STUDY AND ANALYSIS OF PARALLEL HUB AND DIVERGING CASING AXIAL ANNULAR DIFFUSER WITH AREA RATIOS OF 4

[1]DEVESH KUMAR, [2]B.B.ARORA
[1]Research Schollar [2] Professor
[1]Department of Mechanical Engineering
[1]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 with and without swirl. The equivalent cone angle and area ratio have been taken as 20^0 and 4.The swirl angle has been 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. Longitudinal velocity and swirl velocity profiles have been checked along the different diffuser passage heights x0.1, x0.2, x0.4, x0.6, x0.8 and x0.9 at different traverses along the length. In this present work CFD approach has been used to find the results by using RNG $k-\varepsilon$ turbulence model. The results obtained from CFD have been compared with experimental results. Finally parametric investigations have been carried out by varying swirl angle.

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

INTRODUCTION

A diffuser is a device which decelerates the flow and to regain static pressure or device which increases the pressure energy of a flowing 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 over and around a hub or a central shaft together its bearing and support. 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 one parameter whole setup has to be changed. Performance of diffuser depends upon geometrical and dynamical parameters. Geometrical parameters are inlet length, size of duct, area ratio of the diffuser, angle of expansion, 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. It is very difficult to conduct experimental Work with annular diffuser due to connectivity of different parameters. If we change one parameter whole setup is replaced with new one. Which involves a lot of cost and it is advantageous to carry out numerical work such as Computational Fluid Dynamics.

In annular diffuser maximum pressure recovery is achieved within shortest possible length due to presence of inner surface to guide the flow outwards by varying the different parameters.

Sovran & Klomp (2), Shrinath (3), Hoadley (4), Colodipietro et al (5), Shaalan & Shabaka (6), Kumar (7), Lohmann et al (8), Sapre et al (9), Agrawal et al (10), Singh et al (11), Kochevsky (12), Mohan et al (13), Japikse (14) and Yeung & Parkinson (15) showed that diffuser performance increases with introduction of swirling flow. Pressure recovery of diffuser increases up to a certain point after that it deteriorated. The performance of an annular diffuser also depends on geometrical and flow parameters. 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 (16), Stevens (17), Singh et al (18) developed an annular diffuser without flow separation, however they become successful up to some extent. **Manoj Kumar, B.B.Arora [19]** analyzed the flow in annular diffuser by using CFD technique and they find the effect of swirl on the flow behavior inside annular diffuser and found that the flow was shifted towards the casing with the introduction of swirl flow. They also calculated the effect of swirl on pressure recovery coefficient. **Ozturk Takar, Ali Pinarbasi [20]** analyzed the flow in centrifugal compressor vaneless diffuser. They used the computational Fluid Dynamics approach to analyze the flow and calculated velocity, Pressure and turbulent kinetic energy at different hub sections with the help of CFD. **Rita J Schnipke et al [20]** analyzed a vaned diffuser by applying Finite element analysis. In this method a model geometry and meshing was made 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. The flow was incompressible and diffuser with gradual expansion was used. Here the numerical approach was used to analyze the flow and finally compared with experimental work. In this paper the effect of divergence angle, Reynolds number and aspect ratio on the flow asymmetry were analyzed. **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 calculated the mean velocity, static pressure and total pressure. **Dr. Basarat salim [23]** analyzed the wide angle diffusers experimentally. The effect of area ratio and diffuser divergence angle was 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 was used to find out the results. 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.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=2.74×10^5.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 this 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. Chithambaran et al (1984) and Buice & Eaton (1997) showed that experimental analysis of annular diffuser is not economical 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) using Fluent software has been used for the detailed flow analysis in axial annular diffuser with parallel hub and diverging casing. Equivalent cone angle and area ratio have been taken as 20° and 4. The swirl angle has been varied from 0^0 to 25^0 at the inlet of diffuser. Experimental velocity profiles were obtained with the axial annular diffuser having hub parallel and casing diverging with area ratio of 2.01, and equivalent cone angle of 20.02° from reference [1]. The Computational Fluid Dynamics analysis of the annular axial diffuser with same geometry and flow conditions have been conducted by taking different turbulence models. Due to closeness with experimental results RNG $k-\varepsilon$ turbulence model was adopted. [1]. Finally parametric investigations have been performed by varying swirl angle to calculate the performance of the diffuser. In the paper we calculated the longitudinal velocity and swirl velocity in non-dimensional form at different diffuser passage heights at x0.1, x0.2, x0.4, x0.6, x0.8 x0.9. Then flow development, flow reversals and flow separation through the diffuser walls have been visualized.

CFD MODELLING

Annular diffuser geometry was drawn with proper meshing scheme with the help of ANSYS- 15.0. Here RNG *k*- ε , turbulent model has been used to analyze the flow with different area ratios and swirl intensities for annular diffuser. RNG *k*- ε turbulence model has been used as RNG *k*- ε turbulence turbulent model gives better results closed to experimental results from reference [1]. The results have been compared and validated with the experimental results. 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% and hydraulic diameter as 12.96 cm. 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.

Here k- ε turbulent model was used 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 *k*- turbulence model with moderate mesh size of 0.07 cm from reference [1]. The RNG-based k- ε turbulence model (Choudhury, 1993) is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called RNG methods.

Problem Description: - A 2-D model of annular axial diffuser and meshing is generated by using Gambit software. Analysis of diffuser is done by using FLUENT software.

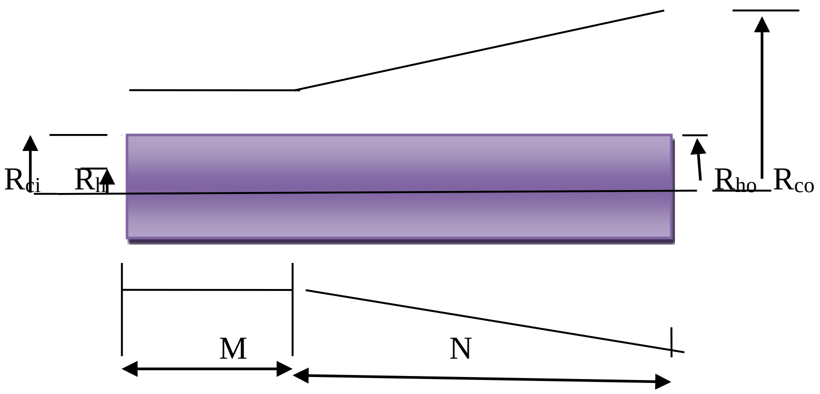

Figure1:- Geometry of the diffuser with parallel hub and diverging casing.

Table1:- Geometric Details of Annular axial Diffuser with area ratio of 4 and Cone angle of 20^0.

S.N.	Details of Parameters	Area Ratio(4)
1.	Hub radius at the inlet (R_{hi})	3.8 cm
2.	Hub radius at the outlet (R_{ho})	3.8 cm
3.	Casing radius at the inlet (R_{ci})	7.75 cm
4.	Casing radius at the outlet (R_{co})	14.3007cm
5.	Diffuser Diverging Length (N)	38.3063 cm
6.	Diffuser inlet Length (M)	3.95 cm
7.	Equivalent Cone angle (2θ)	20^0

Here we take six diffuser passages along x0.1, x0.2, x0.4, x0.6, x0.8 and x0.9 in radial directions at x/L =10%,20%,30%, 40%, 60%, 80% and 90% of distance from the inlet of the annular diffuser along the length.

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 :

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

This is the general equation of mass conservation and can be applied for compressible and incompressible fluid. The source S_m is the mass added to the continuous phase from the dispersed second phase (eg, due to vaporization of liquid droplets) and any user defined sources.

