Computational Investigation of Performance Characteristics through Annular Diffuser

Prasanta K. Sinha*, Bireswar Majumdar**

*Department of Mechanical Engineering, DIATM, Durgapur-71321, WB, India.
**Department of Power Engineering, Jadavpur University, Kolkata, WB, India

ABSTRACT

In the present investigation the distribution of mean velocity, static pressure and total pressure are experimentally studied on an annular curved diffuser of 36° angle of turn with an area ratio of 1.284 at Reynolds number 2.15 x 10^5 based on inlet diameter and mass average inlet velocity. 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 2.0 with change in angle of turn from 30º to 75º.The velocity distribution shows high velocity fluids shifted and accumulated at the concave wall of the exit section due to the combined effect of velocity diffusion and centrifugal action. It also indicates the possible development of secondary motions between the concave and convex walls of the test diffuser. The maximum values of the mass average static pressure recovery and total pressure loss are 33% and 18% compared to the predicted results of 37% and 21% respectively, which shows a good agreement between the experimental and predicted results. From the parametric investigation it is observed that static pressure recovery increases up to an area ratio of 2 and pressure recovery decreases steadily up to angle of turn 75º. The coefficient of total pressure loss almost remains constant with the change in area ratio and angle of turn for similar inlet conditions

Keywords: *Annular curved diffuser Fluent solver, k-ε model,*

NOMENCLATURE

A_r	Area ratio
A_s	Aspect ratio
CC	Concave or inward wall
Cpr	Coefficient of pressure recovery
CV	Convex or outward wall
D	Inlet diameter of the Diffuser
L	Centerline length of the Diffuser
P	Pressure sensed by five hole probe
Re	Reynolds number
U	Velocity of air
ρ	Density of air
Δ β	Angle of turn of the center line
ρ_m	Density of manometric fluid
ζ	Coefficient of pressure loss

Subscript

av	Average
x,y,z	Directions along the 3D Cartesian
e	Exit
S	Static
T	Total

1. 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 annular curved diffuser is one of such design and is an essential component in many fluid handling systems. Annular 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 use of such diffusers mainly depends either on the specific design of the machine or on the space limitation where compactness is desired or both.

The objective of the present study is to investigate the flow characteristics within a circular cross sectioned annular curved diffuser. The performance of an annular curved diffuser is characterizes by static pressure recovery and total pressure loss coefficient and are defined as

$$Cpr = \frac{(P_{Se} - P_{Si})}{0.5\rho U^2 avi} \quad \text{.... (1)}$$

$$\zeta = \frac{(P_{Ti} - P_{Te})}{0.5\rho U^2 avi} \quad \text{.... (2)}$$

An exhaustive survey of the reported literatures on various types of diffusers
reveals that the studies on straight diffusers are available in considerable numbers in open literatures. Literatures for curved diffusers, specially, part turn diffusers are less in number and detailed flow measurements methodologies for these diffusers are very limited in the open literatures.

The earliest work on curved diffuser was reported by Stanitz [1]. He designed the diffuser based on potential flow solution for two-dimensional, inviscid, incompressible and irrotational flow.

The first systematic studies on 2-D curved subsonic diffusers were carried out by Fox & Kline [2]. 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 and at different values of Δβ.

A qualitative measurement of the mean flow quantities in a 40° curved diffuser of rectangular cross section with A_r =1.32 and inlet A_s =1.5 have been reported by McMillan [3]. The result clearly showed the development of strong counter rotating vortices between two parallel walls, which dominate the flow and performance characteristics.

Seddon [4] has made extensive experimental investigations to explain the self-generated swirl within the S-shaped diffuser of rectangular to circular cross-section having Ar =1.338. He attempted to improve the performance of S-shaped diffusing ducts by introducing fences of 10 different configurations within the first bend of the diffuser and observed a significant improvement in the performance and exit flow distribution.

The relation between the shape of curvature and cross sectional area can be understood, if their effects are considered separately. In the early parts of 1980's researchers started working on how to improve the performance by introducing vortex generator, fences etc. within the diffusers to change the magnitude and direction of the generated secondary motion.

Vakili *et al.* [5] reported the experimental studies in an S-shaped diffusing duct of Δβ = 30°/30° having circular cross-section and A_r =1.5. They observed that there is a significant improvement in the exit flow distribution and pressure recovery by introducing vortex generator at the inlet.

