HEAT TRANSFER AND FLUID FLOW ANALYSIS OF FLAT SURFACE WITH ARTIFICIALLY ROUGHENED SQUARE TRANSVERSE WIRE RIB FOR APPLICATION IN SOLAR AIR HEATER

R. Thundil Karupaaraj[1], Y. Raja Sekhar[1], G. Edison[1], B. Sai praneeth[1] and T. Elango[2]
[1]SMBS, Energy Division, VIT University, Vellore, India
[2]Dhanalakshmi Srinivasan College of Engineering, Perambalur, India
E-Mail: thundil.rajagopal@vit.ac.in

ABSTRACT

Metal duct or channels of rectangular cross section with black selective coating has an application in solar air heaters. Increase in heat transfer rate is achieved by suitably modifying the absorber surface with fins, corrugations, honey comb structure etc. In the present article numerical study is performed using computational fluid dynamics (CFD) on a rectangular metal duct having transverse square wire rib roughness of small diameter created artificially on the absorber surface. The design parameters of the analysis are the flow Reynolds number, hydraulic diameter roughness pitch and height. The 3D analysis is performed using ANSYS FLUENT 12.1 code with Renormalization-group (RNG) k-ε model having the mesh model under five lakh nodes to investigate the heat transfer and fluid flow characteristics. The results of smooth duct are compared with results from literature and found to be in good agreement. The ratio of increase in heat transfer to friction factor referred as energy gain ratio with the use of artificial roughness is estimated for range of parameters and optimum performance condition is arrived. The maximum value of energy gain ratio is about 1.31 times more compared to the plain duct for the range of operating parameters investigated.

Keywords: solar air heaters, artificial roughness, energy gain ratio, CFD, heat transfer enhancement.

INTRODUCTION

The conversion efficiency of most of the solar thermal devices are found to be less because of poor heat transfer rate between the surface absorbing the radiant energy from sun to the fluid to be heated. A solar sir heater is thermal device used for space heating, drying agricultural products, curing of industrial raw material etc., One important aspect of solar air heaters are that they do not require continuous maintenance because of its simple design [1]. For enhancing the heat transfer performance between the fluid and the hot surface, passive techniques were used by researchers since no external source is required in the system except additional power [2].

The heat exchange process can be increased by different passive techniques such as rough surfaces, treated surfaces and extended surfaces along with insertion of turbulent promoters in the annulus of heat transfer region [3]. Joule [4] was the first to use the concept of artificial roughness for heat transfer enhancement and the technique was used by many researchers in their experimental investigations specifically in the area of gas turbines, electronic equipment, heat exchangers, etc. Kays [2] studied the thermal characteristics using thin wires of small diameter having relative pitch in the range of 10 to 20. Similarly, heat transfer studies with modified absorber plate configurations were carried out by Gupta and Garg [5] to study the performance of different solar air heaters. The effect of heat transfer coefficient using transverse ribs is studied by Prasad and Mullick [6] and tried with experimental studies to determine an optimum roughness parameter for attaining maximum heat transfer. V-shaped ribs were used by Momim *et al*., [1] in their experimental studies to study the heat transfer and fluid flow of solar air heaters having rectangular duct and ribs placed underneath the absorber plate. Many researchers has under gone experimental studies on application of artificial roughness to absorber surface of the solar air heater but a few researchers have performed simulation studies on increase of the heat transfer rate and fluid flow analysis to validate the experimental results [7-16]. Karuppa Raj and Srikanth [17] performed CFD simulation on shell and tube heat tube heat exchanger to study the heat transfer phenomenon by varying the baffle inclination. Sekhar *et al*., [18] studied the heat transfer and fluid flow characteristics inside a pipe using nanofluids and twisted tapes by experimental analysis and showed enhancement with use of passive techniques and nanofluids for solar applications.

Most of the CFD simulations performed considering artificial roughness provided by ribs of different cross sections was of 2-dimensional form. Saha and Acharya [8] studied heat transfer and turbulent flow characteristics of rotating ribbed ducts of different aspect ratios using averaged Navier-Stokes procedure. Numerical investigations in a tube with circular cross-sectional rings were carried out by Ozceyhan *et al*., [9]. The effect of thermal boundary conditions on heat transfer in rib-roughned passages was carried out by Iaccarino *et al*., [10]. Chaube *et al*., [14] reported the results of CFD their studies performed on solar air heaters with absorber surface having provided with artificial roughness to study the effect of roughness on heat transfer. The artificial roughness provided on the absorber plate will affect the heat transfer since the laminar sub-layer formed close to the absorber plate is distrubed and the flow becomes turbulent [15]. Yadav and Bhagorai [19] had presented a detailed review on heat transfer and fluid flow analysis on solar air heaters using artificial roughness and stated that for two-dimensional CFD simulation of solar air heater Renormalization-group (RNG) k-ε model yields the best results compared to other turbulence models. CFD simulation predicts the exact physical phenomenon of fluid flow happening in a duct with great accuracy and a few groups have tried the simulation of solar thermal devices. However, the results of CFD depend on the computational limitations and flow pattern considered.

