CFD ANALYSIS OF FLOW THROUGH AXIAL ANNULAR DIFFUSER AT AREA RATIO 2.5 AND CASING DIVERGING ANGLE 0F 6^0

[1]DEVESH KUMAR, [2] B.B.ARORA
[1]Research Schollar [2] Professor
Department of Mechanical Engineering
Delhi Technological University, Delhi

ABSTRACT

The present work relates to the analysis of flow through the parallel hub and diverging casing annular diffuser. In this analysis development of flow, flow reversals and separation of flow have been visualized through the diffuser walls with and without swirl. The casing angle and area ratio have been taken as 6^0 and 2.5 .The swirl angle was varied from $0^0$to 25^0 at the inlet of diffuser. The analysis was carried out for flow regimes with various experimentally obtained inlet velocity profiles with or without swirl [1]. Longitudinal velocity, swirl velocity have been calculated along the different diffuser passage heights at different traverses along the length of annular diffuser. In the present work CFD approach was used to find the results by using RNG $k-\varepsilon$ turbulence model.

Keywords:-Annular diffuser, Pressure recovery coefficient, Swirl velocity, Area ratio, Casing diverging angle and CFD.

INTRODUCTION

A diffuser is a device which decelerates the flow and to regain total pressure or device which increases the pressure of a fluid at the expense of its kinetic energy. Diffuser is also used to maintain the uniform flow at the exit. For obtaining the uniform flow at the exit the magnitude of the secondary components should be less than 10%.
Annular diffuser is generally used in turbo machines where fluid may have to flow through a hub or a central shaft. It was well demonstrated that the diffusers of annular type are complex in nature due to the presence of inner wall which makes the flow complex through annular diffuser. In the annular diffuser there are many unknown parameters which are interconnected, by changing the one parameter whole setup has to be changed. So it is very difficult to perform experimental works on annular diffuser. To overcome this problem Numerical methods are generally used which are less costly and need less time. So Computational Fluid Dynamic approach is generally used to analyze the flow through annular diffuser.

Performance of diffuser depends upon geometrical as well as dynamical parameters. Geometrical parameters are inlet length, size of duct, area ratio of the diffuser, divergence angle, diffusion length of the diffuser, shape of the exit duct with free or submerged discharge conditions. Dynamic parameters are inlet velocity profile, boundary layers parameters, Reynolds Number and Mach Number etc. In annular diffuser maximum pressure recovery is achieved within shortest possible length due to presence of hub and internal surface to guide the flow outwards.
Sovran & Klomp (1), Shrinath (2), Hoadley (3), Colodipietro et al (4), Shaalan & Shabaka (5), Kumar (6), Lohmann et al (7), Sapre et al (8), Agrawal et al (9), Singh et al (10), Kochevsky (11), Mohan et al (12), Japikse (13) and Yeung & Parkinson (14) showed that diffuser performance depends on swirl velocity and increases with introduction of swirling flow. Pressure recovery of diffuser increases up to a certain point after that it deteriorated with swirling flows. The effectiveness and pressure recovery coefficient of annular axial diffusers decreases as separation occurs on the walls of the diffuser i.e. at the casing and on the hub surface. The separation of the flow can be shifted or delayed from one position to another by introducing the swirling flow. The efforts have been made by Howard (15), Stevens (16), Singh et al (17) developed an annular diffuser without flow separation, however they have successful up to some extent. Manoj Kumar, B.B.Arora [18] analyzed the flow in annular diffuser by using CFD technique and they calculated the effect of swirl on the flow behavior inside annular diffuser and found that the flow was moved towards the casing with the introduction of swirling flow. They also calculated the effect of swirl on pressure recovery coefficient. Ozturk Takar, Ali Pinarbasi [19] analyzed the flow in centrifugal compressor vaneless diffuser. They used the computational Fluid Dynamics approach to analyze the flow and calculated the different parameters such as velocity, Pressure and turbulent kinetic energy at different hub sections. CFD work was performed by using Fluent and Gambit. Rita J Schnipke et al [20] analyzed a vane diffuser by applying Finite element analysis. In this method a model geometry and meshing was prepared in the Gambit, then this mesh model is exposed to Finite Element Analysis. Here continuity and momentum equations are discretized by using Galerkins method. Finally the pressure contours and velocity contours are plotted. Majid Nabavi [21] analyzed the 3-D asymmetric flow through a planar diffuser. Here the flow was incompressible and diffuser with gradual expansion was used. The numerical approach was used to find the results and finally compared with experimental work. In this paper the effect of divergence angle, Reynolds number and aspect ratio on the flow asymmetry were calculated. Manoj Kumar, B.

