



International Journal of Engineering, Business and Enterprise Applications (IJEBA)

www.iasir.net

Cfd Study on Heat Transfer through Solar Air Heater Absorber Plate with Surface Roughness

Dharam Singh¹, Dr. V.N. Bartaria²
^{1,2} Mechanical Engineering Department
Lakshmi Narain College of Technology
Bhopal, Madhya Pradesh- 462041, INDIA

Abstract: Solar air heater is solar energy collection devices and this collected solar energy is used for low temperature heating purpose, agriculture drying purpose etc. The heat conversion rate of solar energy to heat energy is low in solar air heater because air is flowing fluids and air has low heat transfer coefficient, which is directly influence the rate of heat transfer from the absorber plate to air. This fluid property heat transfer coefficient can be increase by creating turbulence in flow field and the turbulence is created by providing different shape artificial roughness on absorber plate. In this paper the study of heat transfer in a rectangular duct of a solar air heater having trapezoidal rib roughness on the absorber plate is done by using Computational Fluid Dynamics (CFD). The effect on Nusselt number, heat transfer coefficient, velocity, and temperature parameter are analyzed. This CFD simulation is done by fluent 6.3 software using K- ϵ model

Keywords: Solar air heater, Heat transfer, Absorber plate, Trapezoidal rib, CFD

I. Introduction

Energy is a basic need for human being; it is a prime agent in the generation and economic development. Energy resources may be classified in two ways conventional and non-conventional energy resources. Solar energy is available abundance on earth in the form of radiation. Solar energy is used for heating application and converts it into thermal energy. Solar air heater is the cheapest way of converting solar energy into thermal energy. Thermal performance of solar air heaters is comparably poor from solar water heaters. Thermal performance may be increased by increasing convective heat transfer coefficient. There are two way for increasing heat transfer coefficient either increase the area of absorbing surface by using fins or create the turbulence on the heat transferring surfaces. 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. Various investigators have studied different types of roughness geometries and their arrangements.

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. Yadav and Bhagoria [2] 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. Ravigururajan and Bergles [3] 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 [4] 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. Prasad and Saini [5] investigated effect of relative roughness height (e/D) and relative roughness pitch (p/e) on heat transfer and friction. Authors reported that increase in relative roughness height results decrease in rate of heat transfer enhancement although rate of increase of friction factor increases. And increase in relative roughness pitch (p/e) results decrease in rate of heat transfer and friction. Momin et al. [6] experimentally investigated effect of geometrical parameters of V-shaped ribs on heat transfer and fluid flow characteristics in rectangular duct of solar air heater. It has been reported by authors that rate of increase of Nusselt number with an increase in Reynolds number is lower than rate of increase of friction factored.

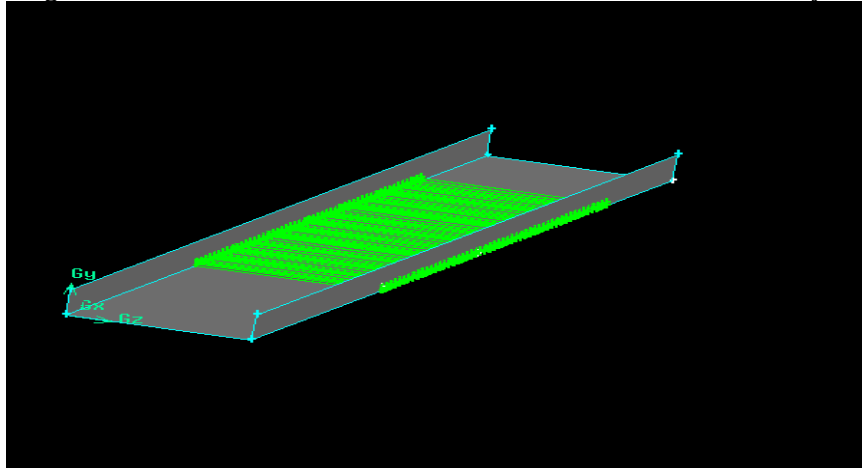
Use of artificial roughness to increase heat transfer coefficient has been studied using CFD (Computational Fluid Dynamics) by various investigators. Suman Saurabh [7] investigate triangular rib on absorber plate by CFD ANSYS Fluent 6.3 and found that triangular rib roughness gives 1.3 to 1.4 time enhancement in nussult

number as compared to smooth duct and also found that maximum Nusselt number is achieved at relative roughness pitch 4. Satya Prakash Nayak [8] presents a CFD study on rectangular roughness rib in a solar air heater duct by computational fluid dynamics (CFD) and evaluates that by generating the flow separation regions (vortices) on each side of the rib, results in enhancement in heat transfer. Chaube *et al.* [9] carried out CFD based analysis for investigating heat transfer and friction characteristics of rib roughened rectangular air duct.

