



# International Journal of Emerging Technologies in Computational and Applied Sciences (IJETCAS)

[www.iasir.net](http://www.iasir.net)

## CFD Based Performance Analysis of Artificially Roughened Solar Air Heater

Arun Kumar Yadav; Amitesh Paul; G. R. Selokar  
Mechanical Engineering Department

Sri Satya Sai Institute of Science and Technology, Sehore, MP 466001  
INDIA

**Abstract:** solar air heaters, because of their simplicity are cheap and most widely used collection devices of solar energy, has great potential for low temperature applications, particularly for drying of agricultural products. The thermal efficiency of a solar air heater is significantly low because of the low value of the convective heat transfer coefficient between the absorber plate and the air, leading to high absorber plate temperature and high heat losses to the surroundings. This paper presents the study of heat transfer in a solar air heater by using Computational Fluid Dynamics (CFD). The effect of Reynolds number on Nusselt number and friction factor are investigated. A commercial finite volume package ANSYS FLUENT 12.1 is used to analyze and visualize the nature of the flow across the duct of a solar air heater. CFD simulation results are found to be in good agreement with experimental results.

**Keywords:** Solar Energy, Solar Air Heater, Heat transfer, Pressure Drop, CFD

### I. Introduction

A solar collector absorbs incident solar radiations and transforms them into useful heat for heating the collector fluid such as water and air. Solar air heaters, being inherently simple and cheap, are most widely used collection devices. Solar air heaters find several applications in space heating, seasoning of timber and crop drying. Efficiency of flat plate solar air heater is low because of low convective heat transfer coefficient between absorber plate and flowing air which in turn increases the absorber plate temperature, leading to higher heat losses to the environment. The low value of heat transfer coefficient is generally attributed to the presence of a viscous sublayer, which can be broken by providing artificial roughness on the heat-transferring surface. Due to this artificial roughness, turbulent flow is created which helps in better enhancement of heat transfer.

Bhushan and Singh [1] presented data on the influence of Reynolds number on heat transfer coefficient distribution on the surface having staggered array of protrusion geometry. In some cases, enhancement in heat transfer rate is about 2.5 times than smooth surface value over a range of Reynolds number. Varun et al. [2] presented the experimental investigation on combination of geometries (i.e. transverse and inclined discrete ribs) which give better heat transfer enhancement as compared to single roughened geometry. An attempt has also been carried out to optimize the thermal efficiency using Taguchi method under similar condition [3]. Sahu and Bhagoria [4] experimentally investigated the effect of 90° broken ribs as roughness elements and found that thermal efficiency lies in the range of 51–83.5%. Karwa et al. [5] investigated the effect of rib chamfered angle (°), duct aspect ratio on heat transfer and friction factor using integral chamfered ribs. Chamfer angle of 15° gave highest friction factor as well as Stanton number.

Solar air heaters, because of their simplicity are cheap and most widely used collection devices of solar energy, has great potential for low temperature applications, particularly for drying of agricultural products. The thermal efficiency of a solar air heater is significantly low because of the low value of the convective heat transfer coefficient between the absorber plate and the air, leading to high absorber plate temperature and high heat losses to the surroundings. It has been found that the main thermal resistance to the heat transfer is due to the formation of a laminar sub-layer on the absorber plate heat-transferring surface. An artificial roughness on the heat transfer surface in the form of projections mainly creates turbulence near the wall or breaks the laminar sub-layer and thus enhances the heat transfer coefficient. However, the energy for creating such turbulence has to come from the fan or blower. It is therefore; the turbulence must be created only in the region very close to the heat transferring surface, i.e. in the laminar sub-layer only. Several investigators have attempted to design a roughness element, which can enhance convective heat transfer with minimum pumping losses. Ravigururajan and Bergles [6] used four types of roughness, namely semicircular, circular, rectangular and triangular ribs to develop the general statistical correlations for heat transfer and pressure drop for single phase turbulent flow in internally ribbed surface. Liou and Hwang [7] tested three shapes of rib roughness, namely square, triangular and semicircular ribs to study the effect of rib shapes on turbulent solar system is never designed to meet the complete energy demand of the application for which it is installed. Providing a system to supply 100 percent of the energy requirement at all times would result in a system which is oversized for most of the time. The resulting initial cost would make

