Optimization of Heat Transfer through Rectangular Duct

Ravi Teja[1], Pathan F Z[2], Mandar Vahadne[3],

[1] Assistant Professor, Mechanical Department, S R E S College of Eng. Kopargaon, Maharashtra, India
[2] Assistant Professor, Mechanical Department, S R E S College of Eng. Kopargaon, Maharashtra, India
[3] Assistant Professor, Mechanical Department, Cummins College of Eng., Pune, Maharashtra, India

---***---

Abstract - *This study comprehensively simulate the use of laminar and k-ε* ***model for predicting flow and*** *heat transfer with measured flow field data in a stationary duct which sheds light on the detailed physics encountered in the fully developed flow region, and the sharp 180° bend region. Among the major flow features predicted with accuracy are flow transition at the entrance of the duct, the distribution of mean and turbulent quantities in the developing, fully developed, and sharp 180° bend, the development of secondary flows in the duct cross-section and the sharp 180° bend, and heat transfer augmentation . Flow intensities in the sharp 180° bend are found to reach high values and local heat transfer comparisons show that the heat transfer augmentation shifts towards the wall and along the duct. Therefore, understanding of the unsteady heat transfer in sharp 180° bends is important. The design and simulation are related to concept of fluid mechanics, heat transfer and thermodynamics. Simulation study has been conducted on the response of turbulent flow in a rectangular duct in order to evaluate the heat transfer rate along rectangular.*

Key Words: Key word1, Key word2, Key word3, etc...

1. INTRODUCTION

The performance of heat transfer and laminar or turbulence flow models in predicting the velocity and temperature fields of relevant industrial flows has become importance during the last few years. This requirement for improved predictive performance is also true for turbulent duct flows which occur frequently in many industrial applications such as compact heat exchangers, gas turbine cooling systems, recuperates, cooling channels in combustion chambers, inter-coolers, nuclear reactors and others.

The cross-section of these ducts might be orthogonal square or rectangular or non-orthogonal such as trapezoidal, in which the generated flow is extremely complex .Sometimes, the ducts are can be wavy or corrugated in the stream wise direction and might be manufactured with ribs in order to achieve faster transition to turbulence. Turbulent flows can be created by introducing geometrical elements periodically in the channel to enhance heat transfer between the fluid and the surface .Turbulent flow can control the temperature along the duct.

Prabal Talukdar et al (1) and M.D. Islama (2) have done research work with rectangular fins of different patterns and arrangements to study heat transfer enhancement in a rectangular channel. A detailed analysis of heat transfer from an end wall with arrays of short rectangular fins of different patterns (co-angular, zigzag, co-rotating and co-counter rotating) in the duct flow has been studied by M. A. Ebadian (3) . Among the four types of fin pattern, the co-rotating fin pattern is found to be the best configuration for heat transfer enhancement. Heat transfer enhancement in arrays of rectangular blocks in channel has been investigated by other researchers. K. Hooman et al (4) and Vazquez et al (5) studied the heat transfer and pressure drop characteristics of arrays of rectangular modules commonly encountered in electronic equipment and determined the thermal behaviour of the arrays in different situations. Yuan, J et al (6) has discussed heat transfer enhancements exceeding a factor of two were achieved by the use of multiple fences like barriers, with the inter barrier spacing and the barrier height being varied parametrically along with the Reynolds number. Azli Abd. Razak et al (7) experimentally studied the heat transfer at the entrance region of an array of rectangular heated blocks and presented empirical correlations of the heat transfer for the array.

In this paper we have discussed the heat transfer through rectangular duct, and optimization of rectangular ducts cross section and its ribs by using ANSYS FLUENT as Computational fluid dynamics platform for Heating ventilation and air conditioning application

2. TURBULENCE MODEL

In this modeling one of the classical models are in this two-equation model Standard k-ε model and Realizable k-ε model. The realizable k-ε model has been choosing as it is consistent with the physics of turbulent flows.

