RESEARCH ARTICLE   OPEN ACCESS

Determination Of Heat Transfer Coefficients and Friction Factors For Turbulent Flow Through Rectangular Ducts With Triangular Ribs

Abhishek Kumar*, Mayank Kumar Dwivedi**

*(Assistant Professor, Mechanical Engineering Department, KNP College of Science and Technology, Misrod, Bhopal MP)
** (M.Tech. Scholar, Mechanical Engineering Department, KNP College of Science and Technology, Misrod, Bhopal MP)

--************************----------------------------------

Abstract:

Computational study was carried out to determine average heat transfer coefficients and friction factors for turbulent flow through rectangular ducts with triangular ribs. A commercial finite volume package ANSYS FLUENT 16 is used to analyze and visualize the nature of the flow across the ribbed duct. The fluid in the duct was air, and the average heat transfer coefficients were determined by measuring the overall heat transfer coefficients of the duct. To attain fully developed conditions at the entrance and exit, the duct was built with additional lengths before and after the test section. The walls of the air side were ribbed with an array of short ribs. The second wall of the duct was well insulated and can be considered adiabatic. The results are presented in dimensionless form, in terms of Nusselt numbers and friction factors as functions of the Reynolds number. It was found that the ribs increase both the heat transfer and the pressure drop compared with a smooth duct.

Keywords **—Roughened Duct, Nusselt Number, Heat transfer, Pressure Drop, CFD**

--************************----------------------------------

I. INTRODUCTION

The rectangular duct geometry is commonly employed in engineering applications, such as compact heat exchangers, solar collectors, and others. A survey of the available published literature on the heat transfer and pressure drop characteristics of rectangular ducts reveals a great lack of information about turbulent conditions. With regard to laminar heat transfer and pressure drop, the situation is better. Turbulent heat transfer and fluid flow characteristics of air in rib-roughened tubes, annuli, ducts and between parallel plates have been studied extensively because of their important applications. Such a heat transfer enhancement method is widely used in cooling passages of gas turbine blade, compact heat exchangers, fuel elements in advanced gas-cooled reactor electronic cooling devices, etc. Taking the cooling passages is usually approximated by a rectangular duct with a pair of opposite rib-roughened walls.

Applications of heat transfer enhancement concepts can be found in gas turbine airfoils, solar air heaters, electronic cooling, etc. A common technique to enhance heat transfer is by installing turbulence promotors (''rib turbulators'') on the smooth walls. The rib turbulators generate secondary flows which increase near-wall shear in the vicinity of the ribs and these secondary flows also interact with channel side walls to increase turbulent transport of energy from relatively hotter walls by forming vortex or vortices. One application of rib turbulators as heat transfer enhancement technique is found in gas turbine airfoils. The gas turbine airfoils are subjected to elevated heat loads on both pressure and suction side walls. Hence rib turbulators are installed on pressure and suction side internal walls to increase heat transfer in order to increase the heat transfer rates between internal walls and coolant. Rib

turbulators also result in increase in wetted surface area which enhances the overall conductance. Several studies have been carried out in the past on heat transfer enhancement by various cooling designs, such as, ribbed channel with bleed holes, ribbed channel with grooves, rib dimpled compound channels, jet impingement, jet impingement with effusion holes, dimpled channel, jet impingement onto dimpled target surface etc. The use of artificial roughness on a surface is an effective technique to enhance heat transfer coefficient also has good application in design and development of efficient solar air heaters. Many investigations have demonstrated the effects of different rib configurations on heat transfer coefficient between absorbers plate and air flowing in solar air heaters, numerically and experimentally, in order to improve the heat transfer capability of solar air heater ducts. Heat transfer enhancement due to rib turbulators is affected by several parameters such as rib angle-of-attack, channel aspect ratio, rib pitch-to-height ratio, blockage ratio, and rib shape. Investigations on these aspects of rib turbulator design have been reported. In the past, researchers have studied the flow characteristics of ribbed duct, both, experimentally and numerically. Some studies focused on the relative arrangement of ribs, e.g. parallel, staggered, and criss-cross. Several investigations have been carried out in the past on rib turbulator as a method of enhancing heat transfer. [1-11]