In the present work, numerical investigation using CFD is carried out on a rectangular duct of a solar air heater having small transverse square ribs of 1mm^2 attached on the absorber plate. Three -dimensional analysis is performed with the fluid flow assumed to be of turbulent nature. The top surface of the duct is subjected to constant heat flux boundary condition while the lower surface is insulated. The square ribs are provided on the surface beneath the top surface of absorber plate and the other three surfaces are assumed to be smooth. The results from the analysis are compared with experimental results available in literature.

The objectives of the present work are:

a. To study the characteristics of heat transfer and fluid flow inside the rectangular duct by varying the roughness pitch, roughness height, relative roughness height and
b. To arrive at an optimal square rib dimension in terms of energy gain ratio.

To the authors best knowledge three dimensional CFD analysis on solar air heating duct having small diameter transverse square ribs below the absorber plate is scarcely available in open literature till now.

CFD SIMULATION

This section explains about the procedure adopted in solving the 3-dimensional simulation of solar air heater duct with artificially roughened transverse square ribs on surface. To perform the CFD simulation ANSYS FLUENT (version 12.1) is used to solve the equations of conservation equations for mass, momentum and energy. The details of the model are as shown in Figure-1. The assumptions considered in the computational analysis are as follows:

a) The fluid flow is turbulent fully developed, steady and three dimensional
b) The thermal conductivity of the duct wall, absorber plate and rib material are independent of temperature
c) The duct wall, absorber plate and rib material are homogenous and isotropic
d) The fluid is assumed to be incompressible in the operating range of the solar air heaters since the density variation along the width and height is very less
e) The walls in contact with the fluid are assigned with no-slip boundary condition
f) The radiation heat loss and other losses are negligible

The rectangular duct of the solar air heater having the height (H) of 20 and 15 mm and width (W) of 100 mm was considered. The depth of the duct has been taken as 640 mm as shown in Figure-1. The solution domain used for the numerical analysis has been generated using ICEM software as shown in Figure-1.

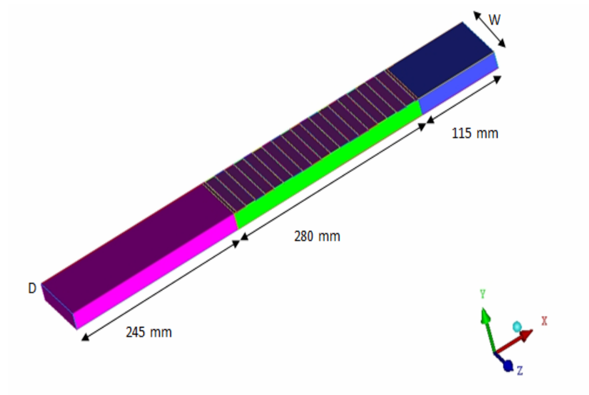

Figure-1. Geometry of three-dimensional computational domain.

Table-1. Range of parameters.

S. No.	Parameter	Range
1.	Reynolds number(Re)	3800-20000 (6 values)
2.	Hydraulic diameter (mm)	26.087,33.33
3.	Duct aspect ratio (W/H)	5-6.67 (2 values)
4.	Heat flux or Insolation(W/m^2)	1000
5.	Roughness height (mm)	1
6.	Relative roughness ratio	0.0333,0.03833

Table-2. Thermophysical properties of fluid and absorber plate considered for the analysis.

Property	Air	Absorber Plate (Aluminium)
Density, ρ (kg/m^3)	1.225	2719
Specific heat,C_p(J/kgK)	1006.43	871
Viscosity,μ (N/m^2)	1.789x10^{-5}	----
Thermal conductivity, K(W/mK)	0.0242	202.4

The duct length was divided into three regions i.e., entry region, test section and exit region as per the ASHRAE 93-2000 recommendations [20]. The top surface of the test section i.e., absorber plate, was subjected to uniform heat flux and 1000 W/m^2 is considered for the computational analysis. Artificial roughness with square ribs was

