Pramit Khatua Journal of Engineering Research and Application *www.ijera.com*
ISSN : 2248-9622, Vol. 8, Issue 8 (Part -III) Aug 2018, pp 10-15

RESEARCH ARTICLE OPEN ACCESS

Investigation of Flow Charactertics within A Curved Square C-Shaped Duct

[1]Pramit Khatua [1]DipyamanGangopadhyay[1]A.K. Biswas, [2]P.K. Sinha
National Institute of Technology, Durgapur, West Bengal, India
Durgapur Institute of Advanced Technology& Management, West Bengal, India
Corresonding Author: Pramit Khatua

ABSTRACT:This paper presents the observations of an experimental work dealing with the measurement of normalized mean velocity and wall static pressure of a 90° C-shaped curved duct. The test duct is made up of transparent perspexsheet to facilitate the flow visualization study. The duct has a centerline distance of 750mm. The Inlet and Outlet aspect ratio of the test duct used is 1.0. Wall pressures are measured with the help of an inclined manometer withthe inclination of 35°. The manometer had two tubes emanating from it: one left open to the atmosphere and the other connected to the steel pipes attached to the four walls of the curved duct. The difference in the readings helped calculate the static pressure and thereby the normalized pressure. Wall pressure distribution along the curved and parallel walls of the duct at 0°, 22.5°, 45°, 67.5° and 90° measuring sections was measured. The investigation for wall static pressure distribution is carried out at the velocity of 40m/s. This paper also presents an experimental work carried out with measurement of normalized mean velocity of the mentioned curved square C shaped duct, taken at the same velocity. The trend of the normalized mean velocity contour development inside the C duct shows continuous decrease of the normalized velocity as we move from the Inlet Section towards the Outlet Section. The distribution of wall static pressure and normalized mean velocity contours are mapped by using SURFER software package. The trend of wall static pressure development on the walls of C shaped duct shows that as the flow proceeds towards the curvature, there exists a high pressure gradient between the Outside Face and Inside Face due the centrifugal force acting along the curvature. This shows the bulk shifting of flow towards the Inside Face. This is due to the generation of secondary motion in a plane perpendicular to the primary flow. The main purpose of this investigation is to show the development of secondary flow which happens when the flow takes place through the bend in the curvature. This secondary flow arises as a result of a centrifugal force acting when the flow moves through the bend.

KEYWORDS:C- shaped curved duct, Secondary flow, contra-rotating vortices, SURFER.

DATE OF SUBMISSION:16-08-2018 DATE OF ACCEPTANCE: 30-08-2018

I. INTRODUCTION

Duct is a very old fluid mechanical device. Study of flow characteristics through a constant area duct is 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. Duct is nothing but a passage way made of sheet metal or other suitable material used for conveying air or other gas and liquid at different pressure. Planning, sizing, optimizing, detailing of duct work and also finding the pressure losses through a duct system is generally termed as duct design. The different types of ducts have been classified as straight, curved, annular, sector, Cshaped, Sshaped, Snake and Coil section, etc. Among all these ducts the common ones are straight, curved, annular and sector ducts, which are used in most industrial applications. A large and important class of engineering problems is represented by internal viscous flows in curved duct. Accurate and reliable calculations of such flows are of great practical interest in the prediction of flow through inlets, nozzles, diffusers and other components of turbo machinery. The flow through a curved tube has attracted considerable attention not only because ofits practical importance in chemical and mechanical engineering, but also because of physically interesting features under the action of centrifugal force caused by curvature of the tube. Internal flow through a duct is essential for obtaining improved performance of the overall propulsion system. In addition to the above, aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. are the main application area of such ducts. Enayet et al., 1982, [1] 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., 1986, [2] 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. The secondary flow in a curved duct of square 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. Secondary flow in semi-circular ducts is reported by Larsson et. al. [4] 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.Three-dimensional turbulent flow in a square duct using a cubic eddy-viscosity model was studied by Gnangaet. al. [5] in 2009. This configuration presents a secondary flow and a significant anisotropy between the Reynolds stress components.Zanounet. al. [6] in 2009, reported flow transition and development in circular and rectangular ducts. 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.Biswas et. al., [7] in 2012, studied the comparison between C shaped constant area duct and diffuser by computational fluid dynamics analysis. They concluded that shifted towards the outer wall due to generation of secondary motion for both Cshaped constant area duct and diffusing duct.Sinha et. al. [8] in 2017, investigated the flow development through a duct and diffuser using computational fluid dynamics. They concluded that shifted towards the outer wall due to generation of secondary motion for both C shaped constant area duct and diffusing duct.