B. Arora [22] analyzed the flow in vaneless diffuser by using CFD approach. They found the effect of inlet swirl and area ratio on the performance of annular diffuser. They presented the results in terms of non-dimensional longitudinal velocity, swirl velocities, static pressure and total pressure. Dr. Basarat salim [23] analyzed the wide angle diffusers experimentally. The effect of area ratio and diffuser divergence angle were checked on the performance of asymmetric rectangular wide angle diffusers. Sparrow et al [24] investigated fluid flows in a conical diffuser with the help of 3-D numerical model. They found that symmetric flow separation occurred for the diffuser angle of 5^0 and Reynolds number less than 2000.Results for the 10^0 and 30^0simulation showed symmetric separation at all investigated Reynolds Numbers (5000-33000). R.Keerthana et al [25] analyzed a series of annular diffusers of divergence angle 9^0, 15^0, 21^0 and 27^0 by using CFD approatch.PRO-E and ANSYS FLUENT were used. Here results showed that pressure recovery increases by increasing the diffuser angle. Stefano Ubertini [26] analyzed experimentally the annular diffuser of an industrial gas turbine, measurements were performed on a scale model of 35% with and without the struts. The results were presented in terms of flow angle, Static pressure, total pressure and wall static pressure. Manoj Kumar gopaliya [27] presented the effect of offset on S shaped diffuser with $90^0/90^0$ turn. The diffuser has rectangular inlet and semicircular outlet with aspect ratio 2.For the analyses a software code on Finite volume Technique using k- ε turbulent model was employed and to modify the flow .The finally results show that outlet pressure recovery decreases with increase in non-uniformity at the exit due to offset.. Manoj Kumar gopaliya [28] presented the effect of horizontal and vertical offset on a Y shaped duct. Here the Y shaped duct has rectangular inlet and circular outlet with area ratio=2.The settling length was 1.5D at the Re=$2.74x10^5$.In this paper a computer based program on finite volume technique using k- ε turbulence model was employed to analysis the problem. The results obtained from this study indicate reduced outlet pressure recovery accompanied with increase in non-uniformity in the flow at the exit contributed by the offset effect Ali pinarbasi[29] measured the level of turbulence experimentally in the different plane of a centrifugal compressor vaneless diffuser .In this paper detailed flow measurements at the inlet of a centrifugal compressor vaneless diffuser are presented .Alysson et al[30] investigated local flow turbulence and velocity profiles by using four turbulent models for a radial air diffuser. For all the turbulence models ANSYS-CFX software was used. It was shown that for the turbulent flow through diffuser shear stress model (**SST**) is a good choice. D. P. Agrawal et al [31] analyzed the flow in a vaned radial diffuser and calculated the velocity distribution in the blade to blade plane. In the present paper both the experimental and numerical methods were performed to calculate velocity variation. Finally results show the good agreement between the Experimental and Numerical methods**. F Frust [32**] calculated the flow of fluid at the exit of planer diffuser with 3:1 suddenly expansion ratio in a duct by using Laser Doppler Anemometry experimentally. They measured the symmetric velocity profiles at Reynolds number of 56, 114 and 252.

Chithambaran et al (1984) and Buice et al (1997) showed that experimental analysis of annular diffuser is costly because it involves precision instrumentation and takes lot of time. Due to this Computational Fluid Dynamics is more economical tool to analyze the flow through annular diffuser accurately.

In the present investigation the computational Fluid dynamics (CFD) approach by using Fluent has been used for the detailed flow analysis in axial annular diffuser with parallel hub and diverging casing. Casing angle and area ratio have been taken as 6° and 2.5. The swirl angle has been varied from 0 to 25^0 at the inlet of diffuser.

In this paper we have calculated the variation of longitudinal velocity and swirl velocity in non-dimensional form at different diffuser passage heights at x0.1, x0.3, x0.5, x0.7 and x0.9 along the axial direction .Then development of flow, flow reversals and flow separation through the diffuser passage heights was analyze. Here RNG k- ε turbulent model have been used to solve the problem numerically due to the closeness of experimental results [1].

CFD MODELLING

Annular diffuser geometry was drawn with proper meshing scheme with the help of ANSYS- 15.0 module. Here *k*- ε , (RNG) and realizable turbulent model was used. The boundary conditions at the inlet is the same velocity profile as experimentally obtained with turbulence intensity of 3% and hydraulic diameter 7.76cm. The outlet boundary condition is zero gauge pressure normal to the outlet boundary with turbulence intensity of 3%. Here the second order up winding scheme is used for momentum, swirl velocity, turbulence kinetic energy and turbulence dissipation rate. The convergence criteria for residuals was 10^{-6} for various parameters involved in the present study such as continuity, axial velocity, radial velocity, swirl velocity, turbulent kinetic energy and dissipation rates.

GOVERNING EQUATIONS

The governing equations for 2D axisymmetric geometries with swirl are given below from the reference [1].

Conservation or Continuity equation may be written as follows :

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = S_m \quad (1)$$

This is the general equation of continuity and it is applicable for compressible as well as incompressible fluid. The source S_m is the mass added to the continuous phase from the dispersed second phase (due to vaporization of liquid droplets) and any user defined sources.

For 2-D ax symmetric Geometries the mass balance equation is as follows [1]

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho v_x)}{\partial x}+\frac{\partial(\rho v_y)}{\partial x}=S_m$$

……………………………………………………………………………………………………… ……(2)

Here x and r are axial and radial directions, v_x is the axial velocity and v_y is the velocity in the radial direction.
Momentum Equation:
Momentum equation in the general form can be written as follows:

$$\frac{\partial(\rho\vec{v})}{\partial x}+\nabla.(\rho\vec{v}\vec{v}) = -\nabla p+\nabla.(\bar{\bar{\tau}})+\rho.\vec{g}+\vec{F} \quad \text{……………………………………………………. (3)}$$