VOL. 9, NO. 5, MAY 2014 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

considered at the inner side of the top surface of the duct while the other three sides were considered as smooth surfaces. The mesh for the model was done by hexagonal meshing with 3D blocking using ANSYS ICEM CFD v12.1 software. The entry and exit regions were created with coarse mesh where as the test section was generated with a fine mesh upto 0.001 mm so as to capture variation of both the thermal and velocity boundary layers. Also, the quality of mesh obtained was checked for all the elements and it was found to be at an angle of 90^0 with determinant as 1 confirming the uniformity in meshing obtained. The entry, test and exit regions were named as fluid inlet, fluid inner core and fluid outlet, respectively. The fluid inlet and exit regions has one lakh and 60000 elements where as the fluid inner core has got 2.8 lakh elements to examine the fluid flow and heat transfer critically at the fluid-rib regions. The simulations were carried out with varying number of grid points ranging from 214345, 398478 and 467566 nodal points, but no appreciable change in result were observed beyond 398478 nodes for square smooth duct. Similar grid independence studies were carried out for the simulations carried out. The fluid and solid interface (FLSO) is defined at the fluid and the rib interface. RNG k-ε turbulence model was considered as suggested by previous researchers, since changes of the velocity stream lines are minimal. The operating range of parameters considered for the analysis and the thermophysical properties of fluid and material are shown in Table-1 and respectively.

DATA REDUCTION

The degree of utilization of input to the system represents the performance of any system. To design a solar air heater, thermal and hydraulic performance is to be analyzed at different operating conditions. The thermal performance corresponds to the heat transfer to the flowing fluid and hydraulic performance concerns to the pressure drop in the duct [21].

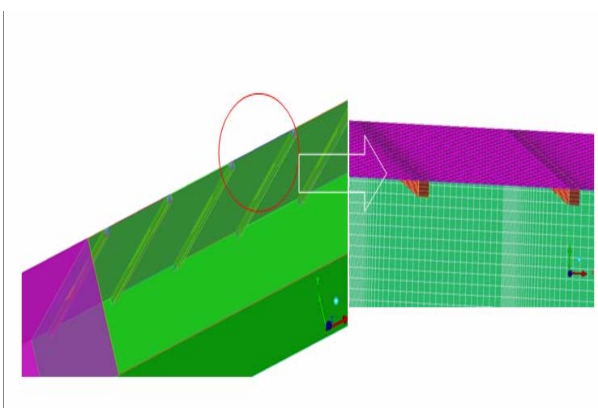

Figure-2. Visualization of the uniform mesh of the computational domain.

The heat gained by the flowing air through the duct is given by $Q_u = \dot{m} C_p (T_o - T_i)$ (1)

The wall heat transfer coefficient at the test section is given by:

$$h = \frac{Q_u}{A(T_{pm} - T_b)} \qquad (2)$$

T_b is the bulk mean temperature of the air in the duct

The Nusselt number is calculated by:

$$Nu = \frac{hD_h}{K_{air}} \qquad (3)$$

The calculated Nusselt number is verified with empirical correlation available in literature for smooth duct of a solar air heater i.e., Dittus-Boelter correlation given by equation (4) below.

$$Nu = 0.023 Re^{0.8} Pr^{0.4} \qquad (4)$$

Friction factor is calculated by using pressure drop, ΔP across the length of the test section and is calculated by:

$$f = \frac{\left(\frac{\Delta P}{L}\right) D}{2\rho\vartheta^2} \qquad (5)$$

In a solar air heater, the heat transfer to the fluid should be maximum to maximize the performance of the system with minimum power consumption to discharge the fluid. The important parameter to evaluate the system having artificially roughened dust to that of the smooth duct is given by Energy gain factor, which gives the thermal enhancement ratio as defined by Webb and Eckert [22].

$$\text{Energy gain factor} = \frac{Nu_R / Nu_S}{(f_r / f_s)^{1/3}} \qquad (6)$$

Where f_S is the friction factor of the smooth duct given by Blasius equation [23]

$$f_S = 0.079 Re^{-0.25} \qquad (7)$$

If the energy gain factor is more than unity, it ensures the effectiveness of the enhancement device and can be used to optimize the design parameters for better performance of the system.

RESULTS AND DISCUSSIONS

The performance of the solar air heater having rectangular duct having artificially roughened square ribs below the absorber surface is studied by varying the Reynolds number of flow using 3-D CFD analysis. The

ARPN Journal of Engineering and Applied Sciences

effect of flow and design parameters on the heat transfer and friction characteristics in a rectangular duct of a solar air heating system is simulated using CFD in this analysis. The enhancement in heat transfer and friction factor values for duct having artificial roughened square ribs is compared with smooth duct under similar operating conditions. To validate the suitability of the present turbulence model considered, turbulence models such as Standard k-ε model, Renormalization group (RNG) k-ε model, and Shear Stress Transport (SST) k-ω models have been tested for smooth duct in order to find out the validity of the models. It is found that the results of Nusselt number and friction factor obtained from RNG k-ε model for smooth tube is in good agreement with the literature results of Yadav and Bhagoria [19] and Dittus-Boelter equation as shown in Figures 3 and 4 respectively thus validating the present model.

