Simulation of Rectangular Duct for Performance Analysis of Trapezoidal Transverse Rib of Different Top Faced Tapered Angle

Atul Sharma, R.K. Agarwal

Abstract--Heat transfer enhancement using artificial roughness attached to the surface of duct is a effective technique in many application. This study presents the comparative change in flow characteristics between trapezoidal shaped of different top face tapered angle artificial roughness in a duct by using CFD. A commercial finite volume package ANSYS FLUENT 12.1 is used to visualize and analyze the nature of the heat transfer and flow phenomenon. The simulations were performed with transversely trapezoidal ribs placed periodically with downstream top face tapered angle of 0^0, 5^0, 10^0, 15^0 and 20^0. Different profile of transverse ribs are compared at fixed p/e, p/d and Reynolds Number(45000). Different profile of the transverse ribs are compared on the basis of pumping power requirement, hot spot region, Nusselt number ratio, friction factor ratio. It is found that Nusselt number ratio is increased on increasing the top face tapered angle from 10^0 to 15^0. Friction factor ratio is decreased on increasing the top face tapered angle. Finally, It is investigated that Performance evaluation parameter is maximum for trapezoidal rib with top face tapered angle of 20^0.

Keywords— Artificial roughness, CFD, Heat transfer enhancement, Rectangular Duct

I. INTRODUCTION

Performance improvement becomes essential especially for a duct with gas or air as flowing fluid because of large thermal resistance of gases. The application of artificial roughness in the form of fine wires and ribs of different shapes has been recommended to enhance the heat transfer coefficient by several investigators. Detailed information about the heat transfer and flow characteristics in ribbed ducts is very important in designing heat exchanger in various heating and cooling industrial applications. To enhance the heat transfer to fluid flowing in a duct artificial rib roughness has been attached to the bottom surface. It has been found that the main thermal resistance to the convective heat transfer is due to the presence of laminar sub-layer on the heat-transferring surface even in the turbulent flow region. The ribs break the laminar sub-layer and create local wall turbulence due to flow separation and reattachment between consecutive ribs, which reduce the thermal resistance and greatly enhance the heat transfer. However, the use of artificial roughness results in higher friction and hence higher pumping power requirements. Therefore, it is desirable that the turbulence should be created in the vicinity of the wall, i.e. only in the laminar sub-layer region, which is responsible for thermal resistance.

Revised Version Manuscript Received on December 30, 2015.

Atul Sharma, Assistant. Professor, Department of Mechanical Engineering, JP Institute of Engineering and Technology, Meerut, Utter Pradesh, India.

Dr. Rajeev Kumar Agarwal, Professor, Department of Mechanical Engineering, Meerut Institute of Engineering and Technology, Meerut, Utter Pradesh. India.

Hence, the efforts of researchers have been directed towards finding the roughness shape and arrangement in order to optimally enhance the heat transfer coefficient most with minimum pumping power requirement. Many experimental investigations have been carried out to determine configurations that produce optimum results in terms of both heat transfer and friction factor. Parameters involved in such experimental studies are duct aspect ratio, pitch ratio P/e; Relative roughness height, e/D, rib angle of attack α and different geometrical shape of roughness element. Several roughness shape have been investigated so far to optimize the performance of duct. On literature survey it is found that various review papers are published, which gives the detailed information about work done on the artificially roughened air duct. Hans at al.[1] and Varun at al.[2] gives the detailed information about the performance evaluation and experimental studies done on artificially roughened air heater. Bhushan at al.[3] review the methodology used in air heater for enhancing the heat transfer rate. A.K.Patil at al.[4] reviewed the roughness geometries and investigation techniques used in artificially roughened solar air heaters. It is also found that various form of roughness such as rib, wire matrix and dimples are investigated out of which ribs are best to enhance heat transfer rate. Han at all [5-7] had done experiment to find the effect of different rib shape, angle of attack, pitch to height ratio etc and found that relative roughness pitch of 10 is better. Bergles [8] developed the general correlation and compared roughness geometries such as semicircular, circular, rectangular and triangular shape ribs. It is found by studies that geometries of rib can be compared on the basis of different parameter such as pitch ratio P/e, Aspect ratio e/D, rib angle of attack[9-11]. A.Murtza at all[12] compares the laminar and turbulent heat transfer in a square duct with transverse and angled rib. It is found by previous experimental studies that for fully developed flow a recirculation region is developed just behind the transverse rib where flow is almost stagnant relative to mainstream in this region. This region is known as hot spot which leads to lower heat transfer coefficient from surface. Therefore, it is of interest to know whether the channel roughened with ribs of different shape can improve the heat transfer rate or not. There have been attempts undertaken to overcome the adverse effect by varying the geometry of ribs. Liou and Hwang [13] investigated the fully developed flow in channels roughened with three rib shapes, namely square, semicircular and triangular cross section. The results showed that the three types of rib channels had comparable thermal performance, but the square-ribbed geometry is the most likely the one to yield hot spots behind the rib. Chandra et al. [14] studied a square

*Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.*

channel with two ribbed walls for five different rib profiles. Their study illustrated that rib turbulators with greater number of sharp corners yield increasingly higher heat transfer coefficient as well as pressure drop. Arman and Rabas [15] investigated numerical code to know the effect of the rib shape on the thermal-hydraulic performance in a circular tube. He compared sine, semicircle, arc and trapezoid rib shape and found that maximum heat transfer in case of trapezoid rib shape. Karwa et al. [16] have done a experiment and develop the correlation of heat transfer and friction, for flow of air in rectangular ducts with chamfered artificial rib. They reported that the Stanton number and friction factor are maximum at chamfer angle of 15^0. Ahn [17] studied the fully developed heat transfer and friction characteristics in rectangular duct roughened by five different rib shapes, i.e., square, triangular, circular, and semicircular geometries. He concluded that the square-shaped roughness geometry has the highest friction factor; meanwhile, the triangular-shaped rib has the highest heat transfer coefficient and efficiency index. It is found by studies that transverse artificial roughness in trapezoidal shape is one of best shape for better thermo-hydraulic performance in a rectangular duct but no studies is found in which trapezoidal shape with different top face tapered angle are compared.

Other than experimental approach investigators also used flow visualization and computational techniques extensively for accurate prediction of flow and heat transfer characteristics in a ribbed duct. It is also found by studies that Computational technique used Reynolds Averaged Navier-Stokes (RANS) turbulence models. Turbulence models used in numerical simulation does not produced accurate results due to different theoretical limitations, however numerical simulation still produced fairly good results and used extensively these days because of the ease and speed the results can be obtained in no time. There are various types of Turbulence of model are available for simulation, which are compared by Oei at all [18]. According to studies v2-f2 turbulence simulation model is best for prediction but it is not suitable for predicting heat transfer problems. Liou et al. [19] used a k-e turbulence model for two dimensional model of ducts. It is found by studies that for two equations model standard k-e, RNG k-e and Realizable k-e model yield good results. Recently, with advent of computer resources and development of CFD software packages, CFD becomes very much popular to predict flow related problems. In computational simulation, a 2d or 3d model of a experimental duct is simulated and after setting the bouandary conditions the flow structure is computed by solving the mathematical equations that govern the flow dynamics. Chaube et al. [20] carried out a computational analysis using Fluent 6.1 software to investigate the flow and heat transfer characteristics of two-dimensional rib roughened rectangular ducts with one wall subjected to uniform heat flux of 1100 W/ m^2. They compared the predictions of different turbulence models with experimental results available in the literature and reported good matching of experimental results and predictions of shear stress transport (SST) K-ω turbulence model. They used SST K- ω turbulence model for analyzing the performance of nine different roughness elements and compared the predictions on the basis of heat transfer enhancement, friction characteristics and performance index. The results obtained from two-dimensional model were reported to be closer to the experimental results and these models required less memory and computational time as compared to three-dimensional models. The highest heat transfer was reported in case of chamfered ribs.

