CFD analysis of solar air heater duct with rectangular rib surface

Surjeet Singh Rajpoot[#1], Dinesh Kumar Koli[*2]

[1] Research Scholar Department of Mechanical Engg. Sagar Institute of Research & Technology, Bhopal, Madhya Pradesh, India-462041

[2] Asst. Professor Department of Mechanical Engg. Sagar Institute of Research & Technology, Bhopal, Madhya Pradesh, India-462041

***ABSTRACT*— This paper presents the study of heat transfer in a rectangular duct of a solar air heater having Rectangular rib with roughness on the absorber plate surface by using Computational Fluid Dynamics (CFD) and the effect on the surface heat transfer coefficient was investigated. 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. A commercial finite volume package ANSYS FLUENT 6.3 and GAMBIT is used to analyze and visualize the nature of the flow across the duct of a solar air heater. 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 is increasing at the outlet side and it reaches to maximum due to the rectangular shape of the duct.**

***Keywords*— Solar Air Heater, Heat transfer, CFD.**

I. INTRODUCTION

Solar air heater is one of the basic equipment through which solar energy is converted into thermal energy. Solar collectors in the system for the utilization of solar thermal energy are widely used in various equipments. Solar collectors (air heaters), because of their simplicity in design, are cheap and most widely used devices for solar systems. The main applications of solar air heaters are space heating, seasoning of timber [4], curing of industrial products and these can also be effectively used for curing/drying of concrete/clay building components.

There seems to be lack of computational works on prediction of heat transfer and fluid flow using CFD in solar air heaters. The advantages of computational simulations are that they can produce extremely large volumes of results at virtually no added expense and it is very cheap to perform parametric studies to optimize performance of equipments. The second reason for such work on computational simulation is that some parameters are difficult for test, and experimental study is expensive as well as time consuming. The results obtained with the CFD are of acceptable quality.

Chaube *et al.* [9] is carried out a 2-D computational analysis to assess the comparative performance of the absorber. Ten different ribs shapes viz. Rectangular, Square, Chamfered, Triangular, Semicircle etc., were investigated at the Reynolds number ranging from about 3000–20000, in which solar air heaters operate normally.

II. 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 [2]. 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. Theses parameters are usually specified in terms of dimensionless parameters, namely, relative roughness height (e/D), relative roughness pitch (P/e), angle of attack (a), relative gap position (d/W), relative gap width (g/e), groove position (g/P) and chamfer angle (v) etc. The roughness elements can be two-dimensional ribs or three dimensional discrete elements, angled ribs, V-shaped continuous or broken ribs, Rib-groove, arc rib, Multi v-rib etc.

III. PERFORMANCE OF FLAT PLATE SOLAR COLLECTOR

Thermal performance of flat plate solar collector was first investigated by Hottel and Woertz reported by Duffie and Beckman Bliss introducing ‘collector heat removal factor’[7], F_R, defined as the ratio of actual useful energy gain to the useful energy gain if the whole collector absorbing surface were at the fluid inlet temperature (Ti).

$$Q_u = A_c . F_R [I(\tau\alpha) - U_L(T_i - T_a)]$$

Or

$$q_u = Q_u/A_c = F_R [I(\tau\alpha) - U_L(T_i - T_a)] \qquad (1)$$

where

$$F_R = mC_p (T_o - T_i) / A_c[I(\tau\alpha) - U_L(T_i - T_a)] \qquad (2)$$

Further, thermal efficiency of a solar air heater can be expressed by the following equation;

