International Journal of Engineering Trends and Technology (IJETT) – Volume 4 Issue 9- September 2013

CFD Analysis of Heat Transfer through Artificially Roughened Solar Duct

Suman Saurav[#1], V.N.Bartaria[*2]

[#]M.E Scholar ,Lakshmi Narain College of Technology, Bhopal ,Madhya Pradesh- 462041(India)
[*] Mechanical Engg. Deptt., Lakshmi Narain College of Technology, Bhopal ,Madhya Pradesh- 462041(India)

***ABSTRACT* - The thermal performance of solar air heater is generally poor due to low heat transfer coefficient between the absorber plate and air flowing in the duct. In order to improve the thermal performance artificial roughness is provided on the underside of absorber plate due to which turbulence is created in the heat transfer zone and ultimately performance of solar air heater improves considerably. This paper presents the study of heat transfer in a rectangular duct of a solar air heater having triangular rib roughness on the absorber plate by using Computational Fluid Dynamics (CFD). The effect of Reynolds number on Nusselt number was investigated to study the heat transfer, friction factor and flow characteristics in a solar air heater having triangular rib roughness on the absorber plate. 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 4000 – 20000 and p/e (4 to 20) has been carried out with k-ε turbulence model is selected by comparing the predictions of different turbulence models with experimental results available in various literature. In present work computational fluid dynamics software (Fluent 6.3 Solver) has been used for simulation. Fluid flow (FLUENT) module has been used in present work. Fluid flow (FLUENT) module predicts the outlet temperature, velocity, flow behavior to great accuracy due to application of thermal loading on the work piece. It has been found that the nusselt number increases with increase in Reynolds number. Maximum value of nusselt number friction factor is obtained at relative roughness pitch of 4.**

***Key Words* – Artificial roughness, Solar air heater, Roughness geometry, Nusselt number, Friction factor, Thermo hydraulic performance, Reynolds number.**

I. INTRODUCTION

Energy plays key role for economic and social development. Demand for energy has been rising rapidly with growing population, transportation and industrialization. Due to continuous use of fossil fuels, not only the energy starvation is felt at global level but another serious problem of environment degradation has also been resulted. The rapid depletion of conventional energy sources has necessitated search for alternative energy sources to meet the energy demand of immediate future and for generations to come. Of the many alternatives, solar energy stands out as the brightest long range promise towards meeting the continually increasing demand for energy. Solar energy is available freely, omnipresent and an indigenous source of energy provides a clean and pollution free atmosphere. The simplest and the most efficient way to utilize solar energy is to convert it into thermal energy for heating applications by using solar collectors. Solar air heaters, because of their inherent simplicity are cheap and most widely used collector devices. There seems to be lack of computational works on prediction of heat transfer and fluid flow using CFD in solar air heater. Number of experimental studies3-5 have been carried out to evaluate performance of solar air heater but very few attempts of CFD investigation have been made so far due to complexity of flow pattern and computational limitations. With the development of computer, hardware and numerical methodology, advanced mathematical models are being used to carry out critical investigations on 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 equipment performance. The second reason for such work on computational simulation is that some parameters are difficult to test, and experimental study is expensive as well as time consuming. In the present work a computational investigation of turbulent forced convection in a two-dimensional duct of a solar air heater having triangular rib roughness on the absorber plate is conducted. The bottom wall is subjected to a uniform heat flux condition while the lower wall and two other side walls, except inlet and outlet are insulated.

II. SOLUTION DOMAIN

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 triangular transverse wire rib roughness on 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 chaube et al. 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 thermally fully developed flow is established in a short length 2-3 times of hydraulic diameter. The exit section is used after the test section in order to reduce the end effect in the test section. For the turbulent flow regime, ASHRAE Standard 93-2003 recommends entrance and exit length of 5√ WH and 2.5√ WH respectively. The top wall consists of a 0.5 mm thick absorber plate made up of aluminum. Artificial roughness in the form of small diameter galvanized iron (G.I) wires is considered at the underside of the top of the duct on the absorber plate to have roughened surface, running perpendicular to the flow direction while other sides are considered as smooth surfaces. Here, select the test length or plate length 480 mm in place of most suitable plate length 1000mm because this is convenient to use for CFD analysis purpose. 1000 mm length increase meshing element in higher amount so finally this increase the computation time and required lager memory. The rib height, 2mm has been chosen so that the laminar sub-layer is of the same order as of roughness height at the lower flow Reynolds number (relevant in solar air heater). A uniform heat flux of 800 w/m^2 is considered for computational analysis.

