International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 07 | July -2017 www.irjet.net p-ISSN: 2395-0072

Turbulent Flow in Curved Square Duct: Prediction of Fluid flow and Heat transfer Characteristics

R Girish Kumar[1], Rajesh V Kale[2]

[1]*M.E Student, RGIT, Andheri, Mumbai, India*
[2]*Professor, Dept of Mechanical Engineering, RGIT, Andheri, Mumbai, India*

Abstract – *Fully developed Turbulent flows in curved square duct with curvature of 30degree numerically and experimentally investigated. Turbulent flows in non circular duct leads to generate the secondary motions in crosswise direction. These secondary motions of second kind are generated due to the gradient of Reynolds Stresses in cross stream in straight duct. But these secondary motions are generated in curved square duct due to imbalance between centrifugal force and radial pressure gradients. This note aims to predict numerically these secondary motions and their effects on fluid flow and heat transfer characteristics in the curved square duct. SST K ω model has been used to simulate the flow. And these results are validated with experimental measurements. It has been observed that there is good agreement between the numerical and experimental results.*

***Key Words*: Turbulent flow, Secondary motions, Reynolds Stresses, Square Duct, Heat transfer**

1.INTRODUCTION

Prediction of Turbulent flow characteristics is of academic interest since few decades. In spite having numerous applications, there are only few exclusive studies which are focussed on secondary motions of turbulent flow. These secondary flows mean velocity will be up to 5-10% of mean velocity of primary flow. Experimental measurements of turbulent flow characteristics in 90˚ curved square duct have been studied by Humphery, White law and Yee [1]. These studies are done with the aid of non disturbing Laser Doppler Velocimetry and they have reported that stronger secondary motions and higher levels of turbulence is experienced near thicker boundary region. Numerous studies undertaken over the past few years done by Lesieur and Métais [2]; Moin [3]; Germano [4] demonstrate that LES is one of the most reliable approaches for the prediction of complex turbulent flows of practical relevance. Gavrilikas investigated the flow pattern inside the straight rectangular duct through numeric simulation and concluded that turbulent flow near the smooth corner is subjected to remarkable structural change, mean stream wise vortices attain their largest values in the region where the viscous effects are significant [5].

In few major applications the continuous change of fluid direction induces the centrifugal effects in fluid flow. Tilak T et al and Humphrey simulated the flow with various boundary conditions and geometries and has given the reason behind formation of secondary flows and relationships between duct geometry, aspect ratios and secondary flows [1, 6]. But there no exclusively devoted study of these secondary flows in curved duct in relationship with heat transfer. Present study focuses on presenting the relationship between heat transfer enhancement and Secondary motions or secondary flows.

2. GOVERNING EQUATIONS, METHODOLOGY AND TESTING DOMAIN

The present case is of unsteady, three dimensional and turbulent in nature. Therefore an averaged continuity equation, Reynolds averaged Navier stokes equation and energy equations are solved.

1. Continuity equation

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0 \qquad (1)$$

2. Navier Stokes equation

$$\frac{\partial u_i}{\partial t} + \frac{\partial(u_i u_j)}{\partial x_j} = \frac{-1}{\rho}\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}\right) - \frac{\partial(\overline{u_i^1 u_j^1})}{\partial x_i} \qquad (2)$$

3. Energy Equation

$$\frac{\partial T}{\partial t} + \frac{\partial(u_i T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\alpha \frac{\partial T}{\partial x_j}\right) - \frac{\partial \overline{u_i^1 u_j^1}}{\partial x_j} \qquad (3)$$

Where 'u' time averaged velocity and $\overline{u_i^1 u_j^1}$ is Reynolds stress

SST K-ω turbulence model is used in predicting the turbulent flow characteristics in the heated Curved squared duct. And this model is validated with DNS results of Gavrilakas(1992) for straight square duct at Reynolds number 4410 and they show good agreement in the results.

Computational domain is as shown in figure.1 consists of 30˚ curvature and all other dimensions are in terms of hydraulic diameter of the duct D_h. It is square duct (Aspect Ratio=1) with 100mm side. Air has been used as working fluid with the properties of kinematic viscosity (υ) = 1.6x10^{-6} m^2/s and Prandtle number (Pr) = 0.7.