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 up consists of a wind tunnel, which is driven by an electric motor of 5.5 kW power. The test piece 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. It is a square 90° curved duct of width 100mm wide and 100mm height with a centerline length of 750mm. It is constituted of four equal segments of 22.5 each. The entire test piece is made of Perspex sheet. Two straight constant 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. Middle points of all the six segments are considered as six sections and they are assigned as Inlet Section, Section A, Section B, Section C, Section D and Outlet Section. The wall static pressure is measured with the help of an inclined multi-tube manometer. At each face of the respective sections twelve locations are selected to determine the wall static pressure. A total of 288 locations on the test duct had been studied. Measuring locations of each selection points for the measurement of wall static pressure is shown in Fig.2. Fig. 3 shows the schematic layout of the experimental setup. Fig. 4 shows the measuring locations for the normalized mean velocity study. There are twelve points of locations on the top faces at six different sections (0°, 11.25°, 33.75°, 56.25° 78.75°, 90°). At each hole 25 readings were taken throughout the 100mm depth. A total of 1800 locations i.e. 300 locations at each of the six sections havebeen studied.

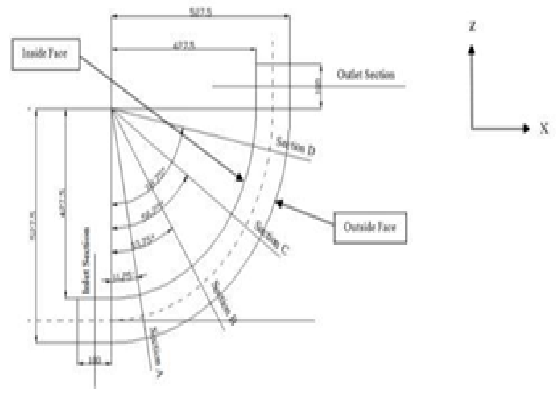

Fig. 1: Geometry of Curved Square C Shaped Duct

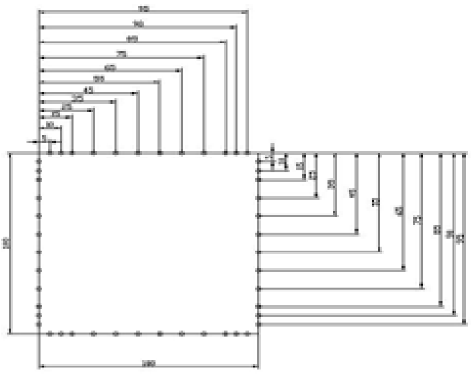

Fig. 2: Measuring locations of drill points for wall pressure measurement

