H (mm)	D (mm)	e (mm)	P (mm)
225	121	115	100	20	33.33	2	10,

Table 2: Range of operating parameters for computational analysis

Operating parameters	Range
Uniform heat flux 'q'	1000 W/m <sup>2</sup>
Reynolds Number 'Re'	3800, 5000, 7200, 10000, 12000, 15000, 18000
Prandlt number 'Pr'	0.7441
Rib hight 'e'	2 mm
Pitch 'P'	10mm
Relative Roughness Pitch 'P/e'	5
Relative Roughness hight 'e/D'	0.06
Duct aspect ratio 'W/H'	5

Table 3: Thermo-physical properties of working fluid (air) and absorber plate (aluminum) for computational analysis

Properties	Working fluid (air)	Absorber plate (aluminum)
Density, 'ρ' (kg m <sup>-3</sup> )	1.225	2719
Specific heat, 'C <sub>p</sub> ' (Jkg <sup>-1</sup> K <sup>-1</sup> )	1006.43	871
Viscosity, 'μ' (Nsm <sup>-2</sup> )	1.7894e-05	-
Thermal conductivity, 'k' (W m <sup>-1</sup> K <sup>-1</sup> )	0.0242	202.4

A typically roughened absorber plate of semi-circular sectioned H transverse ribs has been shown in Figure 2.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

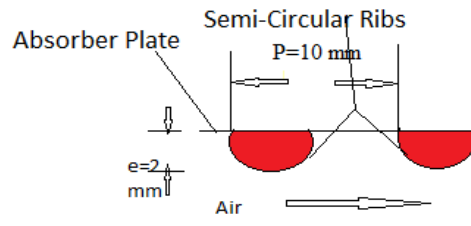


Figure 2: Roughened absorber plate with different arrangement of Semi-circular sectioned transverse ribs.

## Mesh generation

Non-uniform grids are generated for all numerical simulations performed in this work. Non-uniform grids are commonly used in modeling when large gradients are expected. Automatic non-uniform grids are generated for all numerical simulations performed in this work. Grids are generated using ANSYS ICEM CFD v14.5 software. A non-uniform grid contained 65100 quad nodes and 62058 elements with cell size of 0.0002m is used to resolve the laminar sub-layer, as shown in Figure 3. An extensive test for the confirmation of grid independence of the model are carried out by increasing the mesh density and adopting various mesh grading until further refinement shows a difference of less than 1% in two consecutive sets of results.

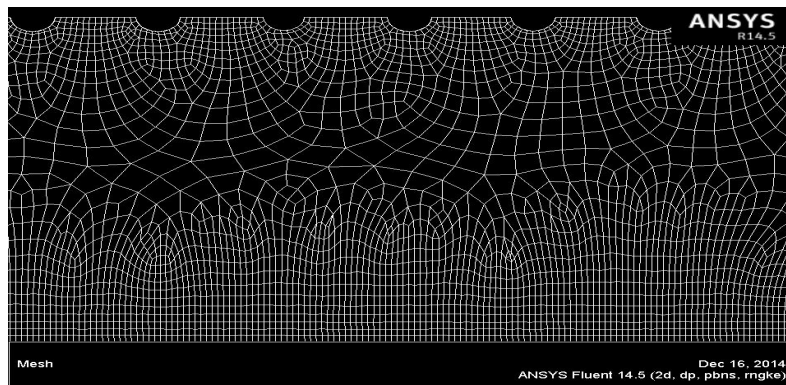


Figure 3: Two-dimensional non uniform meshing.

## Governing equations

The flow phenomenon in artificially roughened solar air heater duct is governed by the steady 2-dimensional form of the continuity, the time-independent incompressible Naviere Stokes equations and the energy equation. In the Cartesian tensor system these equations can be written as:

Continuity equation:

$$\frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (1)$$

Momentum equation:

$$\frac{\partial}{\partial x_i} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial p}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \overline{u_i' u_j'}) \quad (2)$$

Energy equation:

$$\frac{\partial}{\partial x_i} (\rho u_i T) = \frac{\partial}{\partial x_j} \left( (\Gamma + \Gamma_t) \frac{\partial T}{\partial x_j} \right) \quad (3)$$