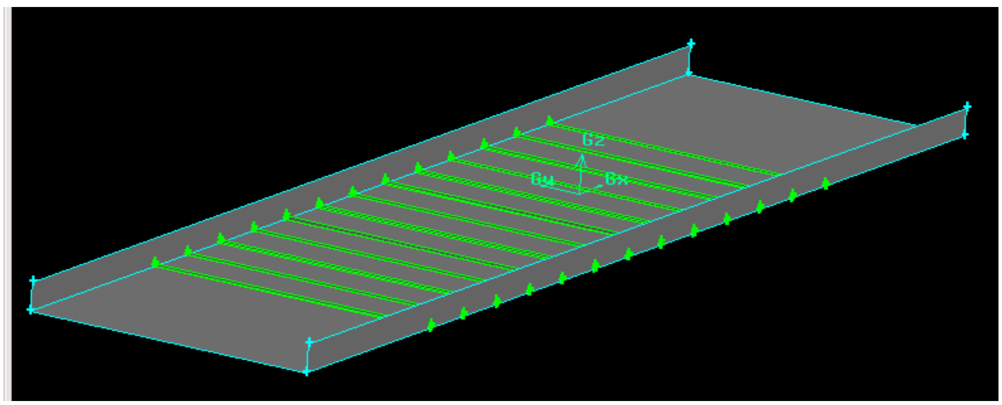

Fig. 1 Schematic of two-dimensional solution domain for CFD analysis.

In the present study, FLUENT Version 6.3 is used for analysis.
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 less.
(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.

III. OPERATING PARAMETERS

The various ranges of geometric and operating parameters are given below in table 1, 2 and 3

TABLE: 1 Geometric parameters of the artificially roughened solar duct

L1 (mm)	L2 (mm)	L3 (mm)	W (mm)	H (mm)	D (mm)	e (mm)	p (mm)
150	480	140	200	20	36.36	2	8,16,24,32,40

TABLE: 2 Range of operating parameters

Operating Parameters	Range
Uniform heat flux ‘I’	800 w/m2

Reynolds number ‘Re’	4000-20000 (5 Values)
Prandtl number ‘Pr’	0.7441
Relative roughness pitch ‘P/e’	4-20 (5 values)
Relative roughness height ‘e/D’	0.0550 Constant
Duct aspect ratio ‘W/H’	10

TABLE: 3 Thermo physical properties of working fluid (air) and absorber plate (aluminum)

Properties	**Working fluid (air)**	**Absorber plate (aluminum)**
Density, ‘ρ’ (kg m^{-3})	1.225	2719
Specific heat, ‘Cp’ (J kg^{-1} K^{-1})	1006.43	871
Viscosity, ‘μ’ (N m^{-2})	1.7894×$^{10-5}$	-
Thermal conductivity ‘k’ (W m^{-1} K^{-1})	0.0242	202.4

IV. MESHING

Meshing of the domain is done using Gambit software. A non-uniform mesh with very fine mesh size is used to resolve the laminar sub-layer and is shown in fig.2, since low- Reynolds number turbulence models are employed, the grids are generated so as to be very fine. For grid independence test, the number of cells is varied from 20000 to 35080 in various steps. After 35080 cells, further increase in cells has negligible effect on the results so in present study non-uniform quadrilateral mesh contained 35080 triangular cells. This size is suitable to resolve the laminar sub layer. Analysis is carried out with 2D mesh, which not only saves computer memory but also lot of computational time.