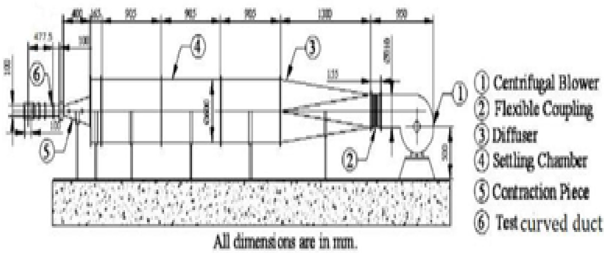

Fig. 3: Schematic layout of the experimental set up

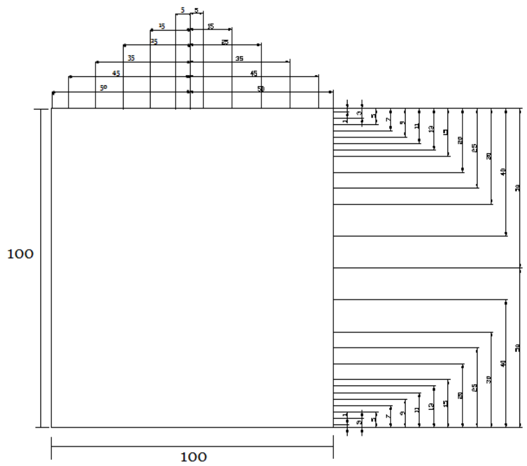

Fig. 4: Measuring locations of drill points for internal pressure measurement

III. RESULTS AND DISCUSSIONS

Fig.5.1(a), 5.1(b), 5.1(c) and 5.1(d) shows the distribution of wall static pressure for Inlet average air velocity of 40 m/s. 5.1(a) clearly indicates a continuous decrease in normalized wall static pressure over the top surface along the direction of flow. The figure also depicts that the high pressure zone is built up near Outside Face which indicates the bulk movement of flow towards the Inside Face due to centrifugal action of the high velocity fluid. This is a probable indication of the development of secondary motion between the curved surfaces from the Outside Face to the Inside Face. Fig. 5.1(b) shows the normalized wall static pressure contours of the Inside Face. The contours show that high pressure zones accumulated near the top and bottom walls lead the movement of the flow from the top and bottom walls towards the mid plane. Fig. 5.1(c) shows normalized wall static pressure distribution over the Bottom Surface along the direction of flow which depicts the same type of observation as indicated in Fig. 5.1(a). Fig. 5.1(d) shows the normalized wall static pressure contours for the Outside Face where the existence of the high pressure zone may be noted at the mid section. This indicates the movement of flow from the mid plane towards the top and bottom surfaces signifying the possible development of counter rotating vortices along the flow passage.