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

Where  $\Gamma$  and  $\Gamma_t$  are molecular thermal diffusivity and turbulent thermal diffusivity, respectively, and are given by the following.

$$\Gamma = \mu / Pr \quad \text{and} \quad \Gamma_t = \mu_t / Pr_t$$

(4)

## Boundary condition:

The solution domain of the considered 2d, rectangular duct flow is geometrically quite simple. It is a rectangle on the x-y plane, enclosed by the inlet, outlet and wall boundaries. No-slip conditions for velocity in solid surfaces are assumed and the turbulence kinetic energy is set to zero on all solid walls. The top wall boundary condition is selected as constant heat flux of 1000 W/m<sup>2</sup> and bottom wall is assumed to be in adiabatic condition. The temperature of air inside the duct is also taken as 300 K at the beginning. At the exit, a pressure outlet boundary condition is specified with a fixed pressure of 1.013×10<sup>5</sup> Pa. A uniform air velocity is introduced at the inlet and the mean inlet velocity of the flow is calculated using Reynolds number.

## V. DATA REDUCTION FOR THERMO-HYDRAULIC PERFORMANCE

The aim of present CFD work is to investigate the effect of relative roughness pitch (P/e) on average Nusselt number, and THPP in artificially roughened solar air heater having semi-circular sectioned transverse rib roughness on the underside of the absorber plate.

Average Nusselt number for artificially roughened solar air heater is computed by

$$Nu_r = \frac{hD}{k} \tag{5}$$

Where h is convective heat transfer co-efficient.

The average friction factor for artificially roughened solar air heater is computed by

$$f_r = \frac{(\Delta p / l) D}{2 \rho v^2} \tag{6}$$

Where  $\Delta p$  is pressure drop across the duct of an artificially roughened solar air heater.

The Reynolds number is defined as

$$Re = \frac{\rho v D}{\mu} \tag{7}$$

A well-known method of THPP evaluation is proposed by Webb and Eckert (1972) [14] and defined by the following equation

$$THPP = \frac{Nu_r / Nu_s}{(f_r / f_s)^{1/3}} \tag{8}$$

For an enhancement scheme to be viable, the value of this index must be greater than unity.

$Nu_s$  represent Nusselt number for smooth duct of a solar air heater and can be obtained by the Dittus–Boelter equation (McAdams 1942) [14].

$$Nu_s = 0.023 Re^{0.8} Pr^{0.4} \tag{9}$$

$f_s$  represents friction factor for smooth duct of a solar air heater and can be obtained by Blasius equation (Fox, Pritchard, and McDonald 2010) [15].

Blasius equation [30]

$$f_s = 0.0791 Re^{-0.25} \tag{10}$$

## VI. RESULT AND DISCUSSION

CFD computations of heat transfer and fluid flow characteristics in an artificially roughened solar air heater provided with semi-circular sectioned transverse rib roughness on the underside of the absorber plate are performed. The effects of grid density, Reynolds number, and relative roughness pitch (P/e) on the average heat transfer for flow of air in an artificially roughness solar air heater are discussed below. The results have been compared with those obtained in case of smooth duct operating under similar operating conditions to discuss the enhancement of heat transfer on account of artificial roughness.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

## Grid independence test:

To study variation of heat transfer and flow friction with change in grid size, grid independence study is carried out. Five sets of grids (element size) with different sizes are used for the simulation to insure that the results are grid independent. A grid independence test is implemented over grids with different numbers of nodes, 53,468, 57,781, 62,305, 65,100, and 68,519 are used in five steps. It is found that the variation in Nusselt number marginally increases when moving from 65,100 cells to 68,519. Hence, there is no such advantage in increasing the number of cells beyond this value. Thus, the grid system of 65,100 cells is adopted for the current computation.