the investment uneconomical. A solar system is thus often used in conjunction with an auxiliary system using conventional energy. The auxiliary sources help to meet unusual high demand situations. It also takes care of situations which arise when solar energy is not available in adequate measure because of adverse weather conditions.

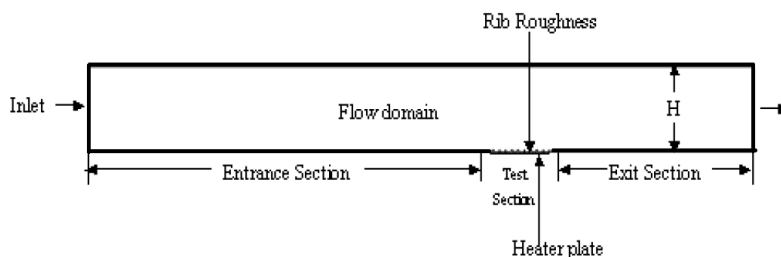
Chaubey *et al.* [8] conducted a CFD based analysis of heat transfer and fluid flow characteristics of an artificially roughened solar air heater. Kumar and Saini [9] performed a CFD analysis of fluid flow and heat transfer characteristics of a solar air heaters having arc shaped artificial roughness on the absorber plate. Karmare and Tikekar [10] carried out CFD investigation of fluid flow and heat transfer in a solar air heater duct with metal grit ribs as roughness elements on the absorber plate. Yadav and Bhagoria [11] conducted a numerical prediction to study only heat transfer behavior of a rectangular duct of a solar air heater having triangular rib roughness on the absorber plate. Yadav and Bhagoria [12] presented the numerical prediction of fluid flow and heat transfer in a conventional solar air heater by CFD. A commercial finite volume package ANSYS FLUENT 12.1 was used to analyze the nature of the flow across the duct of a conventional solar air heater. Yadav and Bhagoria [13] presented a detailed literature survey about different CFD investigations on artificially roughened solar air heater. The aim of our study is to improve the prediction of the flow in the solar air heater. A near-wall function for TKE will be implemented in Computational Fluid Dynamics code Fluent. Second order upwind and SIMPLE algorithm were used to discretize the governing equations. The FLUENT software solves the following mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem.

## II. CFD Modeling and Analysis

Computational Fluid Dynamics (CFD) is the science of determining numerical solution of governing equation for the fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest. The equation can represent steady or unsteady, Compressible or Incompressible, and in viscid or viscous flows, including non-ideal and reacting fluid behavior. The particular form chosen depends on intended application. The state of the art is characterized by the complexity of the geometry, the flow physics, and the computing time required obtaining a solution.

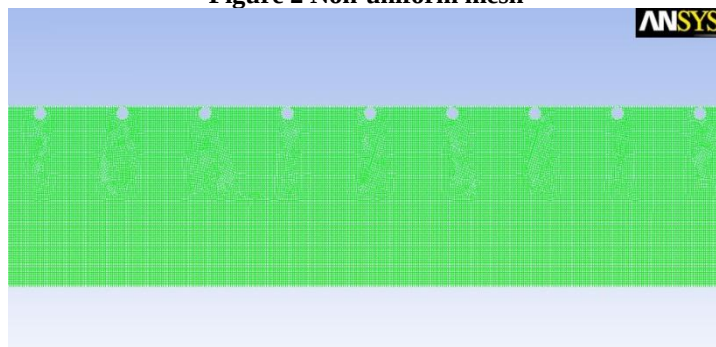
The 2-D computational domain used for CFD analysis having the height (H) of 20 mm and width (W) 100 mm and length of 461 mm as shown in Fig. 1. In the present analysis, a 2-dimensional computational domain of artificially roughened solar air heater has been adopted which is similar to computational domain of Yadav and Bhagoria [14].