2.1 Benchmark Problem

Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here. Paragraph comes content here.

The study of heat transfer and laminar flow through straight rectangular duct. Cross section of the rectangular duct is 200 × 100 mm and length of the duct is 1 m.
Data given:-
Temperature at the inlet of the duct = 323 K
Outside temperature =300 K;
Inlet velocity=0.35 m/s
Mass flow rate through the duct =7.14×10^-3 kg/sec
Analytical solutions
The velocity profile through the duct can be calculated by using following formulas

$$\frac{u}{u_{max}} = \left[1-\left(\frac{y}{b}\right)^n\right]\left[1-\left(\frac{z}{a}\right)^m\right]$$

$$\frac{u_{max}}{u_m} = \left(\frac{m+1}{m}\right)\left(\frac{n+1}{n}\right)$$

Where u=velocity at any point on or inside the duct
Umax=Maximum velocity
Um=mean velocity
y=distance from the center line along width
z=distance from the center line along height
a=half width of rectangular duct
b=half height of rectangular duct

$$n = \begin{cases} 2 & \text{for } \alpha^* \le 1/3 \\ 2+0.3(\alpha^* - 1/3) & \text{for } \alpha^* \ge 1/3 \end{cases}$$

Hear α^*=2b/2a=100/200=0.5;
m=4.33;
n=2.051;
The graph obtained after the analytical solution and Finite element analysis area shown eyes free interaction of user with software and reduce the time delay.

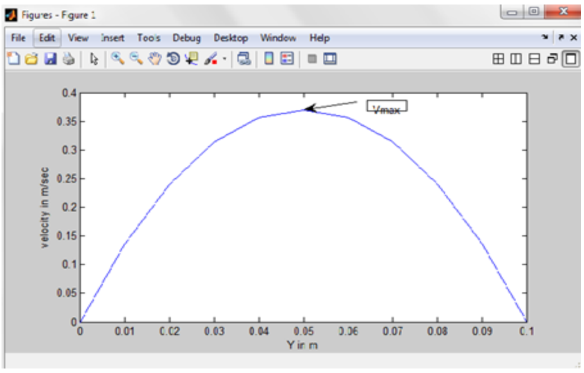


Fig 1. A) Velocity profile plot for square duct

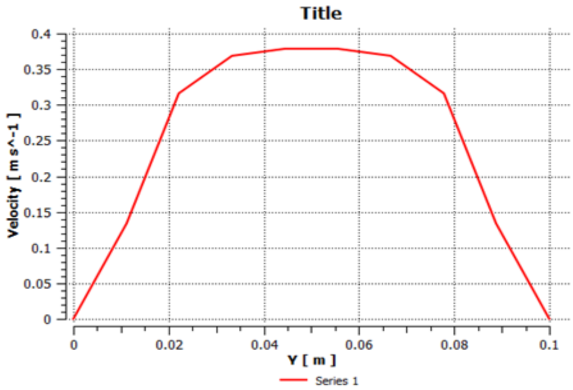


Fig 1. (B) Velocity profile plot obtained using ANSYS Fluent

Fig 1. Shows the velocity profile plots by using theoretical calculation and by using FLUENT ANSYS of square duct.