## Effect of Reynolds number (Re)

Figure 4 shows the effect of Reynolds number on average Nusselt number for relative roughness pitch ( $P/e$ ) of 5 and relative roughness height ( $e/D$ )=0.06. The average Nusselt number is observed to increase with increase in Reynolds number which is due to the increase in turbulence intensity that is caused by the increase in turbulence kinetic energy and turbulence dissipation rate. The heat transfer phenomenon can be observed and described by the contour of turbulence kinetic energy. The contour plot of turbulence kinetic energy is shown in Figure 8. Since the  $k-\epsilon$  models use as the turbulent viscosity, the high values of  $k$  in the flow field leads to high heat transfer predictions. The intensities of turbulence kinetic energy are reduced at the flow field near the rib and wall and a high turbulence kinetic energy 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 9 shows the contour plot of turbulence intensity at a Reynolds number of 10,000 and  $e/D=0.06$  and  $P/e=5$ . The peak value of turbulence intensity is seen on the top of rib front region, and then it decreases with increase in distance from the wall. This can be attributed to the formation of eddies in the vicinity of wall due to the high shear stresses which diffuses in the main flow. Low level of turbulence intensity leads to a low level of heat transfer. Figure 6 shows the contour plot of velocity for Reynolds number of 10,000 and  $e/D=0.06$ ,  $P/e=5$  and it displays stronger vortices because of the presence of semi-circular wire rib roughness, which results in more heat transfer rate. The peak velocity magnitude is occurred at center line of the duct and lowest magnitude is near the wall and between the adjacent ribs. Figure 5 shows the contour plot of static pressure for relative roughness pitch ( $P/e$ )=5 at a fixed value of Reynolds number of 10,000 and relative roughness height ( $e/D$ ) of 0.06. It is found that very low pressure variation is occurred in the duct. Pressure increases only at infront of ribs because working fluid exerts the force on rib roughness. Figure 7 shows the contour plot of static temperature for relative roughness pitch ( $P/e$ )=5 at a fixed value of Reynolds number of 10,000 and relative roughness height ( $e/D$ ) of 0.06. Maximum temperature is occurred at the absorber plate and decreases with increase in distance from absorber plate.

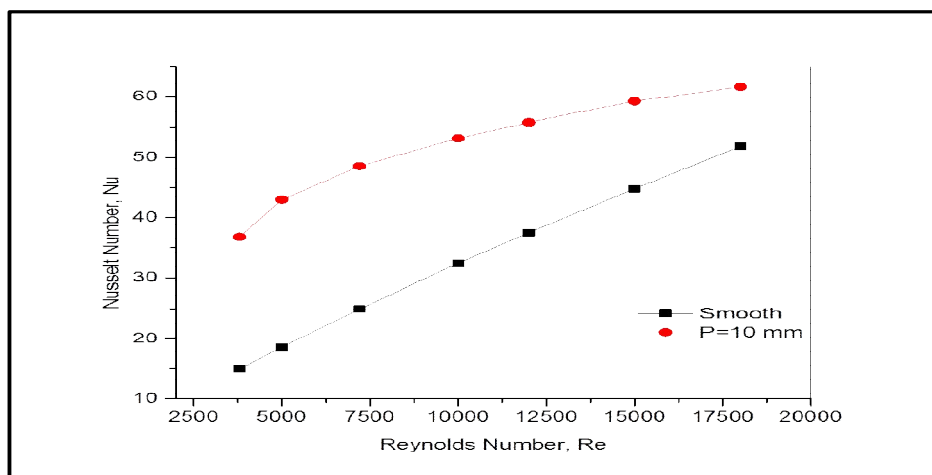


Figure 4: Variation of Nusselt number with Reynolds number for rib height  $e=2$  mm

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

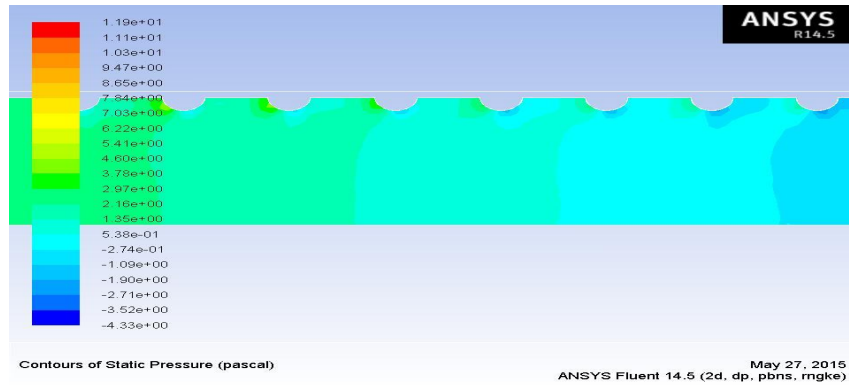


Figure 5: Contour plot for static pressure.

