RESEARCH ARTICLE OPEN ACCESS

Flow Visualization for secondary flow and velocity separation due to curvature effect inside a Curved Double S Duct

[1]Shantonu modak, [2]Subrata Majumdar[1] A.K. Biswas, [2]P.K. Sinha
[1]*National Institute of Technology, Durgapur, West Bengal, India*
[2]*Durgapur Institute of Advanced Technology& Management, West Bengal, India*
Corresponding Author; Shantonu Modak

ABSTRACT
In the present investigation the distribution of mean velocity are experimentally studied on square cross section Double-S duct.The experimental set up Double-S duct having center line length of 750 mm and square cross section of (100×100) made up of transparent Perspex sheet to study flow visualization. Experiment is carried out at mass averaged mean velocity 40 m/s. The velocity distribution 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. Flow at end of the 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.

Keywords:Double-Sduct,Secondary motion,Flow visualization,velocitycontour, Standard k – ε model.

Date Of Submission: 05-09-2019 Date Of Acceptance: 22-09-2019

I. INTRODUCTION

Duct is old engineering mechanical devices. It is nothing but conduit or passage usedfor convey liquid or gases .The duct is extensively used since earlier days. In present time duct is used in Each and every field from engineering to medical.

Firstexperiment and study has done on straight duct. After some year as requirement of curved duct occurs then start Study on curved duct start. Flow characteristics through a constant area duct are the foundation and fundamental research area of basic fluid mechanics since the concept of potential flow and frictional losses during the time of flow were established. The different types of ducts have been classified as straight, curved, annular, sector, C-shaped, S-shaped, Snake and Coil section, etc. But generally duct is classified as straight, curved, annular and sector ducts, which are used in most industrial applications.Many engineering problems is represented by internal viscous flows in curved duct. nozzles, diffusers and other components of turbo machinery. Aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. are the main application area of such ducts.Duct are also used in heating ventilation and air conditioning (HVAC) to deliver and remove air.A duct system is also called ductwork. Planning (laying out), sizing, optimizing, detailing, and finding the pressure losses through a duct system is called duct design.

its huge requirement in chemical and mechanical engineering. Investigation on Internal flow through a curved duct is essential for getting improved performance of the overall propulsion system. Including combustor, aircraft intakes, internal ventilation duct, cooling system of gas turbines, ventilation ducts, wind tunnels etc. are the main application area of such ducts.. They observed that strong secondary flow motion occurs over the entire duct cross-section for the inclined ribs. The flow structures between two consecutive ribs show that the fluid flows along the ribs from one end of the ribs to the other end, and then turns back at the transverse center. An experimental study was carried out by Cioncolini and Santana [2] in 2006 to investigate the transition from laminar to turbulent flow in helically coiled pipes. The main purpose of this investigation is to visualize the development of secondary flow and velocity separation due to curvature effect which is shown by velocity and pressure contour distribution by software package Surfer which happens when the flow takes place through the bend in the curvature. This secondary flow arises as a result of centrifuges. This result extreme help for optimum design of Double-S duct and shown in the form of contours by using the software package

The secondary flow in a curved ductofsquare cross-section is investigated experimentally by Yamamoto et. al. [3] in 2006.Three walls of the duct (except the outer wall) rotate around the center of curvature and an azimuthal pressure gradient was imposed.

The variation of the flow patterns with change of flow parameters is compared with that of numerical calculations and was found to be in good agreement. A study on flow transition and development in circular and rectangular ducts was reported by Zanounet. al. [4] in 2009. The hot-wire anemometer was used to carry out measurements close to the circular duct exit; the Laser Doppler Anemometry (LDA) was utilized for the measurements. Transition criteria in both ducts were discussed, reflecting effects of flow geometry, entrance flow conditions, and entrance length on the transition Reynolds number. A laminar behavior was maintained up to R_e=15.4x10^3 and R_e=2x10^3 in the circular and rectangular ducts, respectively, and the transition was observed to take place at different downstream positions as the inlet flow velocity varied. Secondary flow in semi-circular ducts is reported by Larsson ET. al. [5] in 2011. They found that the secondary flow in semi-circular ducts consists of two pairs of counter rotating corner vortices, with a velocity in the range reported previously for related configurations. Agreement between simulation and experimental results are excellent when using a second moment closure turbulence model, and when taking the experimental and numerical uncertainty into account.