Karmare et al. [21] simulated a rectangular duct with metal grit ribs attached to a one wall of a solar heater. Commercial CFD code FLUENT 6.2.16 and Standard k-€ turbulence model were employed in the simulation. Authors compared the square , triangular and circular shape ribs with different angle of attack and reported that the absorber plate of square cross-section rib with 58∘ angle of attack was thermo hydraulically more efficient. CFD simulation results as good as experimental results. Saini [22] performed three-dimensional CFD-based analysis of an artificially roughened solar air heater having arc shaped artificial roughness on the absorber plate. FLUENT 6.3.26 commercial CFD code and Renormalization group (RNG) k-€ turbulence model were employed to simulate the fluid flow and heat transfer. Yadav and Bhagoria [23] used triangular transverse artificial rib roughness on the absorber plate to study the heat transfer behavior of an artificially roughened solar air heater by simulation CFD method. They adopting k-€(RNG) turbulence model in ANSYS FLUENT 12.1 and at Reynolds no. upto 20000. He observed 1.4 to 2.7 times enhancement in Nusselt number as compared to smooth solar air heater. Yadav and Bhagoria[24] simulated a two dimensional CFD model of artificially roughened solar air heater in which circular transverse rib are attached to the absorber plate. For simulation they are using ANSYS FLUENT 12.1 code. Yadav and Bhagoria[25] also using ANSYS FLUENT 12.1 and RNG k-e turbulence model for analysis the nature of flow. They reported that the simulation results are in good agreement with existing experimental results. Yadav and Bhagoria [26] done numerical analysis of the heat transfer and flow characteristics in an artificially roughened solar air heater having square transverse ribs roughness. It is found that performance evaluation parameter maximize at relative roughness pitch of 10.71. Yadav and Bhagoria[23-27] numerically investigated and reported that flow field, Average Nusselt number, friction factor depend on the relative roughness height and found that the performance evaluation parameter of duct is maximize at roughness height of 0.042. R. Kamali at al [28] done CFD investigation on rib shape effect on local heat transfer and flow friction characteristics of square duct with ribbed internal surface. They prepared algorithm and the computer code are applied to demonstrate distribution of the heat transfer coefficient between a pair of ribs. They reported that trapezoidal rib with decreasing height in flow direction provide higher heat transfer effect and pressure drop.

On the basis of literature review it is found that the trapezoidal rib with decreasing height in flow direction are one of the best option to enhance heat transfer, but no CFD investigation available on which is the suitable angle of trapezoidal rib with decreasing height in terms of heat transfer, pressure drop, hot spot region and performance

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

evaluation parameter. In the present study top face tapered angle of the trapezoidal rib are varies as 0^0, 5^0, 10^0, 15^0, and 20^0. The main purpose of the present study to investigate the effect of angle variation in a trapezoidal rib on friction factor, hot spot region, Nusselt number and performance evaluation parameter. For the present CFD investigation commercial software ANSYS Fluent v12.1 are used.

II. COMPUTATIONAL MODEL AND SIMULATION

A. Geometric Model

In this experiment a rectangular duct is considered, which has uniform inlet and outlet section. For better heat transfer condition ribs of square and trapezoidal geometrical shapes are attached to the bottom surface of rectangular duct. Cross-section of inlet fluid flow is 0.16m x.040 m and length is 0.888m. An isometric view of rectangular duct is shown in the figure. Total ten square ribs are transversely attached to the bottom surface of the rectangular duct.

The three dimensional (3-d) model is avoided to for saving the computer memory and computational time. So a 2D computational model is selected as suggested by Yadav and Bhagoria [11]. They suggests that a suitable 2D computational model is able to simulate well the turbulent flow. Therefore, a 2D computational model is sufficient enough to study the flow characteristics of a artificial roughened rectangular duct having square rib at bottom surface. The solution domain also well versed with ASHARE standard[16].

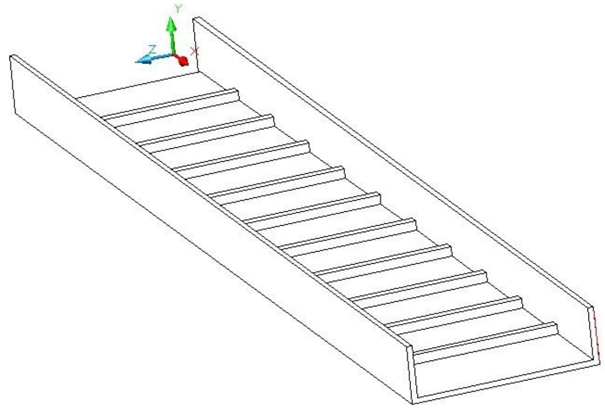

Fig.1. Isometric view of sectioned duct

Rib height (e) and rib pitch (P) are 8mm and 80mm respectively. The key dimensionless parameter such as relative roughness pitch(p/e) and relative roughness height (e/D) for rectangular duct are 10 and 0.125 respectively. These parameter are selected as their optimum value are reported in the literature[30]. A suitable Reynolds number 45000 is selected to find better thermodynamic performance. Ribs are attached at the bottom surface of the duct. Orthographic views of the geometry and computational domain of the rectangular duct is shown in the figure2. Variation of top face tapered angle $\theta = 5^0, 10^0, 15^0, 20^0$ are shown in figure 3. Configuration of artificial roughness used in this study is given in Table (i).

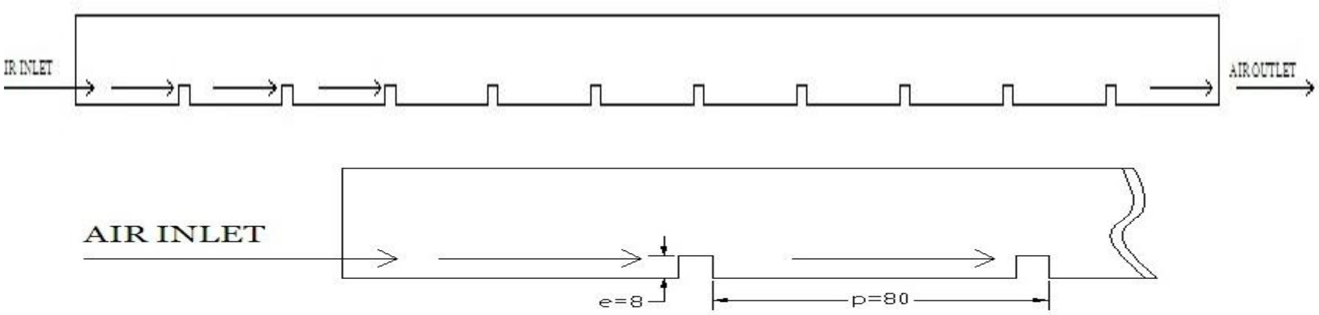

Fig.2.Computational domain of Rectangular duct with square rib

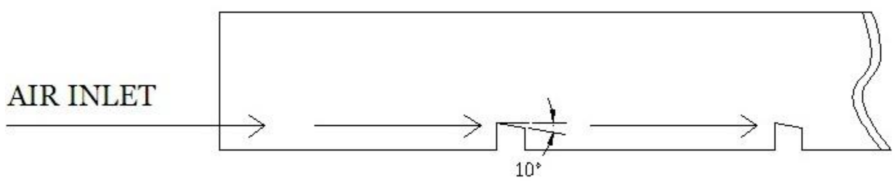

Fig.3. Variation of top face tapered angle

Table (i): Configuration of artificial roughness used in this study

Roughness Configuration	Rib Height, e(mm)	Rib Pitch, P(mm)	Hydraulic Diameter, D(mm)	Relative Roughness pitch, P/e	Relative Roughness height, e/H_d
Square rib (0^0)					
Trapezoidal Rib(5^0)					
Trapezoidal Rib(10^0)	8	80	64	10	0.125
Trapezoidal Rib(15^0)					
Trapezoidal Rib(20^0)					

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

B. Grid Generation

In this study ANSYS Workbench 12.0.1 was used to mesh the 2D computational domain that was modeled in Design Modular. A uniform quad process has been extensively adopted in computational fluid dynamics to save mesh generation time. So, in this study mesh are generated by using uniform quad method. Uniform quad mesh also suitable for performing numerical simulation for the solution of the two dimensional governing equations for mass, momentum and energy as utilized in present work. Visualization of mesh shown in figure 4.