EFFECT OF REYNOLDS NUMBER

Figure-5 shows the variation of average Nusselt number for different hydraulic diameter and Reynolds number of flow. The average Nusselt number is found to increase with increase of Reynolds number due to enhancement in turbulence intensity resulted by increase of turbulence dissipation rate and kinetic energy [19]. Figure-6 shows the variation of friction factor for increasing Reynolds number and various hydraulic diameters of duct and it can be observed that at lower values of hydraulic diameter the friction factor is slightly less. The contour plot of temperature distribution at the outlet section and along the length of the smooth duct dust is shown in Figure-7 at Re=3800. Figures 5 and 6 also depict that there is an enhancement in Nusselt number with provision of artificial roughness at the absorber tube bottom surface but there is also a considerable increase of pressure drop. Figures 5 and 6 reveals that the

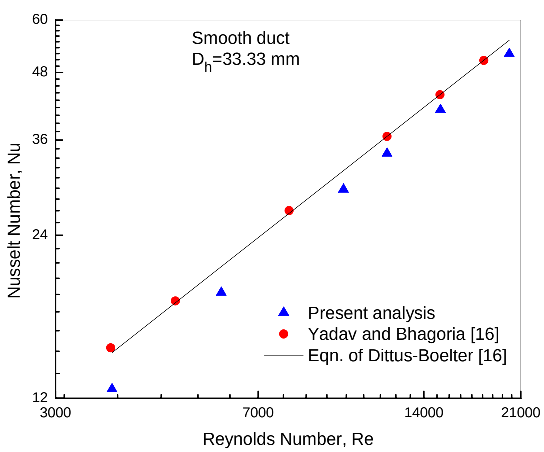

Figure-3. Comparison of Nusselt number for varying Reynolds number with literature for a smooth duct.

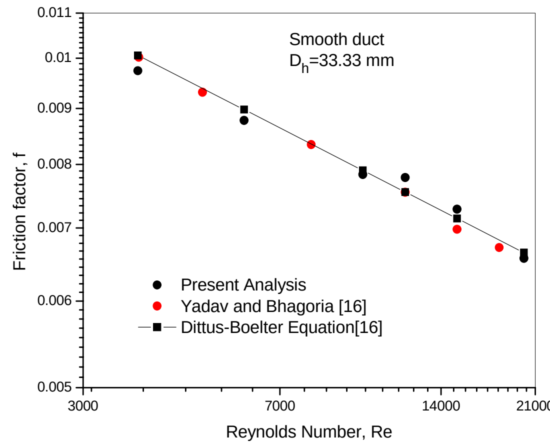

Figure-4. Comparison of friction factor for varying Reynolds number with results of literature for a smooth duct.

increase in Nusselt number enhancement ratio for roughened absorber surface is about 2 to 2. 6 times compared to the smooth duct with an increase of 1.7 to 2 times the pressure drop. It can be noticed from the graph Figure-6 that friction factor reduces with increase of Reynolds number because of suppression of viscous sub-layer near the wall. So, for a particular value of relative roughness pitch and relative roughness height, the value of Nusselt number increases and friction factor decreases thus validating the experimental results of Prasad and Mullick [6] reported for square ribs. A temperature rise of 18^0C at the outlet and 8^0C along the length of the duct from the inlet section is observed from the temperature profiles shown in Figures 7 and 8 respectively. The peak value of temperature at the outlet section is observed at the top-heated wall of absorber plate and then it decrease as the distance from the wall increases. Similarly, the air temperature is high at the end of the duct is observed as the air flows towards the outlet section. However, no change in temperature is observed in the entry section of the duct. Heat transfer characteristics can also be well understood with the help of contours of turbulent kinetic energy. The contour plot of turbulence kinetic energy for smooth duct is shown in Figures 11 and 12 for two different Reynolds number. Since the RNG k-ε turbulence model is used, $\mu_t = \dfrac{\rho C_\mu k^2}{\varepsilon}$ is considered as the turbulence viscosity, where an increase of k value in the flow field also increase the heat transfer rate. It is observed that there is considerable increase in turbulence intensity factor by an order of 30 times for increase in Reynolds number from 3800 to 20000 for smooth duct. The turbulent kinetic energy plot for a duct with artificial roughness at Re=10000 is shown in Figure-15 for a fixed value of roughness height and pitch. Figures 14 and 16 show the temperature and velocity distribution along the length of the duct at Re=10000. It can be observed from the results that the increase in friction factor for roughened

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

absorber is 3 times when compared to smooth duct. The Energy gain ratio, which is the ratio of increase in heat transfer to the increase of friction factor with the use of roughened absorber, is found to decrease with increase of Reynolds number as shown in Figure-17.