II. EXPERIMENTAL SETUP

Experiment is carried out using the facility of wind tunnel at the Aerodynamics Laboratory of National Institute of Technology, Durgapur. The experimental set uphaving consists of a wind tunnels, Double-S duct, traversing mechanism, three hole probe manometer. Thewind tunnel is drivenby an electric motor of 5.5 kW power. The testpiece is connected with the settling chamber via a constant area straight duct to ensure uniform velocity profile at the inlet section of the test piece. The geometry of the curved duct under test is shown in Fig.1 and Fig. 2. It is a square cross section Double-S duct of 100mm wide and 100mm height with a centerline length of 750mm. It is constituted of four equal segments of22.5 each. The entire test piece is made of Perspex sheet. Two straightconstant area ducts of cross sectional area 100mm x 100mm and 100mm long are connected as extension pieces at the inlet and exit of the test piece. hence there is actually six segment are considered as six sections and they are assigned as SURFER.

Inlet Section, Section - A, Section- B, Section - C, Section- D andOutlet Section

The staticand dynamic pressure is pressure is measured by the three hole probe manometer and traversingmechanism. Inside internal pressure of the duct was measured by using the three hole probe. We first cut 10 mm single rectangular channel on upper faces of Inlet, A, B, C, D, and Outlet. And then mark 12 line on each rectangular slice at distance of 5,10,15,25,35,45,55,65,75,85,90,95 respectively from inside wall and then taken 25 pressure difference reading into depth normal direction by the three hole probe at each line. Finally we have taken 300 reading {25*12} in each section . The three hole probe fitted by one fastened device and the projected into the hollow rectangular channel section and make closed all portion other then three hole probe section area and then measured the pressure difference reading by inclined manometer. This process continued for all line and all section

Fig.1 Double –S duct set up

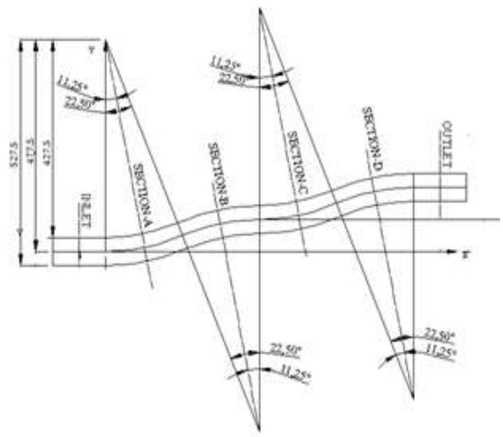


Fig.2Schematic Double –S duct

III. RESULT AND DISCUSSION

III.1. Mean velocity contour

Fig.3(a)to Fig. 3(f) shows the normalized mean velocity distribution for six sections of theDouble S-duct. This distribution hasrevealed more detailed information regarding the flow development within the duct.

At InletSection, Fig. 3(a) shows that velocity is evenly distributed throughout the entire cross-sectional area except some regions very close to bottom wall. From the figure it is clear that the low velocity is accumulated closed to the bottom right and left corner. This indicates generation of weaker flow at the corners and low momentum fluid accumulated toward the bottom surface of inlet.

