



Computational analysis of effect of pitch on heat transfer performance in duct

Rahul Saikhedkar, Yogesh Tembhurne
Mechanical Engineering Department,
Bhabha Engineering Research Institute, Bhopal, MP, India.

Abstract: Enhancing heat transfer surface are used in many engineering applications such as gas turbine blade cooling passages (i.e. channel/duct), air heater, heat exchanger surfaces, gas-cooled reactor fuel elements, ventilation equipment of micro-electronic systems and air conditioning/ refrigeration systems. One of the most important techniques used are passive heat transfer technique. These techniques when adopted in heat transfer surfaces proved that the overall thermal performance improved significantly. In this paper a computational study was carried out to determine average heat transfer coefficients and friction factors for turbulent flow through rectangular ducts with ribs. A commercial finite volume package ANSYS FLUENT 12.1 is used to analyze and visualize the nature of the flow across the ribbed duct. 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: Ribbed duct, Heat transfer, Pressure Drop, CFD

I. Introduction

Heat transfer enhancement by inserting ribs is commonly used application in tubes and ducts. Ribs improve the heat transfer by interrupting the wall sub layer. This yields flow turbulence, separation and reattachment leading to higher heat transfer rates. Due to the existence of ribs effective heat transfer surface increases. Many researchers have been carried out on heat transfer enhancement achieved by different ribs. Rib turbulators are used to improve the convective heat transfer coefficient in many practical applications i.e. gas turbine blade cooling passages, heat exchanger surfaces, gas-cooled reactor fuel elements and ventilation equipment of micro-electronic systems. The use of ribs to enhance heat transfer is very important in the duct of modern advanced gas turbines. One reason is that the ribs increase the fluid flow turbulence near the wall and disrupt the laminar boundary layer to enhance the heat transfer. The other reason is that the ribs also increase the heat transfer area. However, the increase in heat transfer is usually accompanied with a rise in pressure drop. In the design and manufacturing of gas turbines, it is desirable to predict the heat transfer coefficient and friction factor by means of theoretical approaches. When the rib height to hydraulic diameter ratio is small, the repeated-rib surface can be considered as a rough surface geometry.

Turbulence promoter i.e. ribs either in the form of surface roughness or in the form of three dimensional surface protuberances tends primarily to increase the heat transfer coefficient due to disturbance or destruction of the viscous sub-layer near the wall. The key dimensions of the roughness geometry are the relative roughness height, the relative roughness spacing and the shape of the roughness element. The optimal geometry of roughness depends mostly on dynamic conditions in the boundary layer and on the properties of fluid.

Sparrow and Haji-Sheikh [1] report analytical results for laminar fully developed heat transfer in isosceles triangular ducts for a large range of apex angles. The thermal boundary condition was constant heat flux at the triangle walls. In addition, pressure drop coefficients are presented. Schmidt and Newell [2] also give results that can be applied to laminar flows. Several thermal boundary conditions were considered. All the situations were concerned with fully developed flows. One of the most extensive sets of results in noncircular ducts is that of Shah and London [3]. Only laminar flows were studied. With regard to turbulent flows, Eckert and Irvine [4] and Altemani and Sparrow [5] appear to the only authors with information about triangular ducts. In [4], measurements of heat transfer and friction characteristics were made in a narrow isosceles triangular duct with an apex angle of 11.46°. In [5], experiments were performed in an equilateral triangular duct. Entrance-region and fully developed heat transfer characteristics were determined; friction factors were also measured. The heating arrangement was designed to yield the following thermal boundary conditions: uniform heat input per unit axial length and circumferentially uniform temperature on two heated walls, the third wall being insulated. Braga and Saboya reported average heat transfer results and friction characteristics for isosceles triangular ducts with an apex angle of 120° [6] and for equilateral triangular ducts [7]. The thermal boundary conditions in both cases consisted of uniform temperature on two heated walls while the third wall was insulated Chaube et al. [8] conducted a CFD based analysis of heat transfer and fluid flow characteristics of an artificially roughened solar air heater. Kumar and Saini [9] performed a CFD analysis of fluid flow and heat transfer characteristics of a solar

