A REVIEW ON EFFICIENCY PREDICTION OF SOLAR AIR HEATER WITH THE USING OF DIFFERENCE SIZE OF RIBS BY COMPUTATIONAL FLUID DYNAMICS

Santosh Kumar maurya[1], Omshankar Jhariya[2], Anil Verma[3], G.R. Selokar[4]

[1]*Research Scholar, Department of Mechanical Engineering, school of engineering, sssutms, sehore*
[2]*Assistant Professor, Department of Mechanical Engineering, school of engineering, sssutms,sehore,*
[3]*Head of the Department of Mechanical Engineering, school of engineering, sssutms, sehore*
[4]*Registrtar, SSSUTMS, Sehore*

ABSTRACT

This paper presents the study of heat transfer in a rectangular duct of a solar air heater having Different size of square rib roughness on the absorber plate by using Computational Fluid Dynamics (CFD).The effect on the surface heat transfer and velocity were investigated. In this Solar air heater an absorber plate is made of 'Aluminium' and Roughened with difference size ribs due which creates the turbulence in the flow of fluid (air) and increases the heat transfer from the absorber plate to the fluid. The commercial finite-volume based CFD code ANSYS FLUENT 16.0 is used to simulate turbulent airflow through artificially roughened solar air heater. The computations based on the finite volume method with the SIMPLE algorithm have been conducted for the air flow in terms of Reynolds numbers ranging from 3000-18000. 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 temperature & velocity is increasing at the outlet side and also turbulence is increasing due to the different rectangular ribs shape of the duct.

Keywords*-Solar Energy, Solar Air Heater, Heat transfer, CFD*.

---

1. INTRODUCTION

Energy is required to sustain life. Broadly energy resources can be classified into conventional and non-conventional energy resources. The conventional energy resources are soon to deplete in near future. Hence, the quest of mankind is to find alternate energy resources. Non-conventional or alternate energy resources can be divided into renewable and non-renewable energy resources. Renewable sources are those which have the short span of renewal. Although there are many forms of renewable energy resources available to us, solar energy is very user friendly and reaches most of the locations of the planet naturally. Solar energy is the most promising source of energy, available freely and omnipresent. It is indigenous source of energy that provides clean energy. The easiest methodology for making the proper use of solar energy is its conversion to thermal energy using solar collector. Thermal efficiency of solar air heaters is generally considered poor because of low heat transfer capability between absorber plate and air flowing in the duct. In order to make solar air heaters economically viable, their thermal efficiency needs to be improved by enhancing the heat transfer coefficient. In order to attain higher heat transfer coefficient, it is desirable that laminar sub-layer formed during flow of air on surface of absorber plate should be broken and flow at the heat transferring surface is made turbulent. This can be achieved by providing artificial roughness on the surface of absorber plate. Artificial roughness can be produced by several methods such as by wire fixation in the form of transverse continuous ribs, transverse broken ribs, inclined and V-shaped or staggered ribs, rib formation by machining process in the form of chamfered ribs, wedge shaped ribs, combination of different integral rib roughness elements and by using expanded wire mesh. Many experimental investigations have been carried out involving roughness elements of different shapes, sizes and orientations with respect to flow direction.

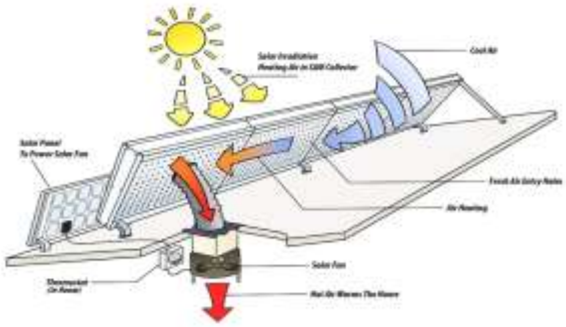

Figure 01. solar air heater

1.1 Solar Air Heater

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, chillies, 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 hosieries, tannery, edible oil, etc.