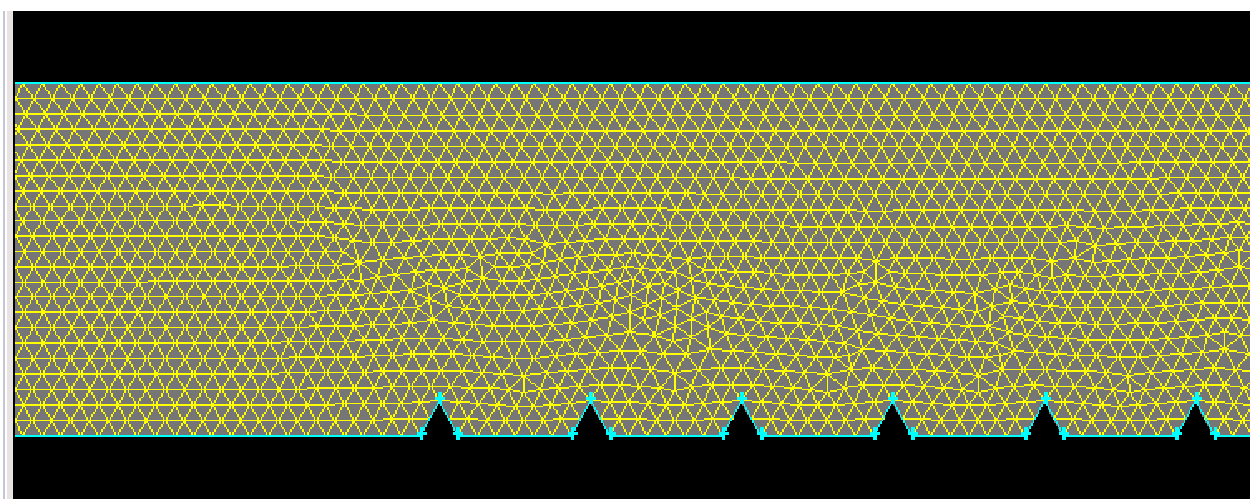

Fig. 2 View of the two-dimensional non-uniform mesh.

V. BOUNDARY CONDITIONS

The boundary conditions selected are listed in table 4.

TABLE: 4 Boundary Conditions

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 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 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. DATA DEDUCTION

The useful heat gain of the air is calculated as:

$Q_u = m'C_p(T_{f0}-T_{fi})$

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 = $(T_{fi} + T_{fo})/2$

The Nusselt number:

$N_{ur} = h.\, D_h /K_{air}$

Where D_h is hydraulic mean diameter of test duct, h is convective heat transfer coefficient, Kair is thermal conductivity of air.

The friction factor was determined from the measured values of pressure drop across the test length:

$F_r = (\Delta P)D_h / (2\rho_{air}Lv^2_{air})$

P is pressure drop in the test duct, ρ is density of air, L is test duct length, V_{air} is average velocity of air

Thermal Performance (overall enhancement ratio)

$(Nu_r / Nu_s)/ (f_r / f_s)^{1/3}$

Where f_s and Nu_s (friction factor and nusselt number for smooth duct) can be calculated as

$fs = 0.085Re^{-0.25}$ and $Nu_s = 0.024Re^{0.8}Pr^{0.4}$

VIII. RESULT AND DISCUSSION

In present work following the simulation methodology and utilizing the boundary conditions as mentioned in detail in Section 3, simulations were completed to obtain following results:

1. Temperature variation in roughened solar air heater.

2. Velocity variation in roughened solar air heater.

3. Heat transfer coefficient.

1. Temperature Variation

The temperature contour of rib roughened plate is shown in the fig. 3and it shows that the temperature in the portion of triangular ribs is high due to the development of wake and form drag in that region which results in lower heat transfer and thus there is rise in temperature.