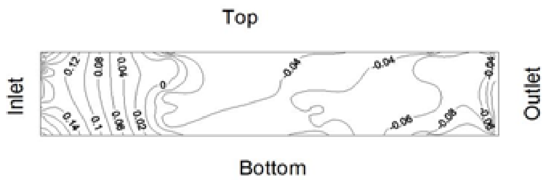

Fig. 5.1(a) Top Face

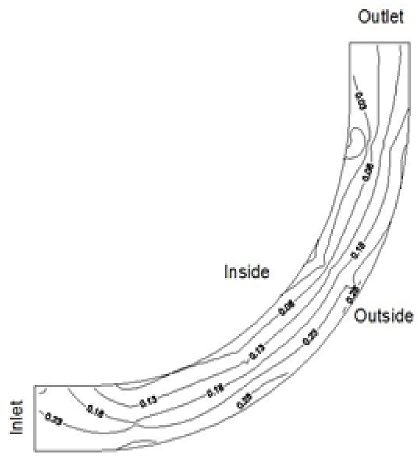

Fig. 5.1(b). Inside Face

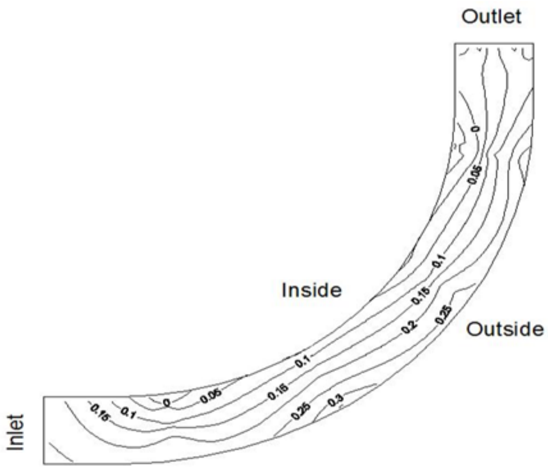

Fig. 5.1(c). Bottom Face

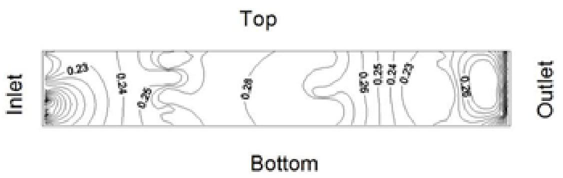

Fig. 5.1(d). Outside Face

Normalized mean velocity is the ratio of velocity at a particular measuring point to that of average velocity measured at Inlet Section. The contours drawn are shown below from Fig. 4.1(a) to Fig. 4.1(f) i.e. Inlet Section, Section A, Section B, Section C, Section D and Outlet Section. It is measured at an average velocity of 40 m/s. The normalized mean velocity contour of Inlet Section is shown in the Fig. 4.1(a). It shows that the high velocity core is distributed throughout 95% of the cross sectional area except some regions very close to the top and bottom surfaces. This proves that there is uniformity of flow throughout the Inlet Section. The normalized mean velocity contour of Section A is shown in the Fig. 4.1(b). Low velocity fluid is seen to be more along wall 1 and the high velocity core is seen to be more along wall 2 of Section A. It can be seen that fluid tries to maintain its flow direction and move towards the wall 1 side due to the centrifugal force coming into effect, which occurs due to moderate curvature of the centre line. The mean velocity contour of Section B is shown in Fig. 4.1(c). The contours in this section indicate slight reduction of normalized mean velocity. It is also observed that high velocity core occupies the major part of the cross sectional area. This happens due to centrifugal force coming into effect, occurring due to moderate curvature of the centre line. The mean velocity contour of Section C is shown in Fig. 4.1(d). It can be observed that the high velocity core has shifted a little towards wall 2 side. Due to the larger angle of turn, low momentum fluid has shifted towards wall 1 side. The mean velocity contour of Section D is shown in Fig. 4.1(e). The contours of Section D show that normalized mean velocity reduces slightly when compared to the previous section. The disturbance generated along wall 1 is seen to be enhanced at this section and spread towards wall 2 occupying more area. The normalized mean velocity contour of Outlet Section is shown in Fig. 4.1(f). The high velocity zone, which was located at wall 2 in the previous Section D, is seen to be moved towards the centre of this section occupying more space throughout the middle part of the section. The zigzag nature of the contour lines in this section depicts the development of secondarymotion.

