RESEARCH ARTICLE OPEN ACCESS

A Numerical study of Flow through Sigmoid Duct

Prasanta K.Sinha[1], S. Mukhopadhya[2], B.Majumdar[3]
[1]Durgapur Institute of Advanced Technology& Management, West Bengal, India
[2]Department of Power Management Institute, National thermal power Corporation, India
[3]Jadavpur University, Kolkata, West Bengal, India

Abstract
Curved diffusers are an integral component of the gas turbine engines of high-speed aircraft. These facilitate 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. In the present investigation the distribution of mean velocity, static pressure and total pressure are experimentally studied on a S-shape Diffusing Duct of 45°/45° angle of turn with an area ratio of 1.65 aspect ratio 3.95 keeping inlet width 55 mm with centre line length 460 mm. The experimental results then were numerically validated with the help of Fluent. The velocity distribution shows that generation of secondary motion in the form of counter rotating vortices within the 1st half of the diffuser. The secondary motion changes their sense of rotation after the inflexion plane of the test diffuser**.** The maximum values of the mass average static Pressure recovery and total pressure loss are 36% and 13% compared to the predicted results of 39% and 11% respectively, which shows a good agreement between the experimental and predicted results.
Keywords: S-shape diffuser, k-ε model, Fluent solver, Five-hole probe

Nomenclature

Ar	Area ratio
As	Aspect ratio
CC	Concave or inward wall
Cpr	Coefficient of pressure recovery
CV	Convex or outward wall
W1	Inlet width of the Diffuser
W1	Inlet width of the Diffuser
L	Centerline length of the diffuser
R_e	Reynold's number
U	Velocity of air
ρ	Mass density of fluid
Δβ	Angle of turn of the curvature
ν	Kinematic Viscosity
μ	Dynamic Viscosity
ρ	Mass density of fluid
ζ	Coefficient of pressure loss

I. 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 sigmoid diffuser is one of such design and is an essential component in many fluid handling systems. sigmoid 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. sigmoid 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 rectangular cross sectioned sigmoid diffuser.

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, sigmoid 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 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. Benin et al.[16], 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 [17] 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.* [18] conducted an experiment on 40° curved diffuser. They measured the mean velocity, static pressure and total pressure along the flow passage of the diffuser. 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. Biswas *et al.* [19] investigated experimentally a 45°/45° constant area S-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 in the first half and inner wall to the outer wall in the second half along the flow passage of curved duct is very instinct. Sinha *et al.* [20] investigated experimentally a 22.5°/22.5° sigmoid diffusing S-duct. They measured the mean velocity and turbulent intensity along the flow passage of the duct. They observed that the velocity distribution shows that generation of secondary motion in the form of counter rotating vortices within the 1st half of the diffuser. The secondary motion changes their sence of rotation after the inflexion plane of the test diffuse*r*. Elsafty and Elazm [21] 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. Ahmed *et al.* [22] 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 diffusers it is apparent that the studies are generally related to straight or curved diffusers with circular cross-section. The present experimental investigation aims for a systematic study on the flow pattern of a sigmoid diffuser through the measurements of mean velocity and Turbulence intensity distribution, under the influence of different geometrical and flow parameters.

II. Material 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 rectangular cross sectioned sigmoid 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 1.65 and aspect ratio 3.95 with the centerline length of .460m. To develop the test diffuser geometry, a straight diffuser of inlet width 0.06m, center line length 0.46m and area ratio 1.65 was first drawn. The first half of the centerline of the straight diffuser was given a circular shape of radius of curvature 0.576m and the 2^{nd} half was then turned in the reverse direction with same radius of curvature to achieve the complete contour of the test diffuser. Top and bottom wall of the test diffuser was kept parallel to ensure 2D geometry of the test diffuser. The test diffuser is made of transparent plexiglass.

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, turbulence intensity and static and total pressure surveys along the entire cross section of curved diffuser, the test piece was divided into seven planes, one at the Inlet section $2W_1$ upstream of the test diffuser, three planes, Section A, Section B and Section C at 18°, 30° and 45° turn along the first half of the diffusing passages and Section D Section E and Section F at -30°,-18° and -0° turn along the second half of the diffusing passages. The number of measuring stations at each section varied from 5 to 7, from inlet to exit depending on the width of corresponding sections as shown in Fig.2.The details of measured planes are shown in Fig.1 and Fig.2.

