International Journal of Mechanical Engineering Research and
Development (IJMERD), ISSN 2248 – 9347(Print)
ISSN 2228 – 9355(Online), Volume 1, Number 2
May-October (2011), pp. 14-23
© PRJ Publication, http://www.prjpublication.com/IJMERD.asp

IJMERD

© PRJ PUBLICATION

NUMERICAL INVESTIGATION OF FLOW CHARACTERISTICS IN A C-SHAPE SUBSONIC DIFFUSER

Prasanta K Sinha
Department of Mechanical Engineering
Durgapur Institute of Advanced Technology & Management
Durgapur-713212.WB,India
E-mail: pksinha_2005@yahoo.com

Bireswar Majumdar
Department of Power Engineering
Jadavpur University,WB, India
bmaj3255@gmail.com

ABSTRACT

In the present investigation the distribution of mean velocity, static pressure and total pressure are experimentally studied on a C-shape diffuser of 30° angle of turn with an area ratio of 1.273 and centerline length was chosen as three times of inlet diameter. The experimental results then were numerically validated with the help of Fluent and then a series of parametric investigations are conducted with same centre line length and inlet diameter but with different area ratios varying from 1.25 to 2 with angle of turn 30° to 75°.

The measurements were taken at Reynolds number 2.25 x 10^5 based on inlet diameter and mass average inlet velocity. Predicted results of coefficient of mass averaged static pressure recovery (38%) and coefficient of mass averaged total pressure loss (14%) are in good agreement with the experimental results of coefficient of mass averaged static pressure recovery (35%) and coefficient of mass averaged total pressure loss (13%) respectively. Standard k-ε model in Fluent solver was chosen for validation.. From the parametric investigation it is observed that for the increase in area ratio from1.25 to 2.0, static pressure recovery increases sharply but with the increase of angle of turn pressure recovery decreases steadily. The coefficient of total pressure loss almost remains constant with the change in area ratio and angle of turn for similar inlet conditions.

Keywords: C-shape diffuser, k-ε model, Fluent solver, Five-hole probe.

INTRODUCTION

Diffusers are used in many engineering application to decelerate the flow or to convert the dynamic pressure into static pressure. Depending on application, they have been designed in many different shapes and sizes. The C-shape diffuser is one of such design and is an essential component in many fluid handling systems. C-shape diffusers are an integral component of the gas turbine engines of high-speed aircraft. It facilitates effective operation of the combustor by reducing the total pressure loss. The performance characteristics of these diffusers depend on their geometry and the inlet conditions. Part turn or curved diffusers are used in wind tunnels, compressor crossover, air conditioning and ventilation ducting systems, plumes, draft tubes, etc.

The objective of the present study is to investigate the flow characteristics within a circular C-shaped diffuser. The research work on curved diffuser was initiated from the study of curved duct. The earliest experimental work on curved ducts has been reported by Williams *et al.* [1] in the beginning of the last century. It was reported that the location of the maximum axial velocity is shifted towards the outward wall of the curved pipe.

The effect of the centrifugal force on the pressure gradient was studied by Dean [2, 3]. He established a relation between viscous force, inertia force and the curvature by a non dimensional number known as Dean Number.

Experimental investigation on circular 90° and 180° turn curved ducts was carried out by Rowe *et al.* [4] and reported the generation of contra rotating vortices within the bends..

Enayet *et al.* [5] 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 the generation of secondary motion within the duct.

Kim and Patel [6] have investigated on a 90° curved duct of rectangular cross section. It was reported that the formation of vortices on the inner wall due to the pressure driven secondary motion originated in the corner region of curved duct.

The earliest work on curved diffuser was reported by Stanitz [7]. The diffuser was designed based on potential flow solution for two-dimensional, invisid, incompressible and irrotational flow. The evaluation of the performance of these diffusers was on the basis of wall static pressure only.

The first systematic studies on 2-D curved subsonic diffusers were carried out by Fox & Kline [8]. The centerline of the diffuser was taken as circular with a linearly varying area distribution normal to the centerline. They established a complete map of flow over a range of the L/D ratio and at different values of $\Delta\beta$.