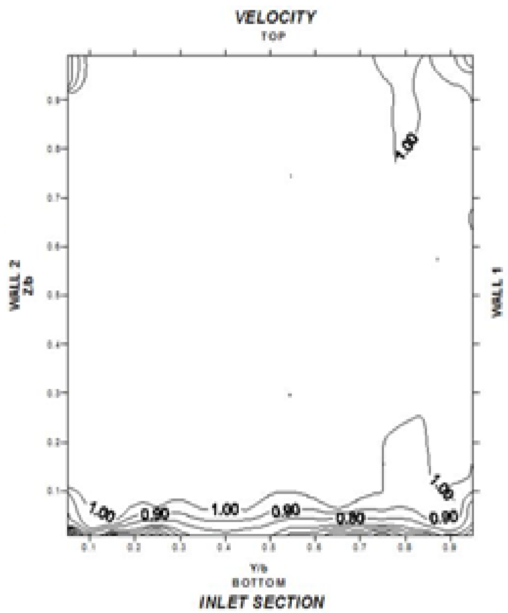

Fig. 6.1(a) Inlet Section

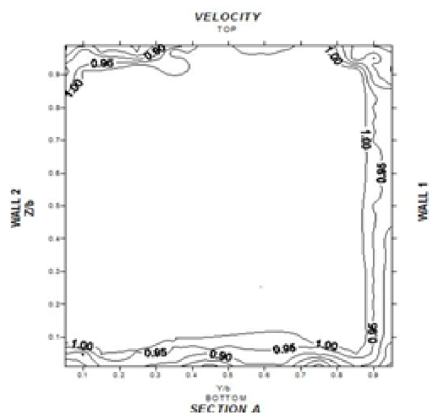

Fig. 6.1(b) – Section A

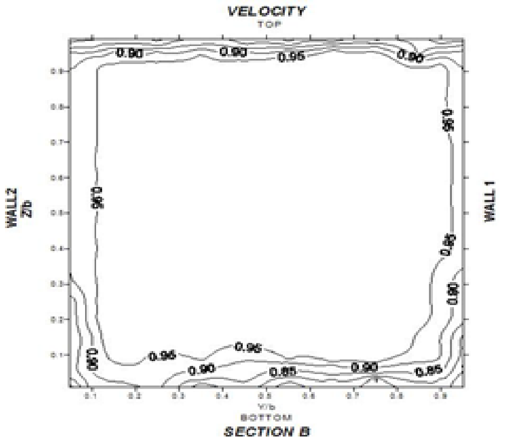

Fig. 6.1(c) Section B

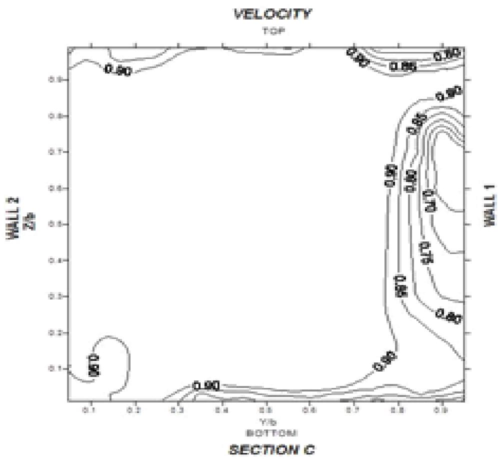

Fig. 6.1(d) Section C

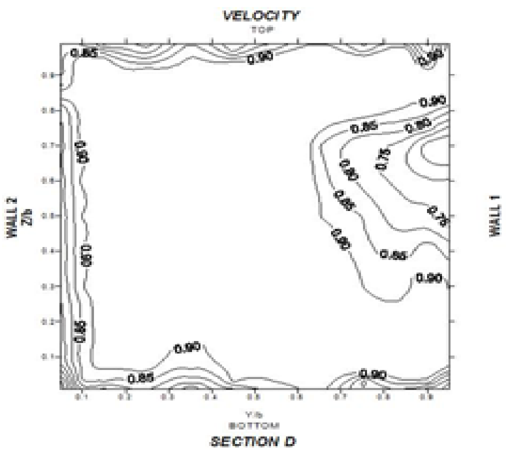

Fig. 6.1(e) Section D

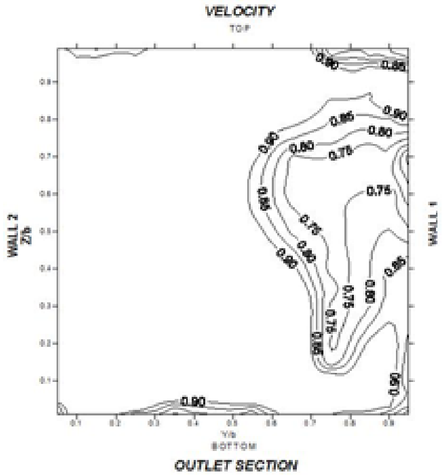

Fig. 6.1(f) Outlet Section

IV. CONCLUSIONS

From the present investigation the following conclusions have been drawn:

a) We observe a continuous decrease of the normalized wall static pressure from the Inlet to Outlet Section of the curved duct for each of the Top, Bottom, Inside and Outside Faces for the velocity of 40m/s. However, the Outside Face records a decrease which is comparatively lesser than other three faces.