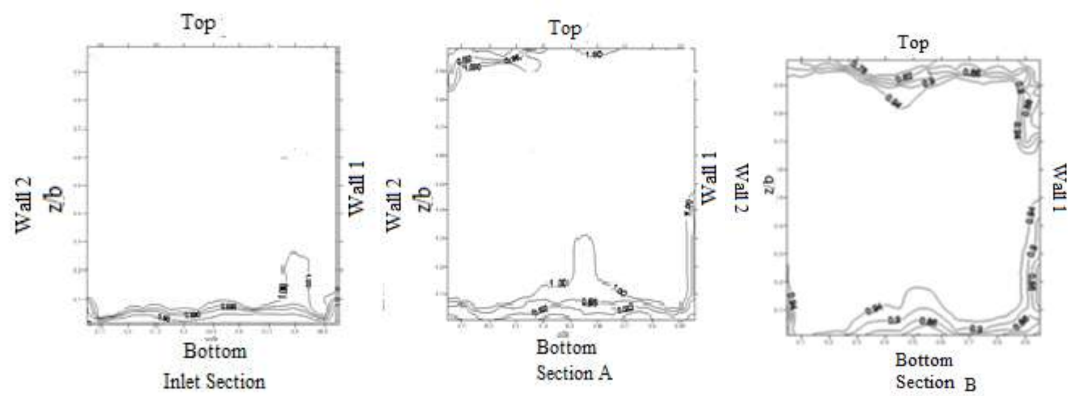
At Section-A Fig.3. (b), shows that here is also uniform distributed throughout the entire cross sectiondue to inertia of high momentum flow. It has also seen that high momentum flow is little bit shifted in bottom wall as compare to inlet section.

At Section-B Fig. 3(c),When high momentum fluid passes from section A to section-B and cross first inflexion plane, High momentum fluid has shifted towards Wall 2 which signifies that it achieve maximum velocity toward Wall 2 side low velocity is achieve toward Wall 1 .Which leads to produce velocity gradient between the wall. It'shappened due to curvature effect which is also called centrifugal effect.

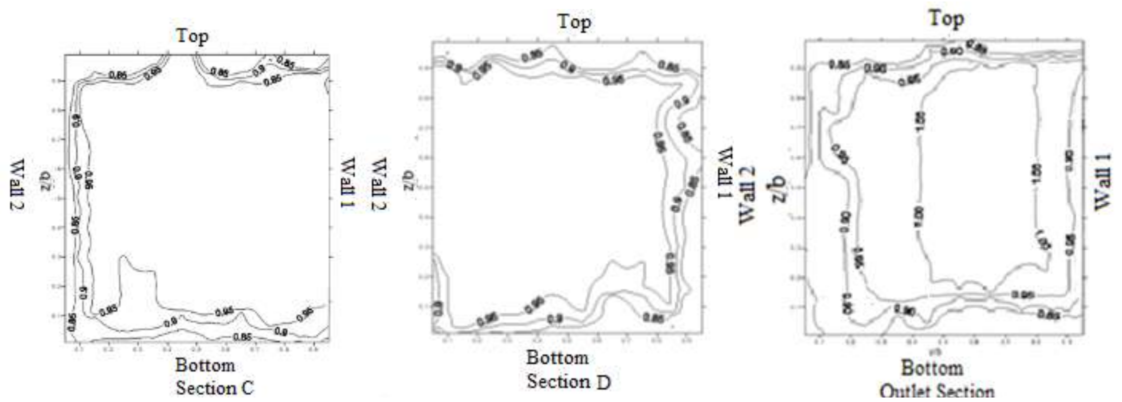
After inflexion, At Section-C Fig.3(d) shows that as flow passage from Section-B to Section-C flow tends toestablished uniformity distribution throughout the entire cross section.

At Section-D Fig. 3(e), it is clearly depict that high velocity is shifted toward Wall 2 because angle of turn of duct which is nothing but curvature effect. This indicates that change in flow direction as the curvature of centerline changes after the third inflexion plane.

At the exit Outlet section Fig.3(f) shows that high momentum flow is little bit shifted toward Wall 1. But more profile distribution along Wall 2. There is marginal flow instability observed in bottom left corner though the flow has been spreading towards the plane Wall 2 and maximum velocity observed at Wall 1.



(a) (b) (c)



(d) (e) (f)

Fig.3 Normalized mean velocity distribution.