C. Grid Independence Test:

The sub domains are called cells or elements, so it is very important to select the suitable size of cells to simulate the turbulent flow in the duct. For this purpose a grid independency test for different size of the cell element is carried out. Different size of cell element 0.30, 0.29, 0.28, 0.27, 0.26 and 0.25 are taken separately. It is found that variation in Nusselt number and friction factor are very low when moving from element size 0.28 to 0.25. So, element size 0.28 is selected for current computation. Final element number for different angle of inclination of rib is shown in Table (ii).

Table (ii). Showing element numbers for different geometries verifying grid independency validation

Geometry	Element Number
Duct with square rib(0^0)	446134
Duct with Trapezoidal Rib(5^0)	447191
Duct with Trapezoidal Rib(10^0)	449399
Duct with Trapezoidal Rib(15^0)	449681
Duct with Trapezoidal Rib(20^0)	450662

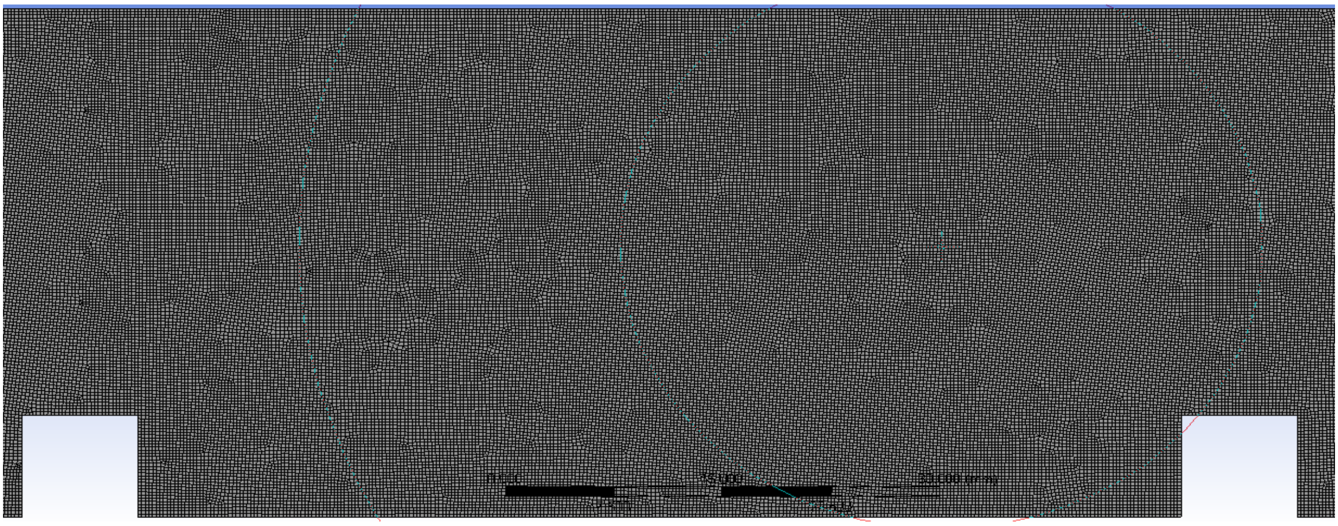

Fig. 4.Visualization of uniform quad mesh distribution in Computational Domain

Table (iii). Range of geometrical and operating parameters for CFD analysis

Geometrical and Operating parameters	Range
Test length of duct 'L'	888 mm
Width of duct 'W'	160mm
Depth of duct 'H'	40mm
Hydraulic diameter of duct 'H_d'	64mm
Duct aspect ratio 'W/H'	4
Reynold's Number, 'Re'	45000
Prandtl Number 'Pr'	0.7

D. Turbulence model and Boundary Conditions:

For selecting suitable turbulence model solution are generated by using two- equations models such as Standard k- ε, RNG k- ε and Realizable K-ε(RKE) . After comparing the results with experimental data it is found that Realizable K-ε(RKE) turbulence model is best for current study.

In the present study, the boundary conditions for the different edges are created during developing the geometry of the mesh. The model has a velocity inlet on one end face and a pressure outlet on the other. A uniform air velocity 10.265m/s (corresponding to value of Reynolds number) is introduced at the inlet, while a pressure outlet condition with fixed pressure of 1.013×10^5 Pa is applied at the exit. Constant velocity of air is assumed in the flow direction. The temperature of air inside the duct is also taken as 300 K at the inlet of the duct. Impermeable boundary and no slip wall conditions have been selected. Top wall and side wall

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

are kept at adiabatic wall condition. The bottom wall is heated with uniform heat flux of 1000 W/m^2 for computational simulation. It is also assumed that the physical properties of air remains constant at bulk mean temperature. The thermo physical properties of working fluid(air) corresponding to temperature T = 300K are shown in table (iv).

Table (iv). Thermo-physical properties of the air for CFD analysis

Thermo-physical Properties	Air
Density, ‘ρ’ (Kg-m^{-3})	**1.225**
Specific Heat, ‘C_p’ (Jkg^{-1}k^{-1})	1006.43
Thermal Conductivity, ‘k’ (wm^{-1}k^{-1})	0.0242
Viscosity, ‘μ’ (Nm^{-2})	1.7894x10^{-5}

E. Solution Method

Now generated mesh for computational domain with defined boundary conditions imported in Fluent Set up 12.0 version, where steady state two dimensional turbulent equations of continuity, momentum and energy are solved for analysis. In the present study pressure-based solver (PBCS) is selected for superior performance. Now governing equations of continuity, momentum and energy are solved by using pressure-velocity linked semi-implicit method for pressure linked equations i. e. SIMPLE algorithms. For discretization of the equation second order upwind is used for 2nd order accuracy. The gradients of solution variables at cell centers is determine by using least square cell based approach. The convergence criteria for all the dependent variables are specified as 0.001. Whenever convergence problems are noticed, the solution is started using the first-order upwind discretization scheme and continued with the second-order upwind scheme. Convergence has been achieved within 200 iterations, where the normalized residual remained constant.

III. VAILIDATION OF SIMULATED MODEL

For validation of simulated model, results obtained by the calculation after post –processing of simulated model of smooth rectangular duct are compared by empirical relations used for finding the friction factor and nusselt number for smooth rectangular duct of same hydraulic diameter. Correlations used for finding Nusselt number and friction factor for smooth duct are following:

Dittus-Boelter equation:

$$Nu_s = 0.023 Re^{0.8} Pr^{0.4}$$

The Blasius equation:

$$fs = 0.0791\ Re^{-0.25}$$

Nusselt number and friction factor for smooth rectangular duct are calculated from simulated model and then compared with the value found by correlations at different Reynolds number. The nature of simulated data and correlations match with a discrepancy of 8% to 18%.

IV. RESULT AND DISCUSSIONS

In the present study trapezoidal rib of decreasing height are attached to the bottom surface transversely, so that the turbulence created in vicinity of lower surface. Heat transfer characteristics for different angles of ribs are compared to gives a better idea of the use of suitable angle of trapezoidal rib on the basis of pressure drop, average surface Nusselt number ratio, friction factor ratio and performance evaluation parameter.