As summarized above, many researchers have investigated internal cooling channels having a variety of ribs over the last few decades. However, the heat transfer performances of many other rib shapes have not yet been reported. In the present work, the flow structures, heat transfer characteristics, and thermal performances of rib-roughened rectangular channels were evaluated using three-dimensional RANS analysis; new rib shapes—fan shaped, house-shaped, reverse cut-trapezoidal, cut-trapezoidal, reverse boot-shaped, boot-shaped, reverse pentagonal, pentagonal, and reverse right-angle trapezoidal—were compared against the rib shapes that have already been tested in previous works—square, isosceles triangular, reverse right-angle triangular, right-angle triangular, right-angle trapezoidal, isosceles trapezoidal, and semi-circular. [12-23]

The present research was undertaken to determine average heat transfer coefficients and friction factors for turbulent flows in ribbed duct. Computational study was performed by using a ribbed duct. The thermal boundary conditions were constant temperature over the surfaces of walls the second wall being insulated. The FLUENT software solves the mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem.

II. COMPUTATIONAL 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 small.

(5) No-slip boundary condition is assigned to the walls in contact with the fluid in the model.

(6) Negligible radiation heat transfer and other heat losses.

The computational domain used for CFD analysis is a simple rectangle. After defining the computational domain, non-uniform mesh is generated. 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. After generating mesh, boundary conditions have been specified. We will first specify that the left edge is the duct inlet and right edge is the duct outlet. Top

edge is top surface and bottom edges are inlet length, outlet length and solar plate. All internal edges of rectangle 2D duct are defined as turbulator wall. Meshing of the domain is done using ANSYS ICEM CFD V16 software. Since low-Reynolds-number turbulence models are employed, the grids are generated so as to be very fine. To select the turbulence model, the previous experimental study is simulated using different low Reynolds number models such as Standard k-ω model, Renormalization-group k-ε model, Realizable -kε model and Shear stress transport k-ω model. The results of different models are compared with experimental results. The RNG k-ε model is selected on the basis of its closer results to the experimental results. The working fluid, air is assumed to be incompressible for the operating range of duct since variation is very less. The mean inlet velocity of the flow was calculated using Reynolds number. Velocity boundary condition has been considered as inlet boundary condition and outflow at outlet. Second order upwind and SIMPLE algorithm were used to discretize the governing equations. The FLUENT software solves the following mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem.

The geometrical and operating parameters for roughened duct are listed in Table 1.

Table 1
Range of parameters for CFD analysis.

Geometrical and operating parameters	Range
Entrance length of duct, 'L_1'	245 mm
Test length of duct, 'L_2'	280 mm
Exit length of duct, 'L_3'	115 mm
Width of duct, 'W'	100 mm
Depth of duct, 'H'	20 mm
Hydraulic diameter of duct, 'D'	33.33 mm
Rib height, 'e'	2 mm
Rib Pitch, 'P'	5-30 mm
Reynolds number, 'Re'	3800-18000 (6 values)

III. RESULT AND ANALYSIS

The computational model studied in the present study includes a segment of the inlet plenum and the test section. Inlet for the computational domain is the inlet to the domain, so that more realistic flow with entry effect can be simulated for the test section. The flow is assumed to leave the test section to ambient with a zero static pressure boundary condition at the outlet.

Fig. 1 shows the effect of Reynolds number on average Nusselt number for different values of relative roughness pitch (P/e) and fixed value of roughness height (e). The average Nusselt number is observed to increase with increase of Reynolds number due to the increase in turbulence intensity caused by increase in turbulence kinetic energy and turbulence dissipation rate. The heat transfer phenomenon can be observed and described by the contour plot of turbulence intensity.
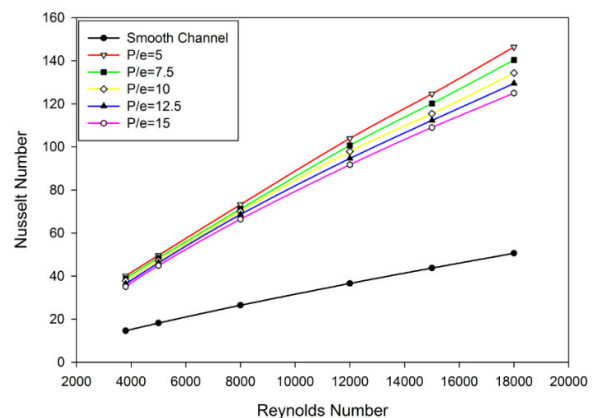