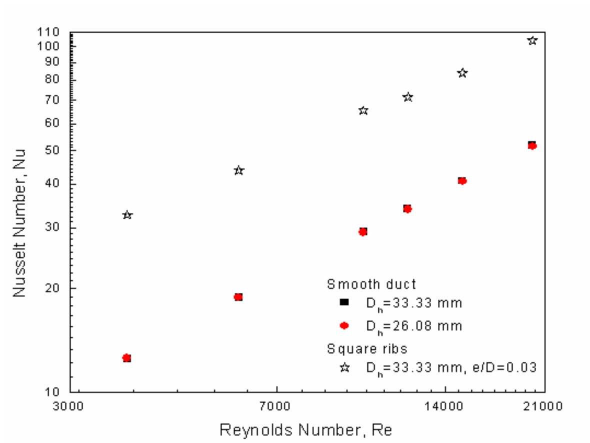

Figure-5. Nusselt number of smooth rectangular duct for varying Reynolds number and hydraulic diameter.

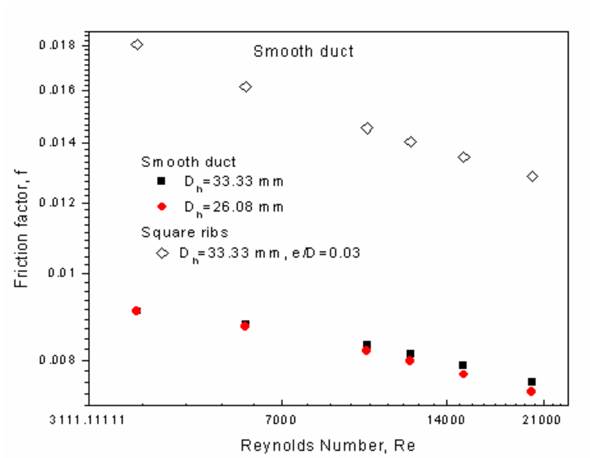

Figure-6. Friction factor of smooth rectangular duct for varying Reynolds number and hydraulic diameter.

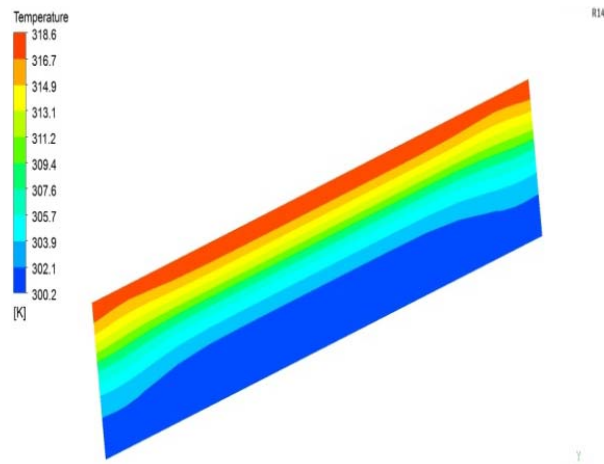

Figure-7. Contour plot of temperature distribution at the outlet section of the duct at Re=3800.

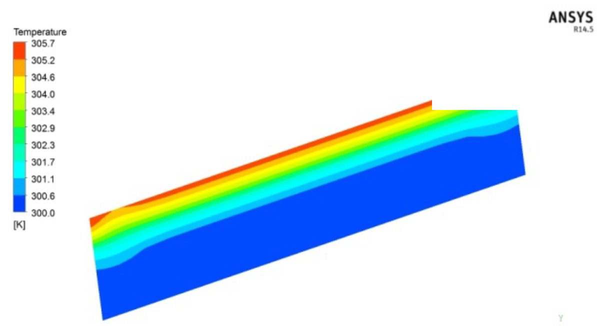

Figure-8. Contour plot of temperature distribution at outlet section at Re=20000 for smooth duct.