II. Detailed geometry of test section

The present CFD analysis is a solar air heater absorber plate of size 100x20x750 and this plate is divided into three sections in the lengthwise direction. First section is inlet section, length 230 mm. Second section which has a trapezoidal rib is the test section, length 402 mm. The third section is exit section, length is 118 mm. The height of the trapezoidal rib is 2 mm and pitch is 8 mm. Inlet and exit length is selected on the basis of ASHRAE Standard 93-2003, which recommends entrance and exit length of $5\sqrt{WH}$ and $2.5\sqrt{WH}$ respectively. A uniform heat flux of 800 W/m^2 is considered for computational analysis.

Figure 1 Schematic of two-dimensional solution domain for CFD analysis



III. CFD Modeling

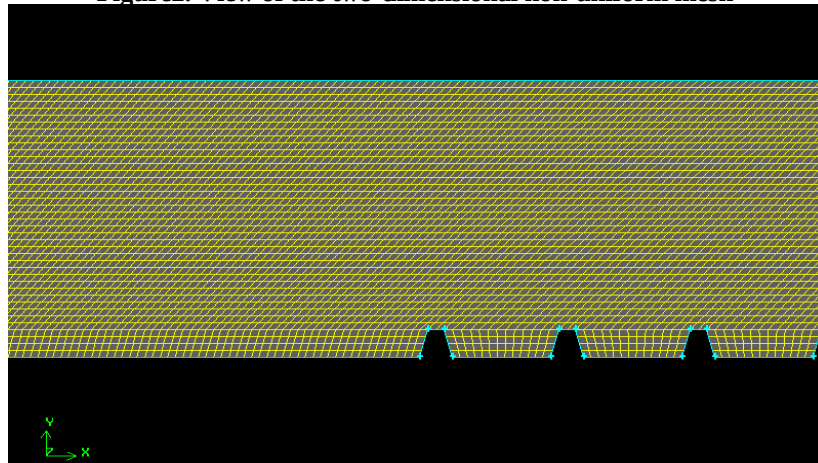
In this investigation, a two-dimensional numerical simulation of the heat transfer, Nusselt number, Velocity, and temperature was conducted using the CFD code FLUENT VERSION 6.3. The CFD modeling involves numerical solutions of the conservation equations for mass, momentum, and energy. These three equations are used to model the convective heat transfer process with the following assumptions;

(a) The flow is steady, fully developed, turbulent, and two-dimensional. (b) Incompressible fluid and flow. (c) The duct wall, absorber plate, and roughness material are homogeneous and isotropic.

IV. Geometry and Meshing

The geometry and the grid were generated using GAMBIT® the preprocessing module of the FLUENT 6.3 and exported into FLUENT. Fig. 2 shows meshing of the test section and the mesh element is triangular and mesh density is 59332 elements.

Figure 2. View of the two-dimensional non-uniform mesh



V. Operating parameter and boundary condition

Table I Show operating parameter

Parameter	Value
Reynold number ranges	4000-20000
Duct depth (H)	20 mm
Duct width (W)	200 mm
Hydraulic diameter (D_h)	36.36 mm
Test length	402 mm
Rib height (e)	2 mm
Relative roughness pitch(p/e)	4-20
Heat flux	800 W/m ² -k

Table II Show Boundary condition

Edge Position	Name	Type
Left	Duct Inlet	Velocity Inlet
Right	Duct outlet	Outflow
Top	Top surface	Wall
Bottom	Inlet length	Wall

VI. Solution Method

In the present simulation governing equations of continuity, momentum and energy are solved by the finite volume method in the steady-state regime. The numerical method used in this study is a segregated solution algorithm with a finite volume-based. The governing equations are solved using the commercial CFD code, ANSYS Fluent 6.3. A second-order upwind scheme is chosen for energy and momentum equations. The SIMPLE algorithm (semi-implicit method for pressure linked equations) is chosen as scheme to couple pressure and velocity. A uniform air velocity is introduced at the inlet while pressure outlet condition is applied at the outlet. Adiabatic boundary condition has been implemented over the top duct wall while constant heat flux condition is applied to the bottom duct wall of test section.