Parson and Hill [9] investigated on three 2-D curved diffusers of $A_s = 10$ of various combination of ratio between the centerline length and inlet width. They observed that the streamline curvature affects the flow development substantially within the curved diffuser.

A qualitative measurement of the mean flow quantities in a 40° curved diffuser of rectangular cross section of $A_r = 1.32$ and inlet $A_s = 1.5$ have been reported by McMillan [10]. The result clearly showed the development of strong counter rotating vortices between two parallel walls, which dominate the flow and performance characteristics.

Majumdar *et al.* [11] experimentally studied the flow characteristics in a large area ratio curved diffuser with splitter vanes installed at different angles to the flow at the inlet of the diffuser. It was observed that splitter vanes deflect the flow towards the convex wall and a pair of contra rotating vertices generated at the flow passage.

Yaras [12] experimentally investigated the flow characteristics of 90° curved diffuser with strong curvature having $A_r = 3.42$ for different values of inlet boundary layer thickness and turbulence intensity. Measurements were taken by the help of seven-hole pressure probe. He observed that the performance parameters were almost independent by the variations in the inlet boundary layer.

Majumdar *et al.* [13] experimentally studied the turbulent characteristics in a curved diffuser. They observed that the stream wise bulk flow shifted towards the outward wall in the downstream of the diffuser, which was mainly due to the influence of centrifugal force. Moreover, one pair of contra-rotating vortices was identified at 30° turns in the flow passage. The overall static pressure recovery was observed as 51%.

Majumdar *et al.* [14] conducted an experiment on 180° bend rectangular diffusing duct. They measured the wall pressure, velocity and turbulence intensity along the flow passage of the diffusing duct. The observation clearly showed the formation of vortical motions between the two parallel walls. The overall pressure recovery was found about 48%

MATERIALS AND METHODOLOGY

A test rig for the present investigation has been constructed at Fluid Mechanics & Machinery Laboratory of Power Engineering Department Jadavpur University to investigate the flow characteristics within a circular cross sectioned C-shape diffuser. The geometry of the test diffuser is shown in Fig. 1. with co-ordinate system and measurement locations. The entire set up was fabricated from mild steel sheet except the test diffuser.

The test diffuser was designed with increase in area from inlet to exit and it distributed normal to the centerline as suggested by Fox and Kline [8]. The test diffuser was designed based on an area ratio of 1.273 and centerline length of 225 mm. The test diffuser is made of fiber glass reinforcement plastic. Centerline was turned at 37.5° from inlet to exit with inlet diameter of 78 mm.

In order to avoid the pressure losses and flow distortion at the inlet and exit, two constant area connectors were attached at the inlet and exit of the test diffuser. A pre-calibrated five-hole pressure probe was used to obtain detailed flow parameters like mean velocity and its components, total and static pressure and secondary motions along the entire length of the diffuser. Ambient air was used as working fluid.

For measuring mean velocity and its components and static and total pressure surveys along the entire cross section of curved diffuser, the test piece was divided into five planes, one at Section H, at Central horizontal plane, the Inlet section one diameter upstream of the test diffuser, two planes, Section A and Section B at 12.5° and 25° turn along the length of the diffusing passages and the fifth plane, Section C is at the mid point of the exit duct. The details of measured planes are shown in Fig. 1. For measurement of flow parameters the five hole pressure probe was inserted through a 8 mm drilled hole provided at four locations, namely , 0°, 45°, 90°, and 315° angle as shown detail in Fig.1.