Turbulence intensity at seven different sections of test diffuser has been measured with a single wire hot wire anemometer. The wire has been place into the flow field at desire location keeping it normal to the flow direction.

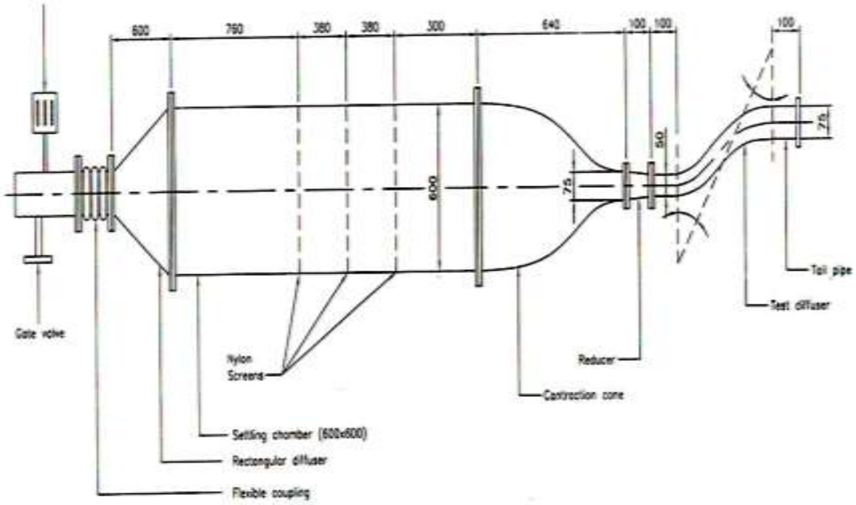


FIGURE 1. Schematic diagram of the experimental set up

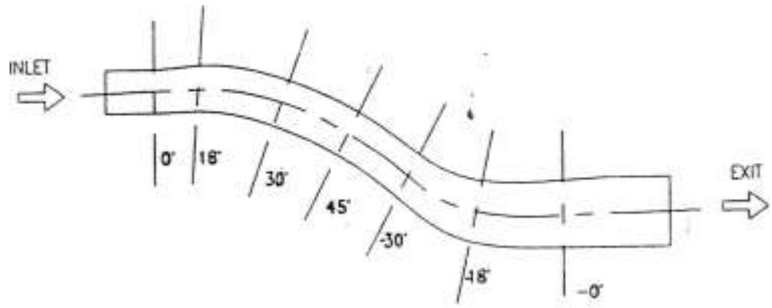


FIGURE 2. Geometry of test diffuser and measuring locations

The pre-calibrated five hole pressure probes was mounted in a traversing mechanism and the probe inserted into the flow field through 6 mm slot provided at the top wall. The probe was placed within 2 mm of bottom wall for the first reading. The probe was then moved Z-direction up 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 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[23].

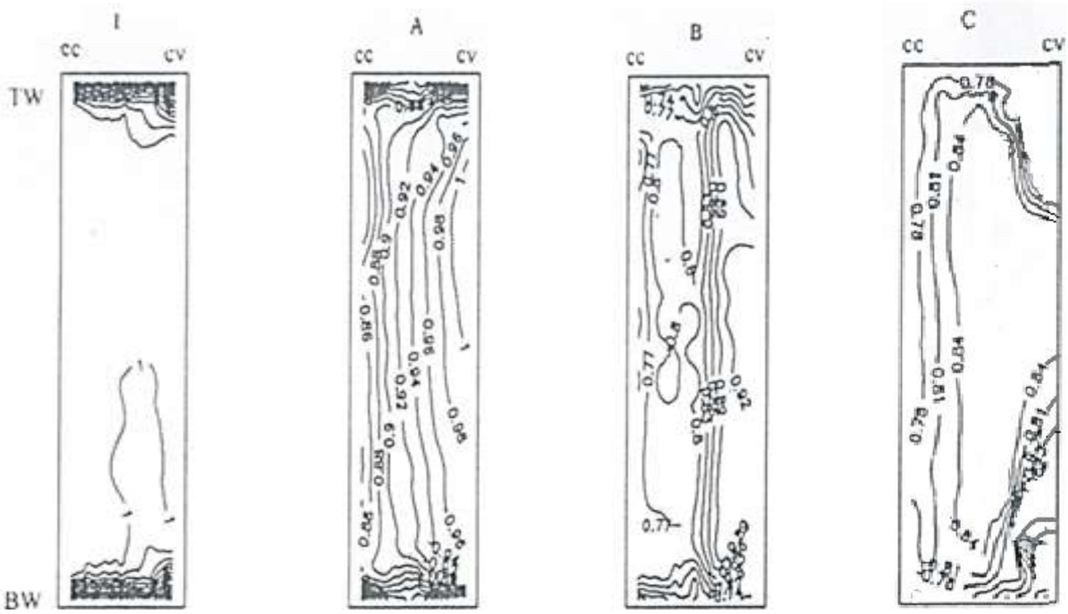
III. Results and discussion

The flow characteristics have been evaluated by mass average mean velocity, between the curved and parallel (top and bottom) walls, turbulence intensity, 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

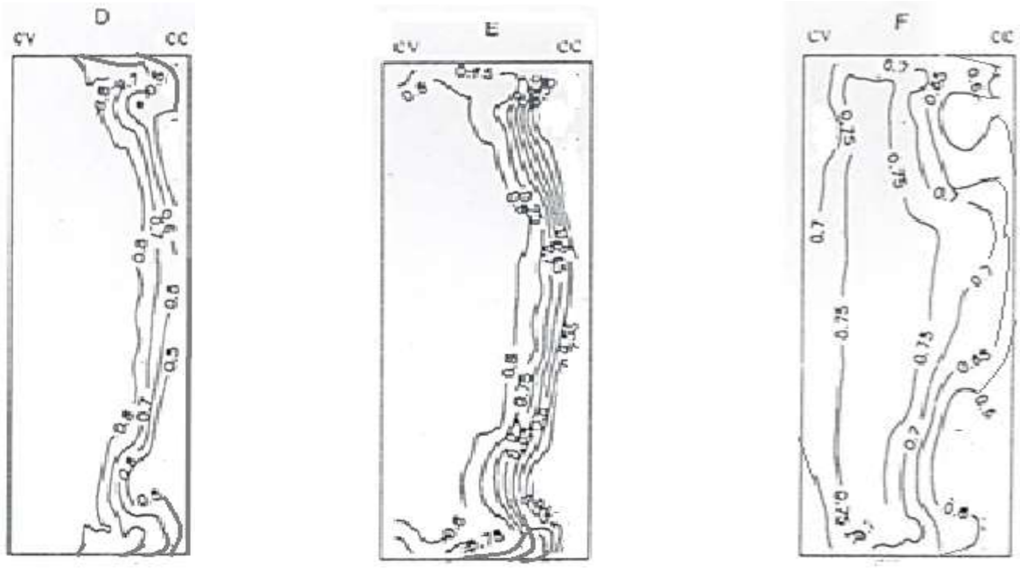
III. 1. Mean velocity contour

The normalized mean velocity distribution in the form of contour plots at at seven measuring sections of different angle of turn for this diffuser has been discussed here and is shown in Fig.3.

At section I, Fig (a) velocity distribution is uniform throughout the section except very close to the top and bottom walls. This is mainly due to the corner effect. At section A, 18° turn, Fig.(b) shows that the high velocity zone is located near to the convex side. Here the diffusion of flow is seen more along the concave wall. Diffusion of flow is also seen to be faster along the concave wall, whereas marginal diffusion is seen along the convex wall. The figure also depicts that the high velocity fluid has accumulated to the top half along the convex wall. This may be due to the momentum transfer from one curved wall to other curved wall. At 30° turn, Fig.(c), depicts an overall diffusion of flow throughout the entire cross section, though the high velocity fluid adhere to the convex wall sides as seen in previous section. However, near to the parallel walls and very close to the convex side, the nature of the iso-velocity lines are ending towards the middle. it indicates the generation of flow instability at these zone. As the flow moves downward and reaches the inflexion plane as in Fig.(d) the instability of flow, seen at 30° turn of 1st bend enhances and is clearly visible along the convex side. The high velocity fluid occupied more cross sectional area at this section. The bulk flow has also moved towards the concave side and as a result of which the flow has accelerated along this wall. This is attributed to the transfer of momentum from convex to concave side. Further downstream, at 30° turn of the 2nd bend Fig.(e) the high velocity fluid has completely shifted towards the convex side. The transfer of momentum, seen at inflexion plane, continues at this section even change in centerline curvature. The low velocity fluid has accumulated along the concave side of the 2nd bend and occupied the entire depth of the section along this wall. The bulging out of the contour lines near to the top and bottom parallel walls further indicates the accumulation of more low fluid at this zone compared to the mid plane of the diffuser. This type of phenomenon has also been reported by Majumder et al. [8] for 90°/90° diffuser. Further downstream, at -18° turn of the right bend as per Fig.(f) more low velocity fluid accumulates along concave side, occupying more flow area. This indicates the change in flow direction as the curvature of the centerline changes after the inflexion plane. The flow along the convex side has also accelerated a little at this section. The movement of fluid from concave to convex continues further downstream Fig.(f). The high velocity core has occupied more or less entire exit cross section except at the corners near to the concave side.



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



(e) Section E (f) Section F (g) Section D

FIGURE 3. Normalized mean velocity distribution

III.2. Pressure recovery and loss coefficients

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.4. It was calculated based on the static pressure difference between the succeeding and preceding sections. The figure shows that the coefficient of pressure recovery increases steadily up to the Section A in S-shape diffuser and then the increase takes place in rapidly up to the Section F. This is mainly due to the formation of secondary motions The overall mass average pressure recovery coefficient is nearly 36% for this diffuser.

The mass average total pressure loss coefficient increases sharply in the test diffuser up to the Section A. Then the loss coefficients in curved diffuser increases in steadily up to Section F. The overall mean value in mass average pressure loss coefficient is nearly 13% for this test diffuser.

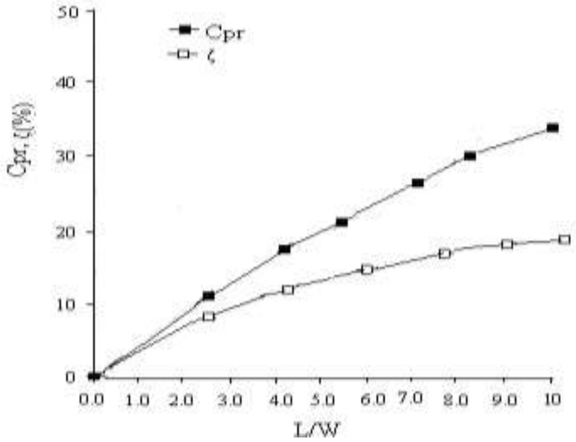


FIGURE 4. Variation of mass average pressure recovery and loss coefficients

III.3. 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:

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

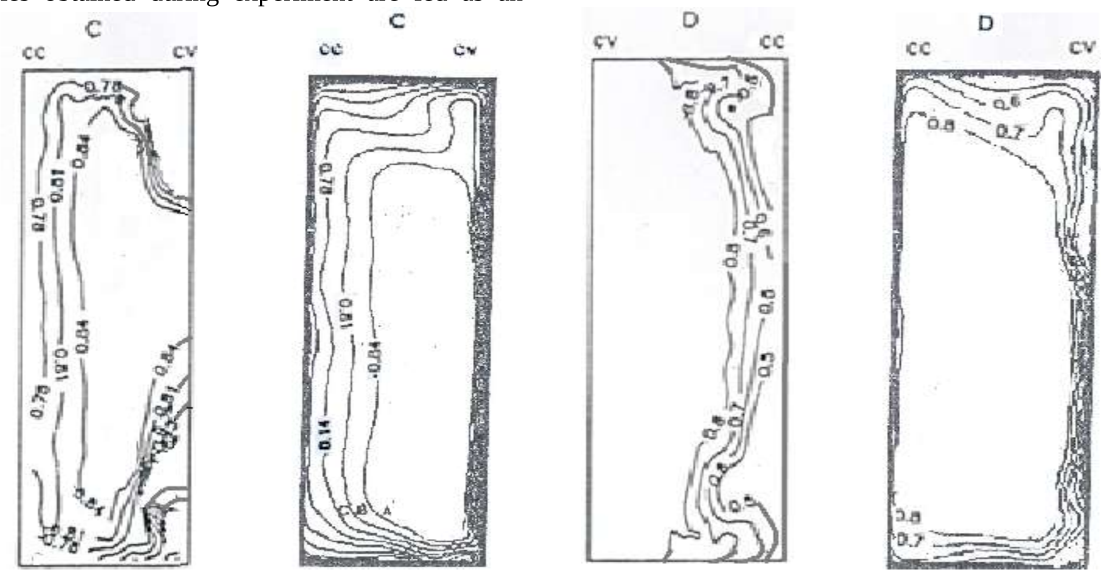
The terms on the right hand side of the Equation from left to right can be interpreted as the diffusion, generation and dissipation rates of ε.

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 normal atmospheric pressure at outlet as outlet condition during the validation with FLUENT Some of the validation figures are shown in Fig.5(a), Fig.5(b), Fig.5(c) and Fig.5(d) respectively

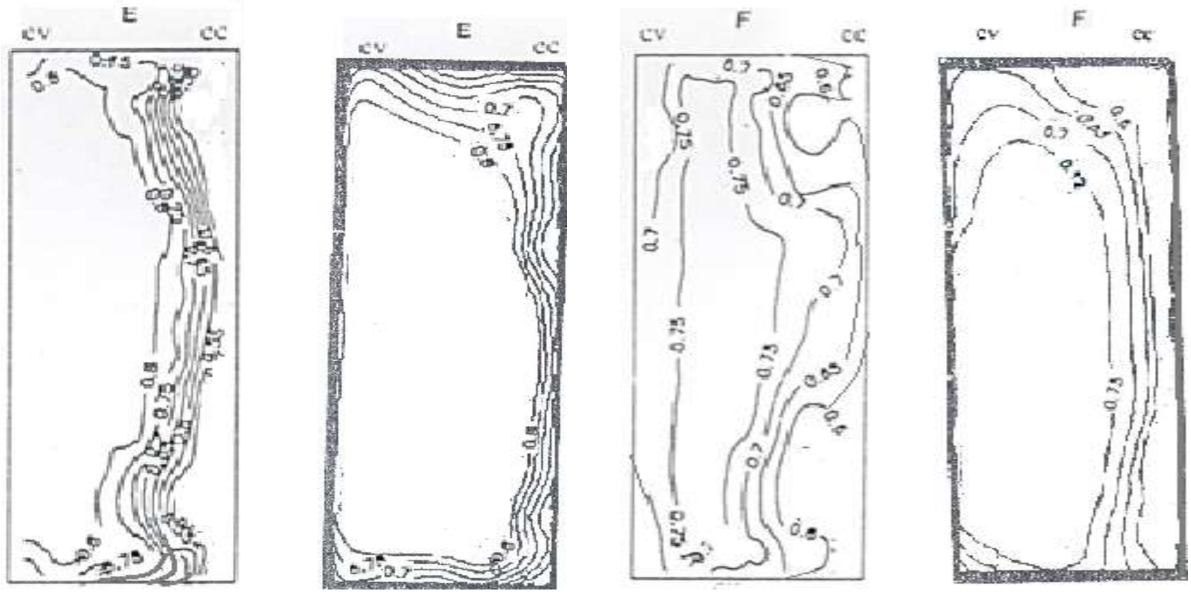
All four figures indicate that the mass averaged mean velocity contours obtained by computational and experimental investigations, which shows a qualitative matching to each other.

The mean velocity distribution at the Section C, Section D, Section E and Section F are shown in Fig. 5(a), Fig. 5(b), Fig.5(c) and Fig.5 (d) show a reasonably good agreement of the computational investigation with the experimental results.



Experimental Computational Experimental Computational

(a) Section C (b) Section D



Experimental Computational Experimental Computational

(C) Section E (d) Section F

FIGURE 5. Comparison of normalized velocity distribution at Section C, Section D, Section E and Section F through Computational and Experimental investigation

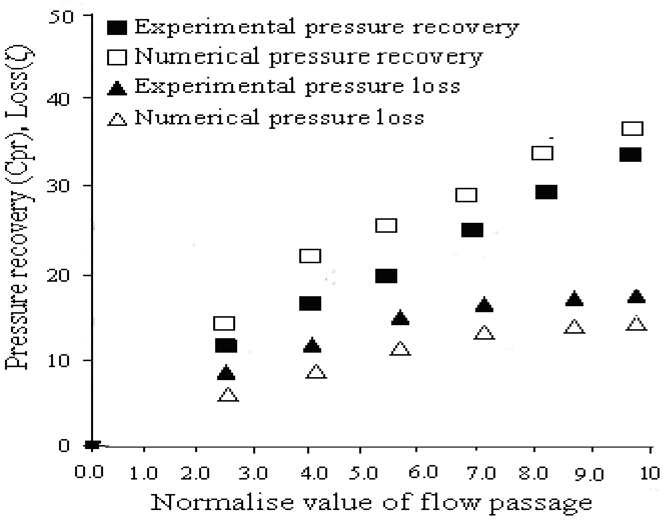


FIGURE 6. Comparison of performance parameters obtained through computational and experimental investigation

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 39% compared to the experimental investigation, which obtained as 36%. Similarly the coefficient of pressure loss is obtained as 11% in computation investigation compared to the 13% 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 $k - \varepsilon$ model can predict the flow and performance characteristics reasonably well for similar geometries with same boundary conditions

IV. Conclusions

From the present investigation the following conclusions have been drawn:

1. High velocity fluids shifted and accumulated at the convex wall of the exit section. .
2. The mass average static pressure recovery and total pressure loss for the curved test diffuser is continuous from Inlet section to Section F.
3. Performance parameter like coefficient of mass average static pressure recovery and coefficient of mass average total pressure loss are 36% and 13% respectively.
4. A comparison between the experimental and predicated results for the sigmoid diffuser show good qualitative agreement between the two.
5. The coefficient of mass averaged static pressure recovery and total pressure loss are obtained as 39% and 11% in predicted results and in the experimental results their values obtained as 36% and 13% respectively, which indicate a good matching between the experimental and predicted results
6. 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 S-shape 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, NACA Report, No. 115, 1953

[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, NASA Contractor Report 3634, 1982.

[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] M .I. Yaras, Effects of inlets conditions n the flow in a Fishtail Diffuser with strong Curvature, Trans ASME, Journal of Fluid Engineering, Vol. 118, 1996, pp 772 – 778.

[7] B.A. Reichert and B.J. Wendt, Improving Curved Subsonic Diffuser Performance with

Vortex Generator, AIAA Journal, Vol.14, 1996, 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 an Annular S-shaped Ducts, Trans ASME, Journal of Turbo Machinery, Vol. 120, 1998, 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] R.G. Dominy, D.A. Kirkham, and A.D. Smith, Flow Development through Inter turbine Diffuser, Trans ASME, Vol. 120, 1998, pp. 298 – 304.

[12] S. Fuji, and T. H. Okiishi, Curved Diffusing Annulus Turbulent Boundary-Layer Development, Authors' Report. No.ISU-ERI-Ames-71033, National Technical Information Service, Springfield, Va., 22151.as N72-10239, 1997.

[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. Engineer, 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, Trans Prase Worthy Prize, Journal of International Review of Aerospace Engineering, Vol. 3. No. 5, pp. 249-256, 2010.

[16] A. J. Benin, F Gul, and E Pasqualotto,. Loss Correlations for Developing Turbulent Pipe Flow", Trans Prase Worthy Prize, Journal of International Review of Mechanical Engineering, January, 2007.

[17] M. Kahrom, and A. Shokrgozar, Evaluation of Turbulence Models in Predicting Turbulence Penetration into Low Reynolds Number Regions, Trans Prase Worthy Prize, Journal of International Review of International Review of Mechanical Engineering, Vol. 4. n. 1, January, pp. 51-59, 2010.

[18] P. K. Sinha, A.K. Biswas and B. Majumdar, Computational Analysis of Flow Structure in a Curved Subsonic Diffusing Duct, Trans Prase Worthy Prize, Journal of International Review of Mechanical Engineering, Vol. 7. No. 4, pp. 734-739, 2013.

[19] A.K. Biswas, Prasanta. K. Sinha., A. N. Mullick and B. Majumdar, "A Computational Analysis of Flow Development through a Constant area S-Duct",Trans Prase Worthy Prize, Journal of International Review of Aerospace Engineering, Vol. 6. No. 4, pp. 145-152, 2013.

[20] P. K. Sinha, A.K. Biswas and B. Majumdar, Computational lysis of Investigation of Flow Development Through 22.5°/22.5° Sigmoid Diffuser, Trans Prase Worthy Prize, Journal of International Review of Aerospace Engineering, Vol. 7. No. 5, pp. 155-163, 2014.

[21] 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 Mechanical Engineering, Vol. 3. n. 6, pp. 796-807, 2009.

[22] 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 Mechanical Engineering, Vol. 4. n. 6, pp. 702-710, 2009.

[23] S. Mukhopadhya, 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.