$$\eta_{th} = F_R [(\tau\alpha) - U_L(T_i - T_a/I)] \qquad (3)$$

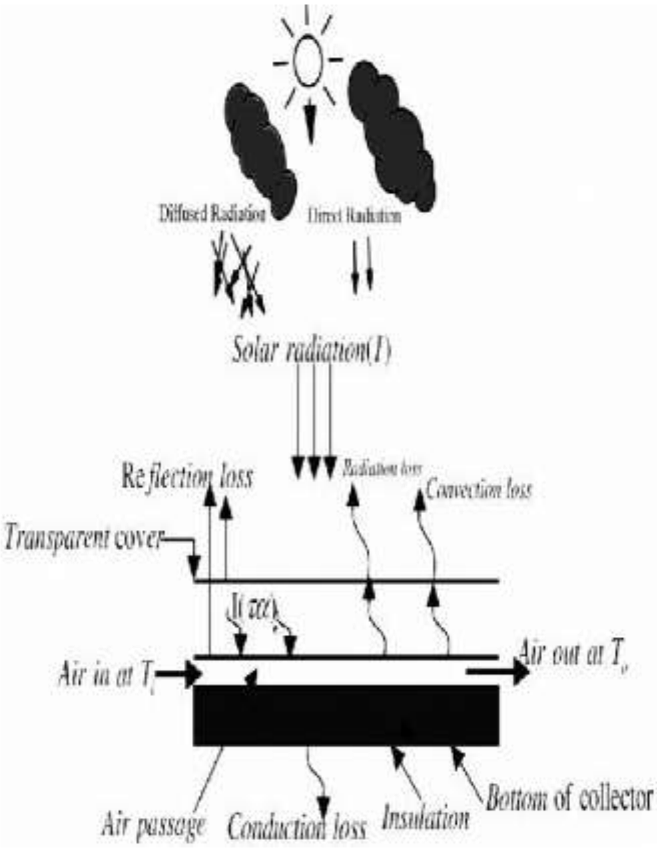


Fig 1 Solar air heater

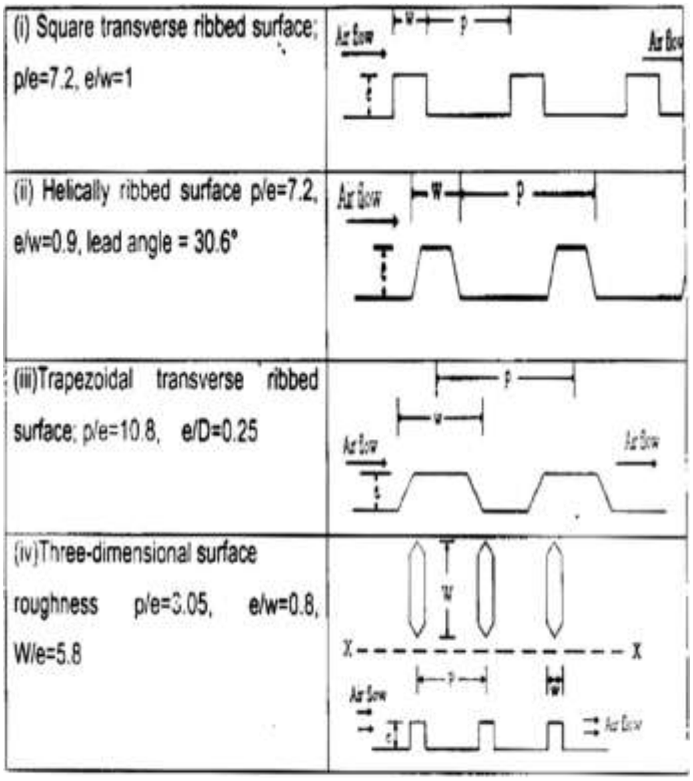


Fig. 2 Rib geometry used by Firth and Meyer

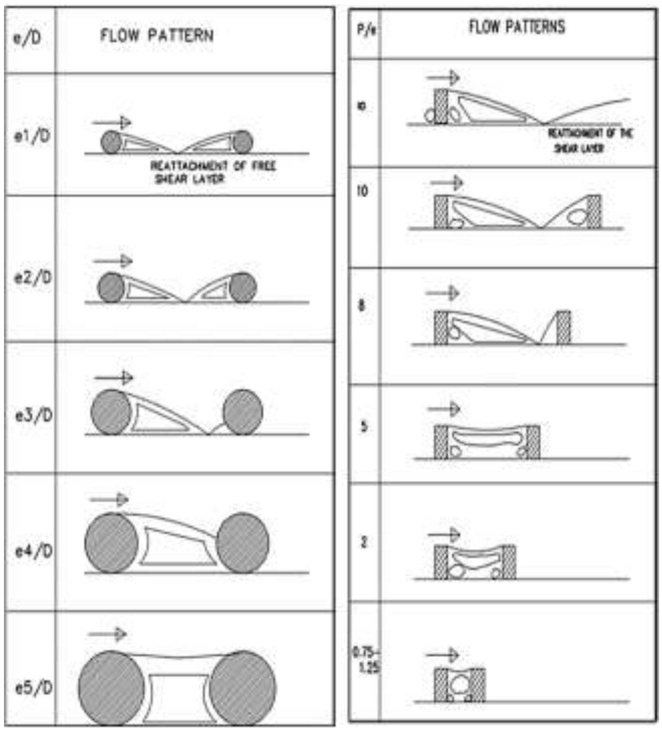


Fig.3. Effect of rib height and pitch on flow

IV. Computational Fluid Dynamics 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.

Dynamics of fluids are governed by coupled non-linear partial differential equations, which are derived from the basic physical laws of conservation of mass, momentum, and energy. Analytical solutions of such equations are possible only for very simple flow domains with certain assumptions made about the properties of the fluids involved. For conventional design of equipment, devices, and structures used for controlling fluid flow patterns, designers have to rely upon empirical formulae, rules of thumb, and experimentation. However, there are many inherent problems with these conventional design processes. Empirical formulae and rules of thumb are extremely specific to the problem at hand and are not globally usable because of the non-linearity of the governing equations. For example, a rule of thumb for designing an aircraft wing may not be applicable for designing a wing mounted on a racing car, as the upstream flow conditions are completely different for the two configurations.