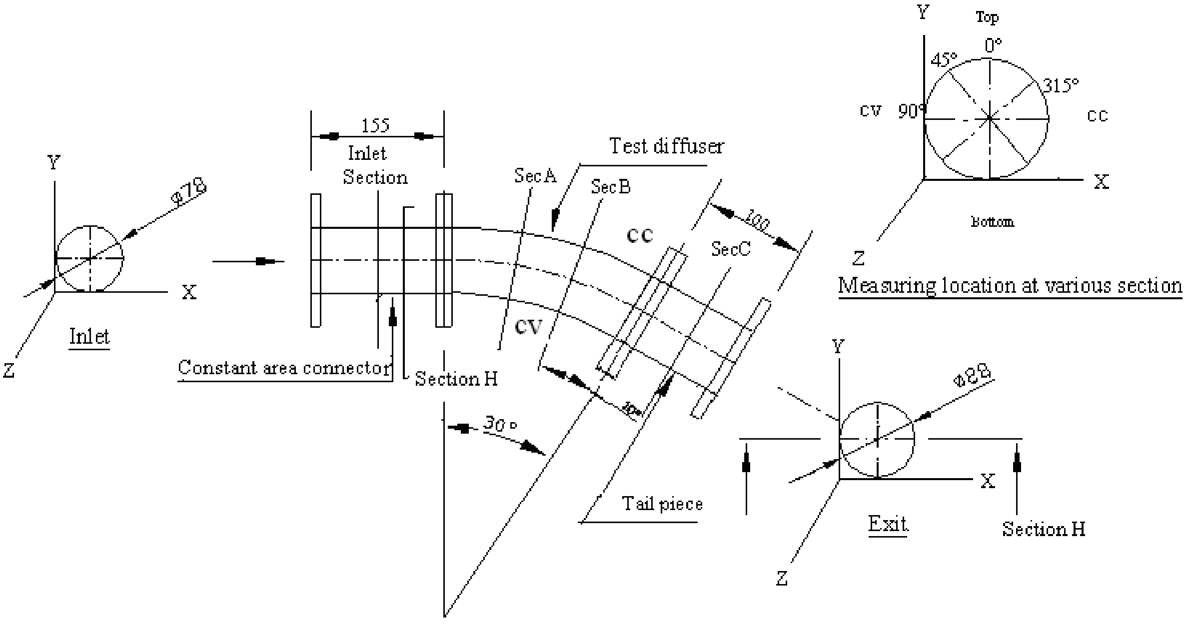


FIGURE 1 Geometry of test diffuser and measuring locations

The pre-calibrated five hole pressure probes was mounted in a traversing mechanism and the probe inserted into the flow field through 8 mm diameter drilled hole provided at the wall. The probe was placed within 1 mm of solid surface for the first reading. The probe was then moved radially and placed at the desired location as shown in Fig. 1.

Instrumentation for the present study was chosen such that the experimental errors are minimum and also to have quick response to the flow parameters.

The pre-calibrated hemispherical tip five-hole pressure probe used for the present study. The probe was calibrated and using non null technique was used to measure the flow parameter.

All the five sensing ports of the probe were connected to a variable inclined multi tube manometer. The readings were recorded with respect to atmospheric pressure. The mean velocity and components of mean velocity distribution have been drawn with the help of SURFER software

The assessment of errors resulting from the readings of the present five hole pressure probe was made as a function of all incidence angles for all flow characteristics in all the probe sectors and discussed in details[15], [16].

RESULTS AND DISCUSSIONS

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

MEAN VELOCITY CONTOUR

The normalized mean velocity distribution in the form of contour plots at various sections of the curved diffuser has been discussed here and shown in Fig.2.