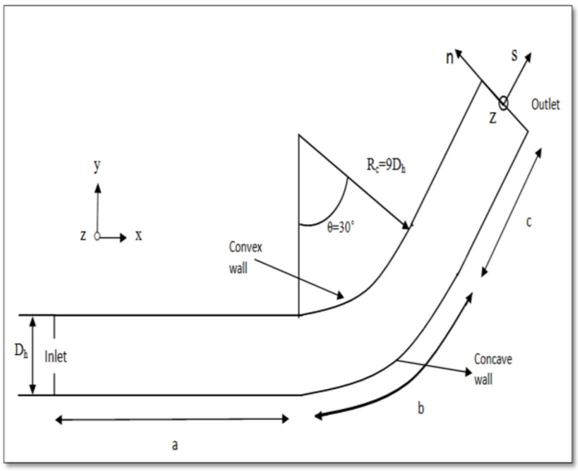

Fig-1: Curved square Duct

To have a fully developed turbulent flow at the beginning of curvature, sufficient length of straight square duct simultaneously simulated in numerical simulation case and experimental apparatus sufficient length straight square duct has been added in front of the curved section of the duct. This length can be determined by using the formula

$$\text{Entrance length } (L_e) = 4.4(R_e)^{(1/6)} D_h \quad (4)$$

Figure 2 shows the grid independent study for different mesh sizes and stream wise velocity is plotted against hydraulic diameter. And it is observed that 780x52x52 is optimum size. And further simulations are carried out with this grid.

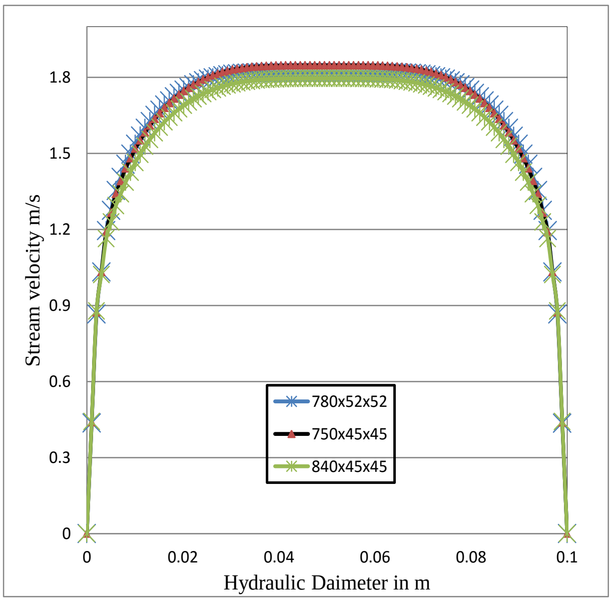

Fig-2: Fully developed Stream wise velocity

No slip boundary conditions are given for velocity near the walls and initially uniform temperature is imposed on the all walls. And in second case concave wall temperature is 3 times higher than the other walls temperature i.e concave wall temperature is 909kelvin when other walls temperature is maintained at 303kelvin. And simulations are carried out at Reynolds numbers 6000 and 10000. Inlet temperature of fluid is given as 303kelvin and outlet pressure is atmospheric pressure.

4.RESULTS AND DISCUSSION

In this section of initial part presents the numerical results, where we can visualize the secondary motions and their effects on the fluid flow and heat transfer characteristics. In second part we present the comparative study between experimental and numerical results.

Fig. no 3 depicts the three dimensional components of velocity. These results are experimentally measured with the help of five hole pressure probe. From the figure it clearly observable that magnitude of normal components of velocity is 3-5% of the axial velocity at initial part of the curvature.