A. Pressure Drop

Static pressure are measured at inlet and outlet section of rectangular duct .It is found that the ribbed wall reduce the flow velocity which results in more pressure drop in comparison to the smooth wall duct. Figure below show that the maximum pressure drop found in case of square rib. The analysis also concludes that the least pressure drop found in case of trapezoidal rib with 20^0 inclination in the same range of Reynolds number. It means that pumping cost increase in case of ribbed duct and minimum increase in pumping cost in case of trapezoidal rib with 20^0 inclination.

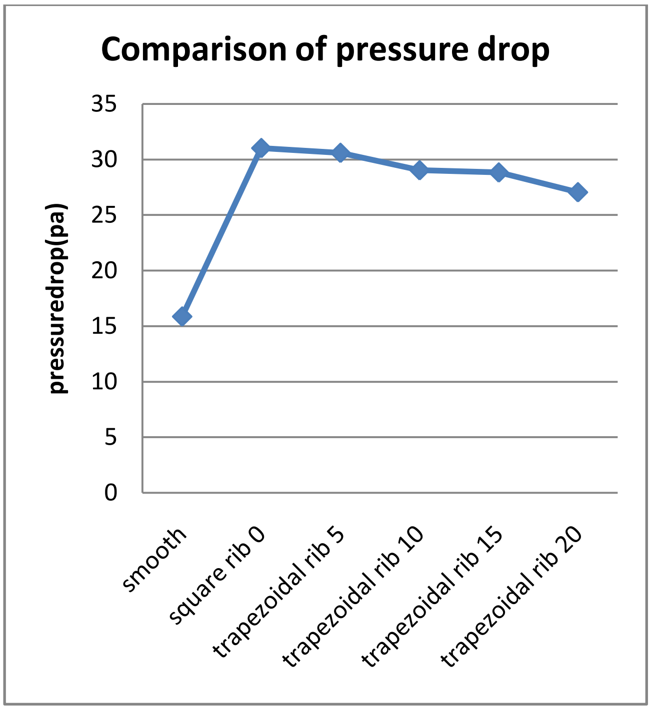

Fig. 5. Pressure drop in duct at fixed Reynolds No.=45000 for different top face tapered angle (a) smooth (b) 0^0 (c) 5^0 (d) 10^0 (e) 15^0 (f) 20^0

*Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.*

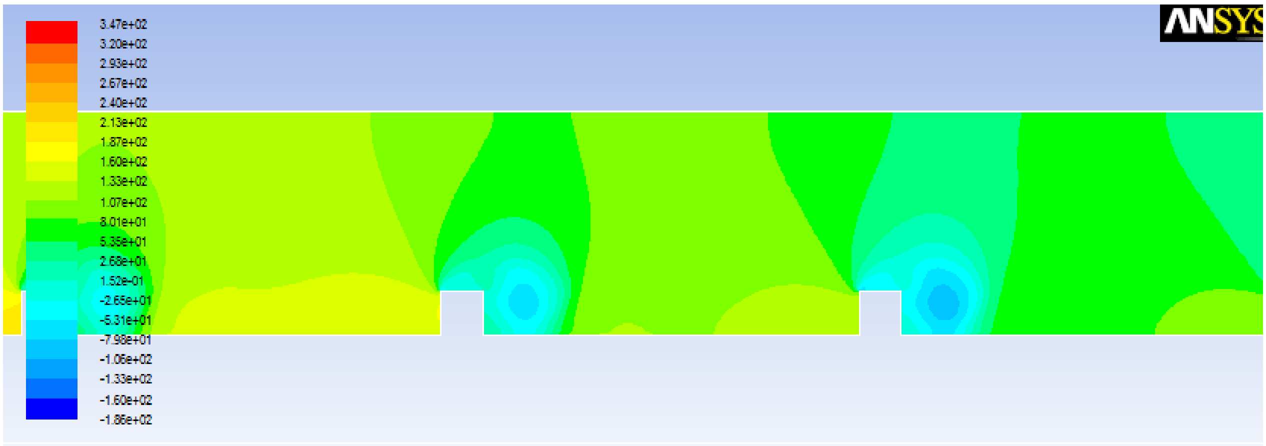

Fig.6(a)

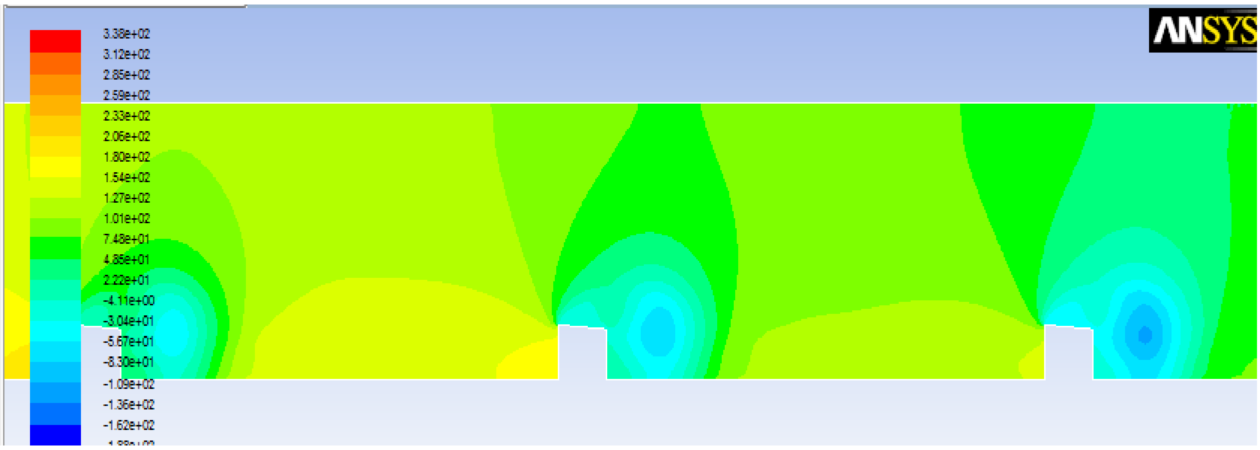

Fig.6 (b)

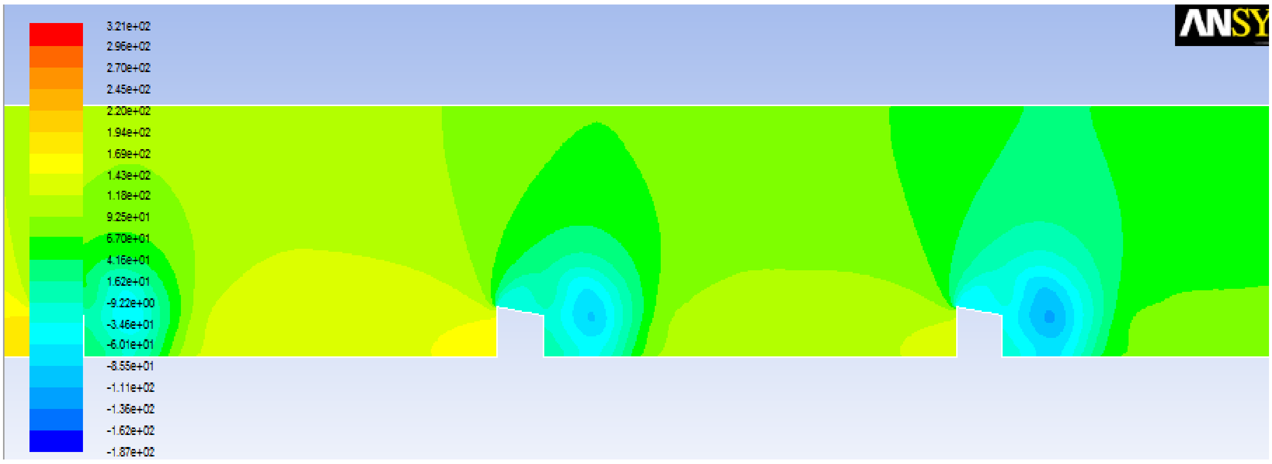

Fig. 6 (c)