air heaters having arc shaped artificial roughness on the absorber plate. Karmare and Tikekar [10] carried out CFD investigation of fluid flow and heat transfer in a solar air heater duct with metal grit ribs as roughness elements on the absorber plate. Yadav and Bhagoria [11] presented the numerical prediction of fluid flow and heat transfer in a conventional solar air heater by CFD. A commercial finite volume package ANSYS FLUENT 12.1 was used to analyze the nature of the flow across the duct of a conventional solar air heater. Yadav and Bhagoria [12] presented a detailed literature survey about different CFD investigations on artificially roughened duct.

The key dimensionless geometrical parameters that are used to characterize artificial roughness are:

1. Relative roughness pitch (P/e): Relative roughness pitch (P/e) is defined as the ratio of distance between two consecutive ribs and height of the rib.
2. Relative roughness height (e/D): Relative roughness height (e/D) is the ratio of rib height to equivalent diameter of the air passage.
3. Angle of attack (α): Angle of attack is inclination of rib with direction of air flow in duct.
4. Aspect ratio: It is ratio of duct width to duct height. This factor also plays a very crucial role in investigating thermo-hydraulic performance.

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.

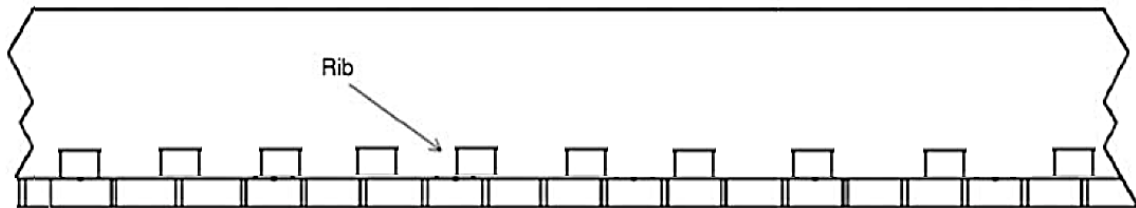
II. CFD Simulation

The following assumptions are imposed for the computational analysis.

- (1) The thermal conductivity of the duct wall, absorber plate and roughness material are independent of temperature.
- (2) The flow is steady, fully developed, turbulent and two dimensional.
- (3) No-slip boundary condition is assigned to the walls in contact with the fluid in the model.
- (4) The working fluid, air is assumed to be incompressible.
- (5) Negligible radiation heat transfer and other heat losses.

The computational domain used for CFD analysis is shown in Fig. 1.

Figure 1 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 of hydraulic diameter. The exit section is used after the test section in order to reduce the end effect in the test section. Rib roughness in the form of small square section wires is considered at the underside of the top of the duct to have roughened surface, running perpendicular to the flow direction while other sides are considered as smooth surfaces. A uniform heat flux of 1000 W/m^2 is considered for computational analysis. After defining the computational domain, non-uniform mesh is generated. After generating mesh, boundary conditions have been specified. All internal edges of rectangle 2D duct are defined as turbulent wall. Meshing of the domain is done using ANSYS ICEM CFD V12.1 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 $k-\omega$ model, Renormalization-group $k-\epsilon$ model, Realizable $k-\epsilon$ model and Shear stress transport $k-\omega$ model. The results of different models are compared with experimental results. The RNG $k-\epsilon$ 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.

III. Results And Discussion

Effect of the relative roughness pitch (P/e) on heat transfer is also shown typically in Fig. 2. It can be seen that the enhancement in heat transfer of the roughened duct with respect to the smooth duct 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 duct with relative roughness pitch of 5 provides the highest Nusselt number ($Nu=146.4$) at a Reynolds number of 18000. The roughened duct with relative roughness pitch of 15 provides the lowest Nusselt number ($Nu=35$) at a Reynolds number of 3800. The maximum enhancement of average Nusselt number is found to be 2.89 times that of smooth duct for relative roughness pitch of 5 at a Reynolds number of 18000.

