International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online)
Vol. 3, Issue 2, pp: (109-116), Month: October 2015 - March 2016, Available at: www.researchpublish.com

Numerical Investigation of Heat Transfer and Fluid Flow Characteristics of Square Type Tabulator Roughness Solar Air Heater

[1]Vikas Kesharwani, [2]Ravi Vishwakarma

[1]Department of Mechanical Engineering, Rajiv Gandhi Prodyogiki Vishwavidyalaya, Bhopal, India

[2]Department of Mechanical Engineering, Rajiv Gandhi Prodyogiki Vishwavidyalaya, Bhopal, India

Abstract: **The applications of artificially roughened duct or surfaces are being used by researchers to enhance heat transfer rate from heat transferring surface to working fluid. There are different techniques to analyze the heat transfer mechanism those are explained in this literature. The present work was undertaken with the objectives of extensive investigation on the 90^0 shaped ribs as artificial roughness on the surface of duct. Experimental setup for heat transfer and pressure lose have been design and developed. Data were collected for local heat transfer and pressure loss of these artificially roughened ducts. Results of artificially roughened duct have been compared with those of a smooth duct under similar flow condition to determine heat transfer and friction factor.**

Keywords: **Heat Transfer, Friction Factor.**

1. INTRODUCTION

This research is a review of general theory of the governing equations for fluid flow. The governing equations of fluid flow are called the Navier-Stokes equations. In this section, concisely we will discuss the principles of the CFD with its components. The liquid crystal thermography theory and the working principle used in experimental analysis will be described

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k U) = div\left(-\overline{p'u'} + 2\mu u' e'_{ij} - \rho \frac{1}{2}\overline{u'_i . u'_i u'_j}\right) - 2\mu \overline{e'_{ij} e'_{ij}} - \rho \overline{u'_i u'_j} . E_{ij} \qquad (1)$$

2. LIQUID CRYSTAL THERMOGRAPHY

During the past 20–30 years or so, liquid crystals have emerged as reliable temperature and colour sensors for heat transfer, fluid flow and biomedical research, and have been applied in a number of situations to visualize the temperature distribution under complex flow fields and applications such as medical imaging, machine vision, astronomy, satellite reconnaissance, and even desktop publishing.

Principle of Liquid Crystal Thermography on the basis of crystal lattice orientations:

A typical liquid crystal substance reflects lights of different wavelength more or less strongly to different directions due to a re-orientation (rotation) of the liquid crystal's lattice depending on the temperature. This selective light reflection usually gives rise to a spectrum of colors on the heat transfer model surface.