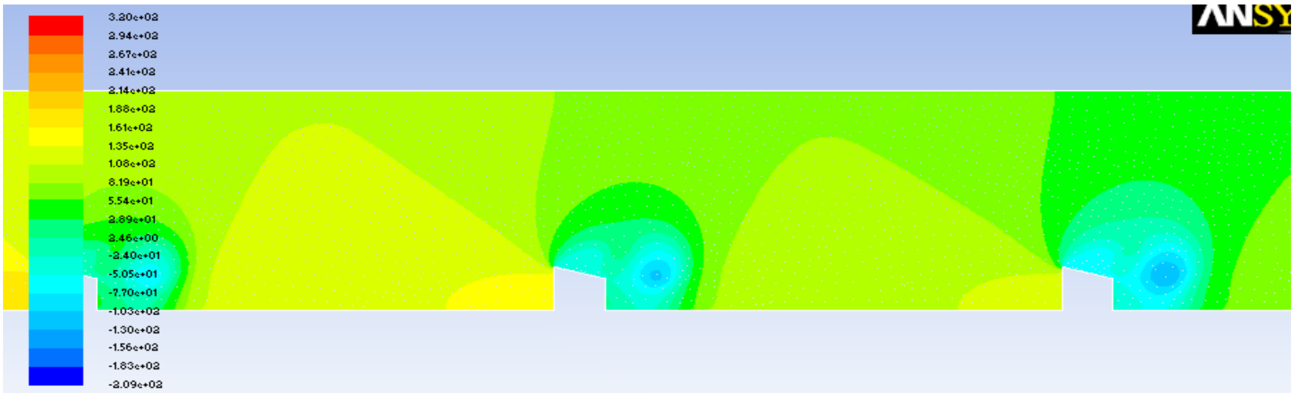

Fig.6 (d)

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

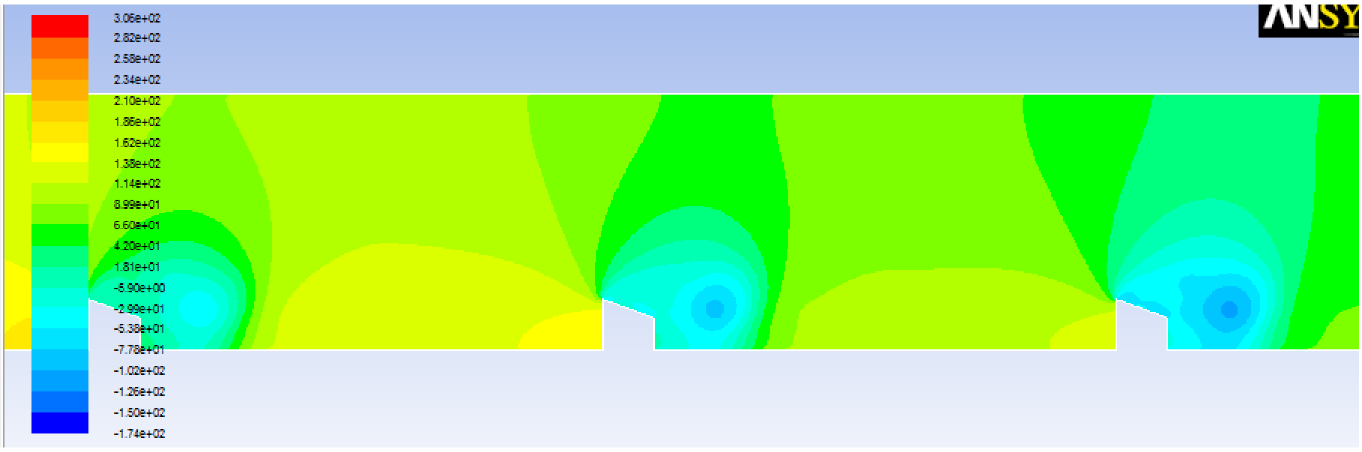

Fig.6 (e)

Fig. 6. Contours of pressure drop between two consecutive ribs for different top face tapered angle (a) 0^0 (b) 5^0 (c) 10^0 (d) 15^0 (e) 20^0

B .Hot Spot Region

Hot spot region is developed just behind the transverse rib in downstream region the transverse rib due to recirculation. In this region fluid flow is almost stagnant relative to mainstream flow in recirculation zone which leads to reduce the heat transfer rate. Therefore, reduced hot spot region is required to improve the heat transfer rate. Reduction in hot spot region for duct roughened with ribs of different chamfered angle can be seen in counters of static temperature as shown in figure 7.

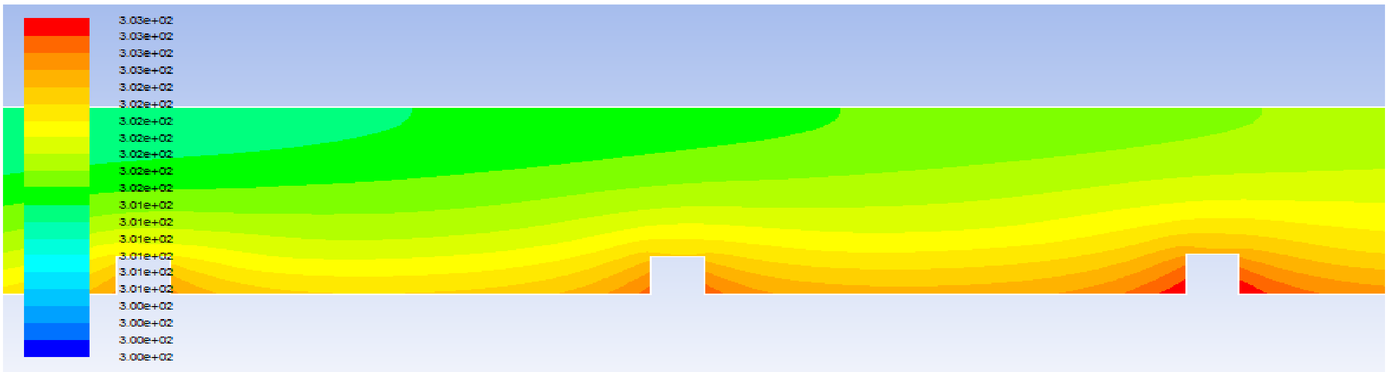

Fig.7(a)

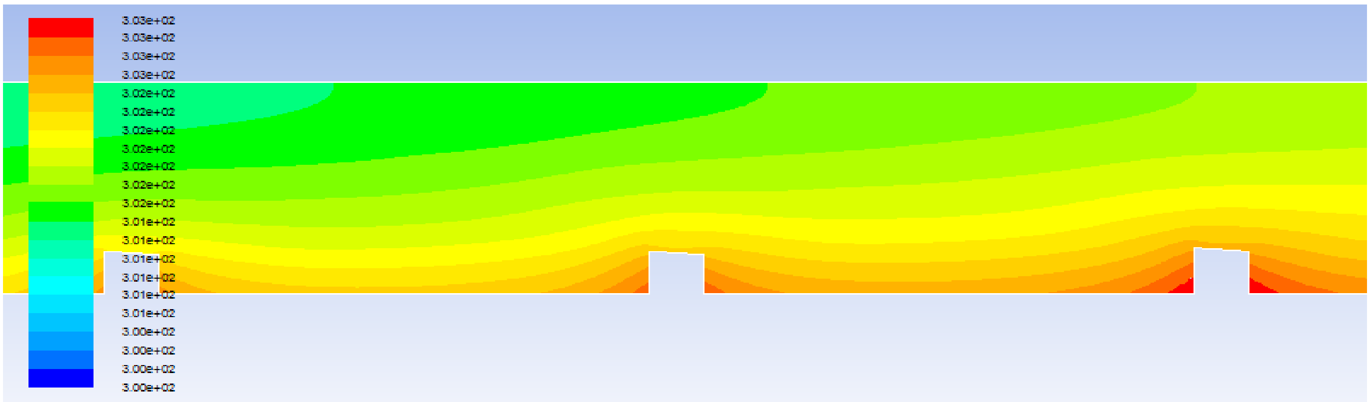

Fig.7(b)