Figure 2 Nusselt number Vs. Reynolds number

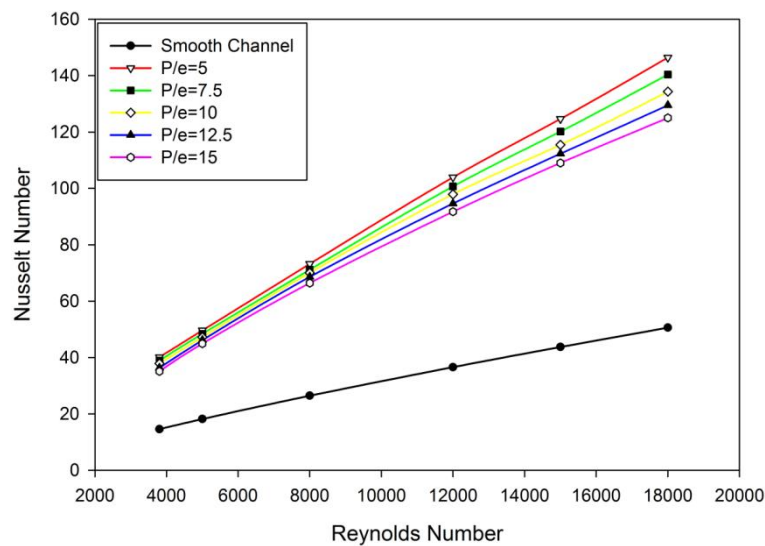


Figure 3 Friction factor Vs. Reynolds number

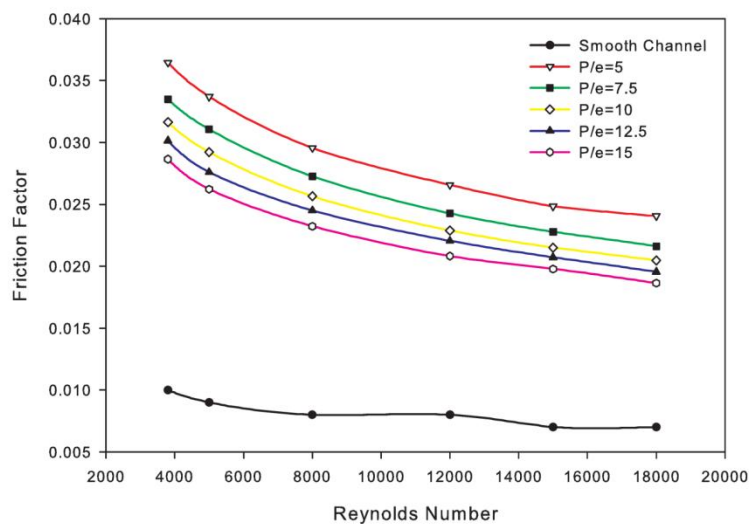
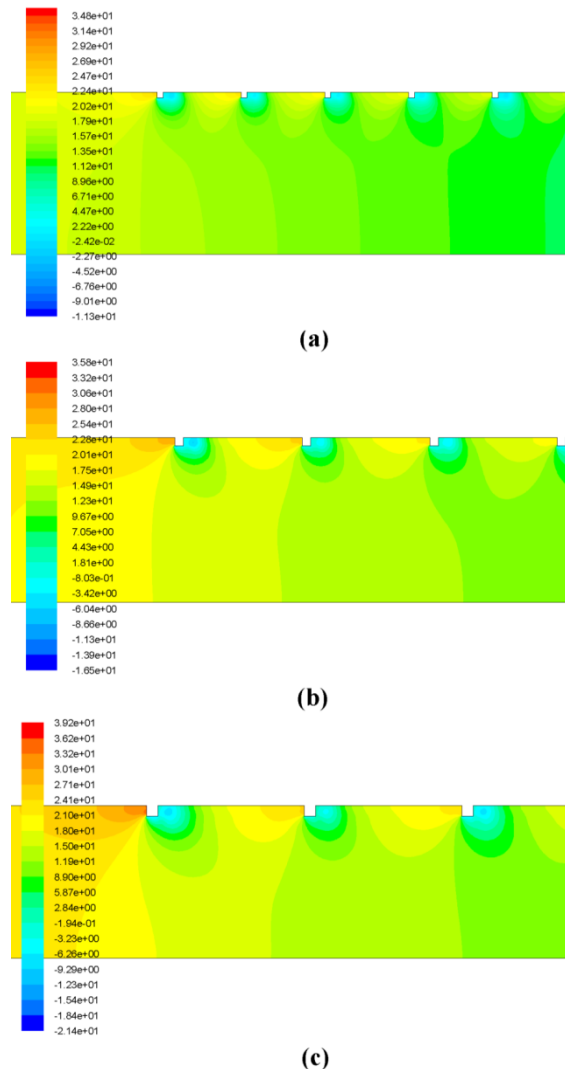