**Figure 1 2-D computational domain**



After defining the computational domain, non-uniform mesh is generated. 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).

**Figure 2 Non-uniform mesh**



After generating mesh, boundary conditions have been specified. We will first specify that the left edge is the duct inlet and right edge is the duct outlet. Top edge is top surface and bottom edges are inlet length, outlet length and solar plate. All internal edges of rectangle 2D duct are defined as turbulator wall.

Meshing of the domain is done using ANSYS ICEM CFD V12.1 software. A non-uniform mesh with very fine mesh size is used to resolve the laminar sub-layer. Since low-Reynolds-number turbulence models are employed,

the grids are generated so as to be very fine. Present non-uniform quadrilateral mesh contained 161,568 quad cells with non-uniform quad grid of 0.22 mm cell size. This size is suitable to resolve the laminar sub-layer. For grid independence test, the number of cells is varied from 103,231 to 197,977 in five steps. It is found that after 161,568 cells, further increase in cells has less than 1% variation in Nusselt number and friction factor value which is taken as criterion for grid independence.

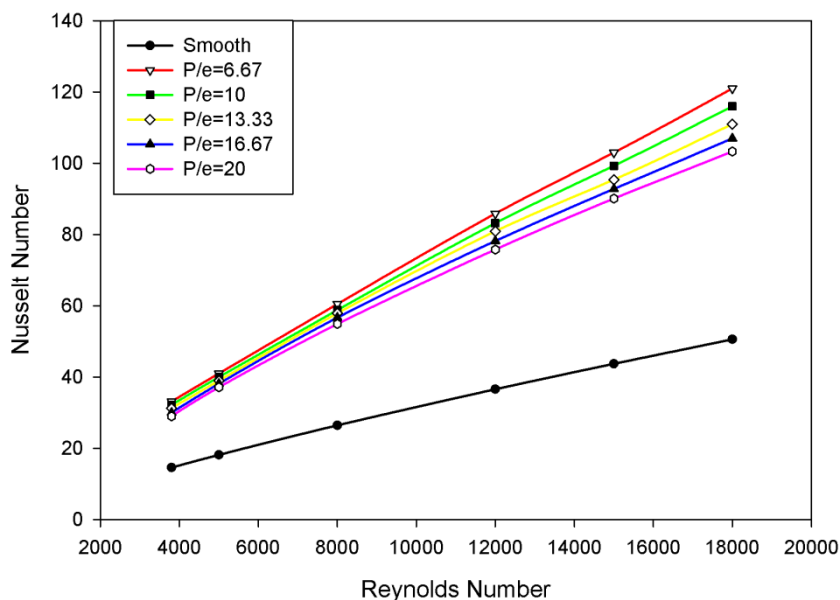
To select the turbulence model, the previous experimental study is simulated using different low Reynolds number models such as Standard  $k-\omega$  model, Renormalization-group  $k\epsilon$  model, Realizable  $k\epsilon$  model and Shear stress transport  $k-\omega$  model. The results of different models are compared with experimental results. The RNG  $k-\epsilon$  model is selected on the basis of its closer results to the experimental results.

The working fluid, air is assumed to be incompressible for the operating range of duct since variation is very less. The mean inlet velocity of the flow was calculated using Reynolds number. Velocity boundary condition has been considered as inlet boundary condition and outflow at outlet. Second order upwind and SIMPLE algorithm were used to discretize the governing equations. The FLUENT software solves the following mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem.

### III. Results And Discussion

Results are presented in form of graphs, representing the average Nusselt number at different Reynolds numbers, and in form of temperature and velocity contours at particular sections for a fixed Reynolds number. Fig. 3 shows the effect of Reynolds number on average Nusselt number for different values of relative roughness pitch ( $P/e$ ) and fixed value of relative roughness height ( $e/D$ ).