VII. Result and Discussion

Result of this analysis is present in the form of velocity and temperature contour. Variation in heat transfer with respected to position is presented in the form graph (fig.3). Fig.3 show that peak of local heat transfer coefficient occurs at the reattachment point it is found that reattachment occurs further downstream for the first inter-rib region compared with the others and the reattachment length decreases rapidly to reach an asymptotic value. The reattachment length decreases to a value after 3–4 ribs, which remains the same for consequent ribs. The peaks of local surface heat transfer coefficient are also found at similar downstream locations, which describe the strong influence of turbulence intensity on heat transfer enhancement.

The temperature contour of rib roughened plate is shown in the fig. 4 and it shows that the temperature in the portion of trapezoidal ribs is high and the temperature is increases from 300 K to 315k this temperature variation is demonstred by temperature contour.

Figure 3. Graph of heat transfer coefficient v/s position

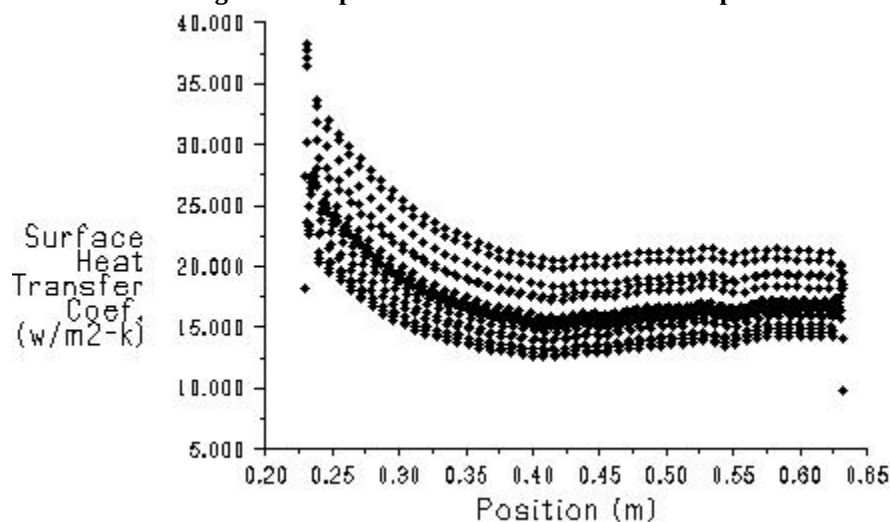


Figure4. Contour of temperature

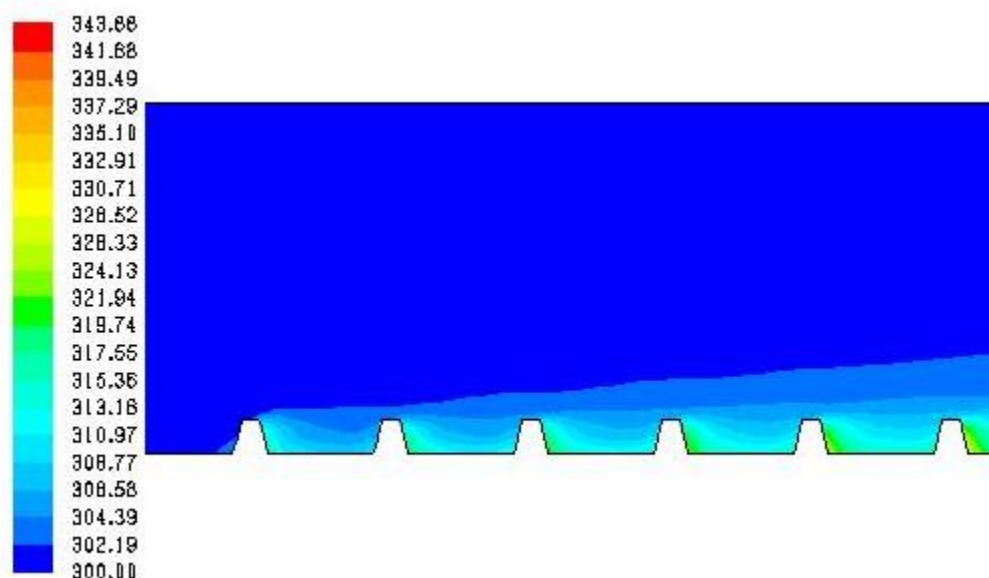


Figure 5. Contour of velocity

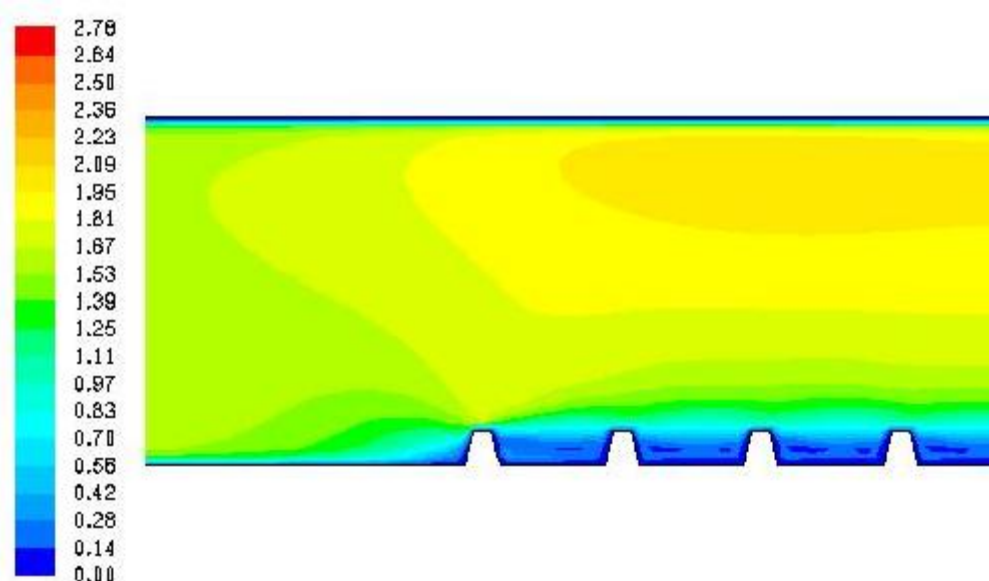


Fig. 5 show the velocity variation on the plate from inlet to outlet. At inlet the velocity is 1.51 and the velocity at outlet is 1.6. On inlet section the velocity is almost zero because at absorber surface, air is in contact and boundary layer formation starts. In test section where artificial roughness in the form of trapezoidal duct is provided the boundary layer formation is disturbed and hence velocity increases with respect to at the initial contact point. This disturbance in the boundary layer increases the heat transfer rate.

VIII. Conclusion