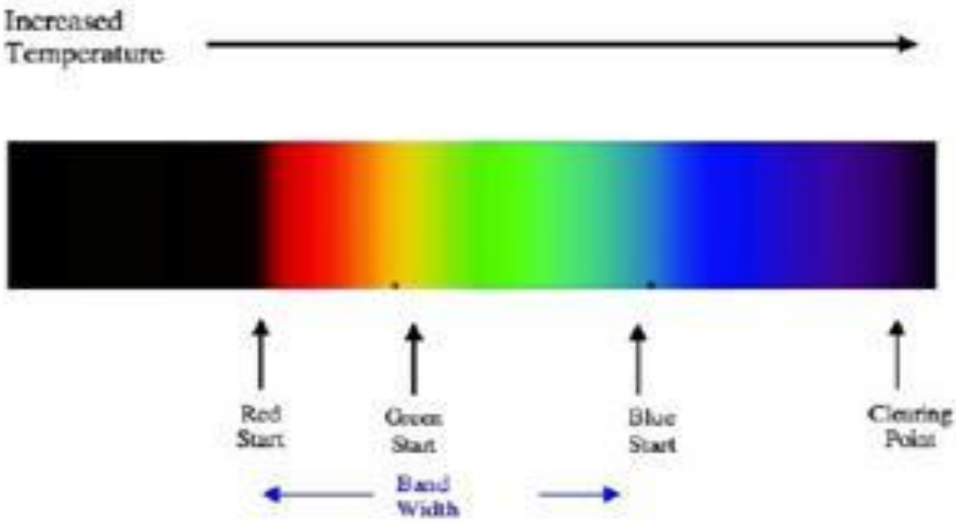

Fig. 1 Different colours corresponding to temperature variation

3. EXPERIMENTAL SET-UP

The experimental schematic diagram set-up including the test section is shown in Fig. 2. The flow system consists of an entry section, a test section, an exit section, a flow meter and a centrifugal blower. The duct is of size 1480mm×171mm×60mm (dimension of inner cross- section) and is constructed from acrylic sheet of 12 mm thickness for three sides and 3mm thickness for upper side. The test section is of length 1480mm (16.67 D). A short entrance length (L/Dh=7.2) was chosen because for a roughened duct the thermally fully developed flow is established in a short length 2-3 hydraulic diameter [24]. For the turbulent flow regime, ASHRAE standard 93-77 [24] recommends entry and exit length of 5√WHand 2.5√WH, respectively.

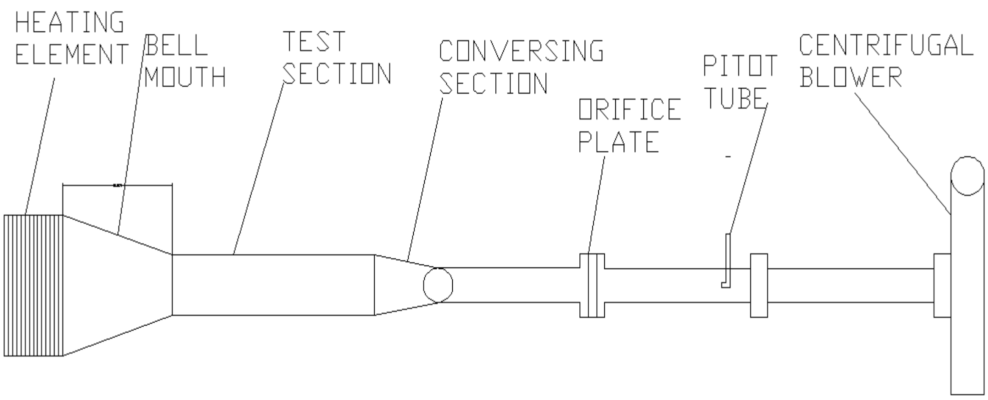

Fig.2 Parts Of Experimental Setup

Parts of Experimental Setup:

1) Rectangular Test Section
2) Heating Element
3) Honeycomb Section(Conversing)
4) Conversing Section to attach Test Section with Pipe
5) Orifice Plate and U tube manometer
6) Pitot Tube
7) Flexible pipe
8) Centrifugal Blower and Motor

Procedure to Perform Experiment:

1) Apply the thermographic liquid crystal sheet to a clean surface of the test section. Liquidcrystal sheets used in this case works in the temperature range of 40˚C to 50˚C.

2) Subject the treated surface to known temperature levels by heater. Heat the incoming air through the heater provided at the front part of experimental set up while maintaining the test section at room temperature so as to notice the color response of liquid crystal sheet applied over the test section.

3) Measure and record the temperature response of TLC. Temperature response of sheet is captured by High Mega Pixel Camera with suitable light source setting and the temperature is recorded by digital thermometer (Cr-Al).

4) After taking temperature and colour image of the different points of test section, we made a relation between colour and temperature.

5) Now the color response of the sheet and their variation with temperature can be used to get exact temperature of any point of the test section.

6) Also record the pressure loss and head difference of U-Tube manometer to get the velocity and mass flow rate of flowing fluid at the pitot tube and orifice meter.

7) Calculations and data analysis on the basis of recorded data and given equations

4. MODEL FORMULATION

Computational Fluid Dynamics Approach:

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.