**Figure 3 Nusselt number vs Reynolds number**

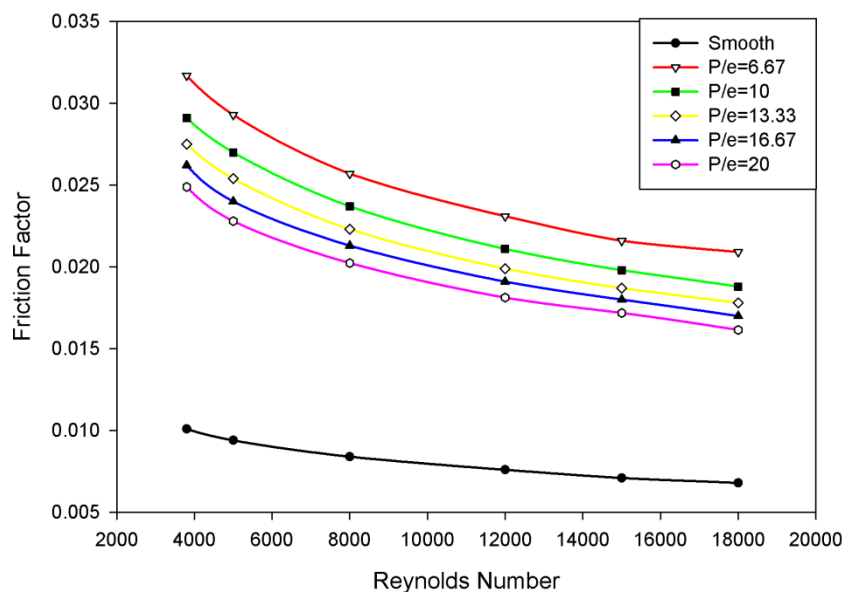


Effect of the relative roughness pitch ( $P/e$ ) on heat transfer is also shown typically in Fig. 3. It can be seen that the enhancement in heat transfer of the roughened duct with respect to the smooth duct also increases with an increase in Reynolds number. It can also be seen that Nusselt number values decrease with the increase in relative roughness pitch ( $P/e$ ) for fixed value of relative roughness height ( $e/D$ ). This is due to the fact that with the increase in relative roughness pitch, number of reattachment points over the absorber plate reduces. The roughened duct with relative roughness pitch of 6.67 provides the highest Nusselt number ( $Nu=121$ ) at a Reynolds number of 18000. The roughened duct with relative roughness pitch ( $P/e$ ) of 20 provides the lowest Nusselt number ( $Nu=29$ ) at a Reynolds number of 3800. The maximum enhancement of average Nusselt number is found to be 2.31 times that of smooth duct for relative roughness pitch of 6.67 at a Reynolds number of 18000.

Fig 4 shows that the friction factor decreases with the increasing values of the Reynolds number in all cases as expected because of the suppression of laminar sub-layer for fully developed turbulent flow in the duct. It can also be seen that friction factor values decreases with the increase in relative roughness pitch ( $P/e$ ) for fixed value of roughness height, attributed to less interruptions in the flow path. The maximum and minimum value of friction factor occurs at relative roughness pitch ( $P/e$ ) of 6.67 and 20 respectively for the range of parameters investigated. It is also observed that the maximum enhancement of average friction factor is found to be 3.14

times that of smooth duct for relative roughness pitch of 6.67 at a Reynolds number of 3800. Figure 4 shows the effect of Reynolds number on average friction factor for different values of relative roughness pitch and fixed value of roughness height.

**Figure 4 Friction factor vs Reynolds number**



#### IV. Conclusion

The effect of relative roughness pitch and Reynolds number on the heat transfer coefficient and friction factor have been studied. CFD Investigation has been carried out in medium Reynolds number flow ( $Re = 3800-18,000$ ). Based on the CFD investigations of heat and fluid flow in a rectangular duct with protrusions as roughness element on one broad wall subjected to a uniform heat flux the following conclusions were drawn.