The following broad conclusions can be drawn from the results and discussion

- (1) An attempt has been made to carry out CFD based analysis using FLUENT 6.2 to fluid flow and heat transfer characteristics of solar air heater having roughened duct provided with artificial roughness. Combined effect of turbulence and reattachment of fluid which was considered to be responsible in the increase of heat transfer rate.
- (2) As it is clear from results and discussion that because of artificial roughness Nusselt number is increasing but simultaneously it will also increase friction factor
- (3) Heat transfer is also an increase because of increase in temperature. Trapezoidal rib is mainly responsible for increase in temperature.
- (4) 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.

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] 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.
- [3] 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.
- [4] 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.
- [5] Prasad, B.N, Saini J.S. "Effect of artificial roughness on heat transfer and friction factor in solar air heater". *Sol Energy*, 1988; 41(6):555-660.
- [6] Momin, Saini, J.S.and Solanki, "Heat transfer and friction in solar air heaterduct with v-shaped rib roughness on absorber plate", *International Journal of Heat and Mass Transfer*, 2002; 45:3383-3396.
- [7] Suman Saurav (2013) "CFD analysis of heat transfer through artificially roughened solar duct" *International Journal of Engineering Trends and Technology – Volume4*.
- [8] Satya Prakash Nayak "Heat transfer analysis of rectangular roughness rib in solar air heater duct by computational fluid dynamics (cfd)", *International Journal of Advanced Research in Science, Engineering And Technology*, Vol.01, Issue 03, Pp. 25-30
- [9] 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.