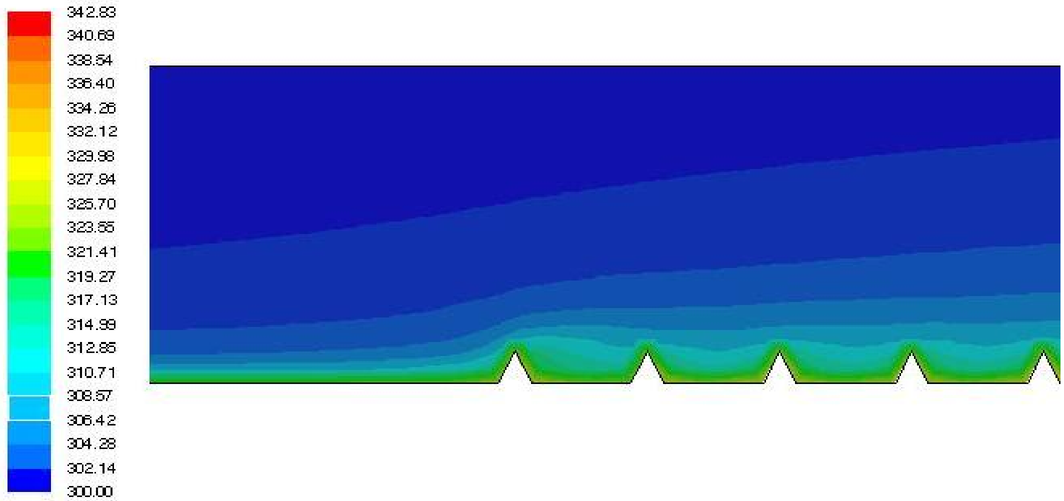

Fig. 3 Contours of temperature

2. Velocity variation in roughened solar air heater

At low reynolds number, the nusselt number for all types of surface approach almost same value. This is due to less disturbance of laminar sub layer. The laminar sublayer offers resistance to heat transfer. To reduce this resistance the laminar sublayer is disturbed by roughened surface at higher Reynolds number. This velocity distribution of air on the roughened duct.

disturbance in the boundary layer increases the heat transfer rate. In addition, local contribution to the heat removal by vortices is originated from the roughness element. Thus Nusselt number and friction factor for roughened surfaces is higher from that of the smooth surface.Fig.4.shows the

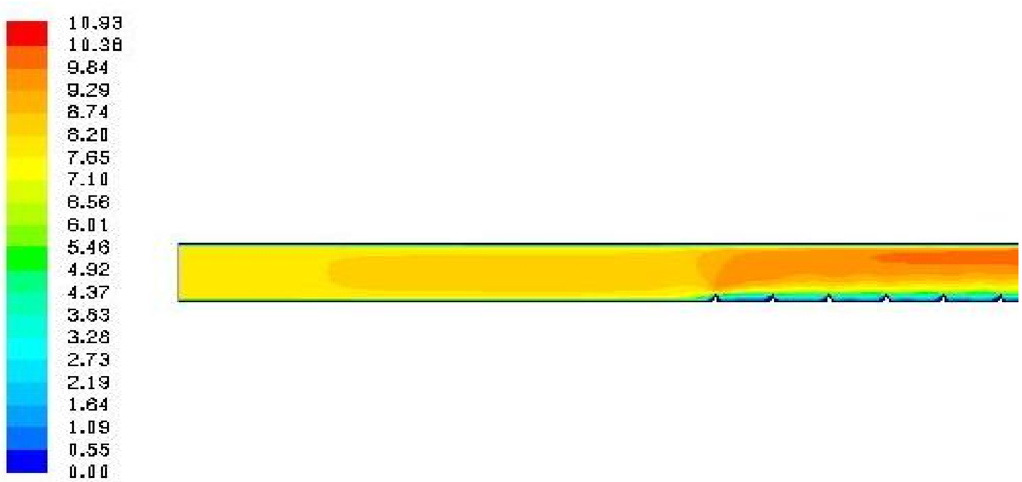

Fig. 4 Contours of velocity

3. Heat transfer coefficient.

Fig.5 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.