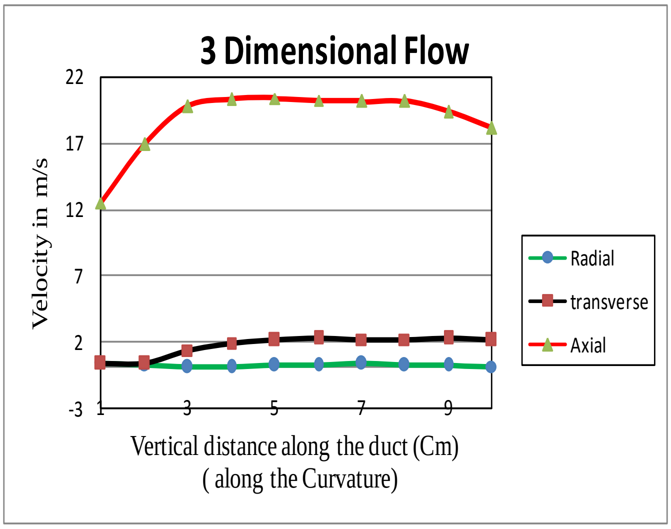

Fig-3: Velocity Components

Fig.4 (a), 4(b) and 4(c) shows the intensity of secondary flows just before the curvature, along the curvature and after the curvature. Intensity (I) is calculated as

$$I = \frac{\sqrt{v^2 + w^2}}{u_{bulk}} \quad (5)$$

From the figures it is clearly observable that at the beginning of the curvature intensity is higher near the concave wall as the whole fluid is pushed towards concave wall due to the centrifugal force effect. As the fluid flows downstream of the curvature secondary flows intensity increases near the lateral walls as the flow reversal is going to take place. And near the out let of duct intensity becomes maximum near the bottom wall due to the presence of the curvature of the duct.

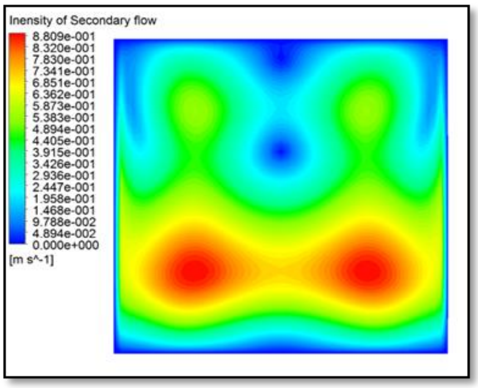

Fig-4(a): Intensity of Secondary flow at beginning of the Curvature

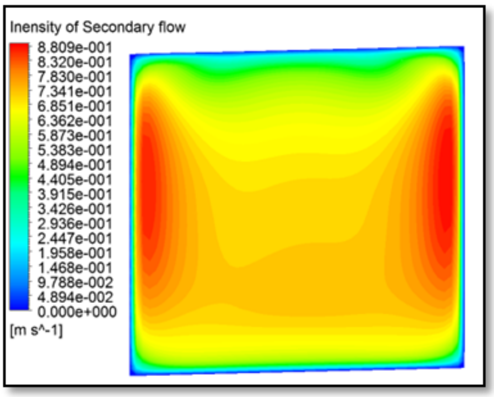

Fig -4(b): Intensity of Secondary flow along the curvature

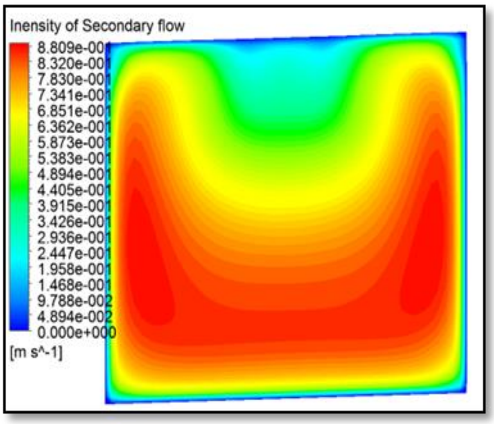

Fig -4(c): Intensity of Secondary flow after the Curvature

As the fluid near the concave wall has to be travel longer distace comapared fluid near the convex wall, fuid near concave wall attains higher velocities due to law of conseration of mss and it can be numerically visualised as shown in figure (5). Where we can observe that down stream of the curvature the fluid attains maximum velocities slightly near to the bottom wall rather than being the middle of the duct.

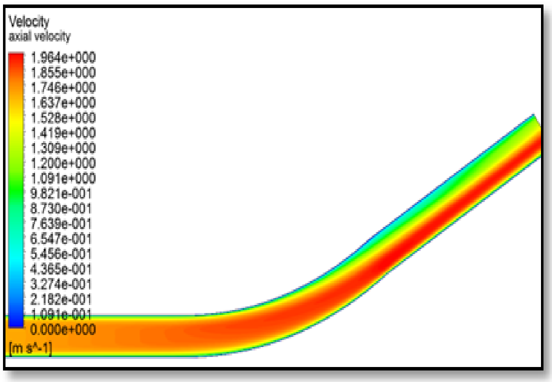

Fig -5:velocity contour in a streamwise direction in curved duct

As already mentioned the main aim of this study is to investigate the effects of secodary flow due to curvature on heat trasfer characteristics, different temperature boundary conditions are imposed on the duct walls numerically and experimentally. Figures 6(a), 6(b) and 6(c) depicts the temperature variation contours before, along and after the curvature. And it is found that temperature variation is uniform untill the curvature. As the fluid flow starts flowing along with the curvature it clearly observable that temperature diffusion becomes non uniform in crosswise direction. Temperature near the convex wall is higher compared to the concave wall, because more heat has been carried away by higher velocity flow near the concave wall. And presence of secondary flows, enhanced the mixing process, becasue of which hot fluid near the walls is brought into the core cold fluid flow, which inturn results in enhanced heattransfer.

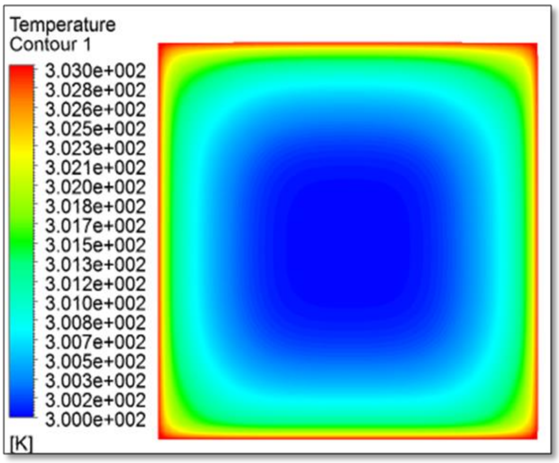

Fig -6(a): Temperature Contour before the Curvature

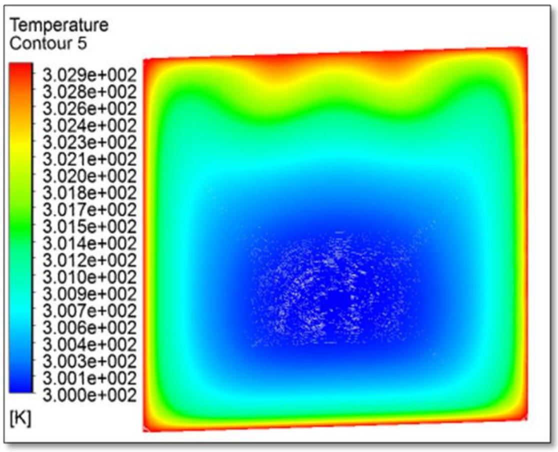

Fig -6(b): Temperature contour alog the Curvature

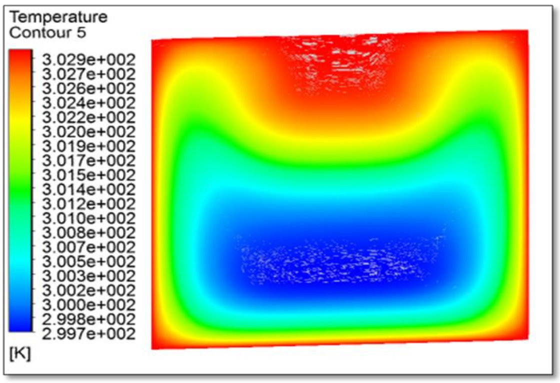

Fig -6(c): Temperature contour after the Curvature

Figures 7 and 8 presents the comparive study of CFD(numrical) and Experimental results on Streamwise velocity and Temperature. It is witnessed that temperatures near the convex wall is higher compared to the region near to the concave wall due to lower magnitudes of velocities.

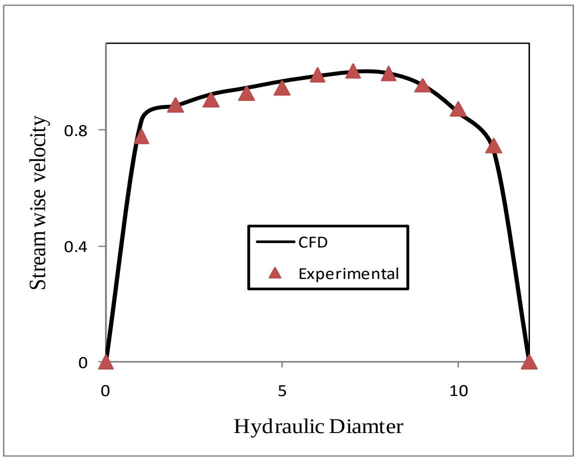

Fig -7: Stream wise velocity profile in transverse direction

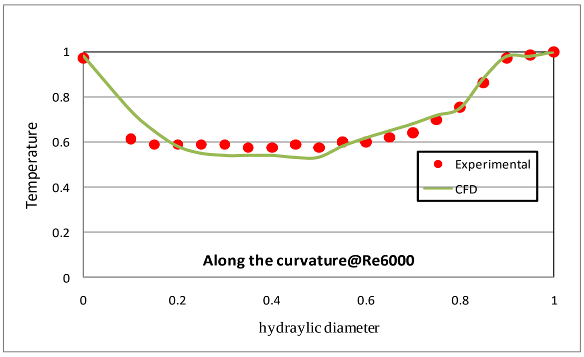

Fig -8: Temperature Profile in transverse direction

5. CONCLUSIONS

Turbulent fluid flow and heat transfer characteristics in the curved square duct is investigated numerically and experimentally. All 3 dimensional components of velocities are compared with each other and it is found that normal components of velocities are in the range of 3-5% of axial component. Velocity, temperature and Pressures are measured at different cross section positions and validated with experimental results at different Reynolds numbers. And it is observed that velocities near concave wall regions are higher compared to convex wall as the fluid has to travel longer distance. Because of which more heat is being carried away near the concave wall. Secondary flows which are generated due to imbalance between the centrifugal force and radial pressure gradients enhances the heat transfer by carrying the heat from the near wall region to core cold fluid. These Secondary motions intensity is higher near the lateral walls and enhances mixing process of fluid which increases the heat transfer rate

ACKNOWLEDGEMENT

All credit goes to the Supreme Personality of Godhead 'Krishna' and his devotees like Sri Vaishnava (Dr.Rahul Trivedi) by their causeless mercy only I could complete this work. Thank you my Lord.

REFERENCES

[1] Humphrey, J., Whitelaw, J., and Yee, G., 1981, Turbulent flow in a square duct with strong curvature. J. Fluid Mech. 103, pp. 443-463.
[2] Moin., P., 1998, Numerical and physical issues in large eddy simulation of turbulent flows, JSME Int. J., Ser. B 41(2), pp 454 – 463.
[3] Germano., M., 1998, Fundamentals of large eddy simulations. In: Advanced Turbulent Flow Computations. Springer-Verlag, pp 1760-1765.
[4] Temmerman, L., Leschziner, M., A., Mellen, C., P., and Frohlich., J., 2003, Investigation of wall-function approximations and subgrid-scale models in large eddy simulation of separated flow in a channel with

streamwise periodic constrictions, Int. J. of Heat and Fluid Flow, Vol. 24, pp 157-180.

[5] Ghosal, S., Lund T. S., Moin P., and Akselvoll, K., 1995, A dynamic localization model for large-eddy simulation of turbulent flows. J. Fluid Mech., 285, pp. 229 -255.

[6] Camarri, S., Salvetti, M. V., Koobus, B., and Dervieux, A., Large-eddy simulation of a bluff body flow on unstructured grids. Int. J. Numer. Meth. Fluids, 2002, 40, pp.1431-1460.