There are various types of solar air heater, which are used for enhancement of thermal performance:

- Simple at plate collectors
- Finned plate collector
- Corrugated plate collector
- Corrugated plate collector
- Matrix type collector
- Overlapped transparent plate type collector
- Transpiration collector
- Matrix air heater
- Honeycomb porous-bed air heater

1.2 Artificial Roughness

In order to obtain higher value of heat transfer coefficient it is desirable that the air flow over the heat transfer surface is to be made turbulent. However, energy for creating such turbulence has to come from the fan or blower and the excessive turbulence leads to excessive power requirements. It is therefore desirable that the turbulence must be created only in the region which is very close to the heat transfer surface i.e. in the laminar sub-layers where the heat exchange takes place and the flow should not be unduly disturbed so as to avoid excessive friction losses. This can be done by keeping the height of the roughness element small in comparison to the duct dimensions. Although there are several parameters that characterize the arrangement and shape of the roughness, the roughness element height (e) and pitch (P) are the most important. The roughness elements can be two-dimensional ribs or three dimensional

1.3 Performance of Solar Air Heater

The performance of solar air heater is the efficiency of the heater to absorb the solar radiations and transferring the heat absorbed to the air flowing over the absorber plate. The conventional solar air heaters have poor efficiency due to the lower heat transfer between the absorber plate and the air. This is primarily because of the formation of laminar sub layer above the absorber plate. The efficiency of the solar air heater can be improved if somehow this laminar sub layer could be broken.

2. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a computer-based simulation method for analyzing flow of fluid, transfer of heat, and related phenomena such as reactions carried out in chemicals. This project is using CFD for analysis of fluid flow and heat transfer. Some examples of application areas are: aerodynamic lift and drag (i.e. aerofoil's or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analysis carried out in the various industries is used in R&D and manufacture of aircraft, combustion systems, as well as many other industrial products.
It can be advantageous to use CFD over traditional experimental-based analyses, since CFD is very cost efficient and results can be produced at practically no added expense. In this way, parametric studies to optimize equipment are very low cost with CFD when compared to experiments.
This section briefly describes the general concepts and theory related to using CFD to analyze flow of fluid and transfer of heat, as relevant to this project. It begins with a review of the tools needed for carrying out the CFD analyses and the processes required, followed by a summary of the governed equations and turbulence models and finally a discussion of the discretization schemes and algorithms is presented.

2.1 CFD Computational Tools

This section describes the CFD software required for carrying out a simulation for analysis and the process one follows in order to solve a problem using CFD. The hardware required and the three main elements of processing CFD simulations: the pre-processor, processor, and post-processor are described.

Pre-Processor: A pre-processor is used to define the geometry for the computational domain of interest and generate the mesh of control volumes (for calculations). Generally, the finer the mesh in the areas of large changes, the more accurate the solution. Fineness of the grid also determines the computer hardware and calculation time needed. The open-source pre-processor used for this project is called Salomé.

Solver: The solver makes the calculations using a numerical solution technique, which can use finite difference, finite element, or spectral methods. Most CFD codes use finite volumes, which is a special finite difference method. First the fluid flow equations are integrated over the control volumes (resulting in the exact conservation of relevant properties for each finite volume), then these integral equations are discretised(producing algebraic equations through converting of the integral fluid flow equations), and finally an iterative method is used to solve the algebraic equations. (The finite volume method and discretisation techniques are described more in the next sections. Open FOAM CFD code is used for solving the simulations in this project.

Post-Processor: The post-processor provides for visualization of the results, and includes the capability to display the geometry/mesh, create vector, contour, and 2D and 3D surface plots. Particles can be tracked throughout a simulation, and the model can be manipulated (i.e. changed by scaling, rotating, etc.), and all in full color animated graphics. Para View is the open-source post-processor used for this project.

2.2 Problem-Solving with CFD

There are many decisions to be made before setting up the problem in the CFD code. Some of the decisions to be made can include: whether the problem should be 2D or 3D, which type of boundary conditions to use, whether or not to calculate pressure/temperature variations based on the air flow density, which turbulence model to use, etc. The assumptions made should be reduced to a level as simple as possible, yet still retaining the most important features of the problem to be solved in order to reach an accurate solution.