Figure 1 Nusselt number vs Reynolds number

It can be seen that the enhancement in heat transfer of the roughened channel with respect to the smooth channel also increases with an increase in Reynolds number. It can also be seen that Nusselt number values decreases with the increase in relative roughness pitch (P/e) for fixed value of roughness height (e). This is due to the fact that with the increase in relative roughness pitch, number of reattachment points over the absorber plate reduces. The roughened channel with lower

relative roughness pitch provides the highest Nusselt number.

The heat transfer phenomenon can be observed and described by the contour plot of turbulence intensity. The intensities of turbulence are reduced at the flow field near the rib and wall and a high intensity region is found between the adjacent ribs close to the main flow which yields the strong influence of turbulence intensity on heat transfer enhancement. (Fig. 2)
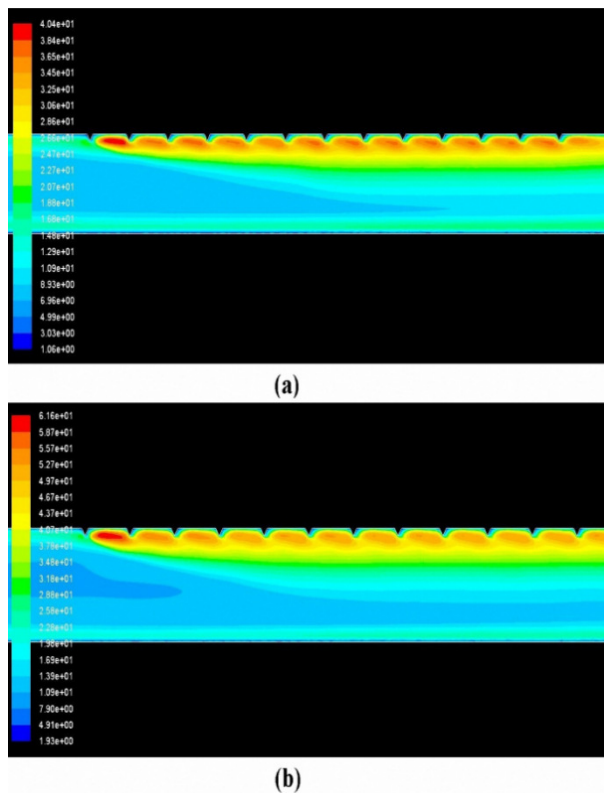

Figure 2 Contour plot of turbulence intensity

IV.   CONCLUSIONS

The following conclusions are drawn from present analysis:

1.   The use of artificial roughness on a surface is an effective technique to enhance heat transfer to fluid flowing in the duct. Artificially ribbed ducts have enhanced rate of heat transfer as compared to the smooth duct.

2.   The RNG k-ε turbulence model gives very close results to the experimental results. RNG k-ε turbulence model has been validated for smooth duct and grid independence test has also been conducted to check the variation with increasing number of cells.

3.   Nusselt number increases with an increase of Reynolds number.

4.   The maximum value of average Nusselt number is found to be at minimum value of relative roughness pitch at a higher Reynolds number.

REFERENCES

[1]   E.Y. Choi, Y.D. Choi, W.S. Lee, J.T. Chung, J.S. Kwak, Heat transfer augmentation using a rib–dimple compound cooling technique, Appl. Therm. Eng. 51 (1) (2013) 435–441.

[2]   S.W. Chang, T.M. Liou, K.F. Chiang, G.F. Hong, Heat transfer and pressure drop in rectangular channel with compound roughness of V-shaped ribs and deepened scales, Int. J. Heat Mass Transf. 51 (3) (2008) 457–468.

[3]   S.V. Ekkad, Y. Huang, J.C. Han, Detailed heat transfer distributions in two-pass square channels with rib turbulators and bleed holes, Int. J. Heat Mass Transf. 41 (23) (1998) 3781–3791.

[4]   S. Kumar, R.S. Amano, J.M. Lucci, Numerical simulations of heat transfer distribution of a two-pass square channel with V-rib turbulator and bleed holes, Heat Mass Transf. 49 (8) (2013) 1141–1158.