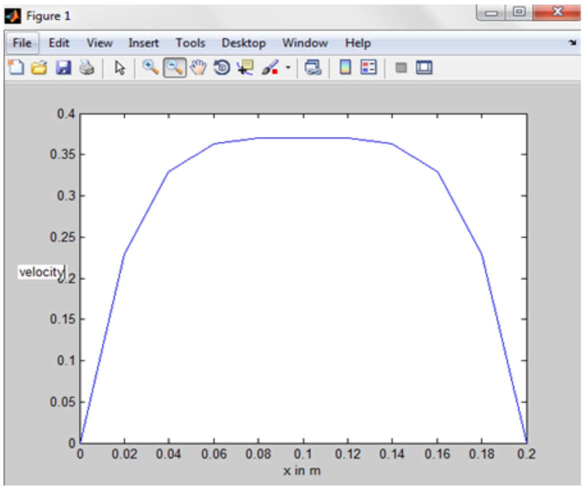


Fig 2. (A) Velocity profile plot for Rectangle duct

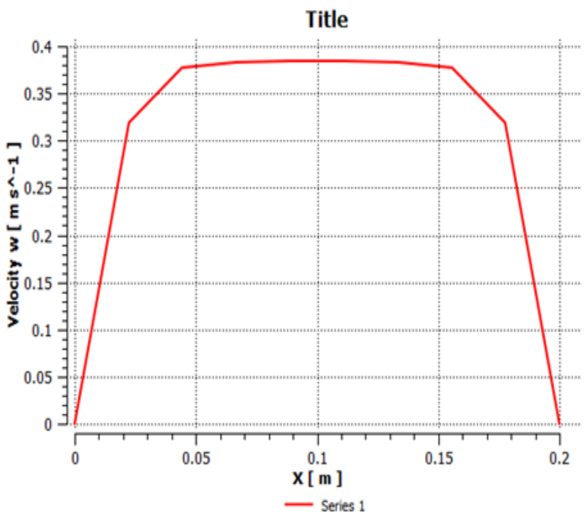


Fig 2 (B) Velocity profile plot for Rectangle duct

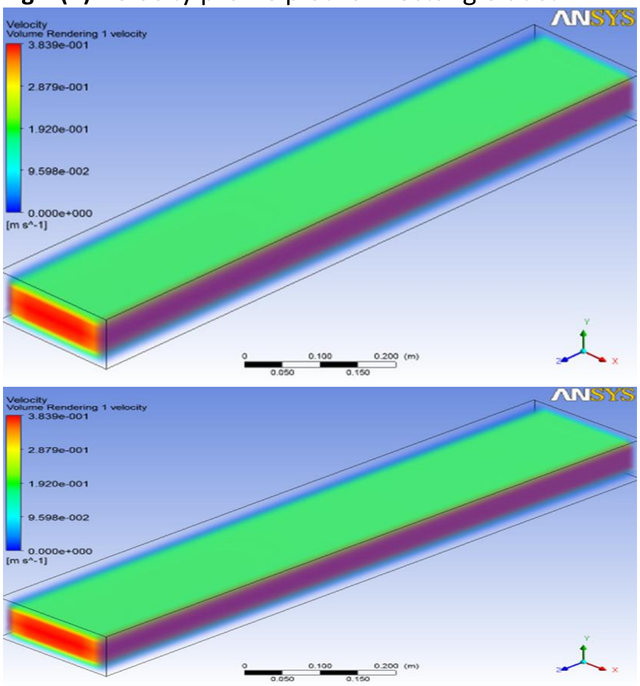


Fig 3 Velocity distribution through rectangular duct

3. PROBLEM DEFINATION

The duct is used for heating and ventilation air conditioning .The rectangular duct of 200×100 mm cross section with 180 degree sharp bend is selected for optimization. Analysis is done by using laminar model and k-Ɛ model of a turbulent flow.

Laminar model

1) Inlet temperature =292 K;
2) Inlet velocity=0.35 m/sec
3) Outlet pressure =1.01325 bar
4) Outside temperature of the duct = 300 K

The insulation material is used for the duct is poly-urethene which is having thermal conductivity of 0.190 W/m K .outer surface of the duct is made up of steel material.