2.3 The Benefits of CFD

- **Insight**
    - Difficult to prototype or test through experimentation
    - Better Details
- **Foresight**
    - Better prediction: In a short time
- **Efficiency**
    - Design better and faster, economical, meet environmental regulations and ensure industry compliance.
    - CFD analysis leads to shorter design cycles and your products get to market faster.
    - In addition, equipment improvements are built and installed with minimal downtime.

- CFD is a tool for compressing the design and development cycle allowing for rapid prototyping.

2.4 CFD governing equation

In this section summarisation of the governing equations are given that are used in to solve the fluid flow and heat transfer mathematically. This solution is based on the principle of conservation of mass, momentum, and energy. CFD Computational equations are given below:-

This equation is a mass conservation equation:-

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

This is the rate of change of momentum equation:-

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_j}{\partial x_i}\right) - \frac{\partial p}{\partial x_j}.$$

This is the Energy equation;-

$$\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_i}\left(\frac{k}{C_p}\frac{\partial u_i}{\partial x_i}\right).$$

Where velocity vector u (with components of the velocities *u*, *v*, and *w* in the direction of *x*, *y*, and *z*), pressure P, density ρ, viscosity μ, temperature T, and heat conductivity k. The changes in these given fluid properties can occur over space because this project works on only steady state condition.

3. LITERATURE REVIEW

[1] A. Yadav et al. employed triangular shaped rib roughness on the absorber plate to predict heat transfer behaviour of an artificially roughened solar air heater by adopting CFD approach. ANSYS FLUENT 12.1 and RNG - turbulence model were employed in their simulation.From1.4 to 2.7 times enhancement in the Nusselt number was observed as compared to smooth solar air heater.

[2]A. Yadav et al. carried out CFD investigation of an artificially roughened solar air heater having circular transverse wire rib roughness on the absorber plate. A two-dimensional CFD simulation was performed using ANSYS FLUENT 12.1 code as a solver with RNG turbulence model. The maximum value of thermal enhancement factor was reported to be 1.65 for the range of parameters investigated.

[3] Chaube et al. conducted two dimensional CFD-based analysis of an artificially roughened solar air heater having ten different ribs shapes, namely, rectangular, square, chamfered, triangular, and so forth, provided on the absorber plate. CFD code, FLUENT 6.1 and SST -turbulence model were used to simulate turbulent airflow. The best performance was found with rectangular rib of size 3 × 5 mm, and CFD simulation results were found to be in good agreement with existing experimental results.

[4] S. Kumar et al. performed three-dimensional CFD-based analysis of an artificially roughened solar air heater having arc shaped artificial roughness on the absorber plate. FLUENT 6.3.26 commercial CFD code and Renormalization group (RNG) -turbulence model were employed to simulate the fluid flow and heat transfer. Overall enhancement ratio with a maximum value of 1.7 was obtained, and results of the simulation were successfully validated with experimental results.

[5] S. Karmare et al. carried out CFD investigation of an artificially roughened solar air heater having metal grit ribs as roughness elements on the absorber plate. Commercial CFD code FLUENT 6.2.16 and Standard -turbulence were employed in the simulation. Authors reported that the absorber plate of square cross-section rib with 580angle of attack was thermos-hydraulically more efficient.

[6] Sethi et al. carried out an experimental investigation to analyse the effects of artificial roughness on heat transfer and frictional characteristics of rectangular duct having dimple shaped elements arranged in angular fashion (arc) as roughness elements. Range of parameters considered were aspect ratio (W/H) of 11, relative roughness pitch (p/e) 10-20, relative roughness height (e/Dh) 0.021- 0.036, arc angle (α) 45-75° and Reynolds number from 3600-18,000. The value of maximum Nusselt number was found to be 135 for values of (p/e) value of 10, arc angle (α) of 60° and (e/Dh) value of 0.03
[7] Gandhi and Singh (2010) 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.