[5]   Y.M. Zhang, W.Z. Gu, J.C. Han, Heat transfer and friction in rectangular channels with ribbed or ribbed-grooved walls, Trans.-Am. Soc. Mech. Eng. J. Heat Transf. 116 (1994), pp. 58–58.

[6]   P. Singh, J. Pandit, S.V. Ekkad, Characterization of heat transfer enhancement and frictional losses in a two-pass square duct featuring unique combinations of rib turbulators and cylindrical dimples, Int. J. Heat Mass Transf. 106 (2017) 629–647.

[7]   P. Singh, B.V. Ravi, S. Ekkad, Experimental investigation of heat transfer augmentation by different jet impingement hole shapes under maximum crossflow, in: ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, American Society of Mechanical Engineers, 2016, pp. V05BT16A018–V05BT16A018.

[8]   P. Singh, S. Ekkad, Effects of rotation on heat transfer due to jet impingement on cylindrical dimpled target surface, in: ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, American Society of Mechanical Engineers, 2016, pp. V05BT16A010–V05BT16A010.

[9]   P. Singh, S.V. Ekkad, Effects of spent air removal scheme on internal-side heat transfer in an impingement-effusion system at low jet-to-target plate spacing, Int. J. Heat Mass Transf. 108 (2017) 998–1010.

[10]   B.V. Ravi, P. Singh, S.V. Ekkad, Numerical investigation of turbulent flow and heat transfer in two-pass ribbed channels, Int. J. Therm. Sci. 112 (2017) 31–43.

[11]   P. Singh, S. Ekkad, Experimental study of heat transfer augmentation in a twopass channel featuring V-shaped ribs and cylindrical dimples, Appl. Therm. Eng. 116 (2017) 205–216.

[12]   Sparrow, E. M., and Haji-Sheikh, A., Laminar Heat Transfer and Pressure Drop in Isosceles Triangular, Right Triangular, and Circular Sector Duct, J. Heat Transfer, 87, 426-428, 1965.

[13]   Schmidt, F. W., and Newell, M. E., Heat Transfer in Fully Developed Laminar Flow Through Rectangular and Isosceles Triangular Ducts, Int. J. Heat Mass Transfer, 10, 1121-1123, 1967.

[14]   Shah, R. K., and London, A. L., Laminar Flow Forced Convection in Ducts, Academic, New York, 1978.

[15]   Eckert, E. R. G., and Irvine, T. F., Pressure Drop and Heat Transfer in Duct with Triangular Cross Section, J. Heat Transfer, 82, 125-138, 1960.

[16]   Altemani, C. A. C., and Sparrow, E. M., Turbulent Heat Transfer and Fluid Flow in an Unsymmetrically Heated Triangular Duct, J. Heat Transfer, 102, 590-597, 1980.

[17]   Braga, S. L., and Saboya, F. E. M., Transport Coefficients for Triangular Ducts (in Portuguese), IX CongressoBrasileiro de EngenhariaMecfinica, FIorian6polis, Brazil, pp. 33-36, December 1987.

[18]   A. Chaube, P.K. Sahoo, S.C. Solanki, Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater, Renewable Energy 31 (2006) 317-31.

[9]   S. Kumar, R.P. Saini, CFD based performance analysis of a solar air heater duct provided with artificial roughness, Renewable Energy 34 (2009) 1285-91.

[20]   S.V. Karmare, A.N. Tikekar, Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD, Solar Energy 84 (2010) 409–17.

[21]   Pitak, P., et al., Turbulent heat transfer and pressure loss in a square channel with discrete broken V-rib turbulators. Journal of Hydrodynamics, 2016. 28(2): p. 275-283.

[22]   Alfarawi, S., S.A. Abdel-Moneim, and A. Bodalal, Experimental investigations of heat transfer enhancement from rectangular duct roughened by hybrid ribs. International Journal of Thermal Sciences, 2017. 118: p. 123-138.

[23]   Singh, P., Y. Ji, and S.V. Ekkad, Experimental and numerical investigation of heat and fluid flow in a square duct featuring criss-cross rib patterns. Applied Thermal Engineering, 2018. 128: p. 415-425.