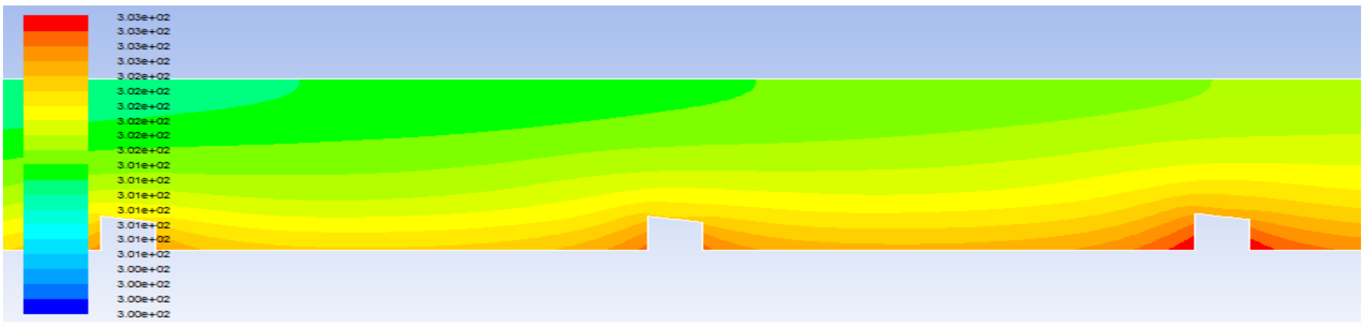

Fig.7(c)

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

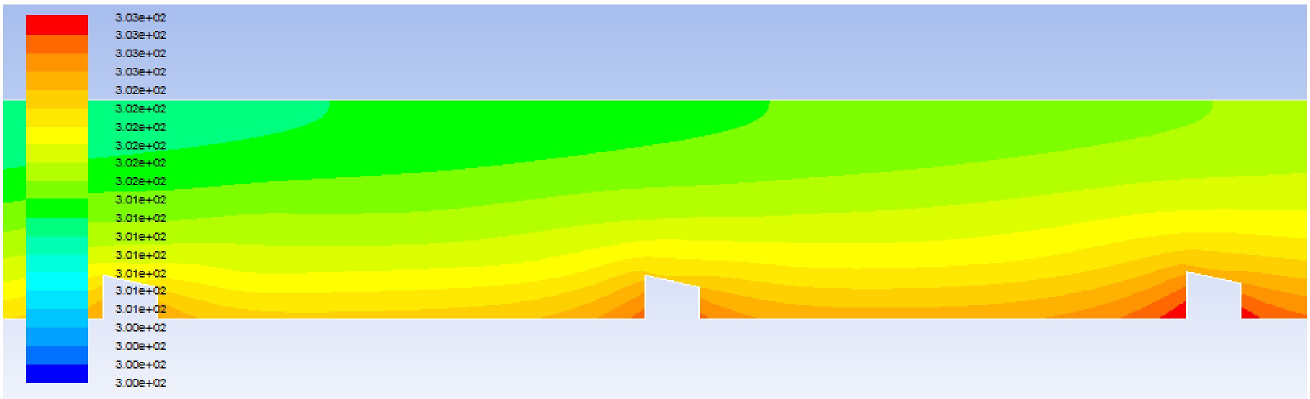

Fig.7(d)

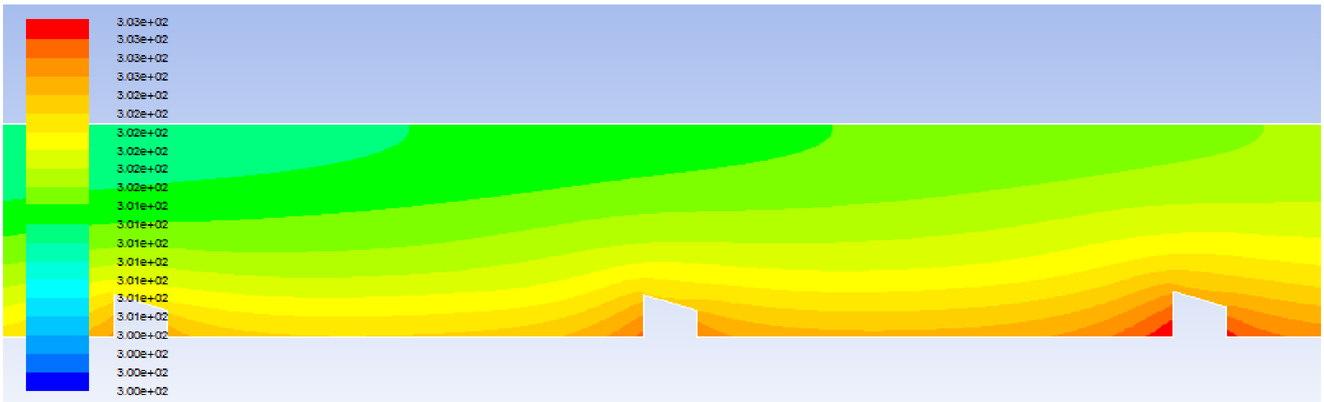

Fig.7(e)

Fig.7. Contours of Static Temperature in duct for ribs of different top face tapered angle (a) 0^0 (b) 5^0 (c) 10^0 (d) 15^0 (e) 20^0

C. Nusselt Number Enhancement Ratio:

Nusselt number represents the enhancement of heat transfer through a fluid layer as result of convection relative to conduction across the same fluid layer. Larger the Nusselt number, the more effective the convection means higher heat transfer coefficient. As discussed earlier that transverse Ribs across the duct break the laminar sub layer & buffer layer and create local wall turbulence due to flow separation, reversal and reattachment between the ribs, which helps to exchange heat among the air particles. So, air particles absorb more heat while flowing in the rib wall than that of the smooth wall.

That is why, duct which is roughed with artificially roughness having higher Nusselt number as compared to smooth duct.

As discussed earlier performance evaluation parameter for artificially roughed rectangular duct is directly proportional to the Nusselt number enhancement ratio. In the present study, local Nusselt number of rectangular duct with artificial roughness are finding at 19 different location between 7^{th} and 8^{th} transverse rib by measuring temperature at these locations. Average of these value is assumed to Nusselt number of the duct.

To know the effect of square sectioned and trapezoidal sectioned rib with variable top face taper angle on Nusselt number ratio a graph is drawn at fixed value of relative roughness pitch, relative roughness height and Reynold's number. Variation of Nusselt number(Nu_r) shown in graph given below in figure 8:

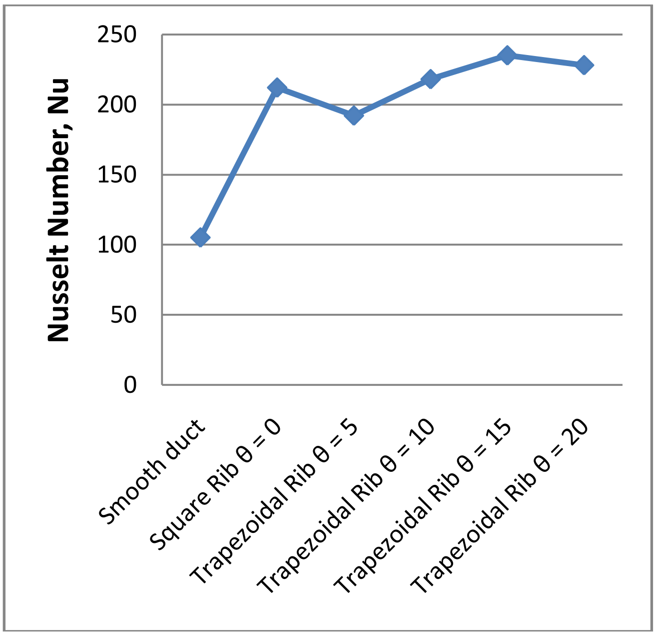

Fig. 8. Nusselt Number (Nur)for duct at fixed Reynolds Number for ribs of varried top face tapered angle (a) Smooth duct (b) 0^0 (c) 5^0 (d) 10^0 (e) 15^0 (f) 20^0