Computational Domain:

The 2-D computational domain used for CFD analysis having the height (H) of 60 mm and width (W) 180 mm and total length of 250mm as shown in Fig. 3.

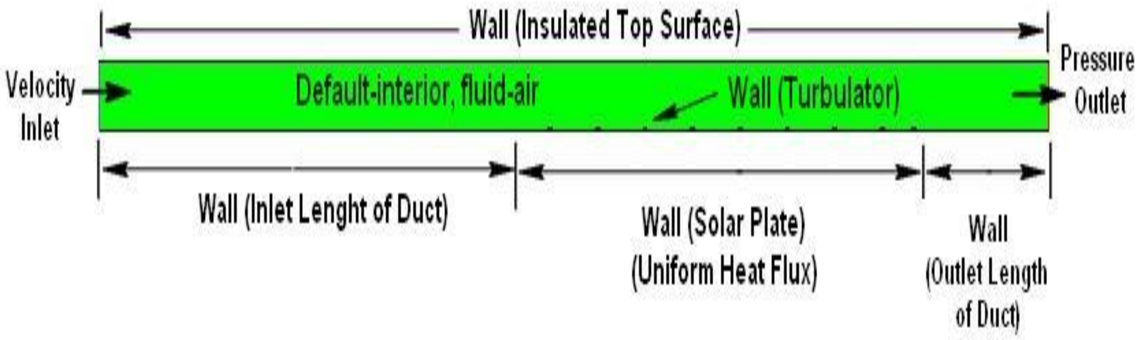

Fig. 3 Two dimensional computational domain

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 hydraulic diameter. The exit section is used after the test section in order to reduce the end effect in the test section. Roughness geometry inclined discrete shape on the roughened absorber plate has been shown in Fig. 4.1 (a,b,c)