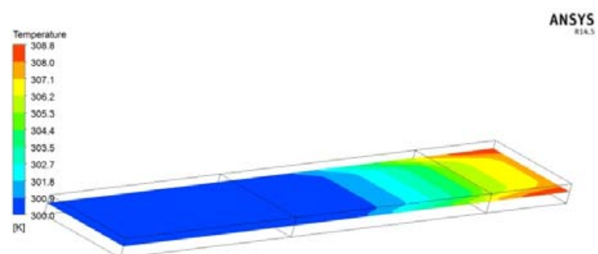

Figure-9. Temperature distribution along the duct at Re=3800 for smooth duct.

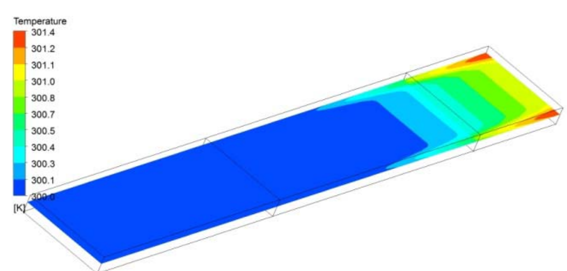

Figure-10. Temperature distribution along the duct at Re=20000 for smooth duct.

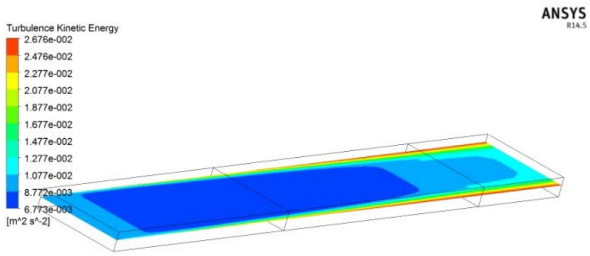

Figure-11. Turbulent kinetic energy distribution at Re=3800 for smooth duct.

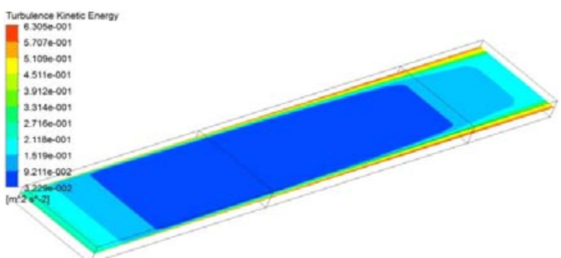

Figure-12. Turbulent kinetic energy distribution at Re=20000 for smooth duct.

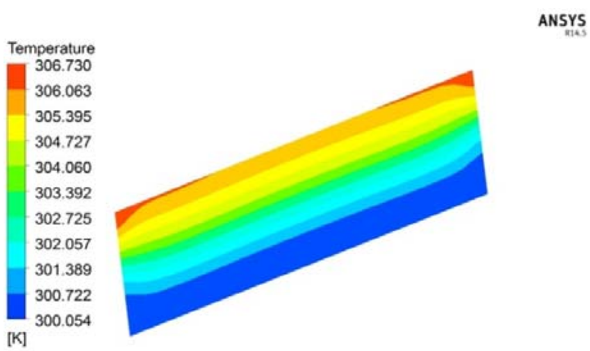

Figure-13. Temperature at outlet Re=10000 square ribbed artificial roughened duct.

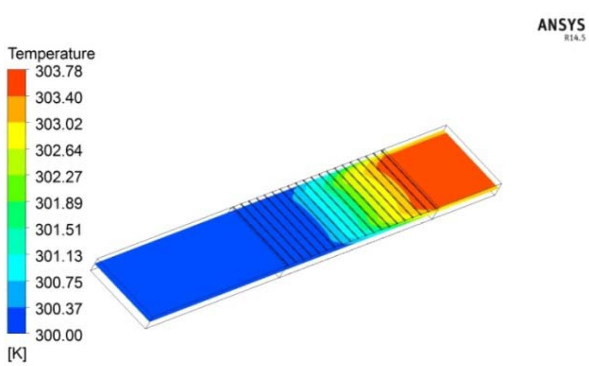

Figure-14. Temperature distribution at Re=10000.

CONCLUSIONS

CFD studies are carried out on a rectangular duct of a solar air heating system having varied hydraulic diameters to analyze the heat transfer and friction characteristics with smooth as well as along the duct length with square ribbed surface below the absorber plate. For the CFD analysis roughness ratio of 0.03 and roughness pitch of 14.29 is considered and heat transfer characteristics are analyzed by varying the hydraulic diameter. The results show that there is considerable enhancement in heat transfer followed by increase in friction factor also.