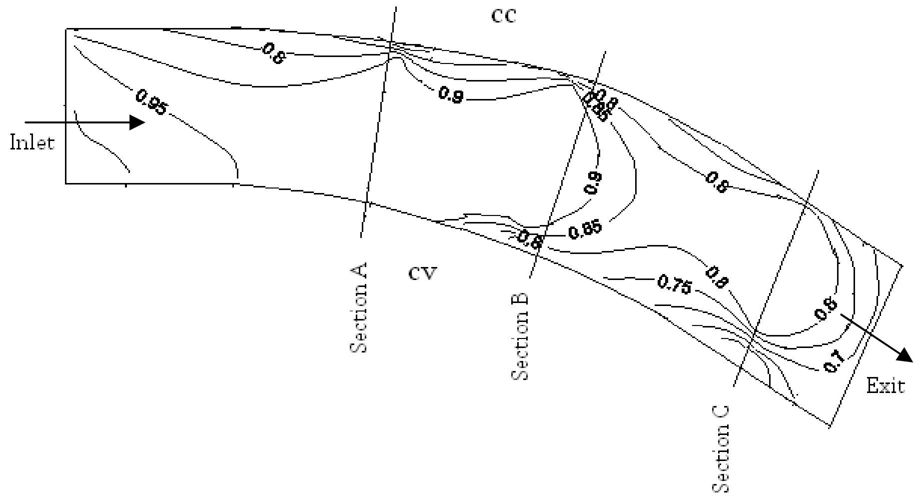
Mean velocity at Section H as shown in Fig.2(a) indicates that the high velocity fluid occupies the flow area close to convex wall (cv) as the flow proceeds from inlet to exit. Flow is also diffused in the downstream direction due to increase in cross-sectional area. Low velocity fluid increasingly accumulates close to the concave wall (cc) indicating a complex flow development dominated by combined effected of flow diffusion and centrifugal force. However, for a better understanding, this flow development can be observed through contour plots at Inlet Section, Section A, Section B and Section C as shown in Fig.2(b), (c), (d), (e).

The mean velocity contour at Inlet Section is shown in Fig.2(b) and it indicates that the flow is nearly symmetrical in nature throughout the entire cross-sectional area.

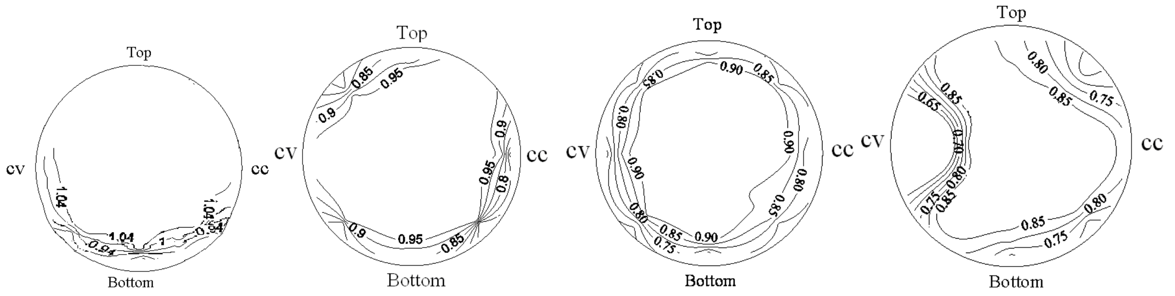
The mean velocity contour of Section A, shown in Fig.2(c), indicates the overall diffusion of velocity particularly near the concave wall of this section and also reveals the shifting of the high velocity fluid close to the convex wall. These are mainly due to combined effect of the inertia force and centrifugal action on the flow.

The mean velocity distribution in Section B is shown in Fig.2(d). The figure shows that the overall diffusion takes place at this section. The flow pattern at this section has changed compared to the preceding section. An appreciable diffusion across the entire section is observed from this figure. It is also observed that the high velocity core has occupied major part of the central area of the cross section and this is mainly due to the tendency of the flow to move towards concave wall due to inertia. As a result, low velocity fluid is pushed towards the convex wall indicating the accumulation of low momentum fluid at this section.

The mean velocity distribution of Section C is shown in Fig.2(e). The figure depicts that the high velocity core is shifted towards concave wall and it occupies a substantial portion of the cross sectional area. The low velocity fluid close to the convex wall, as compared with the observation at Section B, occupies more area at Section C. Comparing the velocity distribution lines about the mid horizontal plane, it can be observed that the flow is symmetrical between top and bottom surfaces; it indicates the development of similar types of flow about the vertical plane. In a closed conduit, this phenomenon is only possible if the directions of flow in the two halves are opposite in nature (counter rotating flows), which is a natural phenomenon of flow through curved duct of any cross-section. Over an above it, increase in the cross-sectional area has further complicated the flow development.