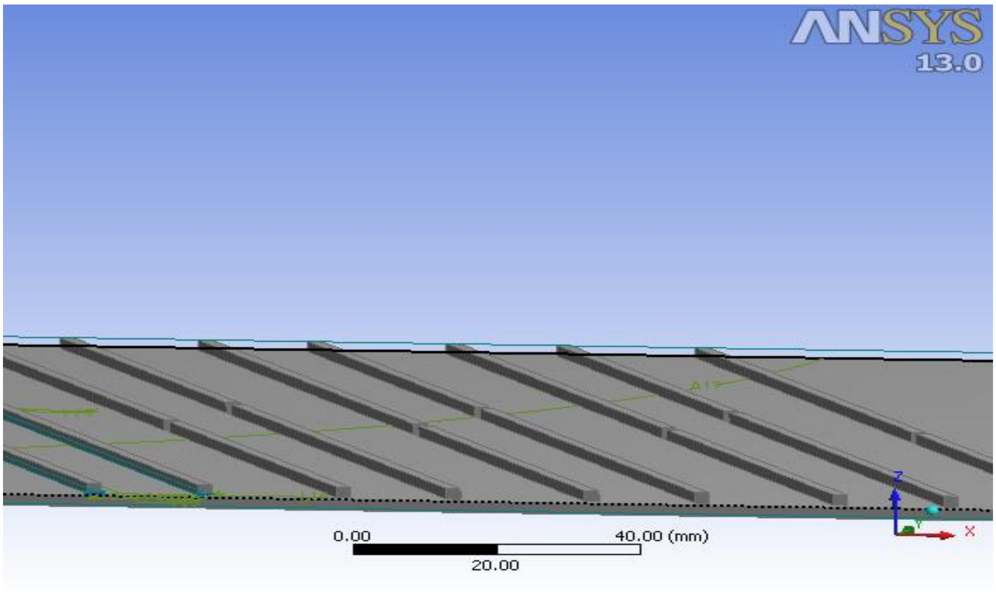

Fig. No. 4.1 (a) Roughness geometry

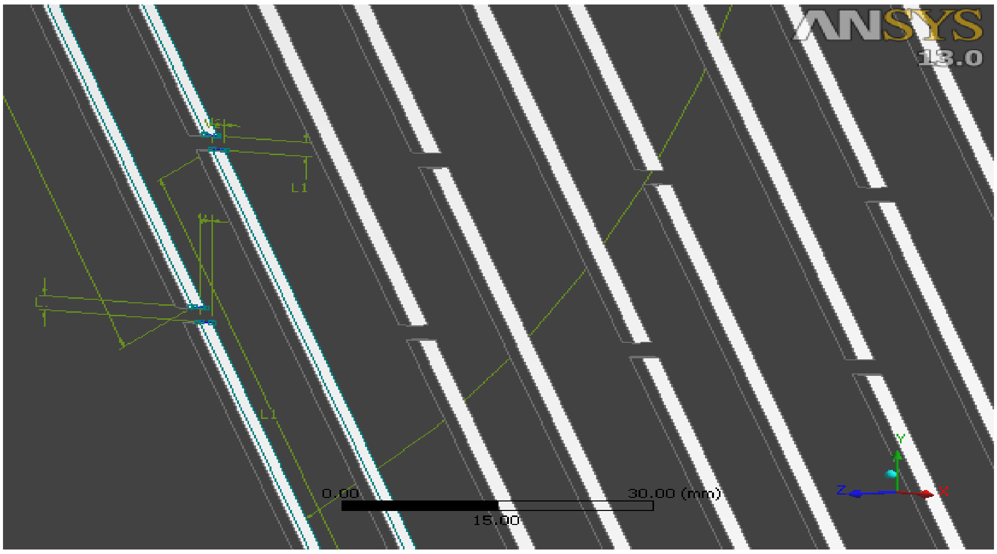

Fig. No. 4.1 (b) Roughness geometry

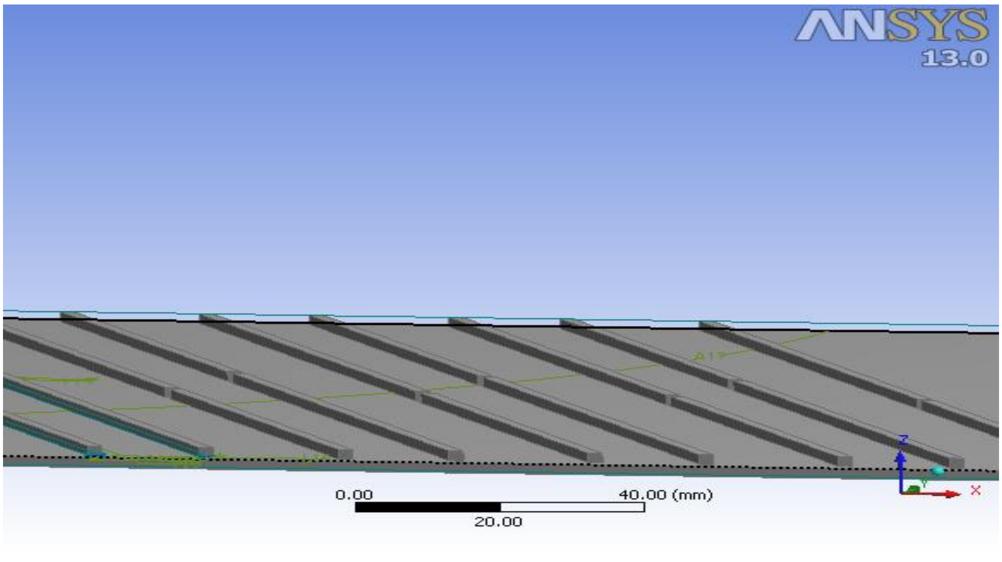

Fig. No. 4.1 (c) Roughness geometry

Table:1 Parameters of the artificially roughened solar air heater duct for computational analysis

PARAMETRE	VALUE
Roughness pitch	20mm
Heat Flux, I	1000 W/m^2
Reynolds number range	3000-15000
Duct depth, H	66mm
Duct width, W	195mm
Hydraulic diameter, D_h	0.09883 m
Rib height, e	4mm
Plate length, L	350mm

In the present study, **FLUENT Version 13. 0** is used for analysis.