The above reasons make experimentation the leading conventional design technique. However, there are many limitations of experimentation techniques as well:

1. Measurement of flow variables may cause these variables themselves to change, might not be possible at all (in very small or unreachable spaces), and may be expensive.
2. Experimentation may take a long time to set up, sometimes lasts for a very short time, and may be very expensive, as in the case of supersonic wind-tunnel runs.
3. Experimental data has limited detail.

V. Computational Domain

Duct height (H) = 20 mm
Rib height (e) = 3 mm (square rib)
p/e=13.3
Inlet length=245 mm
Uniform heat at bottom surface=1100 W/m^2 (the surface below a rib is considered Insulated)
Aspect ratio (AR) = 5
Pitch p = 40 mm
Length of test section = 280 mm
Outlet length = 115 mm
Width of duct = 100 mm

- *GRIDE*

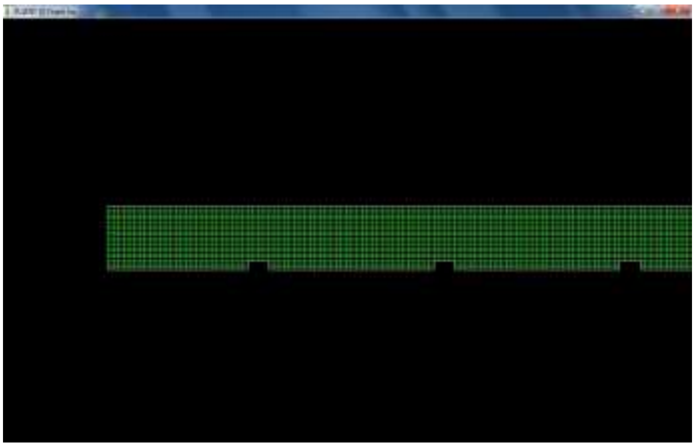


Fig 4: 2D Grid formation of duct

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.

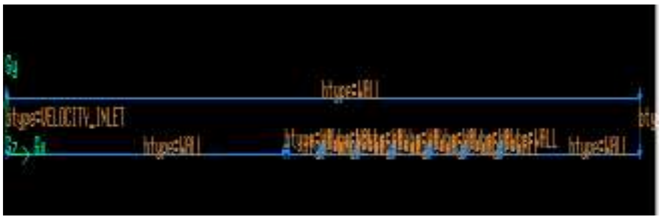
(4) The working fluid, air is assumed to be incompressible for the operating range of solar air heaters since variation in density is very small.

(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.

VI. Detail of Boundary Conditions

Name	Type
wall	WALL
b_wall	WALL
u_wall	WALL
outlet	PRESSURE_OU
inlet	VELOCITY_INLE



- *SOLVER*

FLUENT Version 6.3 is used as a solver with RNG k-epsilon 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.