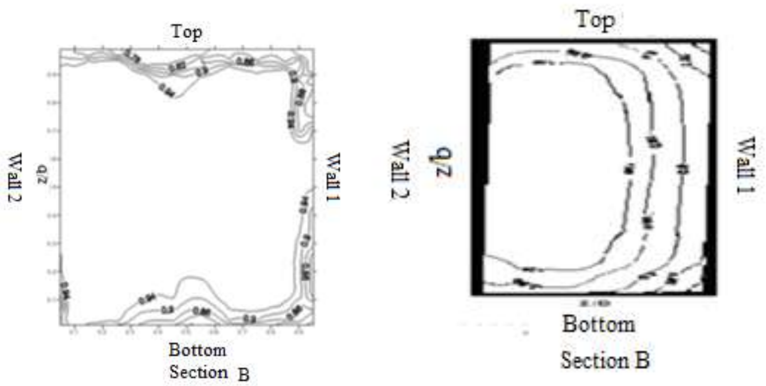
III.2. Numerical Validation

In the present study a preliminary investigation was carried out using different turbulence models available in FLUENT. 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 profiles obtained during experiment are fed as an inlet condition during the validation with FLUENT. Some of the validation figures are shown in Fig 4 (a) to Fig 4 (c) respectively.

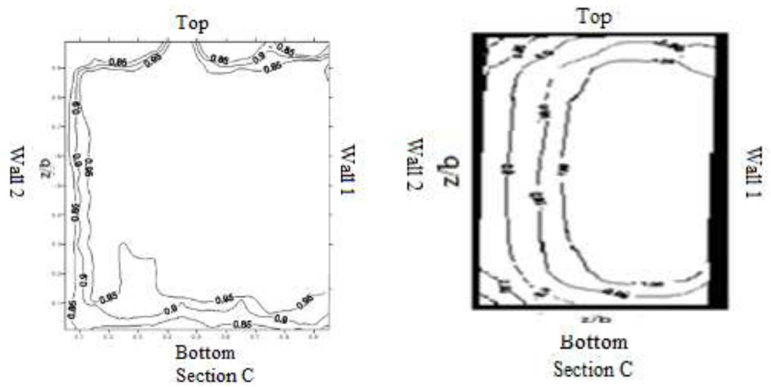
All three figures indicate that the mass averaged mean velocity contours obtained by computational and experimental investigations, which shows a qualitative matching to each other.

However a slight mismatch can be observed at the outlet Section close to the outer wall. This could be due to the complicated nature of flow at this plane, which was not properly predicted by the process of computer simulation.

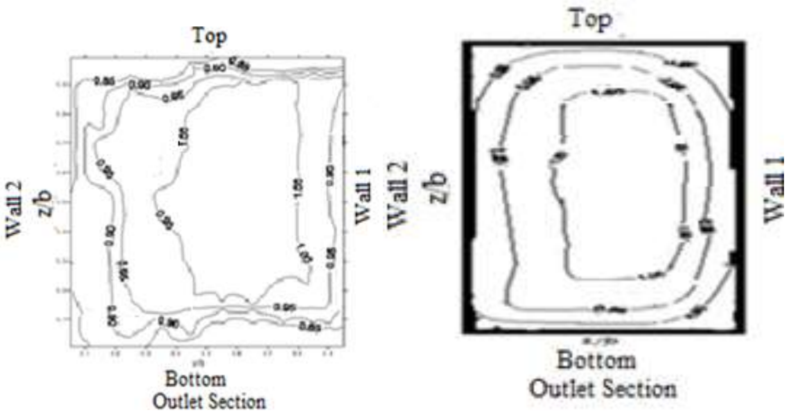
The mean velocity distribution at the Section A, andSection B are shown in Fig 4 (a) and Fig 4(b) show a reasonably good agreement of the computational investigation with the experimental results. These agreements confirm that the CFD code using Standard k – ε model can predict the flow and performance characteristics reasonably well for similar geometries with same boundary conditions.



(a)



(b)



(c)

Fig.4. Comparison of normalized velocity distribution at Section B, Section C and Outlet Section obtained through Computational and Experimental investigation

.