Fig 3 shows that the friction factor decreases with the increasing values of the Reynolds number in all cases as expected because of the suppression of laminar sub-layer for fully developed turbulent flow in the duct. It can also be seen that friction factor values decreases with the increase in relative roughness pitch (P/e) for fixed value of roughness height, attributed to less interruptions in the flow

path. The maximum and minimum value of friction factor occurs at relative roughness pitch (P/e) of 5 and 15 respectively for the range of parameters investigated. It is also observed that the maximum enhancement of average friction factor is found to be 3.6 times that of smooth duct for relative roughness pitch of 5 at a Reynolds number of 3800.

The fluid friction phenomenon can be observed and described by the contour of pressure. The contour plot of pressure is shown in Fig. 4.

Figure 4 Contour plot of pressure for (a) $P/e=5$ (b) $P/e=10$ (c) $P/e=15$ at $Re=15000$



IV. Conclusion

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 $k-\epsilon$ turbulence model gives very close results to the experimental results. RNG $k-\epsilon$ 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. Average Nusselt number increases with an increase of Reynolds number. The maximum value of average Nusselt number is found to be 146 for relative roughness pitch of 5 at a higher Reynolds number, 18,000. The maximum enhancement of average Nusselt number is found to be 2.89 times that of smooth duct for relative roughness pitch of 10.
4. Average friction factor decreases with an increase of Reynolds number. The maximum value of average friction factor is found to be 0.03645 for relative roughness pitch of 5 at a lower Reynolds number, 3800. The maximum enhancement of average friction factor is found to be 3.6 times that of smooth duct for relative roughness pitch of 5.

References

- [1] 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.
- [2] 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.
- [3] Shah, R. K., and London, A. L., *Laminar Flow Forced Convection in Ducts*, Academic, New York, 1978.
- [4] 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.
- [5] 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.
- [6] Braga, S. L., and Saboya, F. E. M., Heat Transfer and Pressure Drop for Triangular Ducts with Laminar and Turbulent Flows (in Portuguese), pp. 207-210, December 1986.
- [7] Braga, S. L., and Saboya, F. E. M., Transport Coefficients for Triangular Ducts (in Portuguese), IX Congresso Brasileiro de Engenharia Mecânica, Florianópolis, Brazil, pp. 33-36, December 1987.
- [8] 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.
- [10] 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.
- [11] A.S. Yadav, J.L. Bhagoria, Heat transfer and fluid flow analysis of solar air heater: a review of CFD approach, *Renewable and Sustainable Energy Reviews* 23 (2013) 60-79.
- [12] A.S. Yadav, 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*, Elsevier, 55 (2013) 1127-1142.