(a) Section H



(b) Inlet section (c) Section A (d) Section B (e) Section C

FIGURE 2 Mean velocity contour

PRESSURE RECOVERY & LOSS COEFFICIENT

The variation of normalized coefficients of mass averaged static pressure recovery and total pressure loss based on the average static and total pressures at different sections of C-shape diffuser are shown in Fig.3. The figure shows that the coefficient of pressure recovery increases continuously up to the Section A, subsequently the increase takes place with a lesser gradient up to the Section B, and then again increases in a steeper gradient. The overall coefficient of mass averaged static pressure recovery is nearly 35% for this test diffuser.

The coefficient of mass averaged total pressure loss increases rapidly in the curved diffuser up to the Section A. Then it increases steadily and marginally up to Section C. The overall mean value of the coefficient of mass averaged total pressure loss is nearly 14% for this test diffuser.

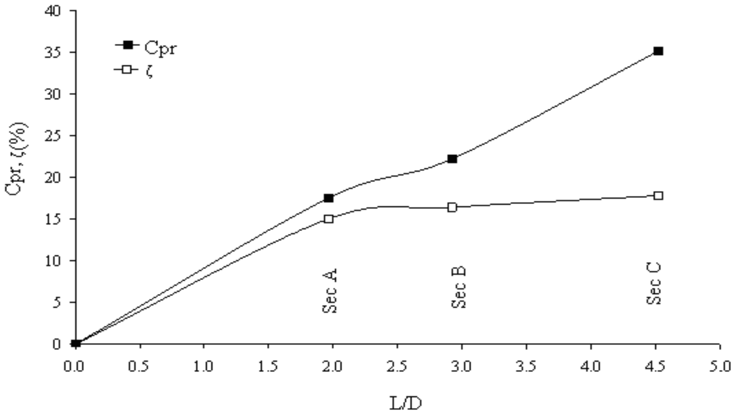


FIGURE 3 Variation of mass average pressure recovery and loss coefficients

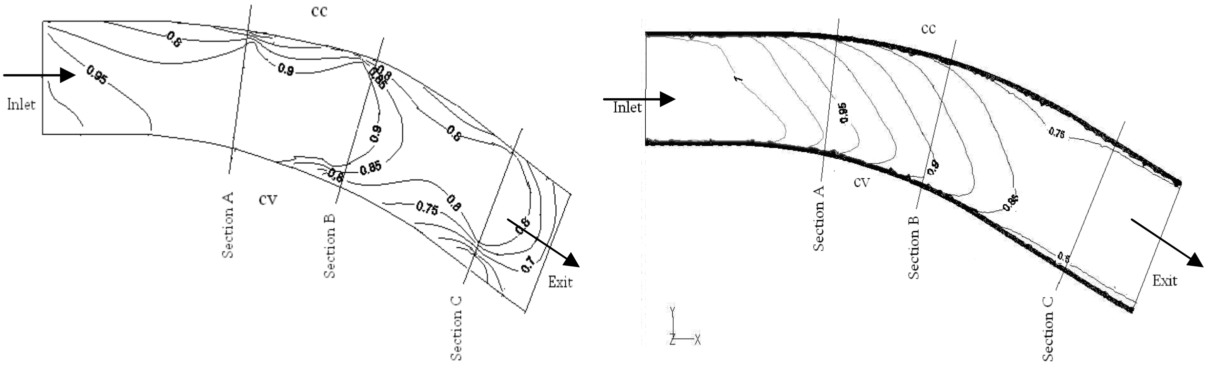
NUMERICAL VALIDATION

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), Fig. 4(b), Fig. 4(c) and Fig. 4(d) respectively.

All four 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 convex side of 10° of Section H and at the 45° plane of Section A close to the top surface.

This could be due to the complicated nature of flow at those planes, which was not properly predicted by the process of computer simulation.