VII. RESULTS AND DISCUSSION

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 technique. 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 an outlet pressure condition is applied at the outlet side. Adiabatic boundary condition has been implemented over the bottom duct wall while constant heat flux condition has been applied to the upper duct wall of test section. Fig. A shows the temperature Contour for the rectangular 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.

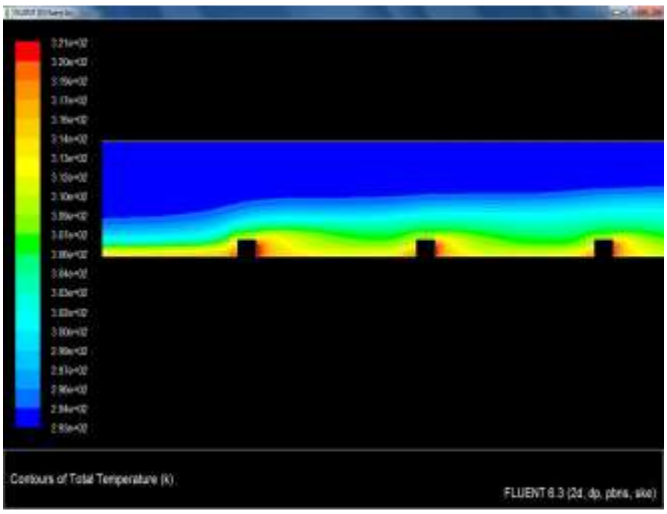


Fig 5 A Temperature contour for the rectangular shape of ribs

Fig. 5 B shows the contour of stream function for the rectangular 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.

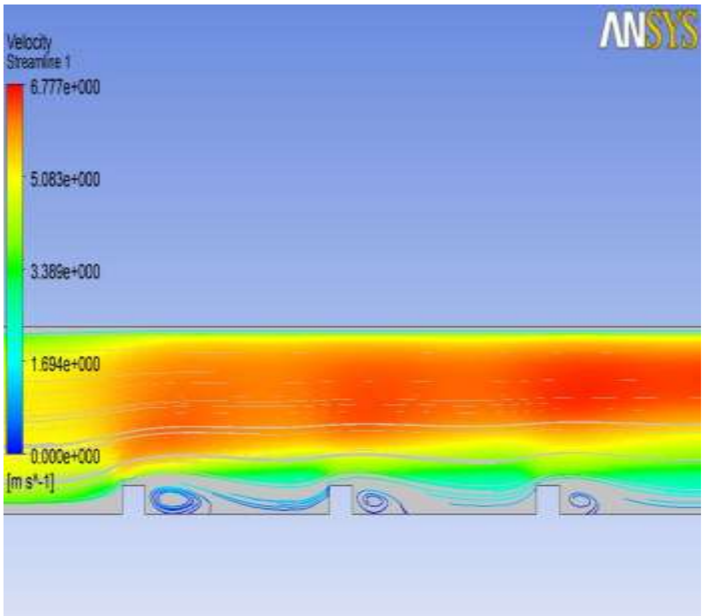
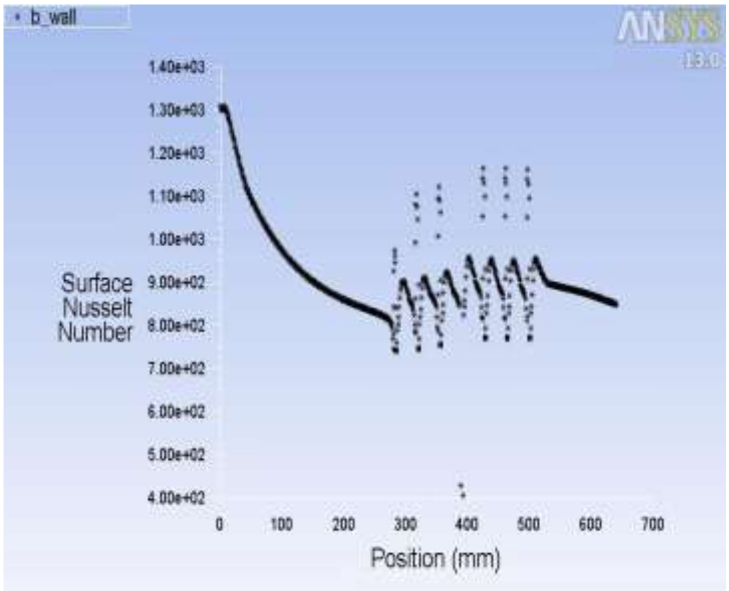
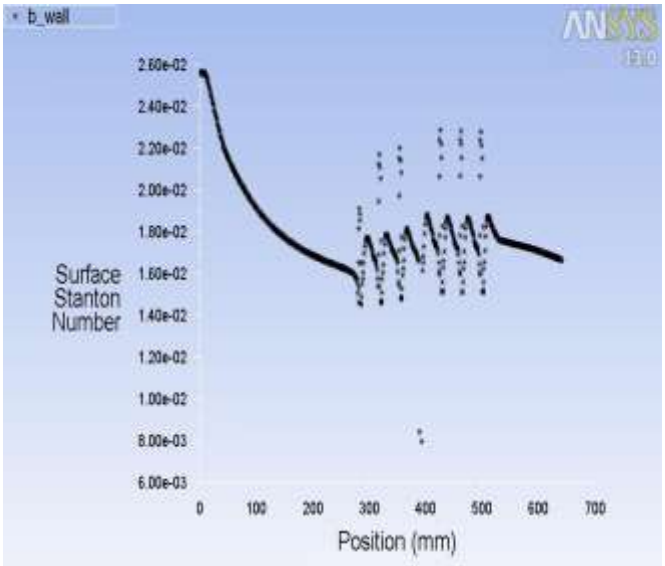