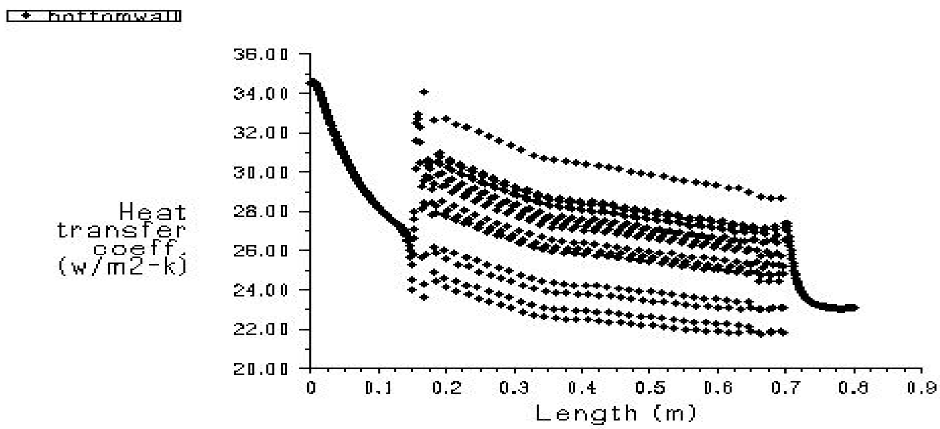

Fig.5 Contours of heat transfer coefficient

The value of Nur, Nus, Fr and Fs calculated are given below in table 5

Table: 5 Value of nusselt number and friction factor at different p/e and Re

Case	p/e	P	Reynold no.	Nur	Nus	Fr	Fs
A	4	8	4000	27	16.239	**1.1066**	0.0106
			8000	36.1883	23.27	0.9064	0.00898
			12000	39.54872	28.31	0.709	0.00764
			16000	41.838	31.31	0.5912	0.0071131
			20000	**44.56**	34.14	0.5429	0.006721
B	8	16	4000	23.746	16.239	0.76085	0.0106
			8000	28.82263	23.27	0.5625	0.00898
			12000	32.715	28.31	0.5033	0.00764
			16000	36.75622	31.31	0.478	0.0071131
			20000	40.46844	34.14	0.4622	0.006721
C	12	24	4000	24.267	16.239	0.7201	0.0106
			8000	29.486	23.27	0.5287	0.00898
			12000	33.453	28.31	0.4775	0.00764
			16000	37.46	31.31	0.4485	0.0071131
			20000	41.11	34.14	0.433	0.006721
D	16	32	4000	24.444	16.239	0.6949	0.0106
			8000	29.59	23.27	0.5053	0.00898

			12000	33.379	28.31	0.459	0.00764
			16000	37.6221	31.31	0.431	0.0071131
			20000	41.0845	34.14	0.431	0.006721
			4000	24.4794	16.239	0.6554	0.0106
			8000	29.652	23.27	0.4685	0.00898
E	20	40	12000	33.427	28.31	0.4106	0.00764
			16000	37.481	31.31	0.3994	0.0071131
			20000	41.112	34.14	0.381	0.006721

Note: Bold value denotes maximum

So from value obtained in table in this study the maximum value of nusselt number and friction factor is obtained at p/e = 4. So finally calculating overall thermal enhancement factor by using the relation $(Nu_r / Nu_s) / (f_r / f_s)^{1/3}$ we get overall thermal enhancement ratio = 0.277.

IX. 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 triangular transverse wire rib roughness. The effect of Reynolds number, relative roughness pitch on the heat transfer coefficient and friction factor have been studied. CFD Investigation has been carried out in medium Reynolds number flow (Re 4000 - 20000). The following conclusions are drawn from present analysis:

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 behavior and performance of a solar air heater.
3. Nusselt number increases with the increase of Reynolds number.
4. Solar air heater with triangular rib roughness gives 1.3 to 1.4 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 4.
6. Friction factor decreases with an increase of Reynolds number. The maximum value of friction factor is found to be 1.1066 for relative roughness pitch of 4 at a lower Reynolds number, 4000.
7. In the inter-rib region, the model predicts well near the central high heat transfer area but it under predicts around ribs.