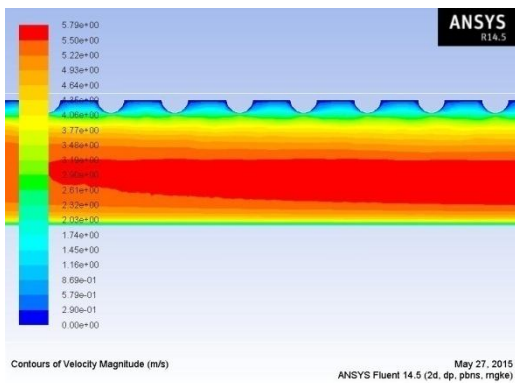


Figure 6: Contour plot for velocity magnitude.

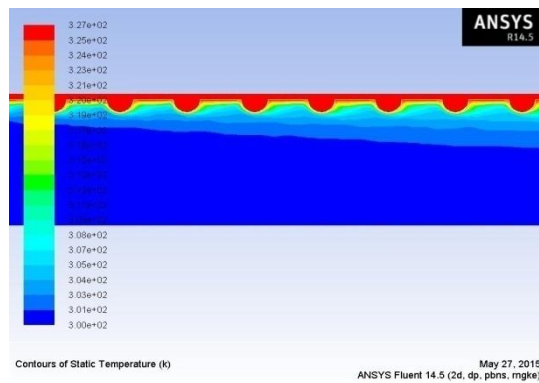


Figure 7: Contour plot for static temperature.

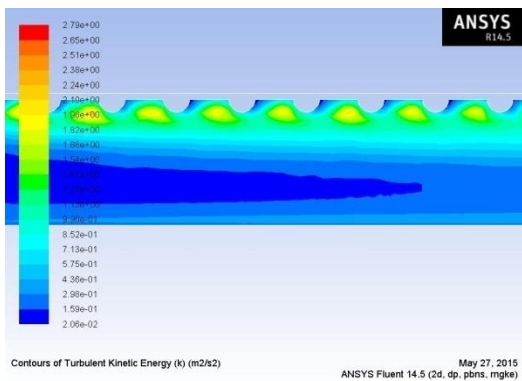


Figure 8: Contour plot for turbulent kinetic energy.

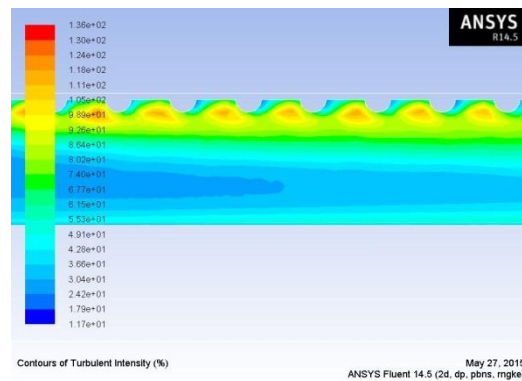


Figure 9: Contour plot for turbulent intensity.



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

Thermal enhancement factor decreases with increase in Reynolds number (Re) and it is found that maximum thermal enhancement factor 2.46 is occurred at Re=3800. Figure 10 shows the variation of Thermal enhancement factor with Reynolds number.

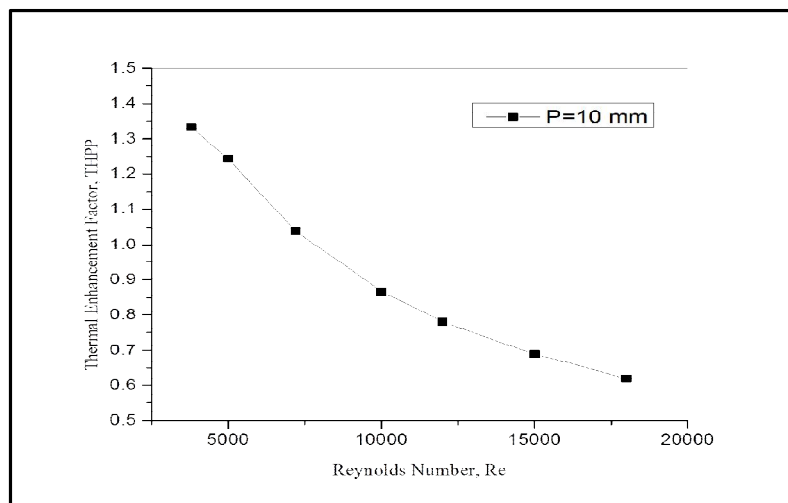


Figure 10: Variation of thermal enhancement factor with Reynolds number.