Here p represents the static pressure, τ shows the stress tensor, $\rho.\vec{g}$ shows the gravitational Force and $\vec{F}$ shows the body force.
The stress tensor is given by the equation:

$$\bar{\bar{\tau}} = \mu[(\nabla\vec{v}+\nabla\vec{v}^{r}-\frac{2}{3}\nabla.\vec{v}I)] \quad \text{…………………………………………………..……………… (4)}$$

Here μ shows the viscosity, I is the unit tensor and the second term on the right hand side is the effect of volume dilation.
For 2D ax symmetric geometries the axial and radial momentum equations can be written as

$$\frac{\partial(\rho v_x)}{\partial x}+\frac{1}{r}\frac{\partial(r\rho v_x v_x)}{\partial x}+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_x)=-\frac{\partial p}{\partial x}+\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left\{2\frac{\partial v_x}{\partial x}-\frac{2}{3}(\nabla.\vec{v})\right\}\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial v_x}{\partial r}+\frac{\partial v_r}{\partial x}\right)\right]+F_x$$

……..…………………………………………………………………………………………………….. (5)

$$\frac{\partial}{\partial t}(\rho v_r)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_r)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_r) = -\frac{\partial p}{\partial r}+\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(\frac{\partial v_r}{\partial x}+\frac{\partial v_x}{\partial r}\right)\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(2\frac{\partial v_r}{\partial r}-\frac{2}{3}(\nabla.\vec{v})\right]-2\mu\frac{v_r}{r^2}+\frac{2}{3}\frac{\mu}{r}(\nabla.\vec{v})+\rho\frac{v_z{}^2}{r}+F_r\right.$$

…….(6)

Where $\nabla.\vec{v} = \frac{\partial v_x}{\partial x}+\frac{\partial v_r}{\partial r}+\frac{v_r}{r}$

Where v_z is the swirl velocity

The tangential momentum equation for 2D swirling flows may be written as

$$\frac{\partial}{\partial t}(\rho v_z)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_z)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_z = \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\frac{\partial v_z}{\partial x}\right]+\frac{1}{r^2}\frac{\partial}{\partial r}\left[r^3\mu\frac{\partial}{\partial r}\left(\frac{v_z}{r}\right)\right]-\rho\frac{v_x v_z}{r}$$

……….. (7)

TURBULENCE MODELLING:

Turbulent flows are described by fluctuating velocity components. These components mix with transported quantities such as momentum, energy and species concentration and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally costly to simulate directly in practical engineering calculations. The instantaneous (exact) governing equation can be time averaged, ensemble averaged or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contains additional unknown variables and turbulence models are needed to determine these variables in terms of known variables.
Different turbulence models are given below:

Turbulence Models

1. Spalart Allmaras model

2. k-ε models
   (a). Standard k-ε models

   (b).RNG .k-ε model

(c).Relizable k-ε models

3. k- ω model
   (a).Standard k- ω model

   (b).Shear stress transport (SST) k- ω models

4. Reynolds stress model

5. Large eddy simulation (LES) model

It is very difficult to choose one model to a particular class of problems. The choice of turbulent model will depend on accuracy required, availability of computational resources and the amount of time available for the simulation work. It is very difficult to state the model which is best suited for a specific problem. Here we used RNG k-ε turbulence model to analyze the flow because it gives the results which were very close to the experimental results[1].

RESULTS AND DISCUSSION

Figures 1.1, 2.2, 1.3, 1.4 and 1.5 show the Non dimensional longitudinal velocity variations for the area ratios of 2.5 respectively. These velocity variations are shown for inlet swirl angles of 0°, 7.5°, 12°, 17° and 25°. Figures 2.1, 2.2, 2.3 and 2.4 represent the swirl velocity profile variations. All the velocity variations are shown in terms of non-dimensional velocity as local longitudinal or swirl velocity divided by local maximum longitudinal velocity. The non-dimensional velocity has been shown as a function of non-dimensional diffuser passage height (y/Ym). The y/Ym = 0 shows hub and y/Ym = 1 shows casing. The graphs are plotted along various diffuser passage heights at different x/L = 0.1, 0.3, 0.5, 0.7 and 0.9 for the area ratio of 2.5 and inlet swirl angles of 0^0, 7.5^0, 12^0, 17^0 and 25^0.

Figure 1.1 shows that the flow is hub generated for no swirl condition. The flow is pushed towards the casing with the introduction of swirl. The peak is shifts towards the casing with introduction of swirl.
From Figures 1.1 to 1.5 It is clear that flow reversal occurs for the following cases

(i). x/L=0.6 and 0.9 at y/Ym =0.12 and 0.22 for 7.5^0 swirl angle and area ratio of 2.5 at hub
(ii). x/L=0.3, 0.5, 0.7 and 0.9 at y/Ym =0.04, 0.2, 0.28 and 0.33 for 25^0 swirl angle and area ratio of 2.5 at hub.

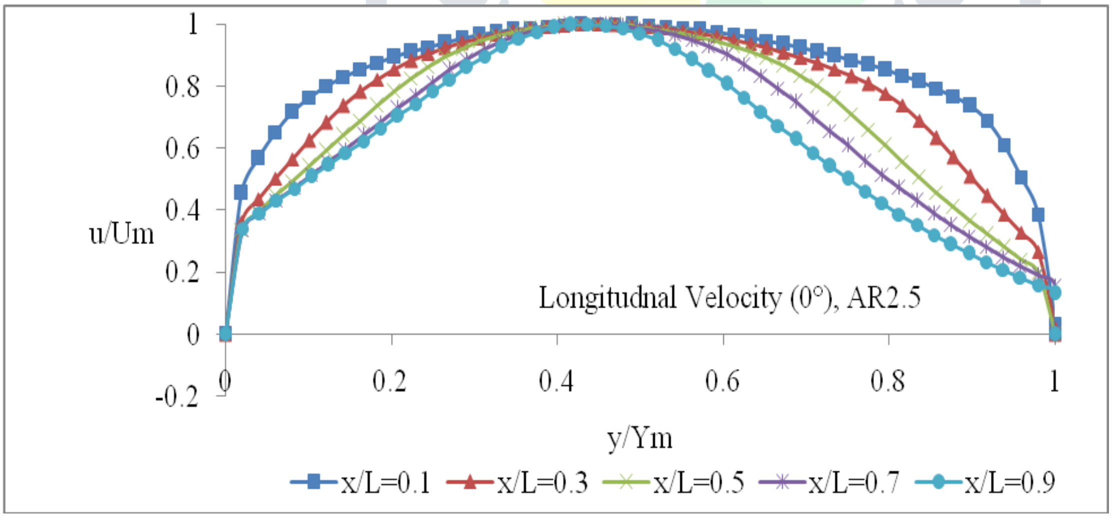

Figure-1.1:- Longitudinal Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α=0^0 at various traverses x/L = 0.1, 0.3, 0.5, 0.7 and 0.9.

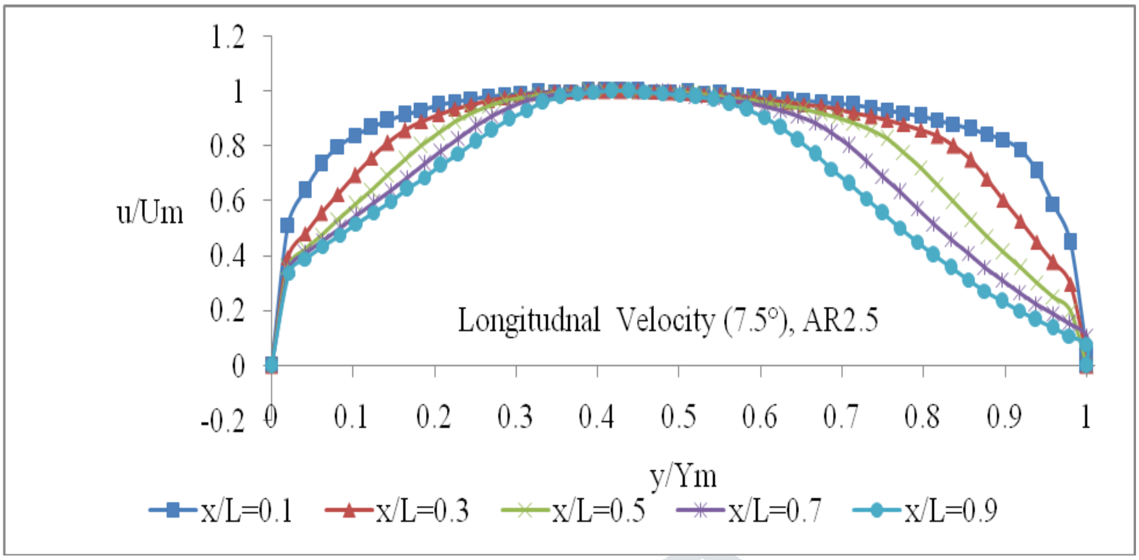

Figure-1.2:- Longitudinal Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α= 7.5^{0} at various traverses x/L = 0.1, 0.3, 05, 0.7, 0.9.

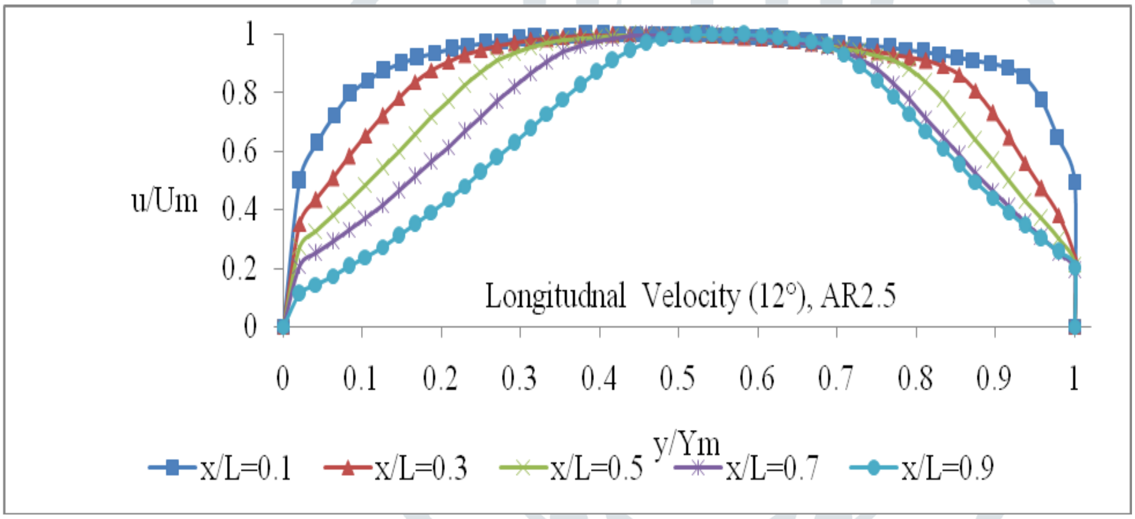

Figure-1.3:- Longitudinal Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α=12^{0} at various traverses x/L = 0.1, 0.3, 05, 0.7 and 0.9.