Yaras [6] 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.

Reichert and Wendt [7] experimentally studied the effect of vortex on the flow field of a diffusing S-duct with Δ β = 30° /30° and A_r =1.5. The objective was to reduce flow distortion and improve total pressure recovery within the diffuser by using the vortex generation at the inlet. They concluded that the mechanism responsible for improved aerodynamic performance is not boundary layer re-energization from shed axial vortices but rather the suppression of detrimental secondary flows by redirecting the flow.

Majumder *et al.* [8] also studied the performance characteristics of a 90° / 90° S-shaped diffuser of rectangular cross section with A_r = 2.0 and inlet A_s= 6.0 by using three-hole pressure probe. They observed a detached flow at the inflexion point and overall pressure recovery in comparison to a straight diffuser is low. Sonoda *et al*. [9] studied the flow characteristics within an annular S-shaped duct. The effect on the flow of a downstream passage is also carried out. They observed that the total pressure loss near the hub is larger due to the instability of the flow, as compared with that near the casing, in the case of curved annular downstream passage. The total pressure loss near the hub is greatly increased compared with the straight annular passage.

Mullick and Majumdar [10] studied the performance of a fully developed subsonic turbulent flow in 22.5° / 22.5° circular cross section S-shaped diffusing duct. The experiment was carried out with Re = 8.4 x $l0^5$ and measurements were taken with the help of a pre-calibrated five-hole pressure probe. The experimental results indicated the generation of secondary flow in the form of a pair of contra rotating vortices in the first half, which changes its senses of rotation in the second half and overall static pressure recovery was 40%.

Numerical Simulation of flow development through turbine diffusers was reported by Dominy *et al*. [11] they performed the experiment in an S-shaped annular duct of inlet hub and core diameters arc 0.286m and 0.421m respectively and A_r =1.5 with inlet Re = 3.9 x 10^5. They used 34 numbers inlet swirl vanes. The results show that the influence of wakes and swirl upon the flow has a significant effect upon the development of flow. The numerical simulations also give a good matching of the flow development within this diffuser.

Fuji *et al* [12] studied the Curved Diffusing Annulus Turbulent Boundary layer Development and concluded a simplified but reliable method for boundary layer calculation with acceptable assumptions for special flow situations.

A numerical and experimental investigation of turbulent flows occurring in a 180° bend annular diffuser with an aperture in front of the bend was reported by Xia *et al.* [13] .They observed that the pressure recovery coefficient increases with increasing blow of mass flow rate and inlet pressure but remains nearly constant if the inlet pressure is higher than about 10 bars. The numerical prediction is compared with the experimental data and excellent agreement is achieved.

Sing *et al*. [14] conducted investigation to select the range of the inlet swirl intensity for the best performance of annular diffusers with different geometries but having the same equivalent cone angle. This is analyzed on the basis of the static pressure recovery and total pressure loss coefficients. The results show that the parallel diverging hub and casing annular diffuser produces the best performance at high-swirl intensities

Sinha *et al*. [15] 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. They also observed that with the increase in area ratio pressure recovery increases upto certain point than with further increase in area ratio Pressure recovery decreases.

Sinha *et al*. [16] investigated an experiment on 37.5° annular diffusing duct. 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 varying area ratio and different angle of turn. They observed that the high velocity fluids shifted and accumulated at the concave wall of the exit section. They also observed that with the increase in area ratio pressure recovery increases but with the increase in angle of turn Pressure recovery decreases.

2. METERIAL 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 annular curved 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 [2]. The test diffuser was designed based on an area ratio of 1.284 and centerline length of 240 mm. The test diffuser is made of fiber glass reinforcement plastic. Centerline was turned at 36° 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° and 24° 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 and Fig.2. For measurement of flow parameters the five hole pressure probe was inserted through a 8 mm drilled hole provided at eight locations, namely , 0°, 45°, 90°, 135°, 180°, 225°, 270° and 315° angle as shown detail in Fig.1.

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.

Resolving the velocity vector in the mutually perpendicular direction can be determined using the following equation.

$$U_{av} = \sqrt{\frac{2}{\rho}(P_T - P_S)}$$

$$U_{av} = \sqrt{\frac{2\rho_m gh}{\rho_{air}}} \qquad \text{...... (3)}$$

$$U_x = U_{av} \cos\alpha \cos\beta$$
$$U_y = U_{av} \sin\alpha \cos\beta \qquad \ldots\ldots (4)$$
$$U_z = U_{av} \sin\beta$$

The mean velocity and components of mean velocity distribution have been drawn with the help of SURFER software

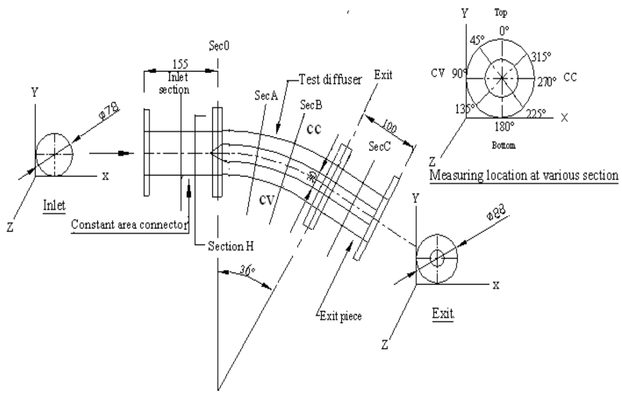


Fig.1 Geometry of test diffuser and measuring locations

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[16], [17].

3. RESULTS AND DISCUSSION

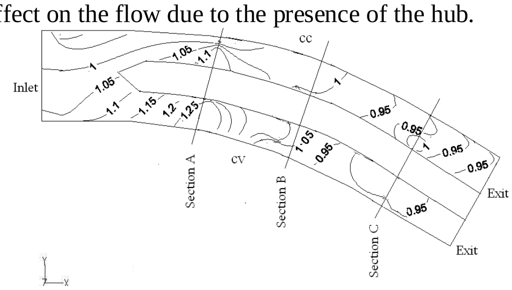
The flow characteristics have been evaluated by mass average mean velocity, between the curved walls, total pressure and static pressure of the flow at various cross sections. Measured flow quantities have been presented in the form of 2-D profiles. All the velocities and pressures were normalized with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

3.1. MEAN VELOCITY DISTRIBUTION

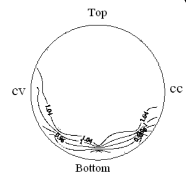
The normalized mean velocity distribution in the form of contour plots at various sections of the curved diffuser has been discussed here and is shown in Fig.2.

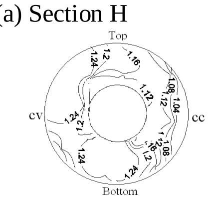
Mean velocity at Section H as shown in Fig.2(a) indicates that the high velocity fluid occupies the flow area close to convex wall (cv) as the flow proceeds from inlet to exit. Flow is also diffused in the downstream direction due to increase in cross-sectional area. Low velocity fluid accumulates more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

Fig.2(b) indicates that the flow is symmetrical in nature throughout the entire cross- sectional area. The high velocity fluid occupies most of the cross-section except close to the bottom surface indicating no upstream effect on the flow due to the presence of the hub.

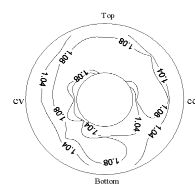


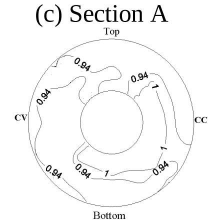
(a) Section H





(b) Inlet section (c) Section A





(d) Section B (e) Section C

Fig.2. Mean velocity Contour

Fig.2(c), of Section A of the annular curved diffuser depicts that the flow is flow is better distributed though the high velocity core is located along the top portion of the plane connecting 45º-225º planes. Marginal variation of flow velocity is observed in the lower portion between 45º-225º planes. Further Fig.2.(c) indicates the overall acceleration of flow compared to the inlet as the effective flow of velocity is reduced due to the insertion of the hub. The mean velocity distribution in Section B is shown in Fig2.(d). This figure shows that the overall diffusion takes place at this section compared to the previous section. It is also observed that the more or less uniform flow occupies most of the cross-sectional area. The curvature effect has more reduced due to the presence of the hub, which restricts the movement of the bulk of the flow to one side of the diffuser. This phenomenon indicates the restriction of the development of the counter rotating flows between the top and bottom surface mainly due to presence of the hub and hence a better uniform flow is seen. However along the central plane of the Section B diffusion is not much and flow is more or less symmetrical in nature. However, the contour line at the bottom surface indicates the generation of secondary motion, which is limited to this only.

The velocity distribution at Section C as shown in Fig.2(e) clearly indicates the further diffusion of flow along the centerline of the flow passage due to increase in cross sectional area. The figure also depicts that the high velocity core is shifted a little towards concave wall though the diffused uniform flow has occupied the more or less the whole cross sectional area. The secondary motion, which observed at previous sections, is not clearly seen at this section indicating a better flow at the exit of the annular curved diffuser.

3.2. PRESSURE RECOVERY & LOSS COEFFICIENT

The variation of normalized mass averaged static pressure recovery and total pressure loss coefficients based on the mass average pressure at different sections of the test diffuser are shown in Fig.3. It was calculated based on the static pressure difference between the succeeding and preceding sections. The figure shows that the coefficient of pressure recovery decreases steadily up to the Section A in annular curved diffuser and then the increase takes place in rapidly up to the Section C. This is mainly due to the formation of secondary motions The overall mass average pressure recovery coefficient is nearly 33% for this diffuser.

The mass average total pressure loss coefficient increases in steadily in the test diffuser up to the Section A. Then the loss coefficients in curved diffuser increases sharply up to Section C. The overall mean value in mass average pressure loss coefficient is nearly 17 % for this test diffuser.

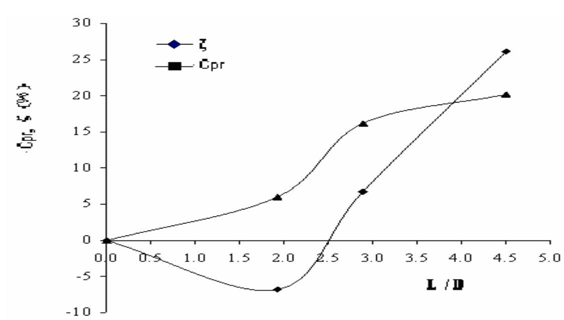
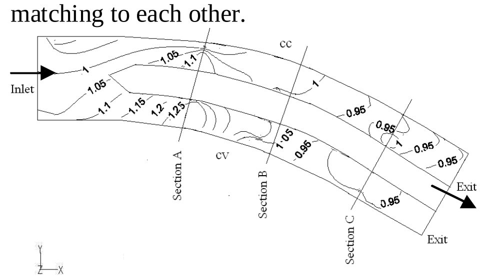


Fig. 3. Variation of mass average pressure recovery and loss coefficients

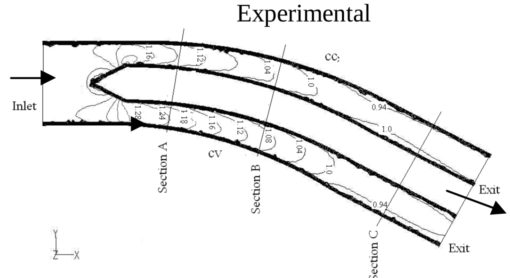
3.3. NUMERICAL VALIDATION

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 and Fig 5 (a), Fig 5 (b) and Fig 5 (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.



Experimental



Computational

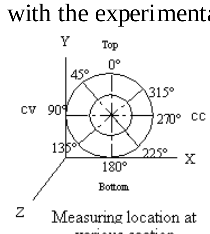
(a) Section H

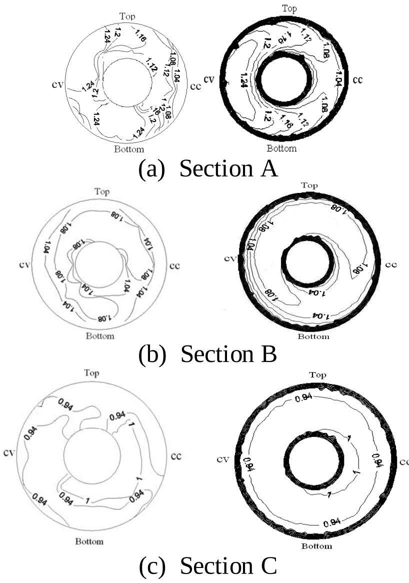
Fig. 4. Comparison of normalized velocity distribution at Section H obtained through Computational and Experimental investigation

However a slight mismatch can be observed at the 135º and 180º plane of Section A close to the bottom and concave 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 5 (b) and Fig 5(c) show a reasonably good agreement of the computational investigation with the experimental results.

Fig.6 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 37% compared to the experimental investigation, which obtained as 33%. Similarly the coefficient of pressure loss is obtained as 21% in computation investigation compared to the 18% of experimental study. This shows very good matching of the predicted results with the experimental one





Experimental Computational

Fig. 5. Comparison of normalized velocity distribution at Section A, Section B and Section C

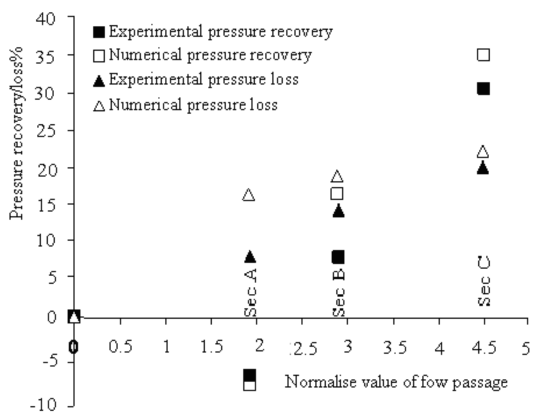


Fig.6. Comparison of performance parameters obtained through computational and experimental investigation

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

3.4. PARAMETRIC INVESTIGATION

To obtain a more insight of the performance parameters an intense parametric study of pressure recovery/loss coefficient for different area ratio diffusers with the angle turn 30º, 40º, 50º, 60º, 70º and 75º.. For this purpose area ratios 1.25, 1.5, 1.75, and 2, with the angle of turn 30º, 40º, 50º, 60º, 70º and 75º annular curved diffusers have chosen. From this investigation it is observed from Fig. 7 that for the increase in area ratio from1.25 to 2, static pressure recovery increases sharply and it was maximum at area ratio 2 with angle turn 30º. But with increase in angle turn pressure recovery decreases steadily except in area ratio 1.25 where with the increase in angle of turn from 30º to 40º pressure recovery increases then with the further increase of angle of turn it decreases steadily. The coefficient of total pressure loss almost remains constant with the change in area ratio for similar inlet conditions

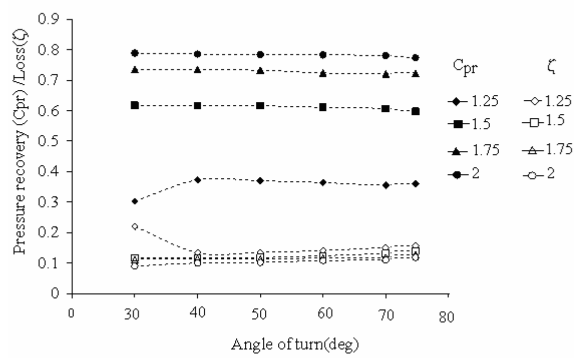


Fig.7. Variation of mass average pressure recovery and loss coefficients.

CONCLUSION

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

High velocity fluids shifted and accumulated at the concave wall of the exit section. The mass average static pressure recovery and total pressure loss for the curved test diffuser is continuous from Section A to Section C. Performance parameter like coefficient of mass average static pressure recovery and coefficient of mass average total pressure loss are 33% and 18% respectively.

A comparison between the experimental and predicated results for the annular curved diffuser show good qualitative agreement between the two. The coefficient of mass averaged static pressure recovery and total pressure loss are obtained as 37% and 21% in predicted results and in the experimental results their values obtained as 33% and 18% respectively, which indicate a good matching between the experimental and predicted results From the parametric investigation it is observed that static pressure recovery increases sharply up to an area ratio of 2 and but it is decreases steadily with the increase of angle of turn up to 75º. The coefficient of total pressure loss almost remains constant with the change in area ratio and angle of turn for similar inlet conditions. Among the different turbulence models within the fluent solver a standard k-ε model shows the best results and predicts the flow and performance characteristics well for annular curved diffusing ducts with uniform flow at inlet.

REFERENCES

1. J.D. Stanitz, *Design of Two Dimensional Channels with Prescribed Velocity Distribution along the Channel Walls*.1953. NACA Report No. 115,
2. R.W. Fox, and S.J. Kline, Flow Regimes in Curves Subsonic Diffuser. *Trans ASME, Journal of Basic Engineering, Vol. 84,* 1962, pp. 303 – 316.
3. O. J. Mcmillan, *Mean Flow Measurements of the Flow Diffusing Bend*.1982.. NASA Contractor Report 3634,

4 J. Seddon, Understanding and Countering the Swirl in S- ducts; Test on the Sensitivity of swirl to Fences. *Aeronautical journal*, 1984, pp.117-127

5. A .D. Vakili, J.M. Wu, P. Liver and M.K. Bhat. Experimental Investigation of Secondary Flow in a Diffusing S- Duct.. University of Tennessee Space Inst, Report *No. TRUTSI 86/14*, University of Tennessee, Tullahoma, TN. 1984
6. Yaras, M .I. Effects of inlets conditions n the flow in a Fishtail Diffuser with strong Curvature. 1996 *Trans ASME, Journal of Fluid Engineering, Vol. 118*, pp 772 – 778.
7. B.A. Reichert and B.J Wendt,. *Improving Curved Subsonic Diffuser Performance with Vortex Generator*.1996. AIAA Journal, *Vol.14,* pp. 65 – 72.
8. B. Majumdar, S.N. Singh and D.P Agrawal. Flow Characteristics in S-shaped Diffusing Duct. *In Journal of Turbo and Heat Engines, Vol. 14,* 1997 pp 45 – 57.
9. T. Sonada, T. Arima, and M. Oana, *The influence of Downstream Passage on the Flow within Annular S-shaped Ducts*. 1998. Trans ASME, Journal of Turbo Machinery, *Vol. 120*, pp. 714 – 722.
10. A. N Mullick and B. Majumdar, Experimental Investigation of flow Parameters in a Mild Curves S-shaped Diffusing Duct. *Journal of Aerospace Science and Technologies, Vol. 58, No. 1,* 2006, pp. 22 – 30.
11. Dominy et al. *Flow Development through Inter turbine Diffuser*. 1998. Trans ASME, Vol. 120, pp. 298 – 304.
12. S. Fuji and T. H. Okiishi, *Curved Diffusing Annulus TurbulentBoundaryLayerDevelopment*.1997..Autho rs'Report. No.ISU-ERI-Ames-71033, National Technical Information Service, Springfield, Va., 22151.as N72-10239,
13. J.L. Xia,, B.l. Smith, T. Zierer, J. Schmidli, and G. Yadigaroglu, Study of Turbulent flow Characteristics in a 180° bend Annular Diffuser with Blow off.. *Int comm. Heat Mass Transfer*, *Vol. 26*, 1999, pp. 685 – 694.
14. S. N. Singh, V. Seshadri, K. Saha, K.K Vempati and S. Bharani, Effect of inlet swirl on the performance of annular diffusers having the same equivalent cone angle. *Proc. Inst.Mech.Enginner, Part G, Journal of Aerospace Engineering Vol. 220, No.2*, 2006, pp.129-143
15. P. K. Sinha, A.N. Mullick, B. Halder and B. Majumdar, A Parametric Investigation of Flow through an Annular Curved Diffuser, *Journal of International Review of Aerospace Engineering, Vol. 3. No. 5*, 2010, pp. 249-256.

[16. P. K. Sinha, A. K. Das and B. Majumdar, Numerical Investigation of flow through Annular Curved Diffusing duct, *International Journal of Engineering & Technology IJET-IJENS, Vol..03, No.03*, 2011, pp.196-196.

17. D. Chowdhoury, *Modelling and calibration of pressure probes*. M.E Thesis, Jadavpur University, 2007.
18. S. Mukhopadhyay, A. Dutta, A. N. Mullick, and B. Majumdar, Effect of Five-hole probe tip shape on its calibration. *Journal of the Aeronautical Society of India, Vol. 53, No.4*, 2001. pp 271 – 279.