Fig.5B Shows the contour of stream function for the rectangular shape

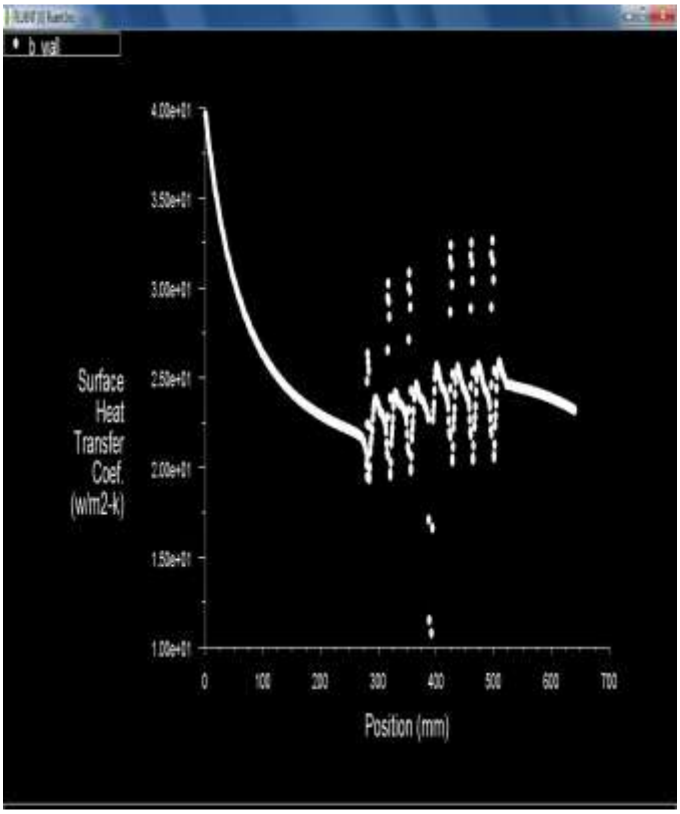
a. Distribution plot of Nusselt number:



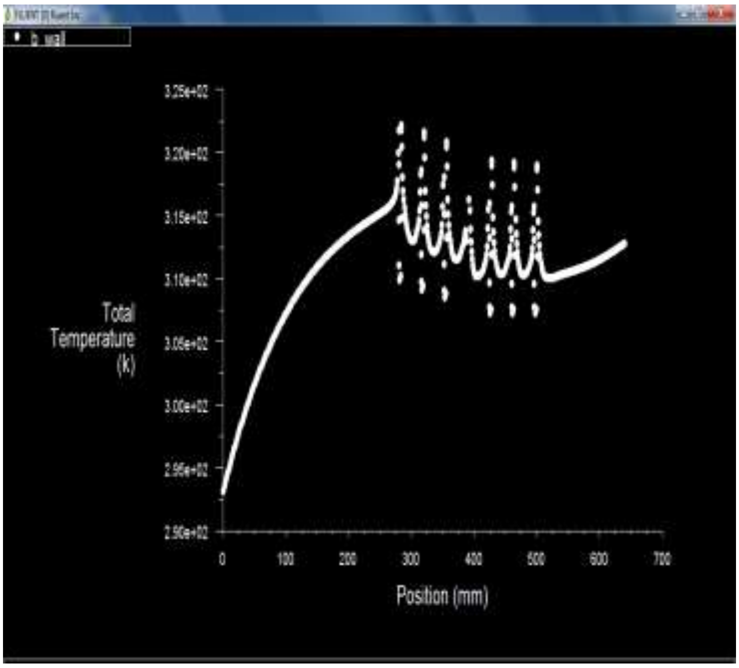
b. Distribution plot of Stanton number:



c. Distribution plot of Heat transfer coefficient:



d. Distribution plot of Heat transfer coefficient:



VIII. RESULTS AND DISCUSSION

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 surface. The following conclusions has been drawn from the present work:

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 studies reported that 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. Nusselt number increases with the increase of Reynolds number.
4. Solar air heater with rectangular rib roughness gives 1.4 to 2.7 times enhancement in Nusselt number as compared to smooth duct.
5. The maximum value of Nusselt number has been found corresponding to relative roughness pitch of 12
6. The maximum temperature has been found on the rib surface 323 K.

ACKNOLEDGEMENT

I would like to express my deepest sense of gratitude and sincere thanks to my highly respected and esteemed guide *Mr. Dinesh Kumar Koli* (Assistant Professor), Department of Mechanical Engineering, Sagar Institute of Research & Technology, Bhopal, MP, India, for suggesting the subject of the work and providing constant support, great advice and supervision during the work of this thesis, which appointed him as a backbone of this work. His cooperation and timely suggestions have been unparalleled stimuli for me to travel eventually towards the completion of this thesis.

REFERENCES

[1] Jaurker, A.R. Saini, J.S. and Gandhi, B.K., (2006). Heat transfer and friction characteristics of rectangular solar air heater duct using rib-grooved artificial roughness, *Solar Energy*, 80, pp. 895 – 907.
[2] Karmare, S.V. and Tikekar, A.N., (2007). Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs, *Int. Journal of Heat Mass Transfer*, 50, pp. 4342 – 4351.
[3] Karwa, R. Solanki, S.C. and Saini, J.S., (1999) Heat transfer coefficient and friction factor correlations for the transitional flow regimes in rib-roughened rectangular duct, *Int. Journal of Heat Mass Transfer*, 42, pp. 1597 -1615.
[4] Karwa, R. Solanki, S.C. and Saini, J.S., (2001). Thermohydraulic performance of solar air heaters having integral chamfered rib roughness on absorber plate, *Energy*, 26, pp. 161 – 176.
[5] Karwa, R., (2003). Experimental studies of augmented heat transfer and friction in asymmetrically heated rectangular ducts with ribs on the heated wall in transverse, inclined, V-continuous and V-discrete pattern, *Int. Comm. Heat Mass Transfer*, 30, pp. 241 – 250.
[6] Kumar, S. and Saini, R.P., (2009). CFD based performance analysis of a solar air heater duct provided with artificial roughness, *Renewable Energy*, 34, pp. 1285 – 1291.
[7] Laohalertdecha, S. Naphon, p. and Wongwises, S., (2007). A review of electro hydrodynamic enhancement of heat transfer, *Renewable and Sustainable Energy Reviews*, 11, pp. 858 – 876.
[8] Layek, A. Saini, J.S. and Solanki, S.C., (2007). Heat transfer and friction characteristics for artificially roughened duct with compound turbulators, *International Journal of Heat and Mass Transfer*, 50, pp. 4845 – 4854.
[9] Alok Chaube, P.K. Sahoo, Renewable Energy 31 (2006) Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater.