ISSN : 2248-9622, Vol. 6, Issue 3, (Part -4) March 2016, pp.115-124

RESEARCH ARTICLE OPEN ACCESS

A Computational Investigation of Flow Structure Within a Sinuous Duct

Prasanta K. Sinha[1], Bireswar Majumdar[2]
[1]Professor & Principal, Durgapur Institute of Advanced Technology & Mangt, Durgapur –713 212
[2]Professor, Department of Power Engineering, Jadavpur University, Kolkata –700098

ABSTRACT
In the present investigation the distribution of mean velocity are experimentally studied on three constant area rectangular curved ducts with an aspect ratio of 2.4. First one is C-shape, second one is S-shape and third one is a DS-shape duct. The experiment is carried out at mass averaged mean velocity of 40m/s for all the ducts. The velocity distribution shows for C-duct, the bulk flow shifting from outer wall to the inner wall along the flow passage and for S-duct, the bulk flow shifting from outer wall to the inner wall in the first half and from inner wall to the outer wall in the second half along the flow passage of curved ducts are very instinct. Due to the imbalance of centrifugal force and radial pressure gradient, secondary motions in the forms of counter rotating vortices have been generated within both the curved duct. For DS-duct the velocity distributions shows the Bulk of flow shifting from inner watt to outer wall in the first bend and third bend of the duct and outer wall to inner wall in the second bend and forth bend of the duct along the flow passage is very instinct. Flow at end of the DS-duct is purely uniform in nature due to non existence of secondary motion. The experimental results then were numerically validated with the help of Fluent, which shows a good agreement between the experimental and predicted results for all the ducts
Keywords: *C-duct, S-duct, DS-duct, Secondary Motion, Wall pressure, k-ε model, Fluent solver*

Nomenclature

Dn	Dean Number
L	Centerline length
R_c	Mean Radius of curvature
R_e	Reynold's number ($U_{av}D/\nu$)
U_{av}	Inlet Average Velocity
W	Inlet Width
b	Height of duct
$\Delta\beta$	Angle of turn of the curvature
ν	Kinematic Viscosity

I. INTRODUCTION

Constant area ducts are used for many engineering applications like small aircraft intakes, combustors, internal cooling system of gas turbines, HVAC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. In order to improve the performance of a duct it is absolutely necessary to design the duct with proper geometry so that the losses due to friction and eddies are minimized. Depending upon its application, the shape of the duct is chosen either straight or curved or annular or polar or sector. As a matter of fact the flow through a curved duct is more complex compared to straight duct due to the curvature of the centerline. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion, which is manifested in the form of a pair of contra-rotating vortices. Depending on the objective, hydro-mechanical systems often demand for the design of ducts with complex geometry albeit with high efficiency. In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition.

Ducts are used in aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. Heat exchangers in the form of curved ducts are used widely in food processing, refrigeration and hydrocarbon industries. Gas turbine engine components such as turbine compressors, nozzle etc. utilise several complex duct configuration. Performance of duct flow depends upon the geometrical and dynamical parameters of the duct. So it is very much essential to design the duct with proper geometry to improve the performance.

Study of flow characteristics through constant area ducts is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. Duct is a part and parcel of any fluid-mechanical system. It is a passageway made of sheet metal or other suitable material used for conveying air or other gases or liquids at different pressures. Depending on its application the shape the duct may be either of straight, curved, annular, polar, sector, trapezoidal, rhombic etc. Flow through curved ducts has practical importance in chemical and mechanical industries in particular. Obviously, compared to a straight duct, flow in a curved duct is more complex due to curvature of the duct axis. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion (normal to primary direction of flow) which is manifested in the form of a pair of counter-rotating vortices. Depending on the objective, fluid mechanical systems often demands for the design of ducts with complex geometry (like inlets, nozzles, diffusers, contractions, elbows etc) albeit with high efficiency. In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition.

Rowe[1], 1970, carried out experiments on circular 90° and 180° turn curved ducts with $R_e=0.4x10^5$ and reported the generation of contra rotating vortices within the bends. Bansod & Bradshaw [2], 1972, studied the flow characteristics within the 22.5°/22.5° S-shaped constant area ducts of different lengths and radii of curvature. They reported the development of a pair of contra-rotating vortices in the low pressure zone at the exit of the duct which was the consequence of the effect of stream wise vortices developed in the first half of the duct. Enayet et al.[3], 1982, investigated the turbulent flow characteristics through 90° circular curved duct of curvature ratio 2.8. It was observed that the thickness of the inlet boundary layer has a significant role on generation of secondary motion within the duct. Azzola et al.[4], 1986, have studied the turbulent flow characteristics through 180° circular bend with curvature ratio of 3.375 through experiments as well as computational methods. They observed a pair of contra-rotating vortices arising out of secondary motion in both experimental and numerical studies. Lacovides et al. [5], 1987, reported the flow prediction within 90° curved duct using numerical simulations based on the experimental investigation by Taylor et al.[6], 1982. They adopted finite volume approach to solve the semi-elliptical form of equation for 3-D flow analysis considering the wall function in the region close to the wall. The result shows a good agreement between the experimental and numerical analysis. Thangam and Hur[7], 1990, studied the secondary flow of an incompressible viscous fluid in a curved rectangular duct by using a finite volume method. They reported that with the increase of Dean Number the secondary flow structure evolves into a double vortex pair for low aspect ratio ducts. They correlated friction factor as a function of the Dean Number and aspect ratio. Kim and Patel[8], 1994, have investigated on a 90° curved duct of rectangular cross-section with aspect ratio 6 using five-hole probe and cross-wire hot wire anemometer. They reported the formation of vortices on inner wall due to the pressure driven secondary motion originated in the corner region of curved duct. Investigation on the turbulent boundary layer on the wall of an S-shaped wind tunnel for various Reynolds numbers ranging from $3.0x10^3$ to $11.0x10^3$ was carried out by Burns et al.[9], 1999. They used hot wire probe to measure mean velocity and Reynolds stresses. They interpreted their results for turbulence response and evaluated Reynolds Stress Transport Equations. Singh et al.[10], 2004, experimentally studied the flow and performance characteristics of a Y-shaped duct having an aspect ratio 1 and 1.66 for two inlet limbs with angle of turn 90°/90°. The average inlet velocities in the two limbs were 29m/s and 24m/s respectively. The longitudinal velocity and static and total pressure were measured by using a 3-hole pressure probe. They observed that the pressure recovery coefficient and loss coefficient increased continuously from inlet to the exit of the diffusing duct and are nearly same. Benin et al.[11], 2007,studeturbulent flows in smooth pipes using CFD analysis and find loss correlations for developing incompressible turbulent flow. They used different k–ε turbulent models and compare the result with experimental data. Kahrom and Shokrgozar [12] studied the flow with distributed turbulence intensity and tested the accuracy by different turbulence models compared the results with the experimental data. They used numerical solutions of one equation Spalart-Almaras Model, two equations high Reynolds number k-ε Model, two equations Shear Stress Transport k- Ω Model and anisotropy Reynolds Stress Models are compared to experimental measurements and results are discussed. Sinha *et al*. [13] conducted an experiment on 30° curved annular diffuser. They measured the mean velocity, static pressure and total pressure along the flow passage of the diffuser. They are also conducted a series of parametric investigations with same centre line length and inlet diameter but with different area ratios. They observed that the high velocity fluids shifted and accumulated at the concave wall of the exit section. It also observed that among the different turbulence models within the fluent solver a standard k-ε model shows the good results and predicts the flow and performance characteristics well for annular curved diffusing ducts with uniform flow at inlet. Biswas *et al*. [13] investigated experimentally a 90° constant area duct. They measured the wall pressure and mean velocity along the flow passage of the duct. They observed that the bulk flow shifting from outer wall to the inner wall along the flow passage of curved duct is very instinct. Elsafty and Elazm [14] studied **i**mproving air quality in enclosed parking facilities using ventilation system design with the aid of CFD simulation. The study draws attention to the effect of exhaust fan height on the carbon monoxide (CO) dispersion in enclosed parking facilities. A mathematical model is presented in a general computer code that can provide detailed information on CO concentrations as well as airflow fields prevailing in three-dimensional enclosed spaces of any geometrical complexity. Abduljabbar *et al*. [15] numerical predicted the effect on air flow rate in the presence of heated obstruction within a room and *tw*o notable points are presented; first, higher flow rate is depending on throw of jet and its effect on the CO2 concentration and temperature distribution in upper zone more than occupied zone with presence the obstruction. The second; in low flow rate buoyancy effect is considerable. Vertical temperature gradient above the obstruction implies that both fresh air and CO2 concentration.

From the available literature on curved ducts it is apparent that the studies are generally related to straight or curved ducts with circular cross-section. The present experimental investigation aims for a systematic study on the flow pattern of a curved C shaped duct through the measurements of mean velocity and wall static pressure distribution, under the influence of different geometrical and flow parameters

II. EXPERIMENTAL FACILITY

Experiments are carried out using the facility of wind tunnel at the Aerodynamics Laboratory of National Institute of Technology, Durgapur. Schematic layout of the experimental set-up is shown in Fig.1. A centrifugal air blower is directly coupled with a three phase, 5.5 kW electric motor of speed 2870 rpm. In order to minimize the vibration transmitted by the air supply unit to the test-piece, a flexible extension made of canvas of dimension 0.155mx0.310m is fitted at the blower outlet. The blower is followed by conical diffuser made of G.I. sheet of length 1.38m having inlet and outlet diameters of 0.310m and 0.600m respectively with a diverging angle of 6.66°. The settling chamber is a straight cylindrical duct through which the discharge from the conical diffuser flows. It is of uniform diameter of 0.600m and length 2.88m. Nylon screens are provided at three locations in the settling chamber in the transverse direction of flow for straightening as well as reducing the turbulence level of the flow. The contraction piece is installed between the settling chamber and the inlet piece of the curved duct. The contraction piece is of hollow truncated pyramid shape made of ply wood symmetric about the centre line with rectangular sections at both of its inlet and outlet. The inlet dimension of the contraction piece is 0.30mx0.30m and outlet dimension is 0.05mx0.10m. This piece when connected to the duct with a straight extension piece ensures uniform velocity profile at the inlet of the duct as well as reduces the resultant turbulence level at its own exit section. The complete geometry of curved ducts under test are shown in fig.2. C-duct is a rectangular 90° constant area duct, S-duct is a rectangular 45°/45° constant area duct and last one is 22.5°/22.5°/22.5°/22.5°double S-shape duct. All the ducts have of width 50mm (W) and height 100mm (b) with a centre line length of 600mm (L). They are constituted of four equal segments each subtending at an angle of 22.25°. The parallel horizontal walls (top and bottom) of the duct are made from 12mm thick transparent perspex sheet whereas the curved vertical walls (convex and concave) are fabricated with 3mm thick perspex sheet. These side walls of the duct are made by bending the sheet and fastened by screws with the top and bottom parallel walls. The radii of curvature of the outer and inner curved walls of the C-duct are 407mm 357mm respectively. The mean radius of curvature of the duct is 382mm (R_c). The radii of curvature of the outer and inner curved walls of the S-duct are 407mm 357mm respectively for both first and second bend of the S-duct. The mean radius of curvature of the duct is 382mm (R_c).The radii of curvature of the outer and inner curved walls of the duct are 407mm 357mm respectively for all first, second, third and forth bend of the DS-duct. The mean radius of curvature of the DS-duct is 382mm (R_c). Two straight constant area ducts of cross-sectional area 50mmx100mm were attached as extension pieces at the inlet and exit respectively. Length of these two extension pieces are 100mm. They help fixing the inlet and outlet conditions of the flow. There are six sections considered at the middle point of all these six pieces of rectangular curved duct. These sections are inlet-section, section-A, section-B, section-C, section-D and outlet-section as shown in Fig.2. These six rectangular sections are separately shown in Fig.3. Wall static pressure on different walls of the duct was measured by using the wall static pressure tapping. Referring to the duct section as shown in Fig.3, there are five holes at a distance 5mm, 15mm, 25mm, 35mm and 45mm from the edge on the top and bottom faces. Similarly on the inside and outside curved faces there are nine holes at a distance 5mm, 15mm, 30mm, 40mm, 50mm, 60mm, 70mm, 85mm and 95mm from the edge. All the holes are drilled with a diameter of 2mm. Hollow stainless steel tubes of length 20mm to 25mm are inserted into the holes such that tubes ends are not projected beyond the inside surface of the walls of the duct. The tubes are fitted into the hole by using adhesive available in the market. For recording any pressure measurement, a particular tap is connected to the inclined tube manometer through flexible tube while all other tapping are plugged by caps, the other limb of differential manometer are kept open to atmosphere. A multi-tube inclined (35° with vertical) manometer has been used to measure the pressure head at different points simultaneously. Kerosene oil is used as manometric liquid. The velocity of air was measured by inserting a Pitot tube at the mid point of the exit section of the duct. Experiment is carried out for air velocity of 40m/s to ensure incompressible flow condition at low Mach number

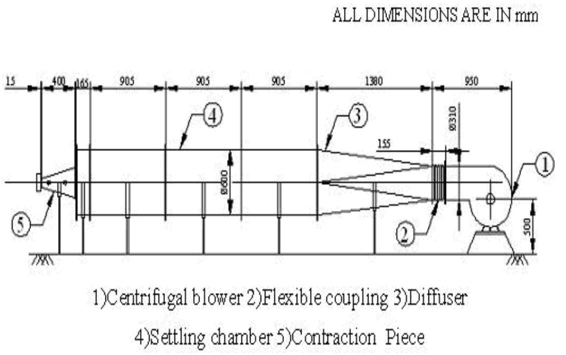
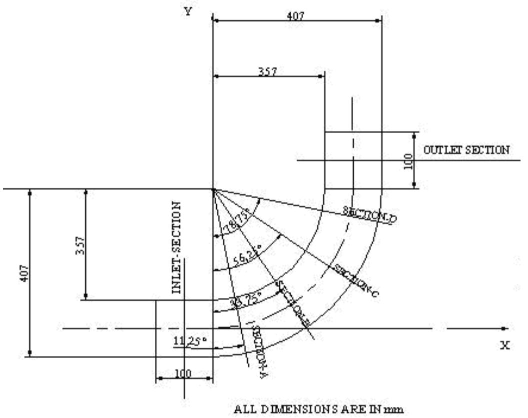
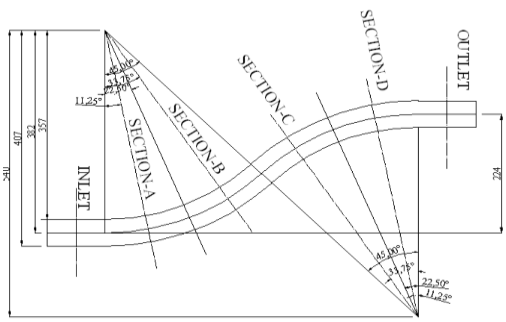


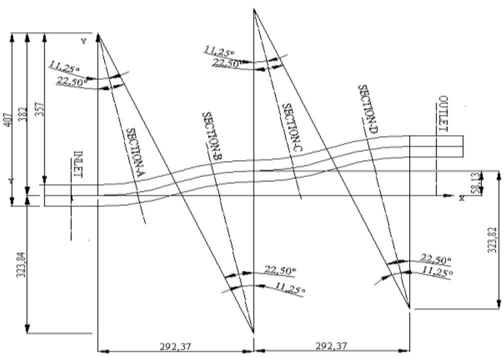
Fig. 1. Schematic Layout of the Experimental Set-up



(a) C-duct



(b) S-duct



(c) DS-duct

Fig. 2. Schematic Diagram of the C-duct, S-duct and DS-duct showing its different planes

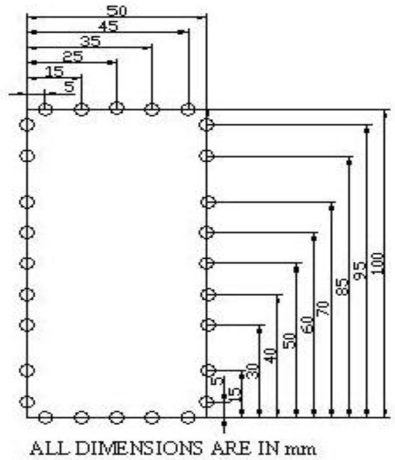


Fig. 3. Measuring Locations of Duct Sections

III. RESULTS AND DISCUSSION

The flow characteristics have been evaluated by variation of normalized mass average mean velocity, between the curved walls of the flow at various cross sections. Measured flow quantities have been presented in the form of 2-D profiles. All the velocities were normalized with respect to the inlet mass average velocity.

III.1. Mean velocity contour

Normalized mean velocity contour at four sections of the curve ducts are shown in Fig.4(a). - 4.(d). The contours are drawn based on the mass averaged mean velocity distribution as shown in Fig.4(a). - 4.(d).

Mean velocity flow distribution at the Inlet for the three ducts is almost similar and uniformly distributed in the entire cross-sectional area.

At B-Section for C-Duct higher velocity cores occupies almost 80% cross-sectional area situated near to wall-2 side and decreases at the wall-1 side. The high velocity core is occupied major part of the cross-sectional area due to combined effect of curvature and centrifugal force which forces the bulk movement of flow from wall-1 to wall-2.

At Section-B for S-Duct high velocity zone occupies very less area compared to B-Section of C-Duct and it is located along the wall-1 side of the section. Here due to larger angle of turn high momentum fluid has shifted towards the wall-1 side and bulk movement of flow taken place in the same fashion as in the case of C-duct.

At Section-B for DS-Duct mean velocity distribution shows high velocity fluid occupies almost entire cross-sectional area. The bulk flow tends to move from wall-2 to wall-1 side and as a result flow has accelerated close to the wall-1.

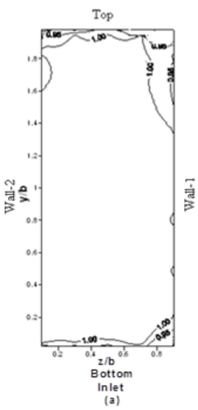
At Section-C for C-Duct high velocity core is shifted a little towards the wall-1 side and normalized velocity gradually decreases close to the wall-1 side and bulk flow has a tendency to move towards the centre of the section which is mainly due to downstream curvature effect and generation of secondary motion.

For S-Duct Section-C is after inflexion. In this section high momentum fluid has shifted towards the wall-2 and occupied middle portion of the section. This is due to the change in direction of curvature of the duct after the inflexion plane and as a result of low momentum fluid occupied near the wall-1.

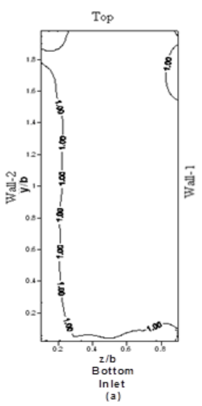
At Section-C for DS-Duct high velocity core fluid completely shifted towards the wall-1 side which is similar to C-duct. Low momentum fluid accumulates near the wall-2 sides of the third bend and occupied entire depth of the section.

At Outlet for C-Duct, high velocity cores with uniform velocity occupied almost 90% of total cross-sectional area and the core is slightly shifted towards the wall-2 but at the outlet section of S-Duct maximum velocity zone occupies very less area compared to the C-Duct. Though the flow has been spread along the mid plane towards wall-2 but instability near the top corner of wall-1 has observed. The bulk flow has diffused substantially at the outlet. In case of DS-Duct high velocity cores occupies 80% of entire cross-sectional area except very close to the wall-1 of the top corner and bottom parallel wall. Flow instability at wall-1 is not visible at the DS-Duct outlet which is clear in the case of S-Duct outlet.

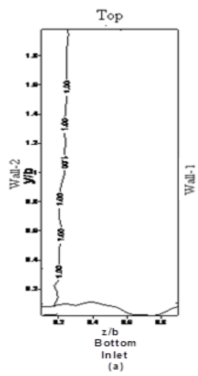
Comparing the three different types of ducts it is evident the normalized mean velocity at the outlet section is more or less uniformly distribute in the cases of C-duct and DS-duct whereas the flow through S-duct is not uniformly distributed which may be due to the flow instability and curvature effect.



C-Duct

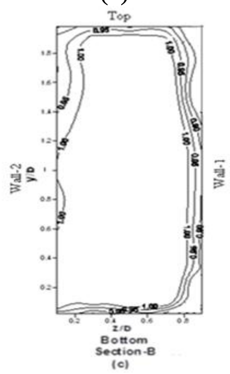


S-Duct

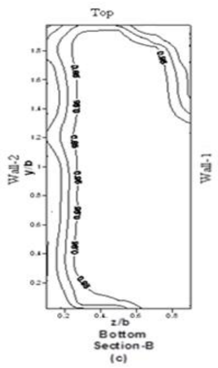


DS-Duct

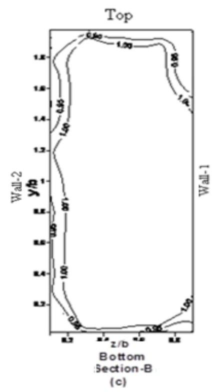
(a) Inlet Section



C-Duct

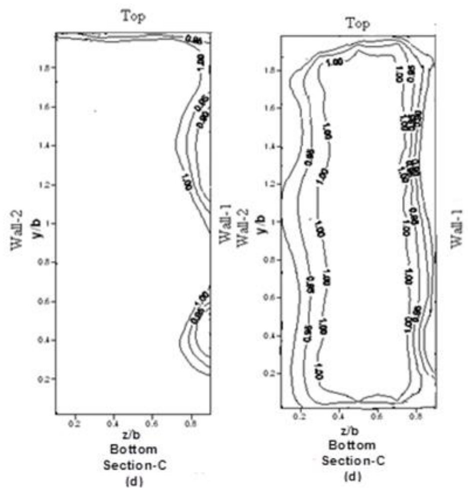


S-Duct

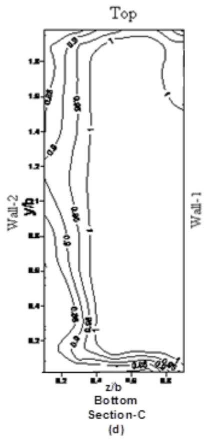


DS-Duct

(b) Section B

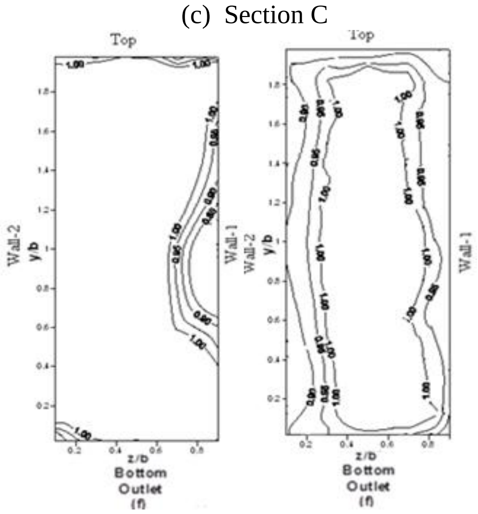


C-Duct S-Duct

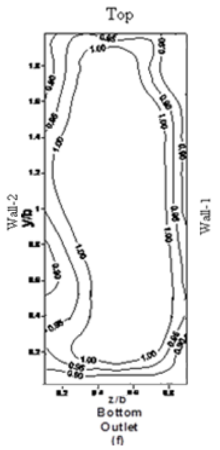


DS-Duct

(c) Section C



C-Duct S-Duct



DS-Duct

(d) Outlet Section

Fig.4. Normalized Mean Velocity at Section B, Section C and Outlet Section for different Ducts

III.2. Numerical Validation

In the present study a preliminary investigation was carried out using different turbulence models available in FLUENT. The main differential equations used here continuity equation, Momentum equation and transport equations used in standard k-ε model.

The law of conservation of mass applied to a fluid passing through a fixed control volume yields the following equation of continuity For a Cartesian coordinate system, where *u, v, w* represent the x, y, z, components of the velocity vector, can be written as

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0$$

The Navier-Stokes equations form the basis upon which the entire science of viscous flow theory has been developed. If the flow is incompressible and the coefficient of viscosity μ is assumed to be constant, The Navier-Stokes equations used in simpler form as

$$\rho \frac{D\bar{U}}{Dt} = \rho f - \nabla p + \mu \nabla^2 U$$

In our investigation flow velocity is 40 m/s i.e. the flow is completely incompressible. Consequently, the temperature gradient is ignored at the time of analysis and the detailed discussion of energy equation is not needed for our study. The transport equations used in the standard k-ε model are as follows:

Turbulent kinetic energy:

$$\rho \frac{D\bar{k}}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_r}{\rho r_k}\right)\frac{\partial k}{\partial x_j}\right] + \left(2\mu_r s_{ij} - \frac{2}{3}\rho \bar{k}\delta_{ij}\right)\frac{\partial u_i}{\partial x_j} - \rho\varepsilon$$

Dissipation rate:

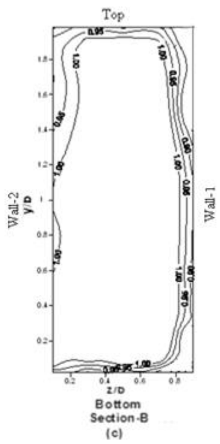
$$\rho\frac{D\bar{k}}{Dt}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_r}{\rho r_k}\right)\frac{\partial k}{\partial x_j}\right]+\left(2\mu_r s_{ij}-\frac{2}{3}\rho\bar{k}\delta_{ij}\right)\frac{\partial u_i}{\partial x_j}-\rho\varepsilon$$

The terms on the right hand side of the Equation from left to right can be interpreted as the diffusion, generation and dissipation rates of ε.

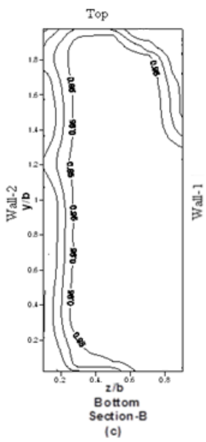
Based on the Intensive investigation it was found that Standard k – ε model of turbulence provides the best result and results obtained from computational analysis match both in qualitatively and quantitatively with the experimental results.

It is to be noted here that the inlet velocity profiles obtained during experiment are fed as an inlet condition and atmospheric pressure at outlet as outlet condition during the validation with FLUENT. Some of the validation figures are shown in Fig 5 (a) to Fig 5 (c) respectively.

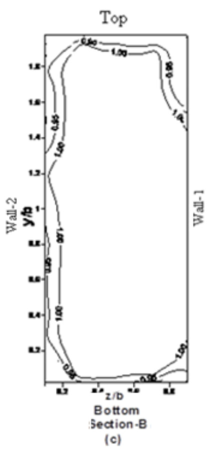
The mean velocity distribution at the Section B, Section C and Outer section are shown in Fig 5 (a), Fig 5(b), and Fig 5(c), show a reasonably good agreement of the computational investigation with the experimental results. These agreements confirm that the CFD code using Standard k – ε model can predict the flow and performance characteristics reasonably well for similar geometries with same boundary conditions.



C-Duct

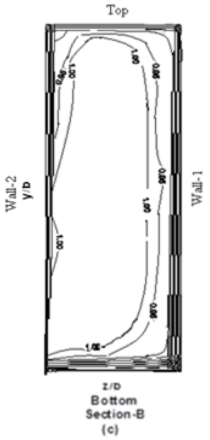


S-Duct

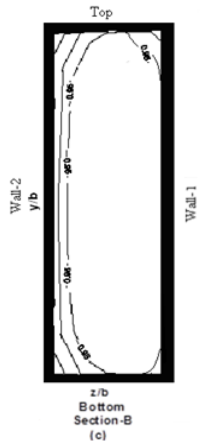


DS-Duct

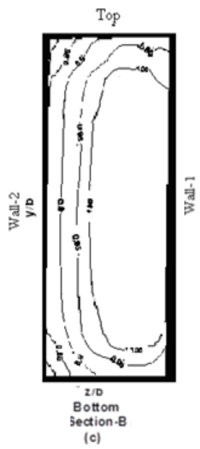
Experimental



C-Duct



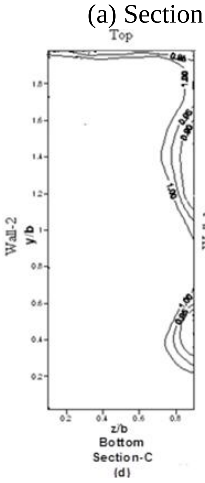
S-Duct



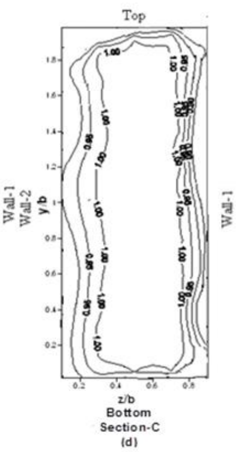
DS-Duct

Computational

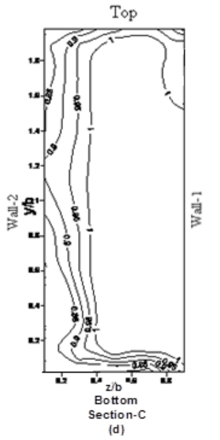
(a) Section B



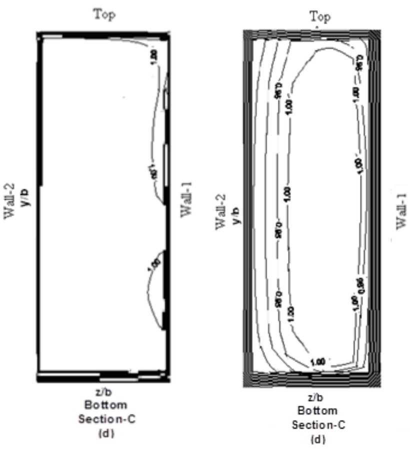
C-Duct



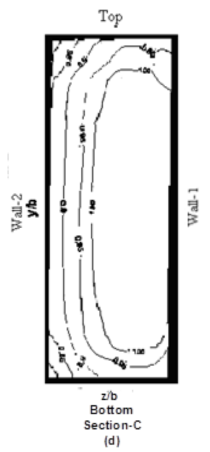
S-Duct



DS-Duct
Experimental

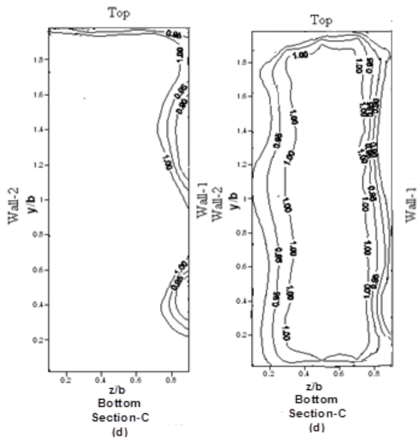


C-Duct S-Duct

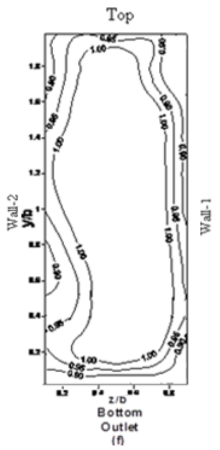


DS-Duct
Computational

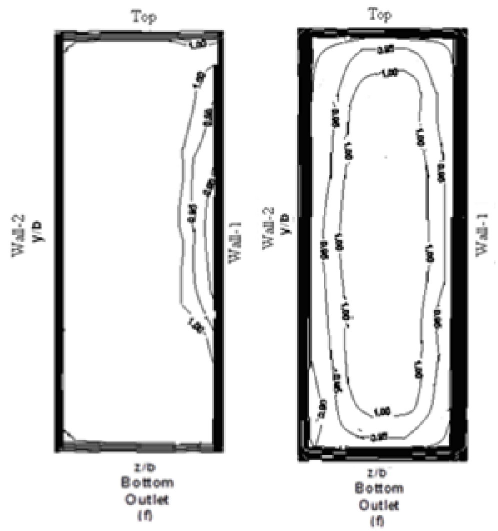
(b) Section C



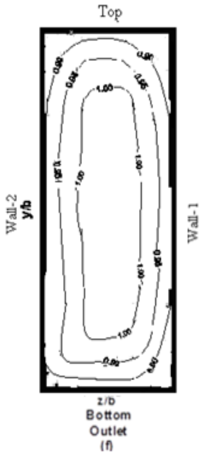
C-Duct S-Duct



DS-Duct
Experimental



C-Duct S-Duct



DS-Duct
Computational

(d) Outlet Section

Fig.5. Comparison of normalized velocity distribution at Section B, Section C and Outlet Section obtained through Computational and Experimental investigation for different Ducts

IV. CONCLUSION

From the present investigation the following conclusions have been drawn:

1. For C-duct, the bulk flow is shifting from outer wall to the inner wall along the flow passage and for S-duct, the bulk flow shifting from outer wall to the inner wall in the first half and from inner wall to the outer wall in the second half along the flow passage of curved ducts are very instinct.
2. Due to the imbalance of centrifugal force and radial pressure gradient, secondary motions in the forms of counter rotating vortices have been generated within both the C-duct and S-duct. This may be termed as pressure driven secondary flow
3. For DS-duct the velocity distributions shows the Bulk of flow shifting from inner wall to outer wall in the first bend and third bend of the duct and outer wall to inner wall in the second bend and forth bend of the duct along the flow passage is very instinct.
4. Flow at end of the C-duct and DS-duct is purely uniform in nature due to non existence of secondary motion.
5. A comparison between the experimental and predicated results for all the Constant area curved ducts show good qualitative agreement between the two.

REFERENCES

[1]. M.,Rowe, Measurements and computations flow in pipe bends, Journal of Fluid Mechanics,Vol 43, n 4, pp. 771-783,1970.
[2]. S Bansod, and P. Bradshaw, The Flow in S-shaped Ducts, Aeronautical Quarterly, Vol.. 23, pp. 131-140, 1972.
[3]. M. M. Enayet, Gibson, M. M., Taylor, A. M. K. P. and Yianneskis, M., Laser Doppler measurements of Laminar and Turbulent Flow in a Bend, Int. Journal of Heat and Fluid Flow ,Vol 3, pp. 211-217, 1982.
[4]. J. Azzola, A. C.H Humphrey, H. Iacovides, and B. E. Launder, Developing Turbulent Flow in a U-Bend circular cross-section: Measurement and Computations, Trans. ASME, Journal of Fluid Engineering, Vol. 108, pp. 214-221, 1986.
[5]. H., Lacovides, B. E. Launder, and P. A. Loizou, Numerical Computation of Turbulent Flow through a Squared Sectioned 90° Bend, Int. Journal of Heat and Fluid Flow, Vol 8,n 4, pp. 320-325,1987.
[6]. A. M. K. P. Taylor, J. H. Whitelaw. and M. Yianneskis, Curved Ducts with Strong Secondary Motion: Velocity Measurements of Developing Laminar and Turbulent Flow, Trans. ASME, Journal of Fluid Engineering, Vol 104, pp. 350-358, 1982.
[7]. S. Thangam, and N. Hur, Laminar secondary flows in curved rectangular ducts, Journal of Fluid Mechanics, Vol 217, pp. 421-440, 1990.
[8]. W. J Kim,. and V. C. Patel, Origin and Decay of longitudinal Vortices in the development of flow in a curved rectangular duct, Trans. ASME, Journal of Fluid Engineering , Vol. 116, n 3, pp. 45-52,1994.
[9]. J. M.,Burns, H. H. Fernholz, and P. A Mankewitz,. An experimental investigation of a three dimensional turbulent boundary layer in an S-shaped duct, Journal of Fluid Mechanics, Vol. .393, pp.175-213,1999.
[10]. [N. Singh, S. N. Singh and V., Seshadri, Flow Characteristics of Asymmetric Y-Shaped Duct, Proceedings of 31st National Conference on Fluid Mechanics and Fluid Power, Jadavpur University, 2004, pp. 749-757
[11]. A. J. Benin, F Gul, and E Pasqualotto,. Loss Correlations for Developing Turbulent Pipe Flow, Journal of International Review of Mechanical Engineering, January, 2007.
[12]. M. Kahrom, and A. Shokrgozar, Evaluation of Turbulence Models in Predicting Turbulence Penetration into Low Reynolds Number Regions, Journal Review of International Review of Mechanical Engineering, Vol. 4. n. 1, January, pp. 51-59, 2010.

[13]. P. K. Sinha., A. N. Mullick, B. Halder and B. Majumdar, A Parametric Investigation of Flow through an Annular Curved Diffuser, Trans Prase worthy Prize, Journal of International Review of Aerospace Engineering, Vol. 3. No. 5, pp. 249-256, 2010

[14]. A.K. Biswas, P. K. Sinha., A. N. Mullick and B. Majumdar, A CFD Investigation of Flow through a Constant area Curved Duct,Trans Prase Worthy Prize, Journal of International Review of Mechanical Engineering, Vol. 6. No. 7, pp. 249-256, 2012.

[15]. A.K. Biswas, P. K. Sinha., A. N. Mullick and B. Majumdar, A CFD Investigation of Flow through a Constant area Curved Duct, Trans Prase Worthy Prize, Journal of International Review of Mechanical Engineering, Vol. 6. No. 7, pp. 249-256, 2012.

[16]. A.K. Biswas, P. K. Sinha., A. N. Mullick and B. Majumdar, “A CFD Investigation of Flow through a Constant area Curved Duct”,Trans Prase Worthy Prize, Journal of International Review of Mechanical Engineering, Vol. 6. No. 7, pp. 249-256, 2012.

[17]. A. F. Elsafty, M. M. Abo Elazm ,. Improving Air Quality in Enclosed Parking Facilities Using Ventilation System Design with the Aid of CFD Simulation, Journal of International Review of International Review of Mechanical Engineering, Vol. 3. n. 6, pp. 796-807, 2009.

[18]. A.M Ahmed,. M. Ahmad and A. Rahim, Numerical Prediction of Effect on Air Flow Rate in the Presence of Heated Obstruction within a Room, Journal of International Review of International Review of Mechanical Engineering, Vol. 4. n. 6, pp. 702-710, 2009.