Turbulent model

1) Inlet temperature =292 K;
2) Inlet velocity= 5 m/sec
3) Outlet pressure =1.01325 bar
4) Outside temperature of the duct = 300 K
5) Turbulent intensity= 3%

Hydraulic diameter is given by the following formula = 4A/P;

6) Hydraulic diameter=0.1333m;
7) Backflow of kinetic energy=1 m^2/sec^2

3.1 Reynolds Number

Re is a dimensionless number that gives a measure of the ratio of inertial forces to viscous forces

$$\mathrm{Re} = \frac{\rho \mathrm{v} D_H}{\mu} = \frac{\mathrm{v} D_H}{\nu} = \frac{Q D_H}{\nu A}$$

Where

DH is the hydraulic diameter of the pipe; its characteristic travelled length, L, (m).

Q is the volumetric flow rate (m3/s).

A is the pipe cross-sectional area (m^2).

V is the mean velocity of the object relative to the fluid (SI units: m/s).

μ is the dynamic viscosity of the fluid (Pa·s or N·s/m^2 or kg/(m·s)).

ν is the kinematic viscosity (ν = μ / ρ) (m^2/s).

In boundary layer flow over a flat plate, experiments confirm that, after a certain length of flow, a laminar boundary layer will become unstable and become turbulent. This instability occurs across different scales and with different fluids, usually when, where x is the distance from the leading edge of the flat plate, and the flow velocity is the freestream velocity of the fluid outside the boundary layer.

3.2 Nusselt Number

In heat transfer at a boundary (surface) within a fluid, the Nusselt number is the ratio of convective to conductive heat transfer across (normal to) the boundary. Named after Wilhelm Nusselt, it is a dimensionless number. The conductive component is measured under the same conditions as the heat convection but with a (hypothetically) stagnant (or motionless) fluid.

A Nusselt number close to one, namely convection and conduction of similar magnitude, is characteristic of "slug flow" or laminar flow. A larger Nusselt number corresponds to more active convection, with turbulent flow typically in the 100–1000 range.

The convection and conduction heat flows are parallel to each other and to the surface normal of the boundary surface, and are all perpendicular to the mean fluid flow in the simple case.

$$\mathrm{Nu}_L = \frac{hL}{k_f} = \frac{\text{Convective heat transfer coefficient}}{\text{Conductive heat transfer coefficient}}$$

Where:

- L = characteristic length
- Kf = thermal conductivity of the fluid
- h = convective heat transfer coefficient

For rectangular ducts with a uniform wall heat flux at four walls under the ~ boundary condition, the fully developed Nusselt numbers can be computed with the following formula

Nu=4.1247;

Convective heat transfer over the duct can be calculated by following formula

h=Nu*K/L

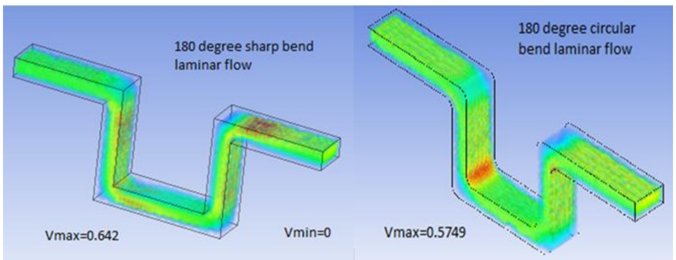
4. DESIGN FORMULATION

Design optimization of the duct is done for 180 bend duct for following condition

1)180 sharp bend
2)180 U bend
3) 180 U bend with 1 rib at the Centre circular curvature
4) 180 U bend with 2 rib inside the circular curvature
5) 180 U bend with 3 rib inside the circular curvature
6) 180 U bend with 2 rib inside the circular curvature at different location

5. RESULTS AND DISCUSSION

The velocity distribution in 180 sharp corner rectangular duct shows that , at the corner points or at the sharp edged corners gives us stagnation point where the velocity at those point is zero hence the heat transfer rate at stagnation point is found to be maximum