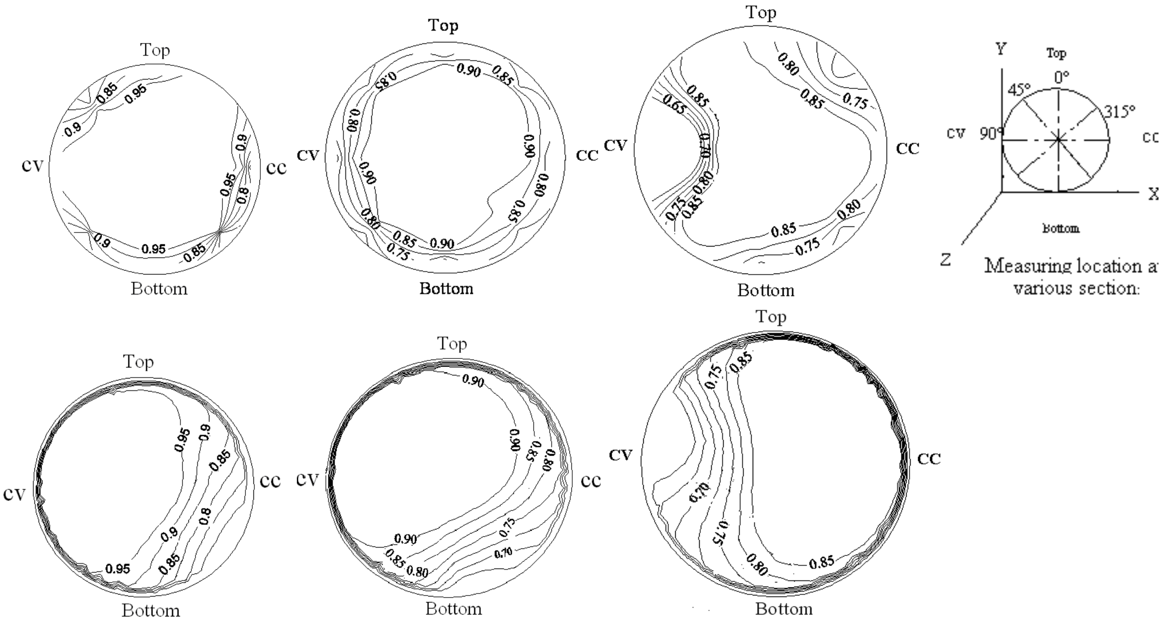
The mean velocity distribution at the Section B and Section C are shown in Fig. 4(c) and Fig. 4(d) show a reasonably good agreement of the computational investigation with the experimental results. These agreements confirm that the CFD code using Standard k-ε model can predict the flow and performance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



Experimental Computational

(a) Section H

Experimental



(b) Section A (c) Section B (d) Section C

Computational

FIGURE 4 Comparison of normalized velocity distribution at Section H, Section A, Section B and Section C obtained through Computational and Experimental investigation.

Figure 5 shows the comparison of performance parameters like coefficient of static pressure recovery and coefficient of total pressure loss obtained through experimental and computational investigation. From the figure it has been observed that coefficient of pressure recovery C_{pr} for the computational investigation was obtained as 38% compared to the experimental investigation, which obtained as 35%. Similarly the coefficient of pressure loss is obtained as 13% in computation investigation compared to the 14% of experimental study. This shows very good matching of the predicted results with the experimental one.

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

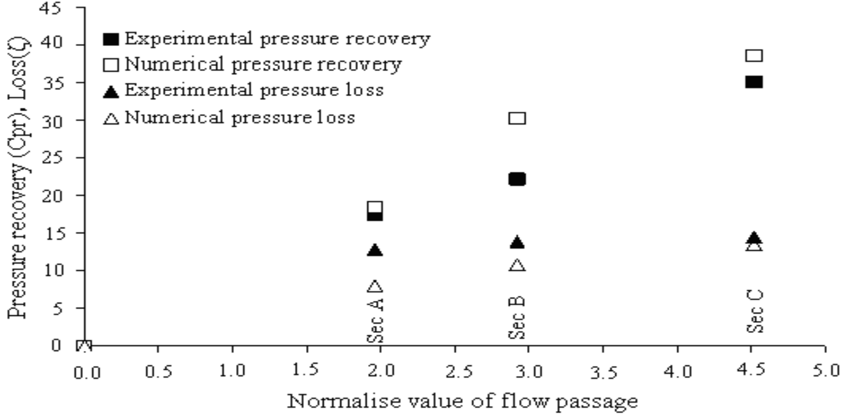


FIGURE 5 Comparison of performance parameters obtained through computational and experimental investigation