It can be seen by the graph that local Nusselt number is reduce in case of trapezoidal rib with top face taper angle of 5^0 and increase in case of trapezoidal rib with top face taper angle of 10^0, 15^0 and 20^0 as compared to square rib. Variation of Nusselt Number enhancement ratio also shown in figure 9.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

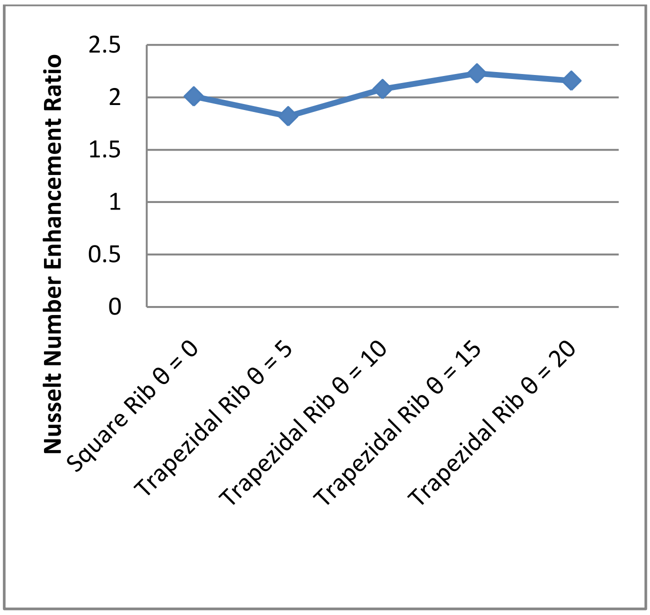

Fig.9. Nusselt Number enhancement ratio for different top face tapered angle

D. Friction Factor

Friction factor is the another important parameter which will effect the performance of the rectangular duct. Duct roughened with artificial transverse rib having higher friction factor as compared to smooth duct. Figure 10 drawn to show the variation in friction factor for the artificial roughened duct with trapezoidal transvese rib of different top face tapered angle at fixed parameter such as relative roughness pitch, relative roughness height and reynolds number. It is seeen that the friction factor value is maximum in case of square sectioned transverse rib than it is decreased in case of trapeozoidal rib of top face tapered angle of 5^0 , 10^0, 15^0 and 20^0 respectively as shown in graph.

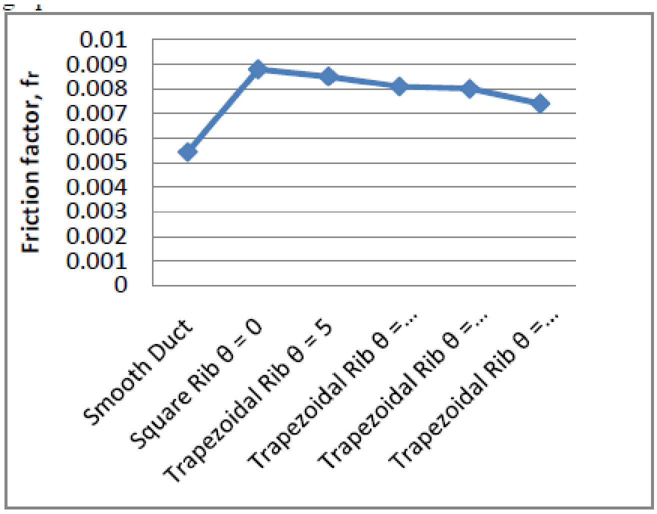

Fig.10. Variation of friction factor, f_r for trapezoidal rib of different top face tapered angle at(a) Smooth duct (b) 0^0 (c) 5^0 (d) 10^0 (e) 15^0 (f) 20^0.

D. Performance Evaluation Parameter

Figure 8 and figure 9 shows the Nusselt Number enhancement ratio and friction factor for square sectioned rib and trapezoidal rib of different top face tapered angle at fixed Reynolds number. It is evident from figure that Nusselt Number ratio is increased on increase of top face tapered angle and friction factor of artificial roughened duct is decreased on increase of top face tapered angle. However, Nusselt Number enhancement ratio decreased in case of trapezoidal rib of top face taper angle of 5^0, than its increase upto 15^0 after this a minor depression shown at an angle of 20^0. Friction factor also slightly increased at top face taper angle of $15^{0.}$ Another well known parameter to compared the rectangular duct with different rib is Performance evaluation parameter as proposed by Webb and Eckert [29]. Figure 11 shows the variation of Performance evaluation parameter at fixed Reynolds number for rectangular duct having transverse rib attached at the bottom surface. Performance evaluation parameter is increased due to application of transverse rib of square shape. It is usually found more than unity. The value of performance parameter varies from 1.57 to 1.97 at the same Reynolds number. It is also observed that the performance evaluation parameter decreased at top face tapered angle of trapezoidal transverse rib of 5^0 as compared to square transverse rib., than it is increased for tapered angle of 10^0 15^0 and 20^0 Finally it is found that the performance evaluation parameter is maximum for trapezoidal rib with top face tapered angle of 20^0. Performance evaluation parameter for trapezoidal rib(20^0) is almost increased by 14% as compared to square rib.

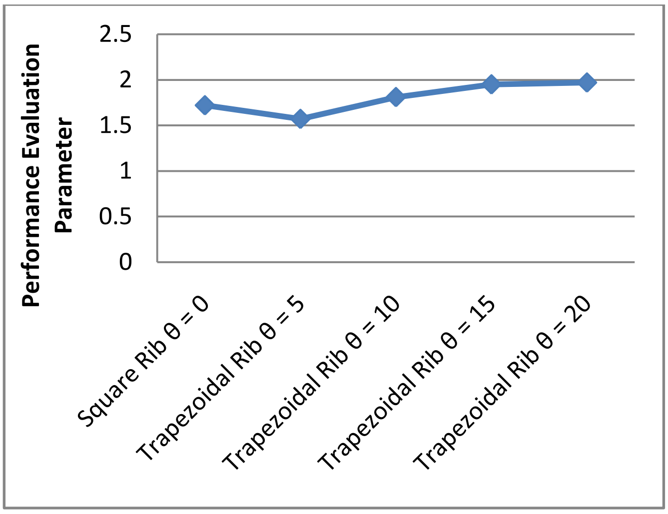

Figure11. Variation of Performance Evaluation parameter for trapezoidal rib of different top face tapered angle at (a) 0^0 (b) 5^0 (c) 10^0 (d) 15^0 (d) 20^0.

V. CONCLUSIONS

In present study, Rectangular duct roughened with transverse rib of trapezoidal shape is simulated using FLUENT. For CFD analysis a two dimensional model is created. Square sectioned and trapezoidal sectioned rib with different top face tapered angle are compared at fixed p/e ratio, p/d ratio and higher Reynolds number (45000). For comparison of flow behaviors characteristics of artificially roughened duct contour maps of turbulent kinetic energy, turbulent intensity , velocity and pressure are presented. Square sectioned and trapezoidal sectioned rib with different top face tapered angle are compared on the basis of pumping power requirement, hot spot region, Nusselt number ratio, friction factor ratio and performance evaluation parameter.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

Following relevant facts are concluded after CFD analysis.

1. By measuring the pressure at inlet and outlet of rectangular duct it is found that the pressure drop is maximum for square sectioned transverse rib. Pressure drop is further decrease on decreasing the top face taper angle. Pressure drop is almost same for tap face tapered angle of 10^0 and15^0 and it is minimum for top face taper angle of 20^0. So it is concluded that the pumping requirement enhancement is minimum for trapezoidal transverse rib of top face taper angle of 20^0 as compared to smooth duct.

