IJAPIE-2016-01-109, Vol 1(1), 33-38

INTERNATIONAL JOURNAL OF ADVANCED PRODUCTION AND INDUSTRIAL ENGINEERING

(A Blind Peer Reviewed Journal)

CFD Analysis of Angle Axial Annular Diffuser with Both Hub and Casing at Diverging Equal Angles

B.B.Arora[1]

[1](Department of Mechanical Engineering., Delhi Technological University, Delhi, India)
[1]prof_bbar@yahoo.com

Abstarct : *The development of flow inside the axial annular diffuser whose hub and casing walls are diverging at equal angles have been predicted with the help of computational fluid dynamics (CFD). The equivalent cone angle for axial annular diffuser has been taken as 20° and area ratio 2 and 3 for the present study has been taken as for The analysis has been carried out for flow regime with various experimentally obtained inlet velocity profiles for area ratio 2 with or without swirl. The flow behaviour inside the annular diffuser was analyzed for the same experimental geometry with CFD and pressure recovery coefficient was computed which agrees reasonably well with the experimental/available data. Further prediction of expected flow behaviour inside the annular diffuser with area ratio 3 has been carried out. Pressure recovery coefficient on the mass average basis has also been computed.*

Keywords: *Annular diffusers; pressure recovery coefficient; swirl, equivalent cone angle*

I. INTRODUCTION

The function of a diffuser is the efficient conversion of kinetic energy into pressure. Many fluid-dynamical systems involve diffusion in annular passages. Need for annular diffusers may arise from a necessity to provide a central core to allow a coaxial shaft in a given situation. Annular diffusers are widely used in engineering, in particular, as outlet devices of pumps and turbines often located downstream of turbo machinery in a number of applications. In aircraft application, annular diffusers often operate downstream of compressors. Such diffusers handle flows having substantial amount of swirl and unsteadiness made up of turbulence and periodic flow components introduced by the turbo machinery. The swirling component of velocity may arise either from the presence of inlet guide vanes or any other components preceding the diffuser, e.g., a compressor, or from rotation of the central shaft through the diffuser. The introduction of presence of swirl alters the flow field considerably and this affects the overall performance of a system.

Swirling flows through annular diffusers have been investigated by many researchers such as Sovran and Klomp [3], Shrinath[4] Hoadley[7], Colodipietro et al. [8], Shaalan et al. [9], Kumar [10], Lohmann et al. [11] and Sapre et al. [13], Agrawal et al.[14], Singh et al.[16, 23], Kochevsky[18], Mohan et al. [19], Japikse, D[20], Kochevsky, A. N[21] and Yeung et.al [22], Manoj et al.[266, 83], Arora et al.[24, 25,29] These investigators found improved diffuser performance with swirl till a point after that it deteriorated. The performance of an annular diffuser apart from swirl is dependent on a large number of geometrical and dynamical parameters. The effectiveness of annular axial diffusers worsens with flow separation. The separation of the flow can be suppressed or shifted from one location to another with the help of swirl. The efforts have been made to design an annular diffuser for no flow separation [2,5,6], however little success has been achieved.

Literature on annular diffusers reveals that earlier studies have been carried out either with parallel hub diverging casing and both hub and casing diverging. The experimental/ analytical data on the pressure recovery coefficient or coefficient of energy losses [1, 12] for a wide range of geometrical parameters and swirl intensities are scant. Experimental studies on annular diffuser [17] require sophisticated instrumentation and complicated time-consuming procedures which is not economically viable and thus has limited the research activity in the field of annular diffusers [12].

The present study is therefore carried out to examine with the help of Computational Fluid Dynamics (CFD), the detailed flow behavior of axial annular diffusers with parallel hub diverging casing for same equivalent cone angle of 20º and area ratio of 2 and 3. Experimental velocity profiles were obtained with the axial annular diffuser having hub parallel and casing diverging and area ratio of the experimental diffuser was 2.01, and equivalent cone angle of 20.02º. CFD analysis of the diffuser with same configuration and dynamic parameters was carried out with different turbulence models. The model which predicted the results more closely with the experimental results was chosen for further investigations. CFD Study has been carried out to predict the effect of experimentally obtained inlet velocity profiles without swirl (0°) and with inlet swirl angles of 7.5°, 12°, 17°and 25° on the performance of annular diffusers.