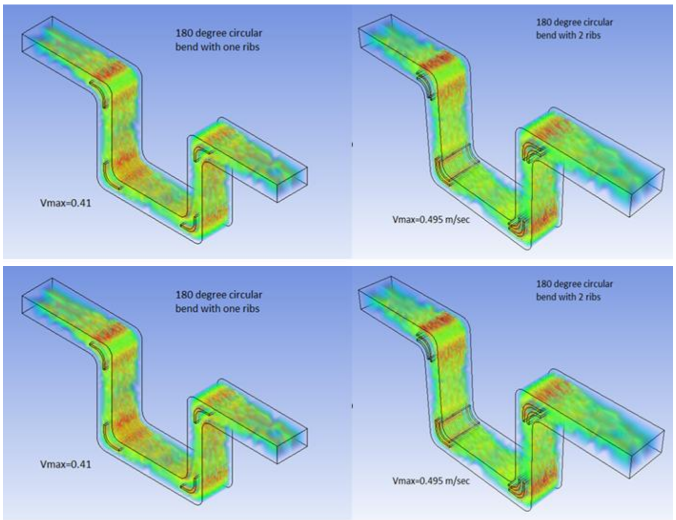


Fig 4 velocity volume rendering of laminar flow

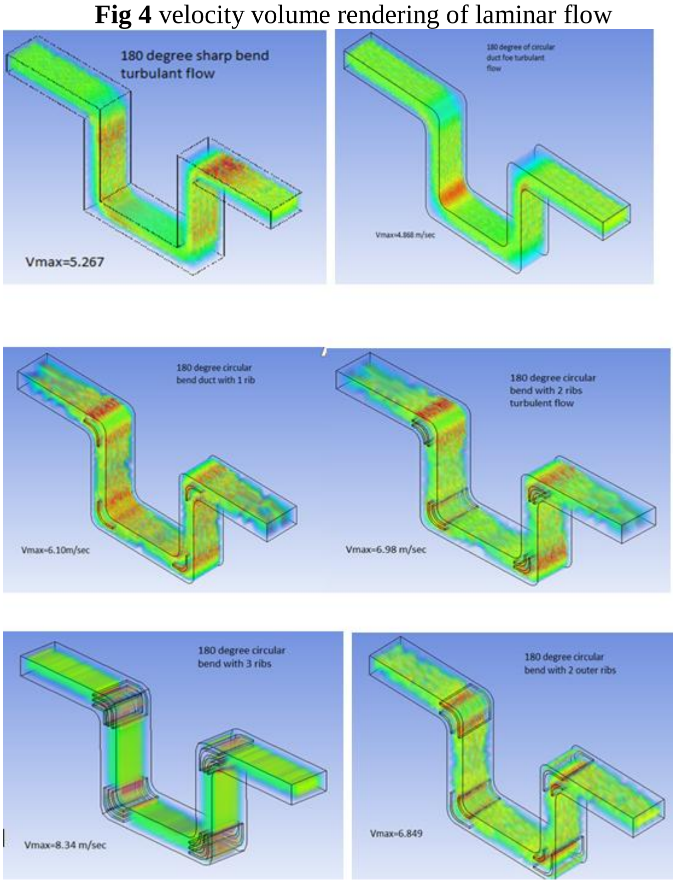


Fig 5 Velocity volume rendering of turbulent flow

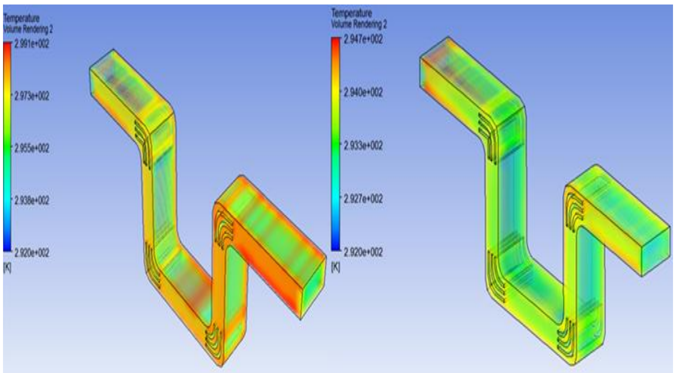


Fig 6 Temperature distribution of 180^0 circular section with 3 ribs

For second design, the circular bends are used instead of sharp edge corner. The velocity distribution shown in Fig 4 and Fig 5, for laminar and turbulent flow which illustrates that at circular points of corner shows uniform velocity and that gives us less heat transfer rate at those point. The Fig 6 shows the temperature distribution for both laminar and turbulent flow.