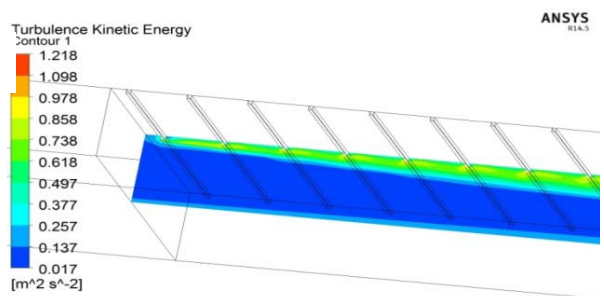

Figure-15. Turbulent kinetic energy along the length at Re=10000 with artificially roughened square ribbed surface.

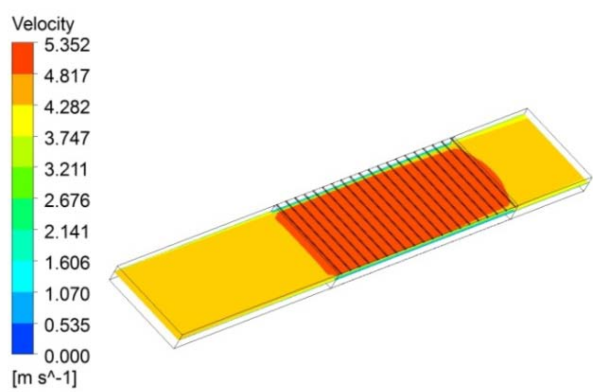

Figure-16. Velocity distribution along the duct for Re=10000 with ribbed surface.

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

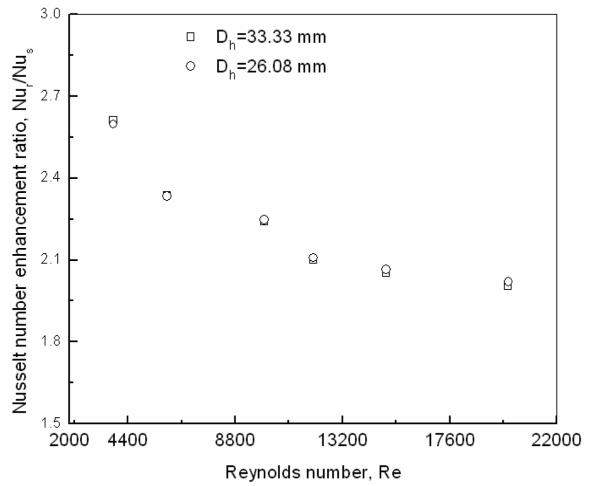

Figure-17. Variation of Nusselt number enhancement ratio with Reynolds number.

An increase in Nusselt number enhancement ratio of 2.6 times for the absorber plate with ribs is observed to that of the smooth duct at Re=3800. Also, for an increase of Reynolds number by 5.26 times, the value of Nusselt number rise by the order of 4.16 times in the case of smooth duct. However, the increase of Nusselt number for the same Reynolds number of flow in the absorber with artificial roughness is observed to be 3.19 times but compared to smooth duct the value is more. One of the important observations from the analysis is that though the hydraulic diameter of the duct is changed no considerable change in the Nusselt number and friction factor is observed as shown Figures 5 and 6. It is found that the energy gain factor values vary between 1.17 and 1.31 for the range of parameters investigated. It is observed that roughened duct gives better energy gain for the studied range of Reynolds number. In order to validate the present numerical model, the results are compared with available experimental results under similar flow conditions. So, it can be concluded that a lot of duct material can be saved for air heating since change in hydraulic diameter could not result any change in heat transfer and provision of artificial roughness at the absorber surface result in better heat transfer.

Nomenclature

A	Surface Area of the duct, m^2
C_p	Specific heat of the fluid, KJ/Kg-K
D	diameter, m
f	Friction factor
h	Heat transfer coefficient, W/m^2K
H	height of the duct, mm
K	Thermal conductivity, W/m^2
L	length of the duct, m
$\mathring{m}$	mass flow rate, kg/s
Nu	Nusselt number
Pr	Prandtl number
Q	heat flux, W/m^2
Re	Reynolds number
T	Temperature, ^0C
W	width of the duct, mm

Greek symbols

ρ	Density, kg/m^3
ΔP	Pressure drop, Pa
ε	Dissipation rate, m^2/s^2
k	Turbulent kinetic energy, m^2/s^2
ϑ	Kinematic viscosity, Ns/m^2
ω	Specific dissipation rate, 1/s
μ_t	Turbulence viscosity

Sub-scripts

b	*fluid bulk*
h	*hydraulic*
pm	plate mean
R	duct with ribbed surface
s	smooth duct
w	water
u	useful

REFERENCES

[1] A.M.E. Momin, J.S. Saini and S.C. Solanki. 2002. Heat Transfer and Friction in Solar Air Heater Duct with V-shaped rib roughness on absorber plate, International Journal of Heat and Mass Transfer 45(3): 383-396.