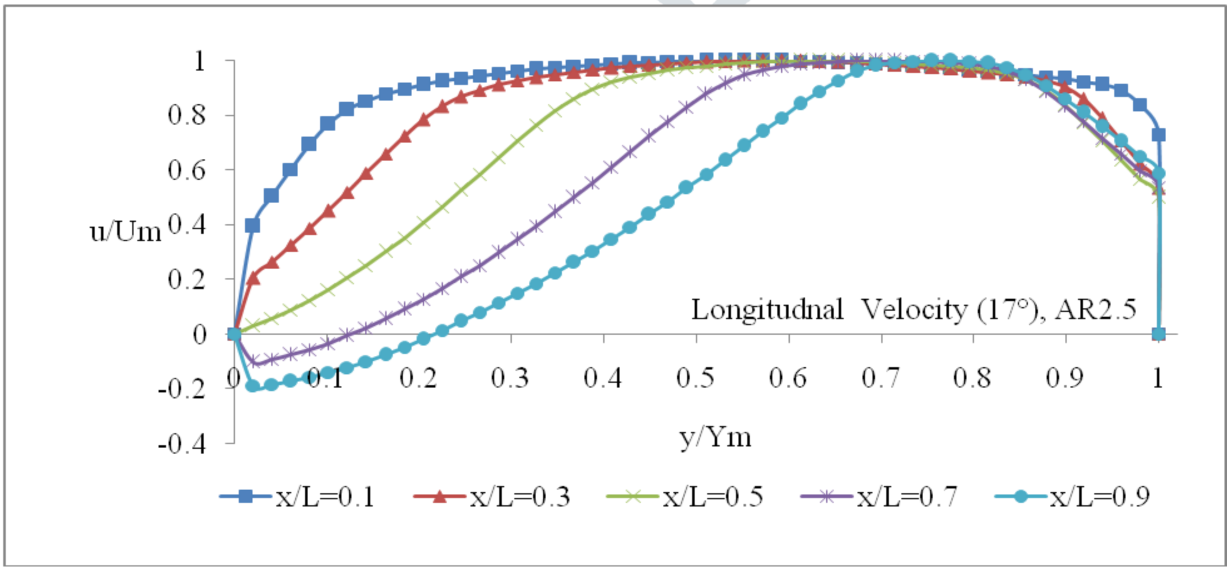

Figure-1.4:- Longitudinal Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α= 17^0 at various traverses x/L = 0.1, 0.3, 05, 0.7 and 0.9.

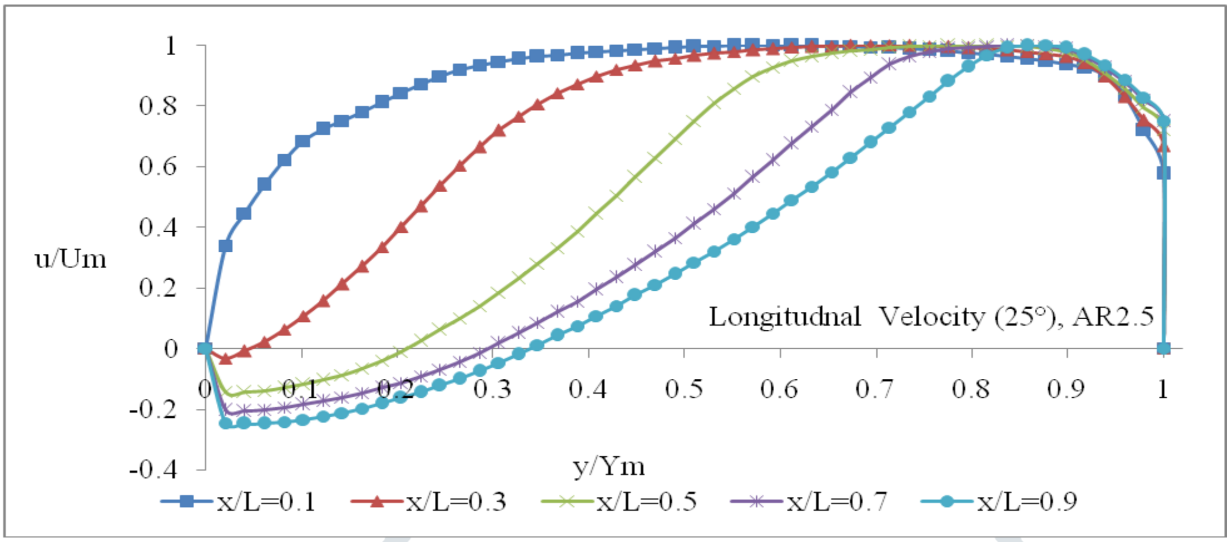

Figure-1.5:- Longitudinal Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α= 25^0 at various traverses x/L = 0.1, 0.3, 05, 0.7 and 0.9.

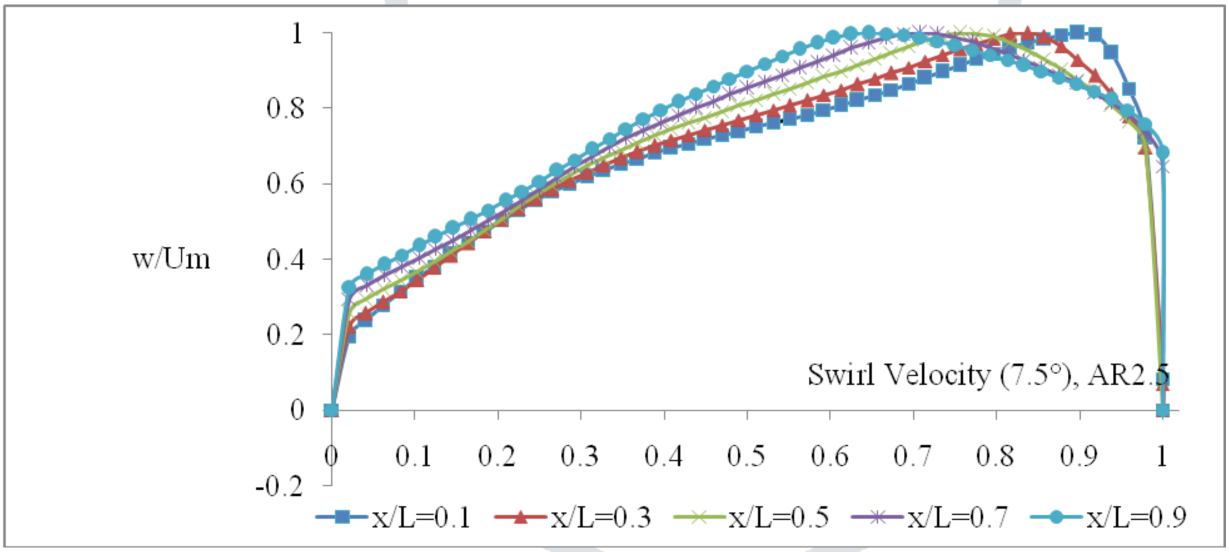

Figure-2.1:-Swirl Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α= 7.5^0 at various traverses x/L = 0.1, 0.3, 0.5, 0.7 and 0.9.

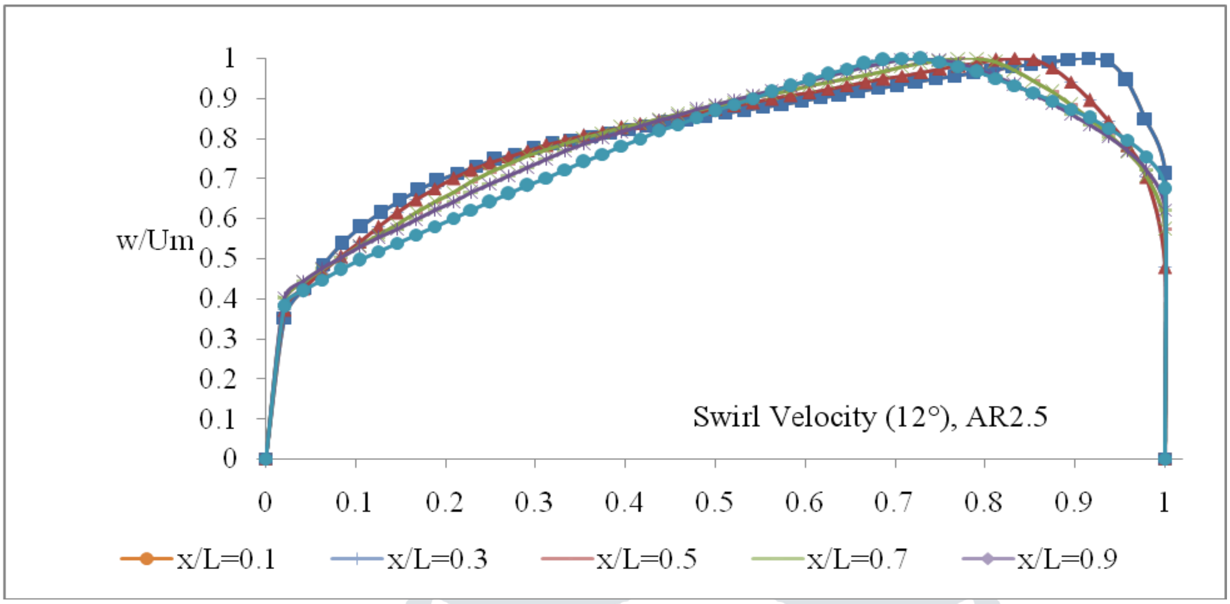

Figure-2.2:-Swirl Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α=12 ^{0} at various traverses x/L = 0.1, 0.3, 0.5, 0.7 and 0.9.

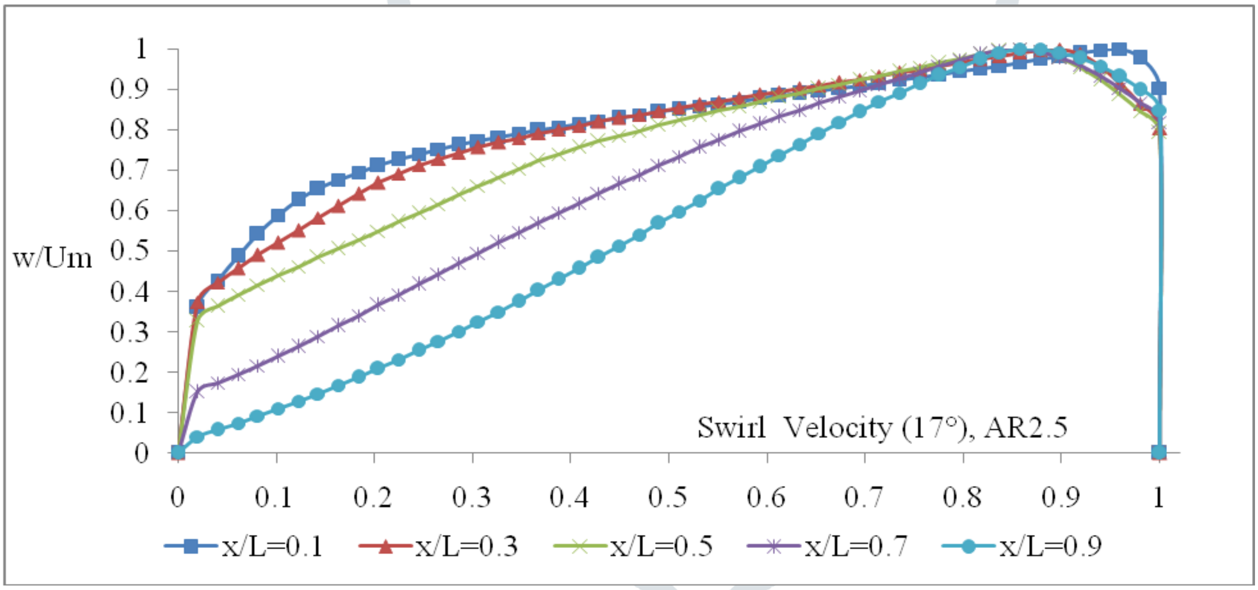

Figure-2.3:-Swirl Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α= 17^{0} at various traverses x/L = 0.1, 0.3, 0.5, 0.7 and 0.9.

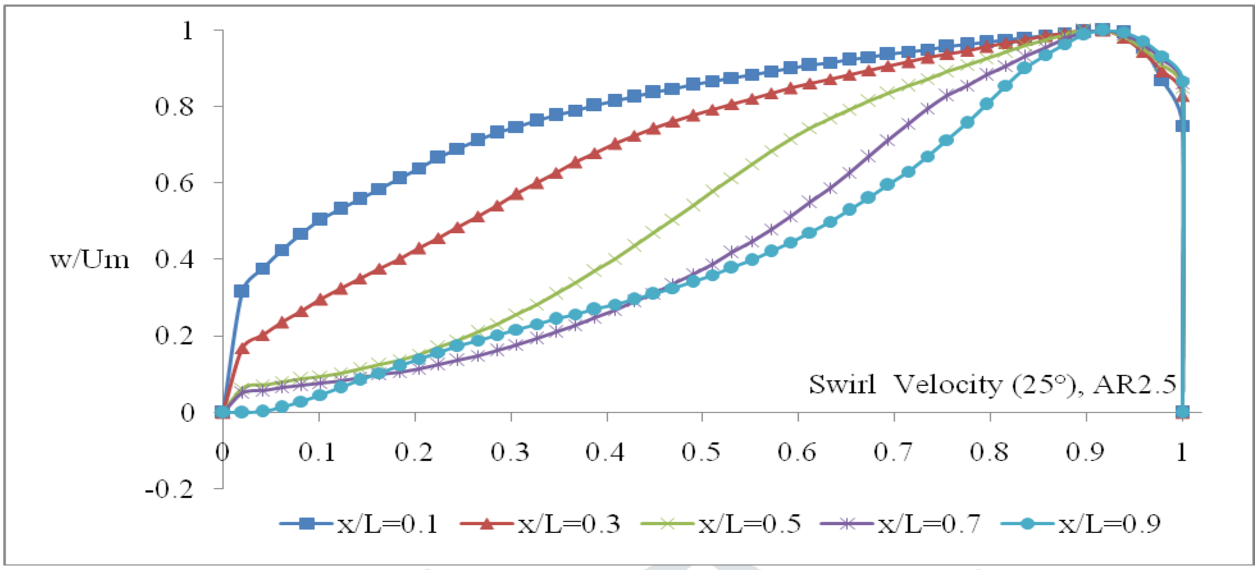

Figure-2.4:-Swirl Velocity variation verses diffuser passage height for AR 2.5 and inlet Swirl angle α= 25^0 at various traverses x/L = 0.1, 0.3, 0.5, 0.7 and 0.9.

CONCLUSIONS

. Following conclusions are drawn for area ratio of 2.5 at various inlet swirl angles:

- As the flow occurs towards downstream, the longitudinal velocity initially increases then decreases continuously in case of swirling or non-swirling flows.
- Velocity profiles have different shapes at different locations x/L of the diffuser passage due to the boundary layer formation.
- The maximum value of longitudinal velocity at any diffuser passage (y/Ym) is not at the center, but it is towards the hub for non-swirling flow, which shifts towards the casing side with the introduction of swirling flow at inlet.
- By the application of swirling flow, the flow is pushed towards casing wall thus making the flow stronger towards casing than hub wall.
- For flow without swirl, there was no flow reversal and separation is found on the hub.
- With the application of rotary flow at the inlet, the flow separation tends to move away from the casing wall and with high value of swirl at inlet, it completely removes the separation on the casing wall.