II. EXPERIMENT SETUP

Figure 1(a) shows the actual experimental setup used for the present investigation. The test rig consists of a single stage centrifugal blower which delivers 1.5m^3/s at 1m water gauge pressure. It draws air from the atmosphere through a very fine mesh filter and delivers it to a settling chamber through a well-designed conical divergence. A symmetrical damper placed at the blower intake controlled and kept the flow rate constant through the system. A piece of heavy fabric serving as flexible coupling was used to seal the gap between the blower and settling chamber in order to prevent the vibrations reaching to settling chamber from the blower. The settling chamber is provided with fine mesh screens and a honey comb section. The purpose of the screens is multifold in serving as flow steadying and straightening, reducing the level of turbulence and losses. It is further connected to a constant-area annular duct made up of two commercial pipes; through a smooth converging section. Smooth transition from the annulus to the conical casing of the diffuser was ensured by inserting suitable metal shims between flanges and the inside was finished off with plasticine. Diffuser hub was made from cast aluminum and machined smooth whereas the casing of the annular test-diffusers was made of transparent Plexiglas. This was done to permit flow visualization inside the annulus so formed. The air flowing through the diffuser was finally exhausted into the atmosphere.

The measurements of static pressure and yaw angle were made manually with the help of Cobra probe, Traversing Mechanism and Manometers.

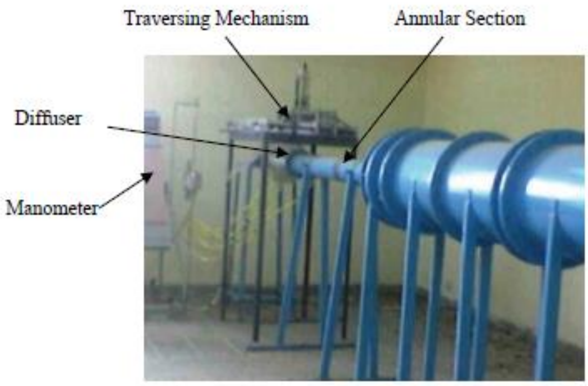

Fig. 1. (a) Actual Diffuser test setup

Figure 2 shows Annular diffuser Geometrical Parameters of the half section as the diffuser has been taken as axially symmetrical.

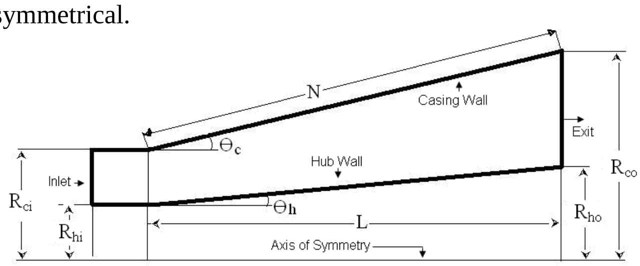

Fig. 2. Annular Diffuser Geometrical Parameters

III. CFD MODELLING

Annular diffuser geometry was sketched with proper meshing scheme with the help of GAMBIT software. In the pre-study *k*- ε turbulence models such as standard, RNG and realizable were tried on the geometries for which experimental data were available. The results obtained were validated with the available experimental results. The boundary conditions fed at the inlet is the same velocity profile as experimentally obtained with turbulence specification of 3% turbulence and hydraulic diameter as calculated from the geometry of the diffuser inlet. The outlet boundary condition is pressure normal to the pressure outlet with turbulence specification of 3% turbulence and hydraulic diameter as calculated from the geometry of the diffuser outlet. The solution controls for momentum, swirl velocity, turbulence kinetic energy and turbulence dissipation rate are second order up winding. The convergence criteria for residuals are 10^{-6} for various parameters involved in the present study such as continuity, velocity components v_x, v_r, and v_Z, swirl, *k* and ε; the results were found to be stable.

The modeling was repeated for various mesh sizes varying from 50000 to 500000 mesh cells to attain the grid independence. It was found that the model which approached more closely to the experimental results was 2D axisymmetric RNG ``renormalization group" *k*- ε turbulence model with moderate mesh size of 0.07cm. The RNG-based *k*- ε turbulence model [15] is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called ``renormalization group" (RNG) methods. The same model has been used for carrying out the analysis for other geometries considered for the present study.

The governing equations for 2D axisymmetric geometries used are the same as used by Arora[ ]

Grid independence Tests were carried out on number of geometries with regards to meshing and turbulence modeling. Various turbulence models and meshes were tested with inlet velocity profile which was obtained experimentally for a fully developed flow on an annular diffuser having an equivalent cone angle of 20° whose both casing and hub were diverging and area ratio of 2. The turbulence model which is in close proximity with the experimental results as shown in Fig 3 is Renormalization-group (RNG) k- ε model.

k-ε RNG model with mesh size of 0.07 was considered for the present CFD analysis to reduce the computational time without foregoing the desired accuracy.

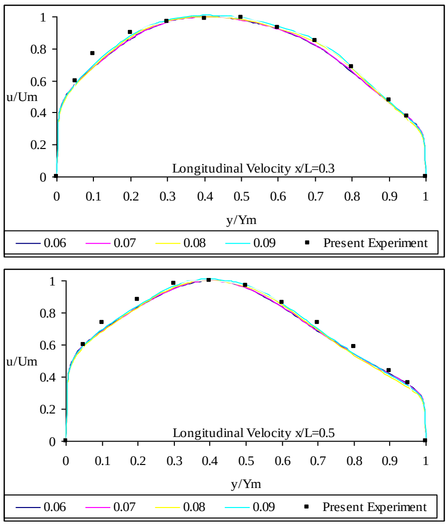

Fig. 3. Grid Independence Test of Longitudinal Velocity (0°) for 20° equivalent cone angle diffuser

Pressure recovery Coefficient

The static pressure rise non-dimensionalised with respect to the diffuser inlet dynamic head is defined as the static pressure recovery [29].

For one-dimensional flow of perfect gas without any energy loss, the ideal pressure rise for given diffuser can be computed by considering energy conservation.

The same turbulence model was used to predict the performance of axial annular diffusers with various geometries and one such model was validated with experimental results as shown in Fig. 4 and Fig. 5

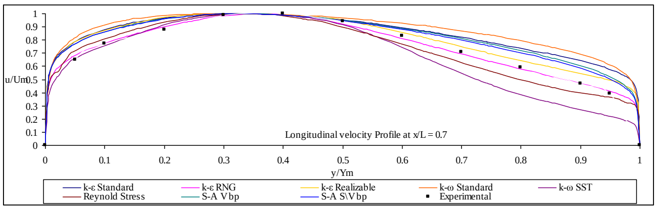

Fig. 4 Validation of Turbulence Model with Experimental Results at x/L = 0.7

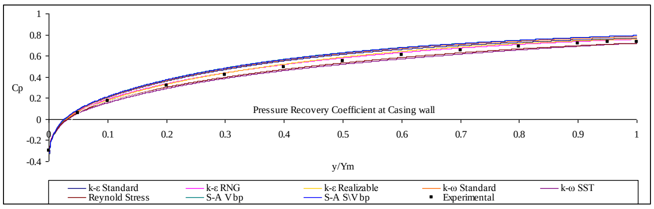

Fig. 5 Validation of Pressure Recovery Coefficient with experimental Results at the casing wall

IV. RESULTS & ANALYSIS

Velocity Profile

Figure 6-8 show the Longitudinal and swirl velocity profiles for the area ratios 2 to 4 respectively. The velocity profiles are shown for inlet swirl angles 0°, 7.5°, 12°, 17°and 25°. Fig.6 (a) to 8(a) represents the Longitudinal whereas 6 (b) to 8(b) represents the swirl velocity profiles All the velocity profiles are shown in terms of non-dimensional velocity as local longitudinal or swirl velocity divided by local maximum velocity of the transverse where velocity is required. The non-dimensional velocity has been shown as a function of non-dimensional diffuser passage height of the particular traverse (y/Ym). The y/Ym =0 is the hub position and for y/Ym =1, it is the casing position of the traverse. The graphs are shown at various traverses of the diffuser passage at x/L= 0.1, 0.3, 0.5, 0.7 and 0.9 for all the area ratios and inlet swirl angles.

Fig.6 (a) to 8(a) illustrate that the flow is hub generated for no swirl condition and the shift increases to larger extent towards hub with the increase in the area ratio for same inlet velocity profile in this case too. The peak of the velocity at x/L =0.9 in the order of area ratio 2,3 and 4 occurs at y/Ym at 0.45, 0.41 and 0.32 for 10° equivalent cone angle and for 20° equivalent cone angle it happens at y/Ym = 0.4,0.34 and 0.29. Moreover the velocity at the casing recedes at faster rate. The flow is pushed towards the casing with the introduction of swirl.

Fig.6 (a) for area ratio 2 illustrates that the reversal of flow takes place at the casing wall for 7.5° inlet swirl at x/L= 0.9 from casing to y/Ym= 0.98, whereas for inlet swirl of 12° and above it takes place at the hub wall. The reversal takes place at x/L= 0.7 for 12° inlet swirl and at x/L= 0.5 and 0.3 for 17° and 25° respectively. For 12° inlet swirl the flow is reversed up to y/Ym= 0.05 and 0.13 for x/L = 0.7 and 0.9 respectively. For 17° inlet swirl the flow is reversed up to y/Ym= 0.05, 0.15 and 0.23 for x/L = 0.5, 0.7 and 0.9 respectively. Whereas for 25° inlet swirl the flow is reversed up to y/Ym= 0.004, 0.09, 0.18 and 0.26 for x/L = 0.3, 0.5, 0.7 and 0.9 respectively. Fig.7 (a) for area ratio 3 illustrates that the reversal of flow takes place at the casing wall for no swirl (0°) and 7.5° inlet swirl at x/L= 0.7 and 0.5 respectively. For no swirl condition it takes place from casing to y/Ym= 0.98 and 0.94 at x/L=0.7 and 0.9 respectively, whereas for 7.5° inlet swirl it takes place from casing to y/Ym= 0.98, 0.91 and 0.87 at x/L=0.5, 0.7 and 0.9 respectively. For inlet swirl of 12° and above it takes place at the hub wall. The reversal takes place at x/L= 0.5 for 12° inlet swirl and at x/L= 0.3 for 17° and 25°. For 12° inlet swirl the flow is reversed up to y/Ym= 0.13, 0.25 and 0.34 for x/L = 0.5, 0.7 and 0.9 respectively. For 17° inlet swirl the flow is reversed up to y/Ym= 0.07, 0.23, 0.33 and 0.38 for x/L = 0.3, 0.5, 0.7 and 0.9 respectively. Whereas for 25° inlet swirl the flow is reversed up to y/Ym= 0.18, 0.32, 0.39 and 0.44 for x/L = 0.3, 0.5, 0.7 and 0.9 respectively.

The graphs demonstrate that the velocity recedes at a faster rate near the hub as well as casing wall as the flow progresses along the diffuser passage due to the growth of boundary layer. The non-dimensional velocity near the hub wall tends to decrease with the introduction of swirl and it increases near the casing wall.

It is also evident from the graphs of non swirling flow (0°), that the location of maximum non dimensional velocity shifts towards the hub for downstream of the diffuser passage. The shift increases to larger extent with the increase in the area ratio for same inlet velocity profile. This is due to the fact that the stall increases at the casing wall with increase in the area ratio for same equivalent cone angle diffusers. The stall tends to shift from casing to the hub wall with the introduction of swirl as observed by examining the Figures 6 and 7. The shift is stronger with the increase in the inlet swirl.

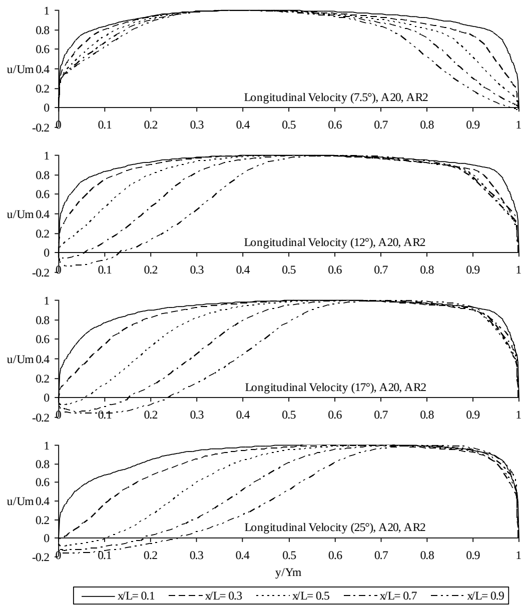

Fig. 6(a) Non-dimensional Longitudinal Velocity versus diffuser passage height for Area ratio 2 for inlet swirl angles (α = 0º to 25º) at various transverses at x/L = 0.1 to 0.9.

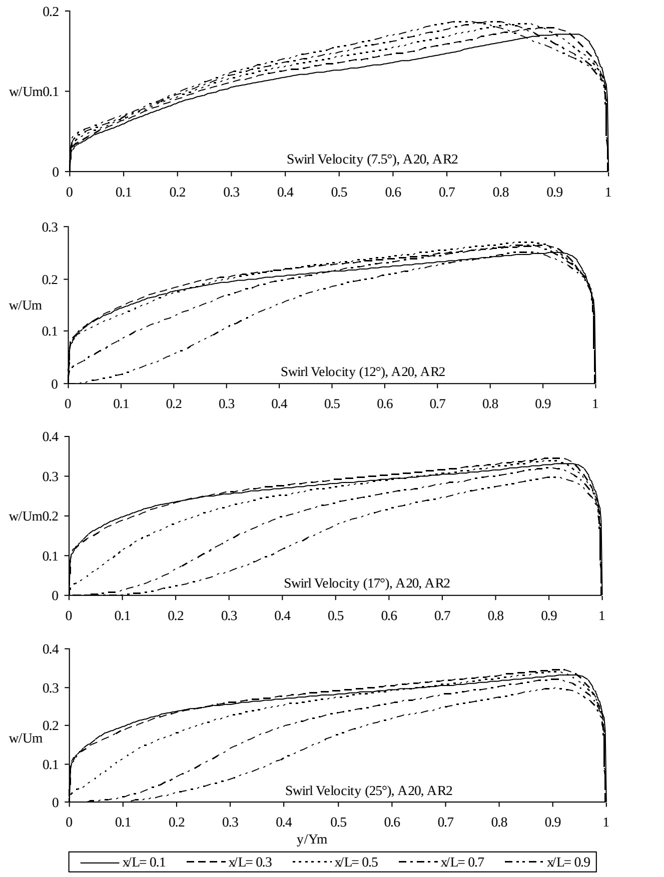

Fig. 6(b) Non-dimensional Swirl Velocity versus diffuser passage height for Area ratio 2 for inlet swirl angles (α = 0º to 25º) at various transverses at x/L = 0.1 to 0.9.

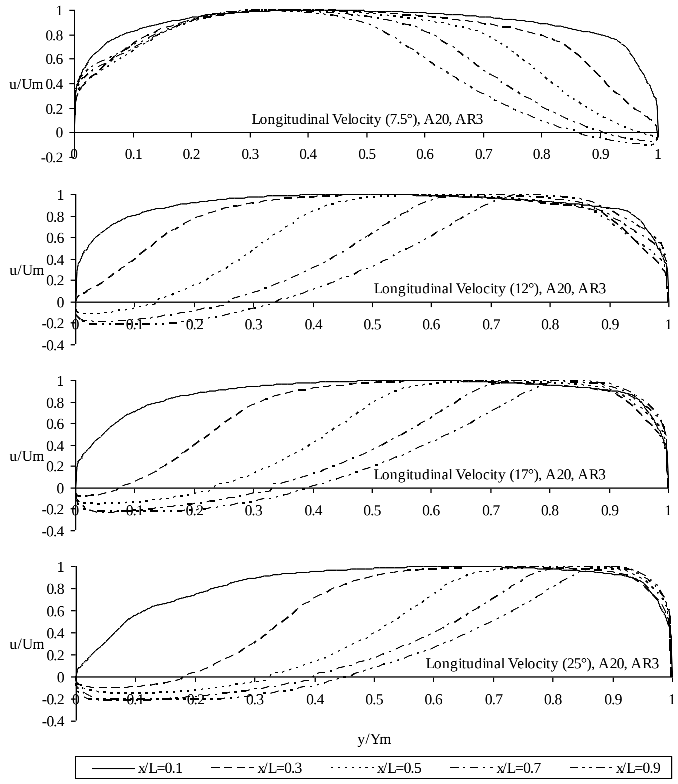

Fig. 7(a) Non-dimensional Longitudinal Velocity versus diffuser passage height for Area ratio 3 for inlet swirl angles (α = 0º to 25º) at various transverses at x/L = 0.1 to 0.9.

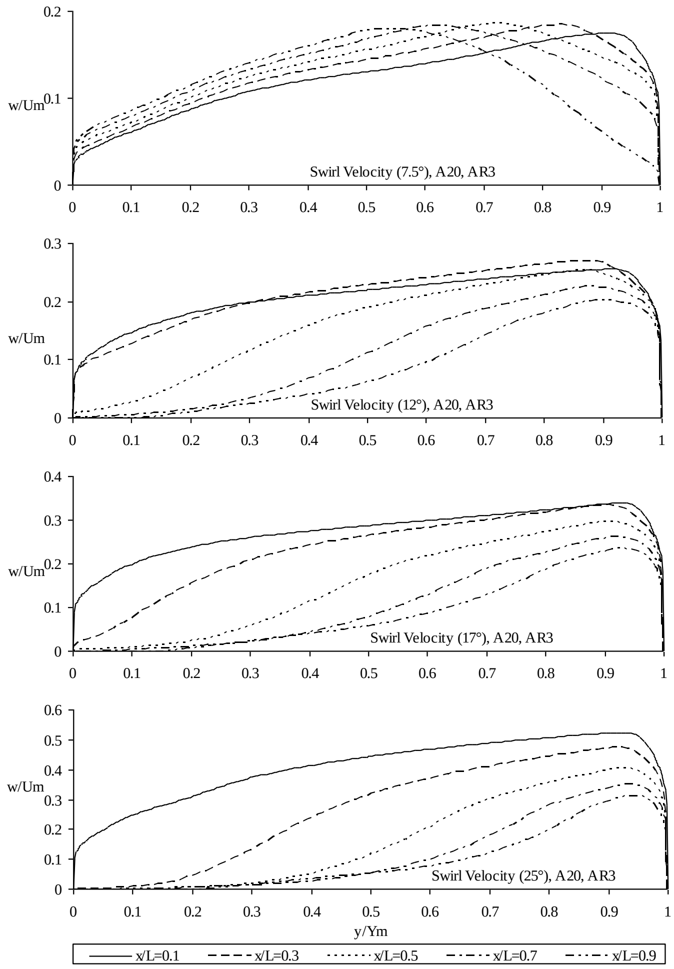

Fig. 7(b) Non-dimensional Swirl Velocity versus diffuser passage height for Area ratio 3 for inlet swirl angles (α = 0º to 25º) at various transverses at x/L = 0.1 to 0.9.

Pressure Recovery Coefficient

Figure 8 indicate pressure recovery coefficient at casing wall (C_p) for diffuser for area ratios 2 to 4 as a function of non-dimensional diffuser passage x/L for various inlet swirl angles 0°, 7.5°, 12°, 17° and 25°. C_p increases with the diffuser passage in each case. The marginal increase in Cp is sharp in the beginning of the diffuser passage and later on it decreases with the diffuser passage.

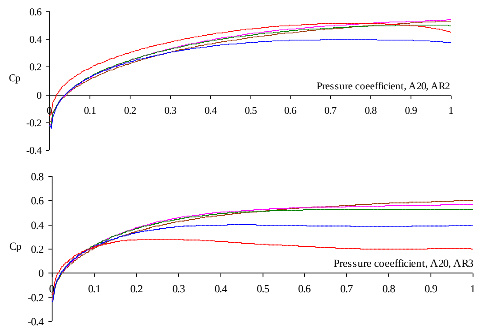

Fig. 8. Pressure recovery Coefficient predicted by RNG *k*- ε model

The pressure recovery coefficient (Cp) increases with the increasing diffuser passage length for flow without swirl. Cp increases with the increasing swirl as compared to the non swirling flow in the beginning of diffuser passage length. However for 7.5°, 12°, 17º and 25º inlet swirl Cp is lower than the flow without swirl beyond x/L =0.82, 0.95, 0.78 and 0.47 respectively in case of Area ratio 2. Cp is highest up to diffuser passage length of 0.36 for 25º inlet swirl. From x/L=0.36 to 0.56 it is maximum for 17º inlet swirl, then from 0.56 to 0.95, it is for 12º inlet swirl and in the last for flow without swirl.
In diffuser with Area ratio 3, the Cp increases as compared to no swirl condition with increasing swirl in the beginning of diffuser passage length and the marginal increase is more than diffuser with Area ratio 2. However for 7.5°, 12°, 17º and 25º inlet swirl Cp is lower than the flow without swirl beyond x/L =0.67, 0.97, 0.60 and 0.25 respectively. Cp is highest up to diffuser passage length of 0.18 for 25º inlet swirl. From x/L=0.18 to 0.28 it is maximum for 17º inlet swirl, then from 0.28 to 0.97, it is for 12º inlet swirl and in the last for flow without swirl. The reason for above phenomenon is that the swirl decays as we move downstream of diffuser passage.

V. CONCLUSIONS

Validated CFD RNG k- ε model has been employed to predict axial annular diffuser performance. Following inferences have been drawn from the predicted computational results for area ratios 2 and 3 for various inlet swirl angles.

1. The longitudinal velocity decreases continuously as the flow proceeds downstream, for both swirling and non swirling. inlet flow
2. Due to the development of boundary layer shapes of velocity profiles are dissimilar at different locations of the flow passage
3. At any diffuser transverse the maxima of velocity profile is not at the middle rather it is towards the hub for non swirling flow. It shifts towards the casing side with the increase of swirl.
4. With the introduction of swirl, the flow is forced towards casing wall which strengthens the flow towards casing wall than the hub wall.
5. Pressure recovery coefficient increases with the diffuser passage. However the marginal recovery decreases with the diffuser passage.
6. With the introduction of swirl the recovery is sooner towards the casing wall. The effect of swirl emerges to decay gradually as the flow proceeds downstream of the flow. Further the recovery is negligible or nil towards the diffuser exit.
7. Stall is not observed on the hub wall for non swirling flow, however it is observed on casing wall for area ratio 3
8. CFD analysis in the pre study is reasonably in good agreement with the experimental data.
9. The RNG k- ε turbulence model used in the present work can be used to predict the flow behaviour in advance and the pressure recovery coefficient and diffuser effectiveness can be computed for a particular type of diffuser.

REFERENCES

[1] Howard, J.H.G., Thorriton Trurnp A.B.and Henseler, H.J, "Performance and Flow Regimes for Annular Diffusers", ASME Paper No. 67 WA/FE 21, 1967.
[2] Sovran Ed. G. Diffusers with Rectangular, Conical or Annular Cross Section", Fluid Mechanics of Internal Flow, , Elsevier Amsterdam, pp.270-319, 1967.
[3] Srinath.T, "An investigation of the effects of swirl on flow regimes and performances of annular diffuser with inner and outer cone angles." M.A.Sc. thesis, University of waterloo, Canada 1968.
[4] Stevens, S.J and Markland, E., "The Effect of Inlet Conditions on the Performance of Two Annular Diffusers", ASME Paper No.68 WA/FE 38.
[5] Stevens, S.J., "The Performance of Annular Diffusers". Proc. Instn. Mech. Engrs., Vol. 182, Part 3D, pp.58-70,1968.
[6] Hoadley, D., "Three Dimensional Turbulent Boundary Layers in an Annular Diffuser", Ph.D. Thesis, Department of Engineering, University of Cainbridge, London, 1970.
[7] Coladipietro, R., Schneider, J.H and Shridhar, K., "Effects ofIfflet Flow Conditions on the Performance of Equi Angular Annular Diffusers", CSME Paper No. 73 84,1974.
[8] Sovran, G and Klomp, E.D., "Experimentally Determined Optimum Geometries for Rectilinear
[9] Shaalan, M.R.A and Shabaka, I.M.M., "An Experimental Investigation of the Swirling Flow Performance of an Annular Diffuser at Low Speed", ASME PaperNo. 75 WA/FE 17,1975
[10] Kumar D.S., "Effect of swirl on flow through annular diffusers", Ph.D. Thesis, I.I.T. Delhi, 1977.
[11] Lohmann, R.P., Markowski, SJ and Prookman, E.T.,"Swirling Flow Through Annular Diffusers with Conical Walls", Journal of Fluids Engineering, Trans. ASME, Vol.101, pp.224 229,1979.
[12] Chithambaran, V.K., Aswatha, Narayana P.A., Chandrashekra Swamy, N.V. "Mean velocity characteristics of plane diffuser flows with inlet velocity distortion." Journal of Indian Institute of Science, 65(A):79 93, 1984.
[13] Sapre, R.N., Singh, S.N., Agrawal, D.P and Malhotra, R.C., "Flow Through Equiangular Wide Angle Annular Diffusers", 15th NCFMFP, Srinagar, July, 1987.
[14] Agrawal, D.P., Singh, S.N., Sapre, R.N and Malhotra, R.C., "Effect of Hub Rotation on the Mean Flow of Wide Angle Annular Diffusers", HydroTurbo 1989, Czechoslovakia, 1989.
[15] Choudhury D. ," Introduction to the Renormalization Group Method and Turbulence Modeling." Fluent Inc. Technical Memorandum TM-107, 1993.
[16] Singh S.N., Agarwal D.P., Sapre R.N. and Malhotra R.C., "Effect of inlet swirl on the performance of wide angled diffusers", Indian journal of Engineering & Materials Sciences, Vol.1.pp 63-69.1994.
[17] Buice, C. U., and Eaton, J. K., "Experimental Investigation of Flow Through an Asymmetric Plane Diffuser", Report No.TSD-107, Thermosciences Division, Department of Mechanical Engineering, Stanford University, Stanford, CA, USA, 1997.
[18] Klomp, E.D., "Performance of Straight-Walled Annular diffuser with Swirling Flow", The Aeronautical Journal, Vol. 101, No. 1010, pp. 467-480, 1997.
[19] Mohan, R., Singh, S.N., and Agrawal, D.P., "Optimum Inlet Swirl for Annular Diffuser Performance Using CFD", Indian Journal of Engineering and Materials Sciences, Vol. 5, pp. 15-21, 1998.
[20] Japikse, D., "Correlation of Annular Diffuser performance with Geometry, Swirl, and Blockage", Proceedings of the 11th Thermal and Fluids Analysis Workshop (TFAWS), Cleveland, Ohio, August 21–25, 2000, pp. 107-118.
[21] Kochevsky, A. N. "Numerical Investigation of Swirling Flow in Annular Diffusers With a Rotating Hub Installed at the Exit of