IV. CONCLUSION

From the present investigation the following conclusions have been observed:

1. The velocity distributions shows the Bulk of flow shifting from inner wall to outer wall in thefirst 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.
2. It is also observed that due to the imbalance of centrifugal force and radial pressure gradient, secondary motions in the form of counter rotating vortices have been generated.
3. The velocity at end of the duct is purely uniform in nature due to non existence of secondarymotion.
4. A comparison between the experimental and predicated results for all the Constant area curved ductsshow good qualitative agreement between the two.

REFFERENCE

[1]. A.M.K.P. Taylor, J. H. Whitelaw and M. Yianneskis, ”'Developing Flow in S-curved Ducts”, Part II-Circular Cross-Section Duct",1984, NASA Contract Report No. 3759.(106).

[2]. Steven R. Wellborn, Bruce A. Reichert and Theodore H. Okiishi, "An Experimental Investigation of theFlow in aDiffusing S-Duct.", NASA Technical Memorandum 105809, AIAA-92-3622, 28th JointPropulsion Conference and Exhibit cosponsored by the AIAA, SAE.

[3]. S. C. M. Yu,”Turbulent flowcalculation in S-shaped diffusing ducts using viscous marching technique”. 6th International conference on computational method and experimental measurements “1993 vol. 1:Heat and Fluid flow, edited by C. A. Brebbia and G. M. Carlamgro, computational mechanics publication, Southamton, Boston, co-published with Eleevier allied science, New York.

[4]. K. M. Britchford, A. P. Manners, J. J. McGuirk, S. J. Stevens, "Measurement and prediction of low in annular S-shaped ducts.", Experimental Thermal and Fluid Science, Volume 9, Issue 2, August 1994,Pages 197–205. (119).

[5]. T. Sonada, T. Arima and M. Oana., "The Influence ofDownstream Passage on the Flow within an Annular S-Shaped Duct", Trans ASME, Journal of Turbo Machinery, Vol. 120, 1998, pp. 714-722.(81).

[6]. H. Lacovides, B. E. Launder, P.A. Loizou and H. H Zhao, “Turbulent boundary layer development around a squaresectionedU-bend:Measurementsand computation.” Trans. ASME ,Journal of Fluid Engineering,1990,vol.112,pp.409-415.(12).

[7]. N. Hur, S. Thangam, C. G. Speziale, “Numerical Study of Turbulent Secondary Flows in Curved Ducts”, Journal of Fluids Engineering. 112(2), 205 (Jun 01 1990). (132).

[8]. G. C. Cheng and S. Farokhi, "On turbulentflows dominated by curvature effects", Trans ASME, Journal of Fluid Engineering, Vol. 114, 1992, pp. 52-57.(65

[9]. P. L. Wilson and F. T. Smith, “Compressible Swirling Flow Through a Significantly Bent duct”, Computersand Fluids, 2005, Volume 34, pp. 891-926.(5)

[10]. V. D. Sakalis, P. M. Hatzikonstantinou, P. K. Papadopoulos, “Numerical Procedure for the Laminar Developed Flow in a Helical Square Duct”, Journal of Fluids Engineering. 127(1), 136 (Mar 22 2005).(129)

[11]. G. J. Harloff, C. F. Smith, J. E. runs and J. R. Debonis, "Navier Stokes Analysis of 3D S-ducts", Journal of Aircraft, 1993, Vol. 30, No.4, pp.526-533.(69

[12]. ZhipengDuan, M. M. Yovanovich, Y. S. Muzychka, “Pressure Drop for Fully Developed Turbulent Flow in Circular and Noncircular Ducts”, Journal of Fluids Engineering. 134(6), 061201 (Jun 11, 2012). (125

[13]. M. Firouzi, S. H. Hashemabadi, "Analytical Solution for Newtonian Laminar Flow Through the Concave and Convex Ducts", Journal of Fluids Engineering. 131(9), 094501 (Aug 14 2009).(126)

Shantonu Modak" Flow Visualization for secondary flow and velocity separation due to curvature effect inside a Curved Double S Duct" International Journal of Engineering Research and Applications (IJERA) , vol. 9, no. 9, 2019, pp. 01-06