REFERENCES

Arora, B.B. 2014.Performance analysis of parallel hub diverging casing axial annular diffuser with 20^0 equivalent cone angle. Australian Journal of Mechanical Engineering, 12(2), 179-194.

Srinath, T.1968.An investigation of the effects of swirl on flow regimes and performances of annular diffuser with inner and outer cone angles. MASc Thesis, University of Waterloo, Canada..Hoadley,D.1970.Three Dimensional Turbulent Boundary Layers in an Annular Diffuser.PhD Thesis, Department of Engineering, University of Cainbridge, London.

.Coladipietro, R., Schneider J. H. & Shridhar, K. 1974. Effects of Inletet Flow Conditions on the Performance of EquiAngular Annular Diffusers.CSME Paper No. 7384.

Shaalan, M. R. A. & Shabaka I. M. M. 1975.An Experimental Investigation of the Swirling Flow Performance of an Annular Diffuser at Low Speed", ASME Paper No. 75WA/FE17.

Lohmann, R. P., Markowski, S. J. & Prookman, E. T. 1979 .Swirling Flow Through Annular Diffusers with Conical Walls. Journal of Fluids Engineering, 101, 224-229.

Sapre, R. N., Singh, S. N.,Agrawal, D. P. & Malhotra, R. C. 1987. Flow through Equiangular Wide Angle Annular Diffusers.15th NCFMFP, Srinagar, July.

Singh, S. N., Agarwal, D. P.,Sapre, R. N. & Malhotra, R. C. 1994.Effect of inlet swirl on the performance of wide angled diffusers. Indian journal of Engineering & Materials Sciences, 1, 63-69.

Singh.S.N., Seshadri.V. et al. 2006.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, 220, 129-143.

Mohan, R., Singh, S. N. & Agrawal, D. P. 1998.Optimum Inlet Swirl for Annular Diffuser Performance Using CFD.Indian Journal of Engineering and Materials Sciences. 5, 15-21.

Japikse, D.2000.Correlation of Annular Diffuser performance with Geometry, Swirl, and Blockage. Proceedings of the 11th Thermal and Fluids Analysis Workshop (TFAWS), Cleveland, Ohio, 21-25, 107-118

Kumar, Manoj, Arora, B.B. et al.2011 .Effect of inlet swirl on the flow behavior inside annular diffuser. International Journal of Dynamics of Fluids, 7(2), (181-188).

Ozturk, Tatar and Pinarbasi,Ali.2008. Flow Analysis in Centrifugal Compressors Vaneless Diffusers. Journal of Scientific and Industrial Research,67,348-354.

Rita J Schnipke,James G Rice and Ronald D. Flack. 2010.Finite Element Analysis of Viscous Flow in a Vaned Radial Diffuser. Department of Mechanical and Aerospace Engineering,University of Virginia. . ***(For Research paper citation).***

Nabavi , Majid.2010.Three dimensional asymmetric flow through a planer diffuser. Journal of Heat and Mass Transfer, 37 (2010), 17-30.

Manoj Kumar, B.B.Arora, Subhashish Maji and S Maji.2012 .Effect of Area Ratio and Inlet Swirl on the Performance of Annular Diffuser , International Journal of Applied Engineering Research , 713 ,1493-1506.

Dr.Basarat salim.Effect of geometric parameters on the performance of wide angle diffusers.2013. International Journal of Innovative research in science, Engineering and Technology, 2(9), 4178-4191.

Keerthana, R. and Rani, G. 2012. Flow analysis of annular diffusers. International Journal of Engineering Research and applications, 2(3), 2348-2351.

.Stefano Ubertini, Umberto Desideri.2000.Experimental Performance Analysis of an Annular Diffuser with and without Struts. Experimental Thermal and Fluid Science, 22,183-195.

Manoj Kumar Gopaliya, Mahesh Kumar, Shailendra Kumar and Shiv Manjar Gopaliya.2007.Analysis of Performance of S Shaped Diffuser with Offset. Aerospace Science and Technology, 130-135.

Tusi,Y.Y. and Wang, H.W.1995.Calculation of laminar separated flow in symmetric Two Dimensional diffuser. Journal of Fluid Engineering, 117, 612-616.

Manoj Kumar Gopaliya, Chaudhary, K.K.2010 .CFD Analysis of Y-Shaped Diffuser with Combined Horizontal and Vertical offsets. Aerospace Science and Technology, 14,338-347.

Alysson Kennerly Colaciti, Luis Migual Valdes Loez. Numerical Simulation of a Radial Diffuser Turbulent Airflow.2007. Applied Mathematics and Computations.

Ligrani, P.M. and Rouster,M. et al.1983.Measurements in the Vaneless Diffuser of a Radial Flow Journal of Heat and Fluid Flow, 4,103-106.

Agarwal,D.P., and Yahya, S.M..1981. Velocity distribution in blade to blade plane of a vaned diffuser. Int.Journal of .Mech Sci. 23(1981),359-366.