[8] Yadav and Bhagoria (2013) 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.

[9] Singh et al. carried out an experimental investigation of thermohydraulic performance, heat transfer and efficiency, of solar air heater. The range of variation of system and operating parameters was investigated within the limits and against variation of Reynolds number and showed that the relative efficiency was maximum for Reynolds Number equal to 22300.

[10] Kumar et al. carried out an experimental investigation of heat transfer and friction in the flow of air in rectangular ducts having multi v-shaped rib with gap roughness on one broad wall. The investigation encompassed Reynolds number (Re) from 2000 to 20,000.

[11] Saini and Verma carried out an experimental study to investigate the effect of roughness and operating parameters on heat transfer and friction factor in a roughened duct provided with dimple-shape roughness geometry. Correlations for Nusselt number and friction factor have been developed for solar air heater duct provided such artificial roughness geometry. The maximum value of Nusselt number has been found to be 11,600 corresponding to relative roughness height (e/D) of 0.0379 and relative pitch (p/e) of 10. While minimum value of friction factor has been found to be 0.05 corresponding to relative roughness height (e/D) of 0.0289 and relative pitch (p/e) of 10.

[12] Jaurker et al. carried out an experimental investigation on the heat transfer and frictional characteristics of rib grooved artificial roughness. The Reynolds number range used from 3000 to 21,000; relative roughness height 0.0181–0.0363; relative roughness pitch 4.5–10.0, and groove position to pitch ratio 0.3–0.7. The rib-grooved duct with relative roughness pitch (p/e) of 6.0 and position of groove to pitch ratio (g/p) of 0.4 provides the maximum value of the Nusselt number in the order of 2.75 times of the smooth duct and 1.57 times ofribbed duct whereas for ribbed duct with similar rib height and rib spacing provides the Nusselt number values of the order of 1.7 times of the smooth duct for the range of experimentation, the maximum value of the friction factor for rib-grooved duct was 3.61 times that of the smooth duct and 1.17 times that of ribbed duct whereas a ribbed duct with similar rib height and rib spacing results in the friction factor value of the order of 3.00 times that of the smooth duct.

[13] Lanjewar et al. carried out an experimental investigation of heat transfer and friction factor characteristics of rectangular duct roughened with W-shaped ribs on its underside on one broad wall. Range of parameters considered were Duct has width to height ratio (W/H) of 8.0, relative roughness pitch (p/e) of 10, relative roughness height (e/Dh) 0.018-0.03375and angle of attack of flow (α) 30-75°. For relative roughness height of 0.03375 and at angle of attack of 60°, W-shape ribs enhanced value of Nusselt number by 2.21 times over smooth plate at Reynolds number of 14,000.

[14] Bekele et al. investigated the performance of conventional solar air heater for the range of Reynolds number (Re) from 2100 to 30,000, relative obstacle height (e/H) from 0.25 to 0.75, relative obstacle longitudinal pitch (Pl/e) from 3/2 to 11/2, relative obstacle transverse pitch (Pt/b) from 1 to 7/3 and the angle of incidence (α) varied from 30° to 90°.For angle of incidence of 30°, 60° and 90°, the maximum thermohydraulic performance was found to be 3.89, 3.29 and 2.09 respectively.

[15] Alam et al. investigated experimentally the effect of geometrical parameters of the V-shaped perforated blocks on heat transfer and flow characteristics of rectangular duct. Range of parameters considered were relative blockage height (e/H) of 0.4–1.0, relative pitch ratio (P/e) of 4–12 and open area ratio (β) of 5–25% at a fixed angle of attack (α) of 60° for Reynolds number 2000-20,000. The maximum enhancement in Nusselt number and friction factor were found to be 6.76 and 28.84 times to that of smooth duct, respectively.