Mesh Generation:

After defining the computational domain, uniform meshing is done by rectangular elements. 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. 4.2 shows the non-uniform quadrilateral meshing.

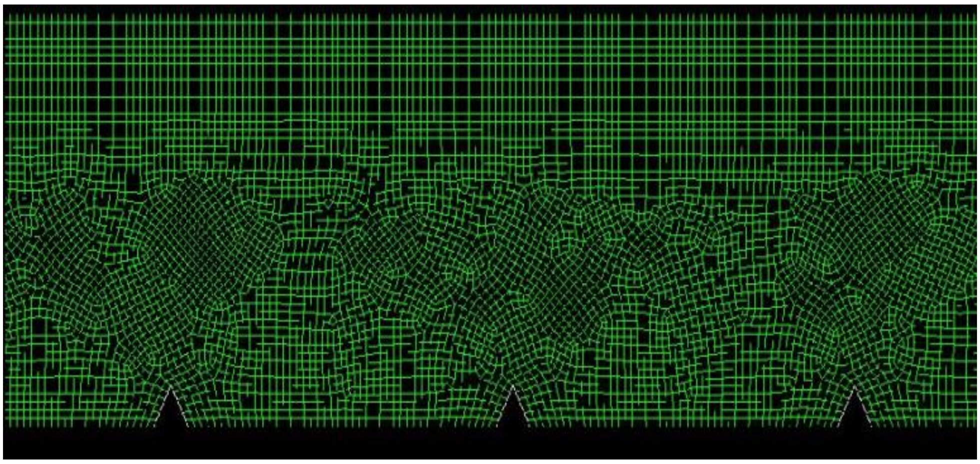

Fig no. 4.2 non-uniform quadrilateral meshing

Boundary condition:

Table 2: Detail of boundary conditions

Edge Position	Name	Type
Left	Duct Inlet	VELOCITY_INLET
Right	Duct Outlet	PRESSURE_OUTLET
Top	Top Surface	WALL
Bottom edge-1	Inlet Length	WALL
Bottom edge-2	Solar Plate	WALL
Bottom edge-3	Outlet Length	WALL
Internal Edges of rectangle	Turbulator	WALL

Solver:

ANSYS FLUENT Version 13.0 is used as a solver with k-epsilon turbulence model. The modeled turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations for k-ε model.

5. RESULT & DISCUSSUIONS

For numerical analysis the basic governing equations i.e. 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 12.1. 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.

From the figures 5.1 (a), 5.1(b) , 5.1(c) and 5.1(d) it is seen that the value of heat transfer coefficient in increases with increase in Reynolds number.

Velocity 1m/s

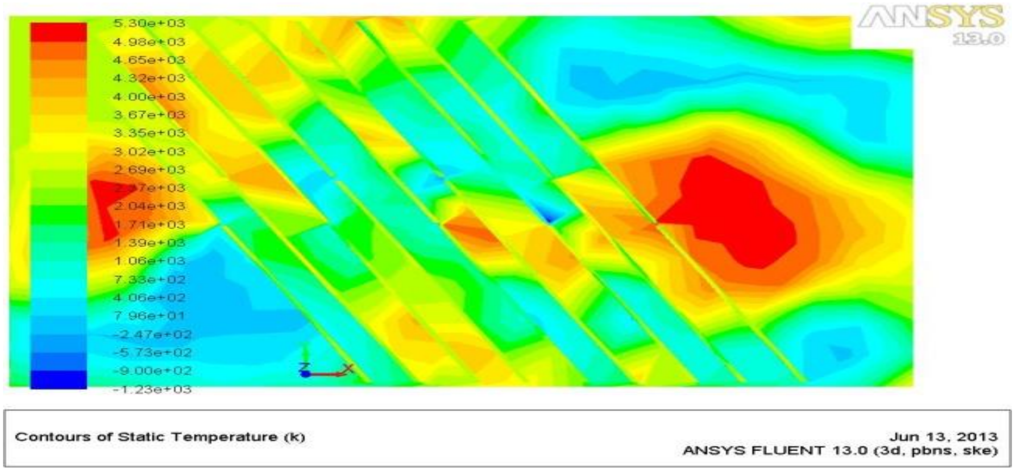

Fig. 5.1 (a) temperature contour for Reynolds number 5234.5

Velocity 2 m/s

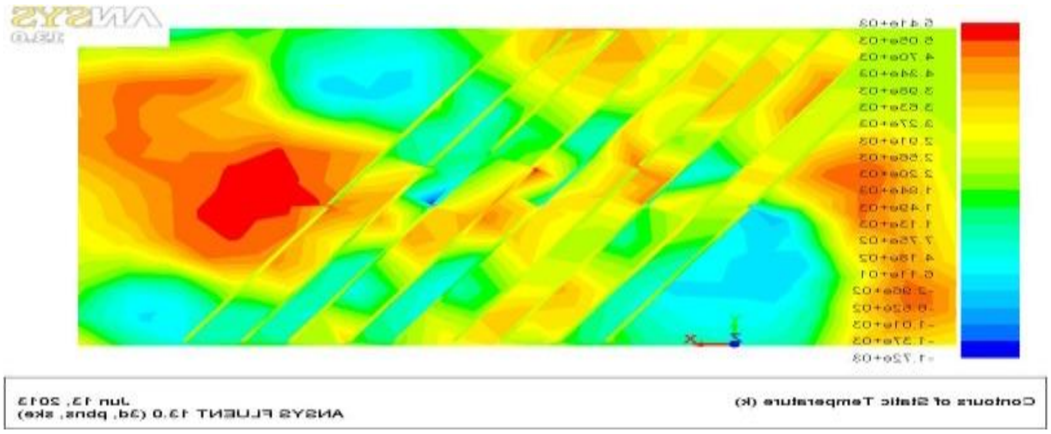

Fig. 5.1(b) temperature contour for Reynolds number 10469

Velocity 3 m/s:

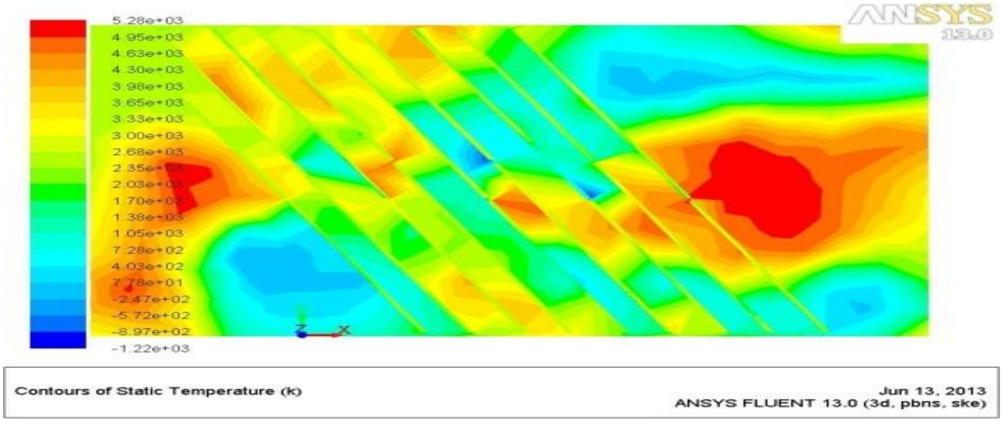

Fig. 5.1(c) temperature contour for Reynolds number 15703.5

Table 3: results for smooth surface

S.no.	Velocity (m/s)	Reynolds Number	Nus	Fs
1	1	5234.5	19.296	0.0092
2	2	10469	33.59	0.0078
3	2.5	13086.3	40.163	0.0074
4	3	15703.5	46.5	0.00706

Table 4: results for rough surface

S.no.	velocity (m/s)	Reynolds Number	Nur	Fr
1	1	5234.5	38.3	0.027
2	2	10469	71.2	0.022
3	2.5	13086.3	98.6	0.019
4	3	15703.5	132.04	0.016

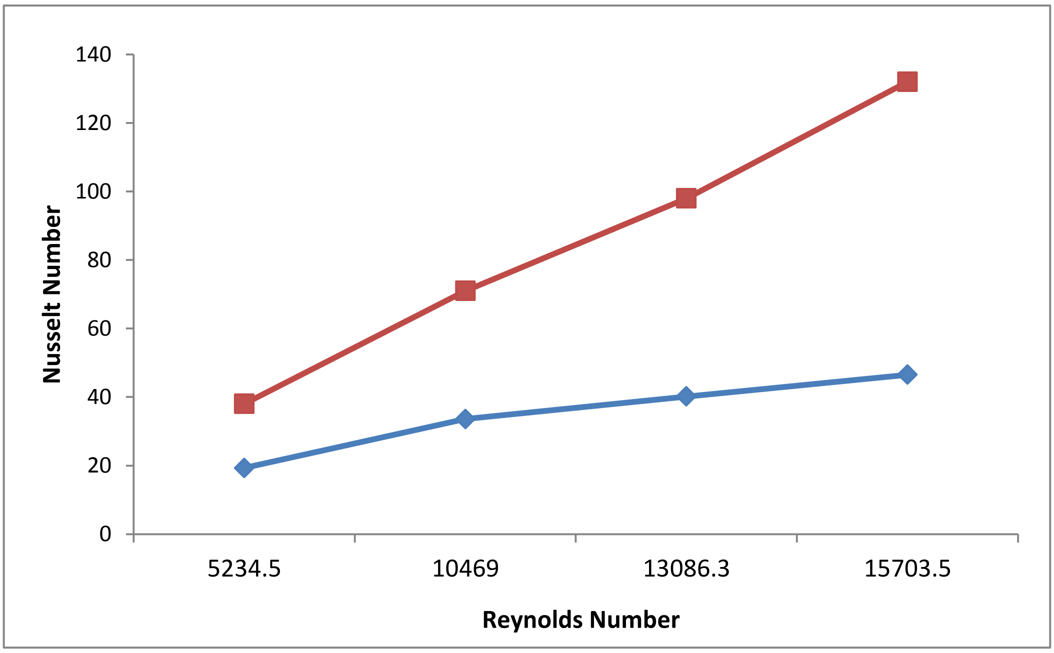

Fig. 5.1(d) Nusselt number variation for smooth and rough surface for different reynold number

6. CONCLUSIONS

The present work was undertaken with the objectives of extensive investigation on the 90^0 shaped ribs as artificial roughness on the surface of duct. Experimental setup for heat transfer and pressure lose have been design and developed. Data were collected for local heat transfer and pressure loss of these artificially roughened ducts. Results of artificially roughened duct have been compared with those of a smooth duct under similar flow condition to determine heat transfer and friction factor. The major conclusions from this investigation are given below:

1.As per the guide lines of ASHRE standard 93-77 an experiment setup comprising artificially roughened duct equipped with measuring and control instruments has been designed and developed to investigate the local heat transfer and pressure loss characteristics.

2. In the present study actual mechanism of heat transfer of smooth and artificially roughened surface has been studied using liquid crystal sheet which shows different colours corresponds to different temperature at different instant of time.

REFERENCES

[1] K.R. Aharwal, B.K. Gandhi, J.S. Saini "Experimental investigation on heat-transfer enhancement due to a gap in an inclined continuous rib arrangement in a rectangular duct of solar air heater", Renewable Energy, Volume 33, Issue 4, Pages 585-596, April 2008.

[2] M.M. Sahu, J.L. Bhagoria "Augmentation of heat transfer coefficient by using 90° broken transverse ribs on absorber plate of solar air heater" Renewable Energy, Volume 30, Issue 13, Pages 2057-2073, 2005.

[3] Varun, R.P. Saini, S.K. Singal "A review on roughness geometry used in solar air heaters", Solar Energy, Volume 81, Issue 11, Pages 1340-1350, 2007.

[4] V. SriHarsha, S.V. Prabhu , R.P. Vedula, Influence of rib height on the local heat transfer distribution and pressure drop in a square channel with 90 continuous and 60 V- broken Applied Thermal Engineering 29 (2009) 2444–2459.

[5] Bruno Facchini, Luca Innocenti, Marco Surace, Design criteria for ribbed channels: Experimental investigation and theoretical analysis, International Journal of Heat and Mass Transfer 49 (2006) 3130–3141

[6] P.R. Chandra, C.R. Alexander, J.C. Han, Heat transfer and friction behavior in rectangular channels with varying number of ribbed walls, International Journal of Heat and Mass Transfer 46 (2003) 481–495.

[7] F. Giglitto, The use of turbulence promoters in gas turbine cooling systems (in italian). Master thesis. University of Florence, 2003.

[8] D. Graham, J. Rhine, The design of transient wall heating experiments for the determination of convective heat transfer using liquid crystal thermography, ASME Paper (2000-GT-658), 2000.

[9] Diego Cavallero, Giovanni Tanda, An experimental investigation of forced convection heat transfer in channels with rib turbulators by means of liquid crystal thermography, Genova, Italy(2001)-16145.

[10] C. Camci, K. Kim, S.A. Hippensteele, A new hue capturing technique for the quantitative interpretation of liquid cry stalimages used in convective heat transfer studies, ASME J. Turbomach. 114 (1992) 765–775.

[11] Gee, D.L., and Webb, R.L., 1980, "Forced Convective Heat Transfer in Helically Rib Roughened Tube", Int. J. Heat & Mass Transfer, Vol21, pp. 127-1136

[12] Han, J.C., 1984, "Heat Transfer and Friction in Channels with Two Opposite Rib Roughened Walls", ASME J. Heat Transfer, Vol 6, pp. 774-781.

[13] Prasad, K., and Mullick, S.C., 1985, "Heat transfer Characteristics of a Solar Air Heater Used for Drying Purposes", J. Applied Energy, Vol 13, pp. 55-64.

[14] Miin, L.T., and Hwang, J.T., 1993, "Effect of Ridge Shapes on Turbulent Heat Transfer and Friction in a Rectangular Channel", Int. J. Heat & Mass Transfer, Vol 30, pp. 931- 940.

[15] Gupta, D., Solanki, S.C., and Saini, J.S., 1997, "Thermohydraulic Performance of Solar Air Heater with Roughened Absorber Plate", J. Solar Energy, Vol 61, No.1, pp. 33-42.

[16] Prasad, B.N. and Saini, J.S., Effect of artificial roughness on heat Transfer and friction factor in a solar air heater, Solar Energy, Vol. 41, No. 6, 555-560, 1988.

[17] Karwa, R., Solanki, S.C., Saini, J.S., 2001, "Thermohydraulic Performance of Solar Air Heater having Integral Chamfered Rib Roughness on Absorber Plates", Int. J. Energy, Vol 26, pp. 161-176.

[18] Momin, A.M.E., Saini, J.S., Solanki, S.C., 2002, "Heat Transfer and Friction in Solar Air Heater Duct with V-Shaped Rib Roughness on Absorber Plate", Int. J. Heat & Mass Transfer, Vol 45, pp. 3383-3396.

[19] Nikuradse, J., 1970,"Laws of Flow in Rough Pipes", NACA, Technical Memorandum 1292.