2. By comparing the contours of static temperature it is found that Hot spot region reduced on reduction of top face taper angle of trapezoidal transverse rib. Hot spot region is maximum in case of square sectioned transverse rib and it is minimum in case of trapezoidal rib with 20^0 angle.

3. The maximum Nusselt number enhancement ratio is found in case of trapezoidal rib with 15^0 top face tapered angle. Nusselt number enhancement ratio is increased on increasing the top face tapered angle from 10^0 to 15^0 but it is slightly decreased after15^0.

4. Friction factor is decreased on increasing the top face tapered angle of trapezoidal transverse rib. Friction factor ratio is found minimum in case of trapezoidal transverse rib with 20^0.

5. It is found that the Rectangular duct roughened with trapezoidal transverse rib with top face tapered angle of 20^0 having the maximum enhancement in Performance evaluation parameter. So, trapezoidal transverse rib of 20^0 inclination can be used for heat transfer enhancement.

REFERENCES

1. Hans V.S., Saini, R.P., Saini, J.S. , "Performance of artificially roughened solar air heaters-A review", Renewable and Sustainable Energy Reviews, 13, 18541869,2009.
2. Varun, Saini R.P., Singal S.K., "A review on roughness geometry used in solar air heaters", Solar Energy, 81,1340-1350, 2007.
3. Bhushan B., Singh R., "A review on methodology of artificial roughness used in duct of air heaters", Energy, 35, 202-212, 2010
4. 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 Research, Vol-2,2012 .
5. Han J.C., Glicksman L.R., Rohsenow W.M., "Heat transfer and friction for rib roughened surfaces" International journal of Heat mass transfer, 21, 11431156,1978.
6. Han J.C., "Heat transfer and friction in channels with opposite rib roughened walls", Trans. ASME Journal of Heat Transfer, 106, 774-781, 1984.
7. Han, J.C., Park, J.S., Lei, C.K., "Augmented heat transfer in rectangular channel of narrow aspect ratios with rib turbulators", International journal of Heat mass transfer, 32/9, 1619-1630, 1989.
8. Ravigururajan T.S., Bergles A.E., "General correlations for pressure drop and heat transfer for single phase turbulent flow in internally ribbed tubes", J. ASME, 52, 9-20, 1985.
9. L. Wang, B. Sundén, Experimental investigation of the effect of rib shape on local heat transfer in a square duct by means of liquid crystal thermography, Proceedings of the sixth World Conference on Experimental Heat Transfer, Fluid Mechanics, and Thermodynamics, Matsushima, Miyagi, Japan, 2005.
10. Y.J. Hong, S.S. Hsieh, Heat transfer and friction factor measurements in ducts with staggered and in-line ribs, ASME J. Heat Transfer 115 (1993) 58–65.
11. M.E. Taslim, T. Li, D.M. Kercher, Experimental heat transfer and friction in channels roughened with angled, V shaped, and discrete ribs on two opposite walls, ASME J. Turbomach. 118 (1998) 20–28.
12. A. Murata, S. Mochizwki, Comparison between laminar and turbulent heat transfer in a stationary square duct with transverse or angled rib turbulators, Int. J. Heat Mass Transfer 44 (11) (2001) 1127–1141.
13. T.M. Liou, J.J. Hwang, Effect of ridge shapes on turbulent heat transfer and friction in rectangular channel, Int. J. Heat Mass Transfer 36 (4) (1993) 931–940.
14. P.R. Chadra, M.L. Fontenot, J.C. Han, Effect of rib profiles on turbulent channel flow heat transfer, AIAA J. Thermophys. Heat Transf. 12 (1) (1998) 116–118.
15. B. Arman, T.J. Rabas, Disruption shape effect on the performance of enhanced tubes with the separation and reattachment mechanism, ASME Symposium, HTD-202, Enhanced Heat Transfer, 1992, pp. 67–75.
16. Karwa R, Solanki SC, Saini JS. Heat transfer coefficient and friction factor correlations for the transitional flow regime in rib-roughened rectangular ducts. International Journal of Heat and Mass Transfer 1999;42:1597–615.
17. S.W. Ahn, The effect of roughness type on friction factors and heat transfer in roughened rectangular duct, Int. Commun. Heat Mass Transf. 28 (7) (2001)933–942.
18. A. Ooi, G. Iaccarino, P.A. Durbin, M. Behnia, Reynolds averaged simulation of flow and heat transfer in ribbed ducts, Int. J. Heat Fluid Flow 23 (2002) 750–757.
19. T.-M. Liou, J.-J. Hwang, S.-H. Chen, Simulation and measurement of enhanced turbulent heat transfer in a channel with periodic ribs on one principal wall, Int. J.Heat Mass Transf. 36 (2) (1993) 507–517.
20. Chaube A, Sahoo P.K., Solanki S.C., "Analysis of heat transfer augmentation and flow characteristics of a solar air heater", Renew Energy, 31,317-331, 2006
21. Karmare S.V., Tikekar A.N., "Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD", Solar Energy, 84, 409-417, 2010.
22. Kumar S., Saini R.P., "CFD based performance analysis of a solar air heater duct provided with artificial roughness", Renewable Energy, 34, 1285-1291, 2009.
23. A. S. Yadav and J. L. Bhagoria, "A CFD analysis of a solar airheater having triangular rib roughness on the absorber plate,"International Journal of ChemTech Research,vol.5,no.2,pp.964–971, 2013.
24. A. S. Yadav and J. L. Bhagoria, "A CFD based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness on the absorber plate," Energy,vol. 55, pp. 1127–1142, 2013.
25. A. S. Yadav and J. L. Bhagoria, "A CFD based heat transfer and fluid flow analysis of a conventional solar air heater," Journal of Engineering Science and Management Education,vol.6,no.2, pp.137–146,2013.
26. A. S. Yadav and J. L. Bhagoria, "Numerical investigation of flow through an artificially roughened solar air heater," International Journal of Ambient Energy,2013.
27. A. S. Yadav and J. L. Bhagoria, "A numerical investigation of turbulent flows through an artificially roughened solar air heater," Numerical Heat Transfer A. In press.
28. R. Kamali and A.R. Binesh, " The importance of rib shape effects on local heat transfer and flow friction characteristics of square ducts with ribbed internal surfaces." Journal of International communications in heat and mass transfer 35(2008) 1032-1040.
29. R. L. Webb and E. R. G. Eckert, "Application of rough surfaces to heat exchanger design," International Journal of Heat and MassTransfer,vol.15,no.9,pp.1647–1658,1972.W. H. McAdams, Heat Transmission, McGraw-Hill, NewYork.
30. Md. Shaukat Ali, Karmveer and B.K.Gandhi, " Measurement of heat transfer coefficient in a rectangular duct with array of rib using liquid crystal photography" International journal of engg. Trends, Issue2, vol.5 july 2012.ISSN 2249-6149

Atul Sharma is presently serving as Assistant Professor in the department of Mechanical Engineering at J.P. Institute of Engineering & Technology, Meerut, India. He received the AMIE degree in Mechanical Engineering from IEI, Kolcata, west Bangal, India in 2005 and M.Tech degree from UPTU, Lucknow, Uttar Pradesh, India. He held 12 years teaching experience. His research interest areas are refrigeration and air conditioning , non conventional energy resources , Computational Fluid Dynamics and Heat Transfer.

Dr. Rajeev Agarwal is presently serving as Professor in the department of Mechanical Engineering at M.I.E.T. Meerut, India. He received B.Tech from M.M.M. Gorakhpur, M.Tech from I.I.T. Roorkee. He received Doctorate in 2011. He held total 27 years of experience. His research interest areas are Energy Conservation and Thermal Science.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.