X. FUTURE SCOPE

1. In present work triangular roughness geometry has been studied. So further performance analysis can be done using various other roughness geometries.
2. In present work for all the roughness geometries under investigation effective efficiency is calculated by varying the relative roughness pitch(p/e), while kept the other operating parameters constant. So there is a large scope for predicting the performance in terms of effective efficiency by varying the other operating parameters such as relative roughness height (e/d), duct aspect ratio (W/H) etc.
3. In present work simulation is carried out to predict the flow behavior along the length of roughened solar air heater but actually there is also a variation in flow behavior along the height of air passage that can be included in further studies.

Nomenclature

D - equivalent diameter of the air passage (m)
d/w - relative gap position
e/D - relative roughness height
fr - friction factor of roughened duct
fs - friction factor of smooth duct
g - heat transfer function (g-function)
g/e - relative gap width
g/p - groove position to pitch ratio
l/s - relative length of metal grit
L/e - relative long way length
S/e - relative short way length
Nu - Average Nusselt number
Nur - Nusselt number for roughened duct
Nus - Nusselt number for smooth duct
P - roughness pitch (m)
p/e - relative roughness pitch
R - roughness function
Re - Reynolds number

Greek Symbols –
α- angle of attack
α/90 – relative angle of attack
φ – wedge angle

References

[1] Momin A-M.E., Saini J.S., Solanki S.C. Heat transfer and friction in solar air heater duct with Vshaped rib roughness on absorber plate. International J. of Heat and Mass Transfer. 2001, 45(16), 3383-3396.
[2] Karwa R., Solanki S.C., and Saini J.S. Thermo-hydraulic performance of solar air heaters having integral chamfered rib-groove roughness on absorber plates. Energy. 2000, 26,161-176.
[3] Bhagoria J.L., Saini J.S. and Solanki S.C.,Heat transfer coefficient and friction factor correlations for rectangular solar air heater duct having transverse wedge shaped rib roughness on the absorber plate, *Renew.Ene.*, 25(3), 341-369, (2001).
[4] Sahu M.M. and Bhagoria J.L., Augmentation of heat transfer coefficient by using 90° broken transverse ribs on absorber plate of solar air heater, *Renew. Ene.*, 30 (13) 2057-2073,(2005).
[5] Jaurker A.R., Saini J.S., and Gandhi B.K. Heat transfer coefficient and friction characteristics ofrectangular solar air heater duct using rib-grooved artificial roughness. Solar Energy. 2005, 80(8),895-907.
[6] Varun, Saini R.P., Singal S.K. Investigation of thermal performance of solar air heater havingroughness elements as a combination of inclined and transverse ribs on absorber Plate. RenewableEnergy. 2007, 33(6), 1398-1405.
[7] Aharwal K.R, Gandhi B.K, Saini J.S. Heat transfer and friction characteristics of solar air heater duct having gap in integral inclined continuous ribs on absorber plate. International Journal of Heat andMass Transfer. Renewable Energy 33 (2007) 585–596.
[8] Karmare S.V., Tikekar A.N. Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs. International Journal of Heat and Mass Transfer. S.V. Karmare, A.N. Tikekar / Solar Energy 83 (2008) 6–13.
[9] Layek A., Saini J.S., and Solanki S.C. Heat transfer and friction characteristics for artificially roughened ducts with compound turbulators. International Journal of Heat and Mass Transfer. A. Layek et al. / Renewable Energy 34 (2008) 1292–1298.
[10] Saini S.K., Saini R.P., "Development of Correlations for Nusselt Number and Friction Factor for Solar Air Heater with Roughened Duct Having Arc Shaped Wire as Artificial Roughness", Solar Energy, 2008, vol.82, pp.1118-1130.
[11] Kumar Thakur Sanjay,Thakur N.S, Kumar Anoop, Mittal Vijay, Heat Transfer and Friction Factor Correlations for Rectangular Solar Air Heater Duct Having 60° Inclined Continuous Discrete Rib Arrangement, British Journal of Applied Science & Technology, 1(3):67-93, 2011.
[12] Kumar Thakur Sanjay,Thakur N.S, Kumar Anoop, Mittal Vijay, "Use of Artificial Roughness to Enhance Heat Transfer in Solar Air Heaters – A Review", Energy, 2010, vol.21, pp.35-51.
[13] Choudhary sachin,Varun,Chouhan Kumar Manish Heat transfer and friction factor characteristics using continuous M shape ribs turbulators International Journal of Energy and Environment (IJEE), Volume 3, Issue 1, 2012, pp.33-48.
[14]Lanjewar.A.M,Bhagoria.J.L,Sarviya.R.M Thermohydaulic Performance Of Solar Air Collector Using W-Shaped Artificial Roughness Journal of Environmental Research And Development Vol. 6 No. 3A, Jan-March 2012.
[15] A.K.Patil, J.S.Saini, K.Kumar A Comprehensive Review on Roughness Geometries and Investigation Techniques Used in Artificially Roughened Solar Air Heaters International Journal of Renewable Energy Resarch, Vol.2, No.1, 2012.
[16] Saini R.P., Verma J., "Heat Transfer and Friction Correlations for a Duct Having Dimple Shape Artificial Roughness for Solar Air Heater", Energy, 2008, vol.33, pp.1277-1287.
[17] Bopche S.B., Tandale M.S., "Experimental Investigation on Heat Transfer and Friction Characteristics of a Turbulator Roughened Solar Air Heater Duct", International Journal of Heat and Mass Transfer, 2009, vol.52, pp.2834-2848.
[18] Varun, Saini R.P., Singal S.K., "A Review on Roughness Geometry Used in Solar Air Heaters", Solar Energy, 2007, vol.81, pp. 1340-1350.
[19] Yadav AS, Bhagoria JL. "Heat transfer and fluid flow analysis of solar air heater:a review of CFD approach", Renewable and Sustainable Energy Reviews 2013;23:60e79.
[20] Chaube A, Sahoo PK, Solanki SC. "Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater",Renewable Energy 2006;31:317e31.
[21] Yadav AS et al. "A CFD Analysis Of A Solar Air Heater Having Triangular Rib Roughness on The Absorber Plate", Int.J.ChemTech Res.2013, 5(2).
[22] Yadav AS et al. "A CFD (computational fluid dynamics) 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 (2013) 1127e1142.