Hydraulic Machines", Journal of Fluids Engineering, Trans. ASME, Vol. 123, pp.484-489, 2001.
[22] Yeung, W. W. H. and Parkinson, G. V., "Analysis and Modeling of Pressure Recovery for Separated Reattaching Flows," ASME Journal of Fluids Engineering, Vol. 126, No. 3, pp. 355–361, 2004.
[23] Singh, S. N., Seshadri, V., Saha, K., Vempati, K. K., and Bharani, S., "Effect of Inlet Swirl on the Performance of Annular Diffusers Having the Same Equivalent Cone Angle", Proceedings of the Institution of Mechanical Engineers, Part G, Journal of Aerospace Engineering, Vol. 220, pp. 129-143, 2006.
[24] Arora, B. B., Pathak, B. D. "Effect of Geometry on the Performance of Annular Diffuser," International Journal of Applied Engineering Research, Vol.5, No.20. (2009) pp. 2639-2652, ISSN 0973-4562.
[25] Arora, B. B., Manoj Kumar, Subhashis Maji, " Study of Inlet conditions on Diffuser Performance," International Journal of Theoretical and applied Mechanics, Vol.5, No.2. (2010) pp. 201-221, ISSN 09736085.
[26] Manoj Kumar, Arora, B. B., Subhashis Maji, " Analysis of flow separation in wide angle annular diffusers," International Journal of Applied Engineering Research, Vol.5, No.20. (2010) pp. 3419-3428, ISSN 0973-4562
[27] Arora, B. B., Pathak, B. D. "CFD analysis of axial annular diffuser with both hub and casing diverging at unequal angles," International Journal of dynamics of fluid Vol.7, No.1. (2011) pp. 109-121, ISSN 0973-1784.
[28] Manoj Kumar, Arora, B. B., Maji. Subhashis and Maji, S. "Effect of inlet swirl on the flow behavior inside annular diffuser," International Journal of dynamics of fluid Volume 7, Number 2 (2011), pp. 181-188. ISSN 0973-1784
[29] Arora, B. B. 2014, "Performance analysis of parallel hub diverging casing axial annular diffuser with 20° equivalent cone angle", Australian Journal of Mechanical Engineering, Vol. 12, No. 2, June, pp. 179-194, http://dx.doi.org/10.7158/M11-823.2014.12.2.