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

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho v_x)}{\partial x} + \frac{\partial(\rho v_y)}{\partial x} = S_m \qquad (2)$$

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:

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

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 :

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

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

$$\frac{\partial(\rho v_x)}{\partial x} + \frac{1}{r}\frac{\partial(r\rho v_x v_x)}{\partial x} + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_x) = -\frac{\partial p}{\partial x} + \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left\{2\frac{\partial v_x}{\partial x} - \frac{2}{3}(\nabla.\vec{v})\right\}\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial v_x}{\partial r} + \frac{\partial v_r}{\partial x}\right)\right] + F_x \quad \ldots\ldots\ldots\ldots \quad (5)$$

…

$$\frac{\partial}{\partial t}(\rho v_r) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_r) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_r) = -\frac{\partial p}{\partial r} + \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(\frac{\partial v_r}{\partial x} + \frac{\partial v_x}{\partial r}\right)\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(2\frac{\partial v_r}{\partial r} - \frac{2}{3}(\nabla.\vec{v})\right] - 2\mu\frac{v_r}{r^2} + \frac{2}{3}\frac{\mu}{r}(\nabla.\vec{v}) + \rho\frac{{v_z}^2}{r} + F_r \quad \ldots\ldots (6)$$

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

Where v_z is the swirl velocity

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

$$\frac{\partial}{\partial t}(\rho v_{z)} + \frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_z) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_z = \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\frac{\partial v_z}{\partial x}\right] + \frac{1}{r^2}\frac{\partial}{\partial r}\left[r^3\mu\frac{\partial}{\partial r}\left(\frac{v_z}{r}\right)\right] - \rho\frac{v_x v_z}{r} \quad \ldots\ldots\ldots\ldots (7)$$

TURBULENCE MODELLING

Turbulent flows are identified by fluctuating velocity components. These components combined with momentum, energy and species concentration and produce fluctuations in these quantities. Since these fluctuations can be of small scale and high frequency, they are also 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

Since no single turbulence model is universally accepted as being best for any type of problems. The choice of turbulent model will depend on accuracy required, availability of computational resources and the time available for the simulation. Here we used RNG k-ε model to analyze the flow because it gives good results near to experimental results [1].

PERFORMANCE EVALUATION OF ANNULAR AXIAL DIFFUSER

To evaluate the performance of diffuser certain performance parameters are defined which are as follows:
(a).Static Pressure recovery coefficient:-It represents the extent by which kinetic energy has been converted in to pressure energy due to diffusing action.

$$C_p = \frac{p_2 - p_1}{\frac{1}{2}\rho v_{av1}^2}$$

(b) Diffuser Effectiveness: It represents the pressure recovery capacity of diffuser in comparison with ideal diffuser.

$$\eta = \frac{C_p}{C_{pi}}$$

TURBULENCE MODEL VALIDATION

Turbulence Model selection is a typical process as one needs to handle number of known and unknown parameters to explain the result which should be close to the experimental results. The following models were investigated by [1] in the pre analysis are given below:

- k-ε models – standard, RNG, realizable
- Reynolds stress model
- Spalart-Allmaras model – vorticity-based production, strain/vorticity-based production
- k-ω Models – standard, shear -stress transport.

These models were tested with inlet velocity profile which was obtained experimentally for a fully developed flow through an annular diffuser. The results for longitudinal velocity profiles for different turbulent models are shown in Figure 2 respectively [1]. It is clear from the Figure-2 that the RNG k-ε model is very close to the experimental results .So we have used this model for further parametric investigations.

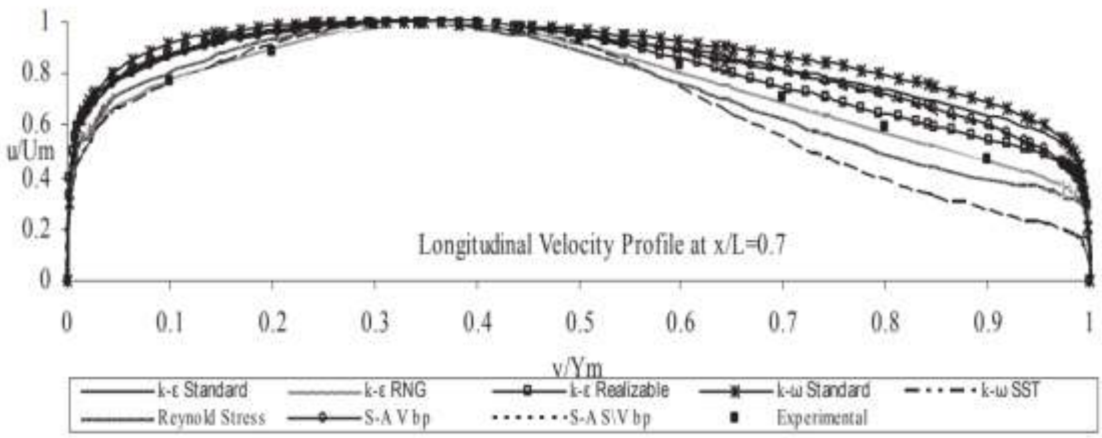

Figure 2: Validation of turbulence model with experimental results at x/L = 0.7. From reference [1]

RESULTS AND DISCUSSIONS:-

Figures 3.1, 3.2, 3.3, 3.4 and 3.5 show the Non dimensional longitudinal and swirl velocity variations for the area ratios of 4 respectively. The velocity variations are shown for inlet swirl angles of 0°, 7.5°, 12°, 17° and 25°. Figure 4.1, 4.2, 4.3, 4.4 and 4.5 represent the swirl velocity 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 of the particular traverse (y/Ym). The y/Ym =0 is location of the hub and for y/Ym = 1 is the location of casing. 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 the area ratio 0f 2 and inlet swirl angles of 0^0, 7.5^0, 12^0, 17^0 and 25^0.

From Figure-3.1, 3.2, 3.3, 3.4 and 3.5 it is clear that the flow reversal occurs for x/L=0.6, 0.8 and 0.9 at y/Ym =0.94, 0.90 and 0.88 for 0^0 swirl angle and area ratio of 4 at casing. It also occurs for x/L=0.6,0.8 and 0.9 at y/Ym =0.88, 0.84 n 0.82 for 7.5^0 swirl angle and area ratio of 4 at casing. It also occurs for X/L=0.4,0.6, 0.8 and 0.9 at y/Ym =0.16, 0.32, 0.38 and 0.42 for 12^0 swirl angle and area ratio of 4 at Hub.It also occurs for X/L=0.4,0.6, 0.8 and 0.9 at y/Ym =0.16, 0.32, 0.38 and 0.42 for 17^0 swirl angle and area ratio of 4 at Hub. It also occurs for X/L=0.2,0.4,0.6, 0.8 and 0.9 at y/Ym =0.16, 0.34, 0.42, 0.48 and0.50 for 25^0 swirl angle and area ratio of 4 at Hub.

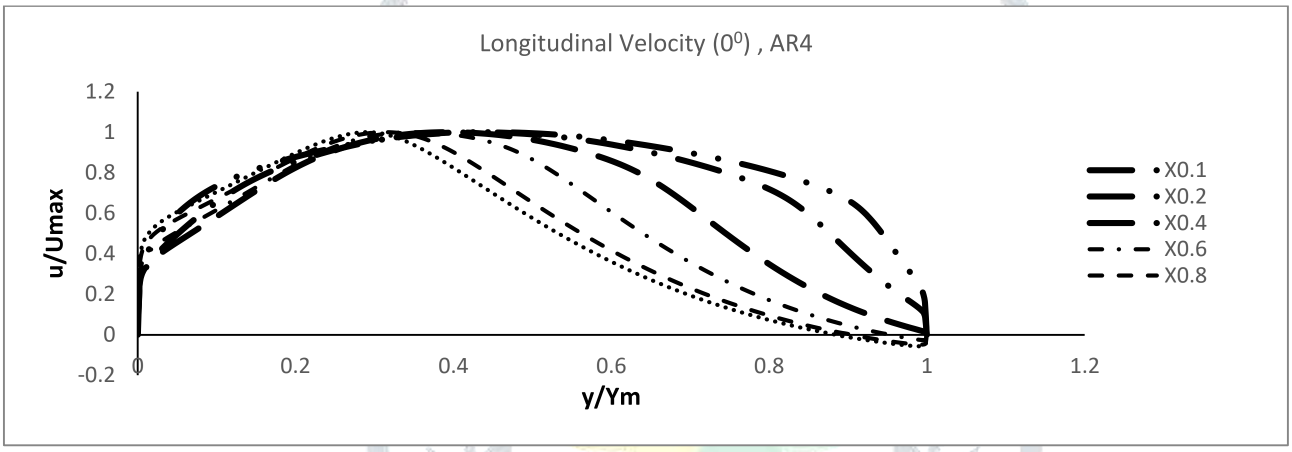

Figure-3.1:- Longitudinal Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle α= 0^0 at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

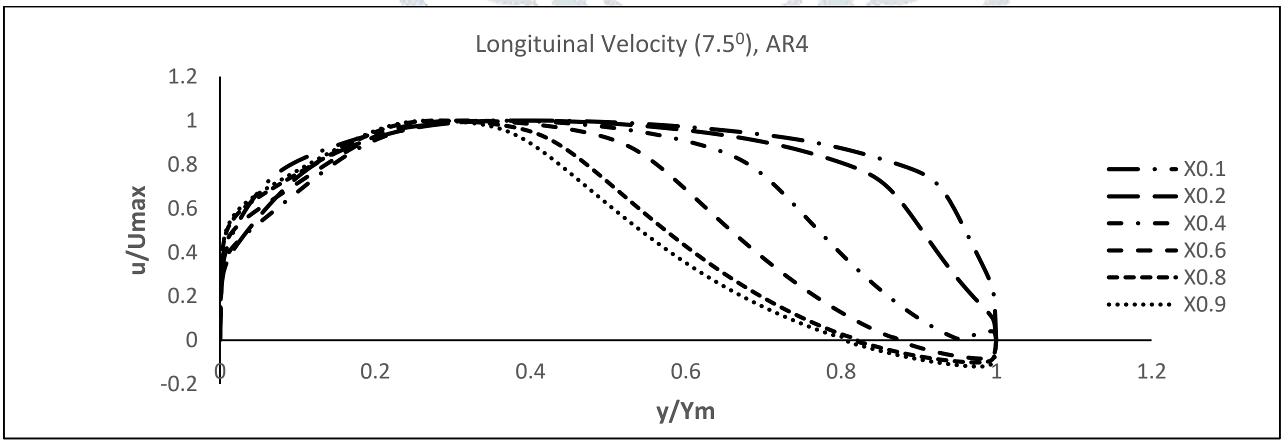

Figure-3.2:- Longitudinal Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle α= 7.5^0 at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

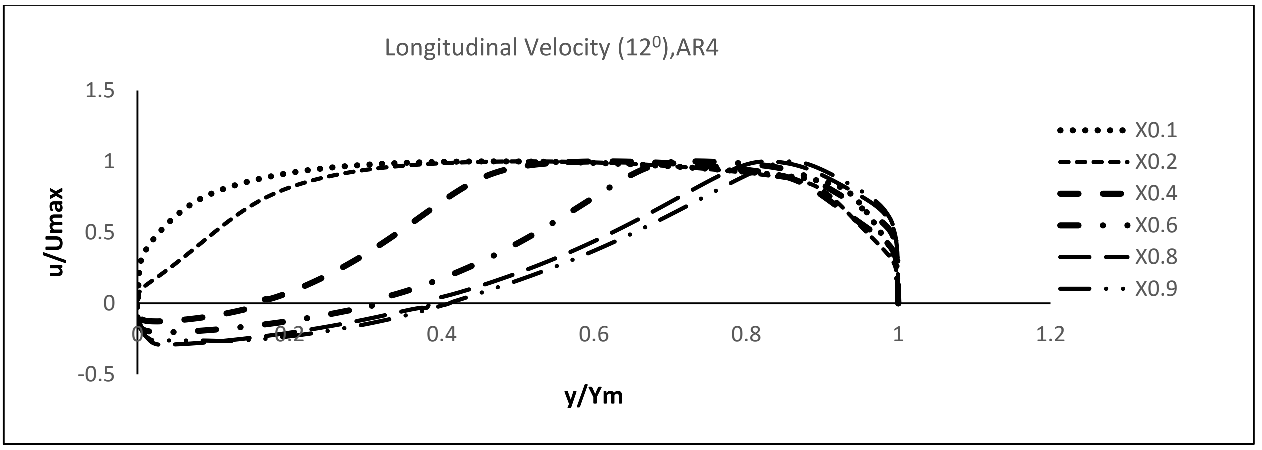

Figure-3.3:- Longitudinal Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle $\alpha = 12^0$ at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