b) The minimum and maximum wall pressures have occurred at the inner and the outer wall respectively under the influence of the radius of curvature and angle of turn of the duct.

c) The bulk flow shifting from outer wall to the inner wall along the flow passage of curved duct is very instinct.

d) Flow at exit is purely non-uniform in nature due to the strong secondary motion.

e) The normalized mean velocity records a continuous decrease of value from the Inlet Section to the Outlet Section along with reinstating the fact that the mean velocity core is higher at outside wall compared to the inside wall.

f) Due to the imbalance of centrifugal force and radial pressure gradient, secondary motions in the form of counter rotating vortices have been generated within the curved duct. This may be termed as pressure driven secondary flow.

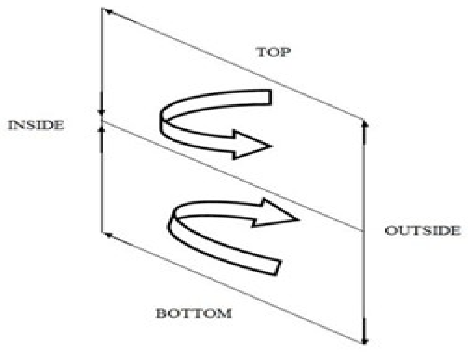

Fig: Counter Rotating Vortices

REFERENCES

[1]. Enayet, M.M. Gibson, A.M.K.P.,M.M Taylor and M. Yianneskis, "Laser Dopplermeasurements of laminar and Turbulentflow in pipe bends", InternationalJournal of Heatand Fluid Flow, 1982, pp.211-217.

[2]. [2] J. Azzola, A.C. Humphrey, H Iacovides, and B.E. Launder, "Developing turbulent flow in a U- Bend Circular cross section: Measurement andComputations", Trans ASME, 1986, Journal of. fluid Engineering ,108 pp. 214-221.

[3]. Kyoji Yamamoto, XiaoyunWub, KazuoNozakib, Yasutaka Hayamizub,"Visualization of Taylor-Dean flow in a curved duct of square cross-section", Fluid Dynamics Research,Volume 38, Issue 1, January 2006, pp. 1-8.

[4]. I.A.S. Larsson, E.M. Lindmark, T.S.Lundström and G.J. Nathan, "Secondary flow in semi-circular ducts", ASME, Journal of Fluid Engineering, 2011, Volume 133, Issue 10, 101206.

[5]. H. Gnanga, H. Naji, and G. Mompean, "Computationof a three-dimensional turbulentflow in a square duct using a cubic eddy – viscosity model", ComptesRendusMécanique, 2009, Volume 337, Issue1, pp.15-23.

[6]. E.S. Zanoun, M. Kito, and C. Egbers, "A study on flow transition andDevelopment in circularand rectangular ducts", Journal of FluidsEngineering, 2009, Volume 131, Issue 6, pp. 061204.

[7]. A.K. Biswas, A.K. Raman, N.K. Das, A.N.Mullick, P. Ray, B. Halder, and B.Majumdar, "A comparison between C shaped constantarea duct and diffuser by CFDanalysis",International ConferenceonRecent Advancesin Mechanical Engineering, 2012, pp.41-46

[8]. P.K. Sinha, A.K. Biswas, A.N. Mullick, andB. Majumdar, "Flow Development through aDuct and a Diffuser Using CFD", InternationalJournal of Engineering Research andApplications,2017, Volume 7 Issue 1, pp.46-54.

Pramit Khatua " Investigation of Flow Charactertics within A Curved Square C-Shaped Duct'International Journal of Engineering Research and Applications (IJERA) , vol. 8, no.8, 2018, pp 10-15