4. CFD APPROACH

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

The 2-dimensional solution domain used for CFD analysis has been generated as shown in fig.1. The solution domain is a horizontal duct with square shaped ribs alternatively placed over the absorber plate at the underside of the top of the duct while other sides are considered as smooth surfaces. In the present analysis, a similar flow domain used for the predictions has been selected as per the details given by a researcher. Complete duct geometry is divided into three sections, namely, entrance section, test section and exit section. A short entrance length is chosen because for a roughened duct. The exit section is used after the test section in order to reduce the end effect in the test section.

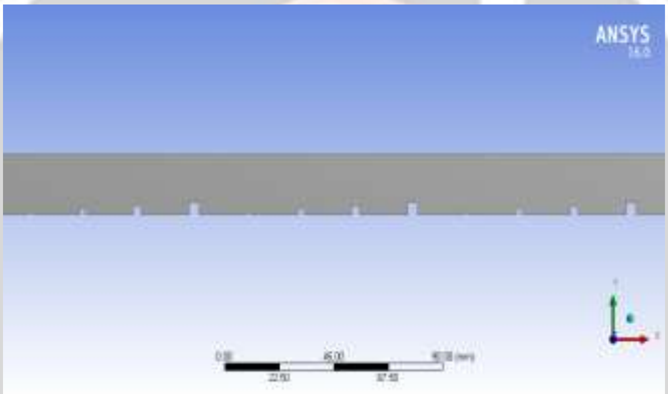

Fig. 1 Square shaped ribs 2-d model

4.1 Basic equations for solar air heater

The useful heat gain of the air is cal culated as:

$$Q_U = mC_p(T_{fo} - T_{fi})$$

Where m is mass flow rate of air through the test duct (kg/sec), Cp is specific heat of air, T_{fo} is fluid temperature at exit of test duct, T_{fi} is fluid temperature at inlet of test duct.

The heat transfer coefficient for the test section is:

$$h = Q_U/A\ (T_{pm} - T_{fm})$$

Where, T_{pm} is the average value of the heater surface temperatures, T_{fm} is the average air temperature in the duct = $(T_{fo} - T_{fi})/2$

The Nusselt number: N_u= h. D_h /K_{air}

Where $D_h$hydraulic mean diameter of test duct, h is is convective heat transfer coefficient, K_{air} is thermal conductivity of air.

Hydraulic diameter: $D_h=\frac{4WH}{2(W+H)}$

Where W is the width of the duct and H is the height of the duct.

Reynolds number: $R_e = \frac{\rho V D}{\mu}$
Where $\mu$the dynamic viscosity of the fluid is, ρ is the density of the fluid and v is the velocity of fluid.

4.2 Computational Domain

Duct height (H) = 20 mm
Rib height (e) = 1, 2, 3&4 mm (square rib)
Inlet length=225 mm
Uniform heat at bottom surface=1100 W/m2 (the surface below a rib is considered Insulated)
Aspect ratio (AR) = 5
Pitch p = 20 mm
Length of test section = 250 mm
Outlet length = 115 mm
Width of duct = 100 mm

The following assumptions are imposed for the computational analysis.
(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 small.
(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.

4.3 Mesh Generation
After defining the computational domain, uniform meshing is done by rectangular elements. 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. Fig. 2 shows the meshing.

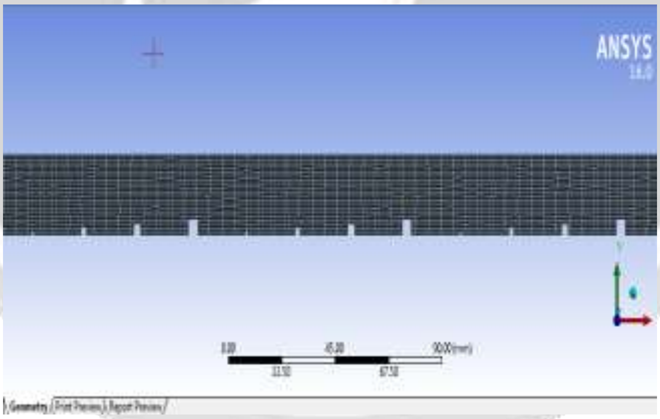

Fig. 2 Meshing Square shaped ribs 2-d model

4.4 Boundary condition

EDGE POSITION	NAME	TYPE
Left	Duct inlet	VELOCITY_INLET
Right	Duct outlet	PRESSURE_OUTLET
Top	Top surface	WALL
Bottom edge-1	Inlet length	WALL
Bottom edge-2	Solar plate	WALL