PARAMETRIC INVESTIGATION

To obtain a more insight of the performance parameters an intense parametric study of pressure recovery/loss coefficient for different area ratio with angle turn diffusers were carried out.

For this purpose area ratios 1.25, 1.5, 1.75 and 2 with the angle of turns 30º, 40º, 50º, 60º, 70º and 75º C-shape diffusers have chosen.

From this investigation it is observed from Fig.6 that for the increase in area ratio static pressure recovery increases sharply upto area ratio 2 but with the increase of angle of turn pressure recovery decreases steadily except for area ratio 1.25 and 1.5 where for increase of angle of turn from 30º to 40º pressure recovery increase sharply then for further increase of angle of turn it decreases steadily.

The coefficient of total pressure loss almost remains constant with the change in area ratio and angle of turn for similar inlet conditions.

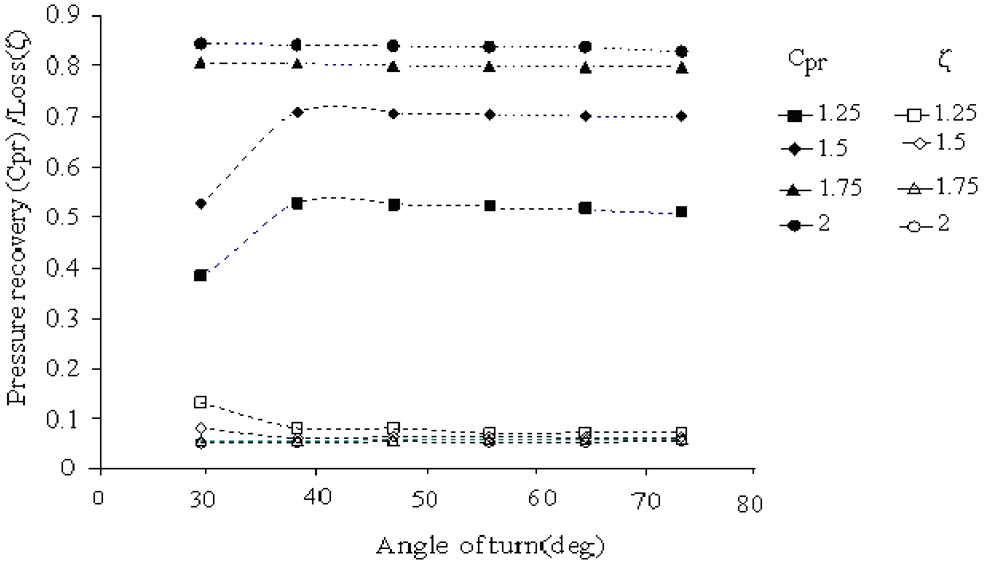


FIGURE 6 Variation of mass average pressure recovery and loss coefficients.

CONCLUSION

Based on the present investigation following conclusion have drawn for the present paper.

i. High velocity fluids shifted and accumulated at the concave wall of the exit section.

ii. The mass average static pressure recovery and total pressure loss for the curved test diffuser is continuous from Inlet section to Section C.

iii. Performance parameter like coefficient of mass average static pressure recovery and coefficient of mass average total pressure loss are 35% and 13% respectively.

iv. A comparison between the experimental and predicated results for the annular curved diffuser show good qualitative agreement between the two.

v. The coefficient of mass averaged static pressure recovery and total pressure loss are obtained as 38% and 14% in predicted results and in the experimental results their values obtained as 35% and 13% respectively, which indicate a good matching between the experimental and predicted results

vi. From the parametric investigation it is observed that for the increase in area ratio upto 2.0, static pressure recovery increases sharply But with the increase of angle of turn pressure recovery decreases steadily except for area ratio 1.25 and 1.5 where for increase of angle of turn 30º to 40º pressure recovery increase sharply then for further increase of angle of turn it decreases steadily.

vii. 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.