[23] Sharma A. K. and Thakur N.S., "CFD based Fluid Flow and Heat Transfer Analysis of a V-Shaped Roughened Surface Solar Air Heater", International Journal of Engineering Science and Technology, 2012, 4(5), 2115-2121.
[24] Karmare S. V. andTikekar A. N. "Analysis of Fluid Flow and Heat Transfer in a Rib Grit Roughened Surface Solar Air Heater using CFD",SolarEnergy, 2010, 84 (3), 409-417.
[25] Kumar S. andSaini, R. P. "CFD based Performance Analysis of a Solar Air Heater Duct Provided with Artificial Roughness", Renewable Energy, 2009, 34 (5), 1285-1291.
[26] Rai G.D., "Non Conventional Energy Sources", 4th ed., Khanna Publishers, 2010.
[27] Shaw C.T., "Using Computational Fluid Dynamics", Prentice Hall, 1992.
[28] Cebeci Tuncer et.al," Computational Fluid Dynamics for Engineers"' © Horizons Publishing Inc., Long Beach, California 2005.
[29] Versteeg H.K.et.al," An introduction to computational fluid dynamics, the finite volume method", Longman group limited 1995.
[30] Nayak Satyapraksh et.al." 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.
[31] Nayak Satyapraksh et.al. "heat transfer and fluid flow analysis of roughness rib in solar air heater duct by computational fluid dynamics (cfd) simulation", International Journal of Advanced Research in Science, Engineering and Technology, Vol.01, Issue 03, pp. 15- 20.