Bottom edge-3	Outlet length	WALL
Internal edges of rectangular	Tabulator	WALL

Table 1. Boundary conditions

FLUENT Version 16.0 is used as a solver with RNG kepsilon turbulence model. The modelled turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations for Renormalization-group (RNG) k-ε model.

5. RESULT AND DISCUSSION

The current simulation that controls equations of continuity, speed and energy is solved with finite volume method in stable-state rule. The numerical method used in this study is a segregated solution algorithm with a finite volume-based technique. The governing equations are solved using the commercial CFD code, ANSYS Fluent 16.0. A second-order upwind scheme is chosen for energy and momentum equations. The SIMPLE algorithm is chosen as scheme to couple pressure and velocity. A uniform air velocity is introduced at the inlet while a pressure outlet condition is applied at the outlet. Adiabatic boundary condition has been implemented over the bottom duct wall while constant heat flux condition is applied to the upper duct wall of test section. Fig. 3 shows the temperature Contour for the squared shape of ribs inserted in a solar air heater duct. The patterns of temperature contour at regions behind and ahead of the rib illustrate the overall temperature field and the degree of heat transfer. CFD predicts temperature contour pattern better at the regions ahead of the rib.

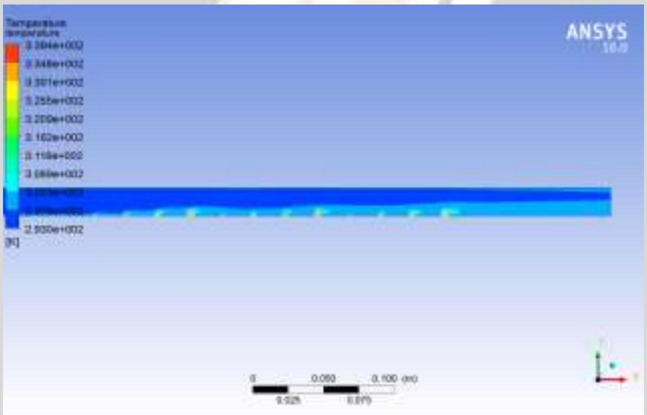

Fig. 3 Shows the temperature Contour for the squared shape of ribs.

Fig. 4 shows the contour of stream function for the different square shape of ribs inserted in a solar air heater duct. An observation of stream function contours reveals that vortex formation at top of the rib surface provides rolling action to the flow and hence reduces the friction.

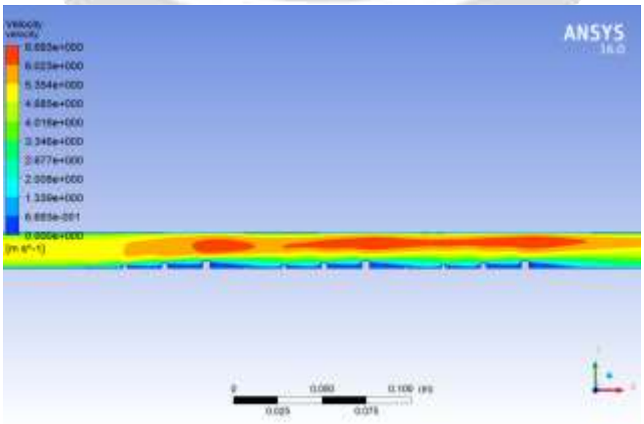

Fig. 4 shows the Velocity of stream function for the different square shape of ribs

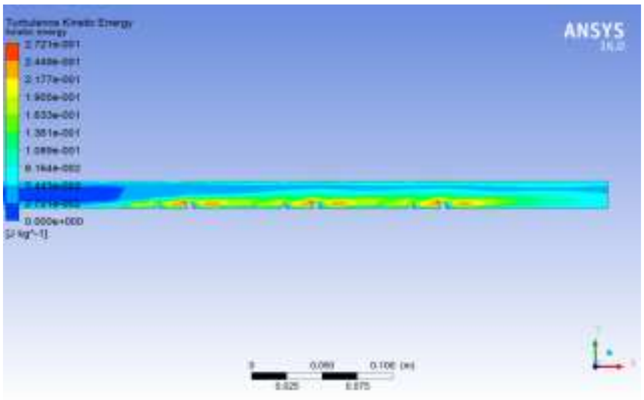

Fig. 5 shown the turbulence kinetic energy

Graph 1 shows the temperature variation from inlet to outlet of the duct. In this case the air entering the duct at a temperature of 293K. It is observed from the plots that the temperature at outlet is more than inlet temperature.

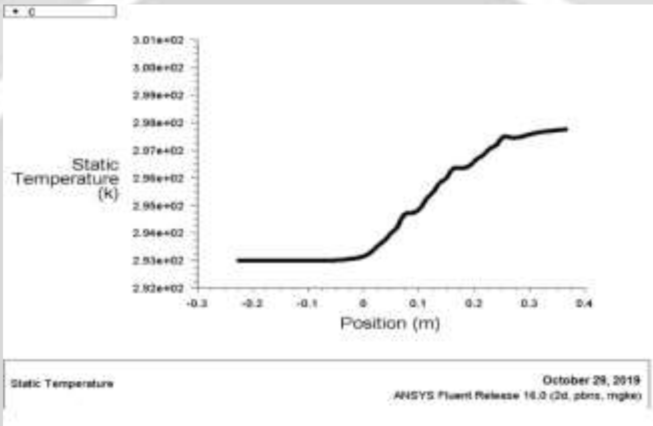

Fig.1 Graph shows the temperature variation from inlet to outlet of the duct

Graph 2 shows the velocity variation from the surface to the top and inlet to outlet of the duct. In this case the air entering the duct at a velocity of 1.5 m/s. It is observed from the plots that the velocity at all location is more than inlet velocity and the outlet velocity is also increasing.

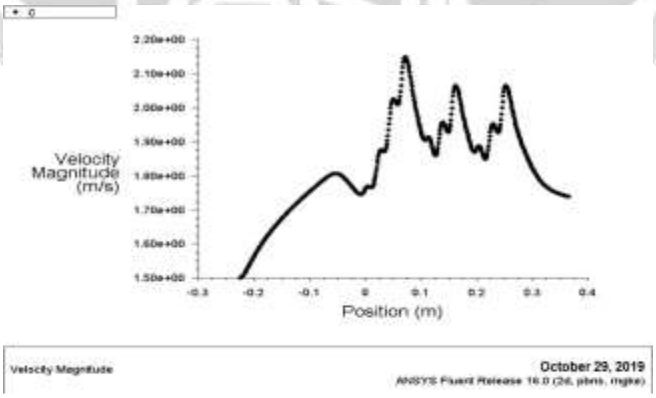

Fig. 2 Graph shows the velocity variation from the surface to the top and inlet to outlet of the duct

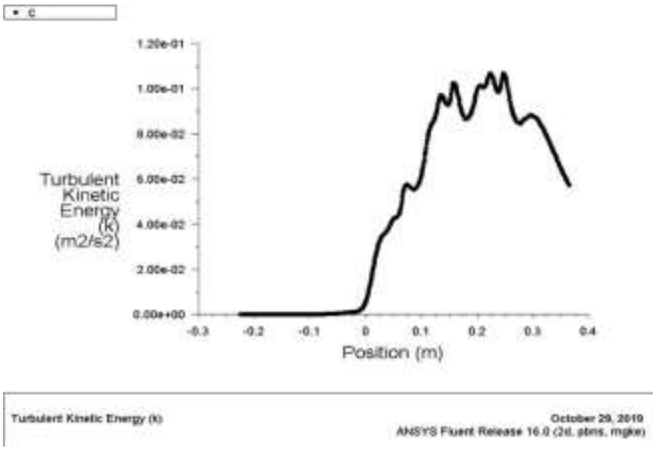

Fig. 3 Graph shows the turbulent kinetic energy of the duct.

6. CONCLUSION

In this present investigation, a numerical prediction has been conducted to study heat transfer and flow friction behaviours of a rectangular duct of a solar air heater having rectangular rib roughness on the absorber plate. The main conclusions are:

1. There is no doubt that a major focus of CFD analysis of solar air heater is to enhance the design process that deals with the heat transfer and fluid flow.
2. In recent years CFD has been applied in the design of solar air heater. The quality of the solutions obtained from CFD simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behaviour and performance of a solar air heater.
3. The air velocity increases at outlet compare to inlet velocity.
4. The maximum turbulence occurred in solar air heater with different size rectangular rib roughness compare to equal size of ribs.
5. The air temperature is increases at outlet.

REFERENCES

[1] Yadav A. S. and Bhagoria J. L., “A CFD analysis of a solar air heater having triangular rib roughness on the absorber plate,” International Journal of ChemTech Research, vol. 5, no. 2, pp. 964–971, 2013.

[2] Yadav A. S. and Bhagoria J. L., “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, vol. 55, pp. 1127–1142, 2013.

[3] Chaube A., Sahoo P. K., and Solanki S. C., “Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater,” RenewableEnergy, vol. 31, no. 3, pp. 317–331, 2006.

[4] Kumar S. and Saini R. P., “CFD based performance analysis of a solar air heater duct provided with artificial roughness,” Renewable Energy, vol. 34, no. 5, pp. 1285–1291, 2009.

[5] Karmareanda S. V. Tikekar N, “Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD,” Solar Energy, vol. 84, no. 3, pp. 409–417, 2010.

[6] Sethi M., Varun, Thakur N.S., “Correlations for solar air heater duct with dimple shape roughness elements on absorber plate”, Solar Energy, Vol. 86, pp. 2852 – 2861.

[7] 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.

[8] 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.

[9] Singh A.P., Varun, Siddhartha, (2014), "Effect of artificial roughness on heat transfer and friction characteristics having multiple arc shaped roughness element on the absorber plate", Solar Energy, Vol. 105, pp. 479 – 493.

[10] Kumar A., Saini R.P., Saini J.S., "Development of correlations for Nusselt number and friction factor for solar air heater with roughened duct having multi V-shaped with gap rib as artificial roughness", Renewable Energy, Vol.58,pp.151–163.

[11] Saini R.P., Verma J., (2008), "Heat transfer and friction factor correlations for a duct having dimple shaped artificial roughness for solar air heaters", Energy, Vol. 33, Issue 8, pp. 1277 – 1287.

[12] Jaurker A.R., Saini J.S., Gandhi B.K., (2006), "Heat transfer and friction characteristics of rectangular solar air heater duct using rib-grooved artificial roughness", Solar Energy, Vol. 80, Issue 8, pp. 895 – 907.

[13] Lanjewar A., Bhagoria J.L., Sarviya R.M., (2011), "Heat transfer and friction in solar air heater duct with W-shaped rib roughness on absorber plate", Energy, Vol. 36, pp. 4531 – 4541.

[14] Bekele A., Mishra M., Dutta S., (2014), "Performance characteristics of solar air heater with surface mounted obstacles", Energy Conversion and Management,Vol. 85, pp.603–611.

[15] Alam T., Saini R.P., Saini J.S., (2014), "Experimental investigation on heat transfer enhancement due to V-shaped perforated blocks in a rectangular duct of solar air heater", Energy Conversion and Management, Vol.81,pp.374– 383.

[16] El-Sebaii A.A., Aboul-EneinS.,Ramadan M.R.I., Shalaby S.M., Moharram B.M., (2011), "Investigation of thermal performance of- Double pass-flat and V-corrugated plate solar air heaters", Energy, Vol. 36, pp. 1076 – 1086.