REFERENCE

[1] Williams, G. S., Hubbell, C.W. and Fenkell, G.H., 1902, “Experiments at Detroit, Mich, on the effect of curvature upon the flow of water pipes”, Trans. ASCE, Vol.47, pp.1-196.

[2] Dean, W. R., 1927, “Note on the Motion of Fluid of Curved Pipe”, Philosophical Magazine, Vol.20, pp.208-223.

[3] Dean, W. R., 1928, “The Streamline Motion of Fluid in a Curved Pipe”, Philosophical Magazine, Vol.30, pp.673-693.

[4] Rowe, M., 1970, “Measurement and computations of flow in pipe bends”, Journal of Fluid Mechanics, Vol.43, pp.771-783.

[5] Enayet, M. M., Gibson, M.M., Taylor, A.M.K.P. and Yianneskis, M., 1982, “Laser Doppler measurements of Laminar and Turbulent Flow in a Bend”, Int. Journal of Heat and Fluid Flow, Vol. 3, pp.211-217.

[6] Kim, W.J. and Patel, V.C., 1994, “Origin and Decay of Longitudinal Vortices in the Development of flow in a Curved Rectangular Duct”, Trans ASME, Journal of Fluid Engineering, Vol.116, pp.45-52.

[7] Stanitz, J.D., 1953, “Design of Two Dimensional Channels with Prescribed Velocity Distribution along the Channel Walls”, NACA Report No. 115,

[8] Fox, R.W. and Kline, S.J., 1962, “Flow Regimes in Curves Subsonic Diffuser”, Trans ASME, Journal of Basic Engineering, Vol. 84, pp. 303 – 316.

[9] Parson D.J and Hill P.G1973, “Effect of Curvature on Two Dimensional Diffusing flow”, Trans. ASME, Journal of fluid engineering, Vol. 95. pp. 1–12.

[10] Mcmillan, O. J., 1982, “Mean Flow Measurements of the Flow Diffusing Bend”, NASA Contractor Report 3634,

[11] Majumdar, B., Singh, S.N. and Agrawal, D.P., “Flow characteristics in a large area curved diffuser”, Proc. I Mech E, Journal of Aerospace Engineering, Vol. 210, 1996, pp. 65 – 75.

[12] Yaras, M .I., 1996, “Effects of Inlets Conditions in the Flow in a Fishtail Diffuser with Strong Curvature”, Trans. ASME, Journal of Fluid Engineering, Vol. 118, pp 772 – 778.

[13] Majumdar, B., Mohan, R., Singh, S.N. and Agrawal, D.P., 1998, “Experimental study of Flow in a high Aspect ratio 90° Curved Diffuser”, Trans. ASME, Journal of Fluids Engineering, Vol. 120, pp. 83 – 89.

[14] Majumdar, B., Singh, S. N. and Agrawal, D.P., 2007, “Flow Structure in a 180° Curved Diffusing Ducts”, The Arabian journal of science and technology, Vol. 24, 1999, pp. 79 –87.

[15] Chowdhoury, D., 2007, “Modelling and calibration of pressure probes”, M.E Thesis, Jadavpur University.

[16] Mukhopadhyay, S., Dutta, A., Mullick, A. N., and Majumdar, B., 2001, "Effect of Five-hole probe tip shape on its calibration", Journal of the Aeronautical Society of India, Vol. 53,No.4, pp. 271-277.

NOMENCLATURE

A_r	Area ratio	D	Inlet diameter of the Diffuser
A_s	Aspect ratio	L	Centerline length of the Diffuser
CC	Concave or outward wall	Re	Reynolds number
C_{PR}	Coefficient of pressure recovery	$\Delta\beta$	Angle of turn of the center line
CV	Convex or inward wall		