[2] W.M. Kays. 1966. Convective Heat and Mass Transfer. 1st Ed. McGraw-Hill, New York, USA.

[3] S. Eiamsa-ard and K. Wongcharee. 2009. Sripattanapipat, 3-DNumerical Simulation of swirling flow and convective heat transfer in a circular tube induced by means of loose fit twisted tapes. Int. J. communications in heat mass transfer. 36: 947-955.

[4] J.P. Joule. 1861. On the Surface-Condensation of Steam. Philosophical Transactions of the Royal Society of London 151: 133-160.

[5] C.L. Gupta and H.P. Garg. 1967. Performance Studies on Solar Air Heaters. Solar Energy. 11(1): 25-31.

[6] Prasad. K and S.C. Mullick. 1983. Heat Transfer Characteristics of a Solar Air Heater used for drying purposes. Applied Energy. 13: 83-93.

[7] Ph. Grenard, V. Quintilla-Larroya and E. Laroche. 2007. Numerical study of heat transfer on a dimpled surface with CEDRE code. In: 2nd European conference for aerospace sciences, May.

[8] Arun K. Saha and Sumanta Acharya. 2006. Turbulent Heat Transfer in Ribbed Coolant Passages of Different Aspect Ratios: Parametric effects. J. Heat Transfer 129(4): 449-463.

[9] Ozceyhan, Veysel and Gunes, Sibel, Buyukalaca, Orha and Altuntop, Necdet. 2008. Heat Transfer enhancement in tube using circular cross-sectional rings separated from wall. Applied Energy. 85(10): 988-1001.

[10] A. Ooi., G. Iaccarino, P.A. Durbin and M. Behnia. 2002. Reynolds averaged simulation of flow and heat transfer in ribbed ducts. International Journal of Heat and Fluid Flow. 23: 750-757.

[11] T.J. Craft, H. Iacovides and N.A. Mostafa. 2008. Modelling of Three-Dimensional Jet Array Impingement and Heat Transfer on a Concave Surface. International Journal of Heat and Fluid Flow. 29: 687-702.

[12] G. Iaccarino, A. Ooi, P.A. Durbin and M. Behnia. 2002. Conjugate Heat Transfer Predictions in two dimensional ribbed passages. International Journal of Heat and Fluid Flow. 23: 340-345.

[13] H.Iacovides, G.Kelemenis and M. Raisee. 2003. Flow and heat transfer in straight cooling passages with inclined rinds on opposite walls: an experimental and computational study. Experimental Thermal and Fluid Science. 27: 283-294.

[14] Chaube Alok, P.K. Sahoo and S.C. Solanki. 2006. Analysis of Heat Transfer Augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater. Renewable Energy. 31(3): 317-331.

[15] Anil Singh Yadav and J.L. Bhagoria. 2013. 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: 1127-1142.

[16] Sharad Kumar and R.P. Saini. 2009. CFD based performance analysis of a solar air heater duct provided with artificial roughness. Renewable Energy. 34:1285-1291.

[17] Thundil Karuppa Raj, R. and Srikanth G. 2012. Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow. Thermal Science. 16(4): 1165-1174.

[18] Y. Raja Sekhar, K.V. Sharma, R. Thundil Karupparaj and C. Chiranjeevi. 2013. Heat Transfer Enhancement with Al2O3 Nanofluids and Twisted Tapes in a Pipe for Solar Thermal Applications. Procedia Engineering. 64: 1474-1484.

[19] Anil Singh Yadav and J.L. Bhagoria. 2013. Heat transfer and fluid flow analysis of solar air heater: A review of CFD approach. Renewable and Sustainable Energy Reviews. 23: 60-79.

[20] ASHRAE Standard 93. 2003. Method of Testing to Determine the Thermal Performance of Solar Collectors. American Society of Heating, Refrigeration and Air Conditioning Engineers, Atlanta, GA30329.

[21] Duffie J.A. and Beckman W.A. 1980. Solar engineering of thermal processes. 2nd Ed. New York: Wiley.

[22] R.L. Webb and E.R.G. Eckert. 1972. Application of rough surface to heat exchanger design. International Journal of Heat and Mass Transfer. 15(9): 1647-1658.

[23] W. Fox, P. Pritchard and A. McDonald. 2010. Introduction to fluid mechanics. John Wiley and Sons, New York, USA.

[24] J.A. Duffie and W.A. Beckman. 1980. Solar Engineering of Thermal Processes. Wiley Interscience, New York, USA.