1. The Renormalization-group (RNG)  $k-\epsilon$  turbulence model predicted very close results to the experimental results, which yields confidence in the predictions done by CFD analysis in the present study. RNG  $k-\epsilon$  turbulence model has been validated for smooth duct and grid independence test has also been conducted to check the variation with increasing number of cells.
2. Average Nusselt number increases with an increase of Reynolds number. The maximum value of average Nusselt number is found to be 121 for relative roughness pitch of 6.67 at a higher Reynolds number, 18,000. The maximum enhancement of average Nusselt number is found to be 2.31 times that of smooth duct for relative roughness pitch of 6.67 at a Reynolds number, 18,000.
3. Average friction factor decreases with an increase of Reynolds number. The maximum value of average friction factor is found to be 0.0317 for relative roughness pitch of 6.67 at a lower Reynolds number, 3800. The maximum enhancement of average friction factor is found to be 3.14 times that of smooth duct for relative roughness pitch of 6.67.

#### V. References

- [1] B. Bhushan, R. Singh, Nusselt number and friction factor correlations for solar air heater duct having artificial roughened absorber plate, *Solar Energy* 85 (2011) 1109–1118.
- [2] Varun, R.P. Saini, S.K. Singal, Investigation of thermal performance of solar air heater having roughness element as a combination of inclined and transverse ribs on the absorber plate, *Renewable Energy* 33 (2008) 1398–1405.
- [3] Varun, Amar Patnaik, R.P. Saini, S.K. Singal, Siddhartha, Performance prediction of solar air heater having roughened duct provided with transverse and inclined ribs as artificial roughness, *Renewable Energy* 34 (2009) 2914–2922.
- [4] M.M. Sahu, J.L. Bhagoria, Augmentation of heat transfer coefficient by using 90 broken ribs on absorber plate of solar air heater, *Renewable Energy* 30 (2005) 2057–2073.
- [5] R. Karwa, S.C. Solanki, J.S. Saini, Heat transfer coefficient and friction factor correlation for the translational flow regime in rib roughened rectangular ducts, *International Journal of Heat and Mass Transfer* 42 (1999) 1597–1615.
- [6] Ravigururajan T. S. and Bergles A. E. (1996). “Development and Verification of General Correlations for Pressure Drop and Heat Transfer in Single-Phase Turbulent Flow in Enhanced Tubes”. *Experimental Thermal and Fluid Science*; 13: pp. 55-70.
- [7] Liou T. M. and Hwang J. J. (1992). “Turbulent heat transfer augmentation and friction in periodic fully developed channel flows”. *ASME/Journal of heat transfer*. Vol. 114, pp 56-64.

- [8] A. Chaube, P.K. Sahoo, S.C. Solanki, Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater, *Renewable Energy* 31 (2006) 317-31.
- [9] S. Kumar, R.P. Saini, CFD based performance analysis of a solar air heater duct provided with artificial roughness, *Renewable Energy* 34 (2009) 1285-91.
- [10] S.V. Karmare, A.N. Tikekar, Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD, *Solar Energy* 84 (2010) 409–17.
- [11] A.S. Yadav, J.L. Bhagoria, A CFD analysis of a solar air heater having triangular rib roughness on the absorber plate, *International Journal of ChemTech Research* 5(2) (2013) 964-71.
- [12] A.S. Yadav, J.L. Bhagoria, A CFD based heat transfer and fluid flow analysis of a conventional solar air heater, *Journal of Engineering Science and Management Education* (2013), (Article in press).
- [13] A.S. Yadav, J.L. Bhagoria, Heat transfer and fluid flow analysis of solar air heater: a review of CFD approach, *Renewable and Sustainable Energy Reviews* 23 (2013) 60-79.
- [14] 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, Elsevier*, 55 (2013) 1127-1142.