## VII.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 semi-circular transverse wire rib roughness. The following conclusions are drawn from present analysis:

1. The maximum Nusselt number ratio is found to be 2.462 corresponding to relative roughness pitch (P/e) of 5 at a Reynolds number of 3,800.
2. It is found that the solar air heater roughened with semi-circular sectioned transverse rib roughness on the absorber plate with relative roughness pitch,  $P/e = 5$  provides the better THPP of 1.33 at a Reynolds number of 3,800.
3. The results predicted by the present CFD investigation are much closer to previous experimental results. It can, therefore, be concluded that the present numerical results are demonstrated the validity of the proposed system. Thus, it is possible to establish a validated model for the prediction of heat transfer in artificially roughened solar air heater roughened with semi-circular sectioned transverse rib roughness on the absorber plate.

## REFERENCES

- [1] J.A. Duffie, W.A. Beckman, "Solar engineering of thermal processes", New York: Wiley, 1980.
- [2] J.P. Joule, "On the surface condensation of steam", Philosophical Transactions of the Royal Society of London, 151:133-60; 1861
- [3] W. Nunner, "Heat transfer and pressure drop in rough pipes", vol. 22. VDI-Forsch, 1956, 445-B, English Trans, AERE Lib / Trans; 786; 1958
- [4] J. Nikuradse, "Laws of flow in rough pipes", NACA Technical Memorandum, 1292; 1950.
- [5] D.F. Dippery, R.H. Sabersky, "Heat and momentum transfer in smooth and rough tubes at various Prandtl number", International Journal of Heat and Mass Transfer, 6:329-53; 1963.
- [6] K. Prasad, S.C. Mullick, "Heat transfer characteristics of a solar air heater used for drying purposes", Applied Energy, 13(2):83-93; 1983.
- [7] R. Karw, S.C. Solanki, J.S. Saini, "Thermo-hydraulic performance of solar air heaters having integral chamfered rib roughness on absorber plates", Energy, 26:161-76; 2001.
- [8] R.P. Saini, J. Verma, "Heat transfer and friction factor correlations for a duct having dimple-shaped artificial roughness for solar air heaters", Energy, 33; 1277-87; 2008

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 6, June 2015

- [9] S. Singh, S. Chander, J.S. Saini, "Heat transfer and friction factor correlations of solar air heater ducts artificially roughened with discrete V-down ribs", *Energy*, 36:5053-64; 2011.
- [10] 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*, 25:341-69; 2002.
- [11] B.K. Gandhi, K.M. Singh, "Experimental and numerical investigations on flow through wedge shape rib roughened duct", *The Institution of Engineers (India) Journal-MC*, 90(January):13-8; 2010.
- [12] A.S. Yadav, J.L. Bhagoria, "A CFD Based Heat Transfer and Fluid flow Analysis of a Solar Air Heater Provided with Circular transverse Wire Rib Roughness on the Absorber Plate", *Energy*, 55: 1127-1142; 2013.
- [13] A.S. Yadav, J.L. Bhagoria, "Modeling and simulation of turbulent flow through a solar air heater having a square-sectioned transverse rib roughness on absorber plate", 2013.
- [14] W.H. McAdams, "Heat transmission", New York, McGraw-Hill, 1942.
- [15] W. Fox, P. Pritchard, A. McDonald, "Introduction to fluid mechanics", New York, John Wiley & Sons, page754, 2010.

### BIOGRAPHY



**Kanhaiya Jhariya** Pursuing M.Tech Degree from N.I.T Jamshedpur, India in year 2015. His area of interest in Solar Energy and Thermal Energy.



**Rajeev Ranjan** Pursuing Ph.D from N.I.T Jamshedpur. Completed his M.Tech from Sindri. His area of interest is CFD, Solar Energy and Thermal Energy.



**Dr. M.K. Paswan**  
Professor at the Department Of Mechanical Engg.  
National Institute Of Technology  
(Govt. Of MHRD) Jamshedpur, Jharkhand. India