The analysis of 180 degree circular bend with 1 ,2 and 3 gives the more number of stagnation point inside the duct hence the temperature distribution inside the duct are not uniform that shows the heat transfer rate is maximum at those points . This increase in temperature in the air model gives more energy losses.

From this analysis the result suggests that the flows will maintained turbulent along the ducts when the inlet velocity is at least in the range of 0.35 m/s – 5 m/s. A result also shows that the correlation between flow structure and heat transfer is found to be strong. It is found that the onset of flow oscillations is important as it dramatically enhances heat transfer. As mention before the temperature drop about 2.5% at the end of 180° bend while the velocity increases about 27%. Here the significant correlation between the sharp bend of the duct and velocity throughout the duct are shown.

REFERENCES

[1] Prabal Talukdar,Conrad R. Iskra, Carey J. Simonson, “Combined heat and mass transfer for laminar flow of moist air in a 3D rectangular duct: CFD simulation and validation with experimental data”, International Journal of Heat and Mass Transfer 51 (2008) 3091–3102

[2] M.D. Islama,K. Oyakawa , M. Yaga , I. Kubo, “The Effects of duct height on heat transfer enhancement of a co-rotating type rectangular finned surface in duct”, Experimental Thermal and Fluid Science 33 (2009) 348–356

[3] M. A. Ebadian and Z. F. Dong, “Forced Convection, Internal Flow In Ducts” Florida International University

[4] K. Hooman , H. Gurgenci , A.A. Merrikh, “Heat transfer and entropy generation optimization of forced convection in porous-saturated ducts of rectangular cross-section” International Journal of Heat and Mass Transfer 50 (2007) 2051–2059

[5] Vazquez, M.S., W.V. Rodriguez, and R. Issa, Effect of ridged Walls on the heat transfer in a heated square duct International Journal of Heat and Mass Transfer, 2005. 48(10): p. 2050-2063.

[6] Yuan, J., M. Rokni, and B. Sunden, Simulation of fully developed laminar heat and mass transfer in fuel cell ducts with different cross- sections. International Journal of Heat and Mass Transfer, 2001. 44(21): p. 4047-4058.

[7] Azli Abd. Razak, Yusli Yaakob, and Mohd Nazir Ramli , “Computational Simulation of Turbulence Heat Transfer in Multiple Rectangular Ducts”,world academy of science and engineering and Technology 53 2009S. M. Metev and V. P. Veiko, *Laser Assisted Microtechnology*, 2nd ed., R. M. Osgood, Jr., Ed. Berlin, Germany: Springer-Verlag, 1998.

[8] J. Breckling, Ed., *The Analysis of Directional Time Series: Applications to Wind Speed and Direction*, ser. Lecture Notes in Statistics. Berlin, Germany: Springer, 1989, vol. 61.

[9] S. Zhang, C. Zhu, J. K. O. Sin, and P. K. T. Mok, “A novel ultrathin elevated channel low-temperature poly-Si TFT,” *IEEE Electron Device Lett.*, vol. 20, pp. 569–571, Nov. 1999.

BIOGRAPHIES

Mr RaviTeja Singamsetty has completed his MS from Hertfort shire, London. Having an experience of 3 years as Lecturer

Mr Pathan F Z has completed his MS Bits Pilani. Having an experience of 2 years as Lecturer

Mr Mandar Vahadne has completed his ME from University of Pune. Having an experience of 4years as Lecturer