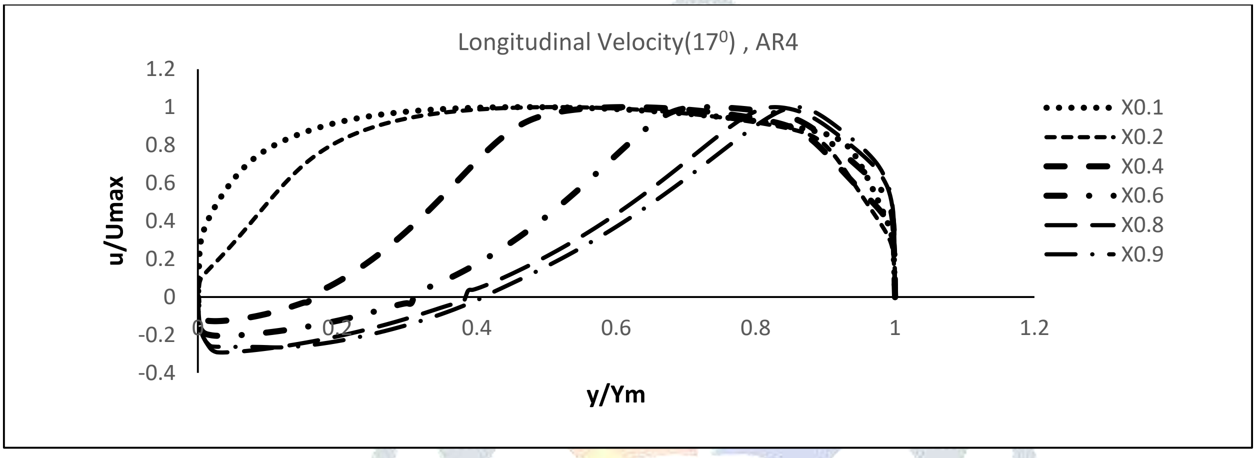

Figure-3.4:- Longitudinal Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle $\alpha = 17^0$ at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

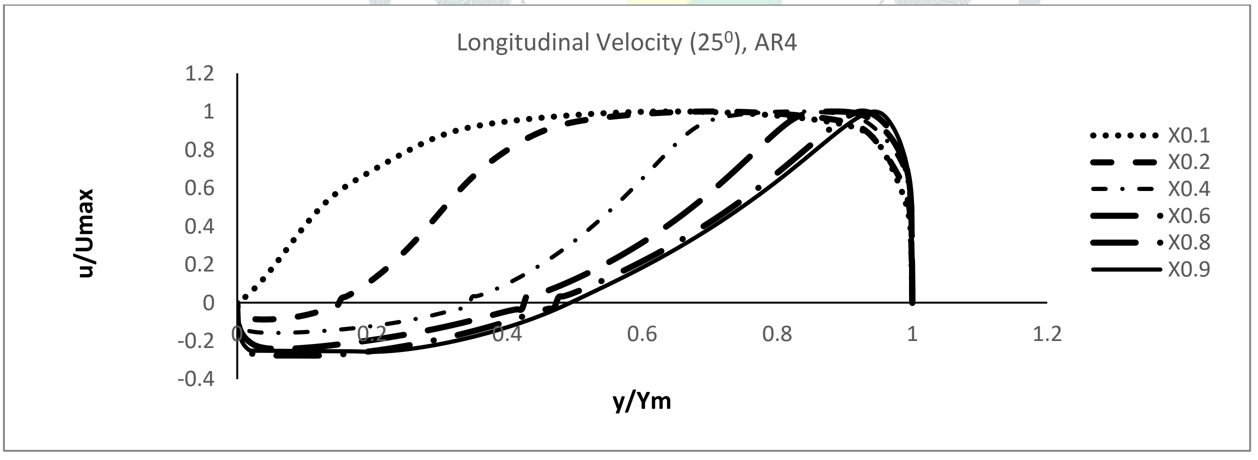

Figure-3.5:- Longitudinal Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle $\alpha = 25^0$ at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

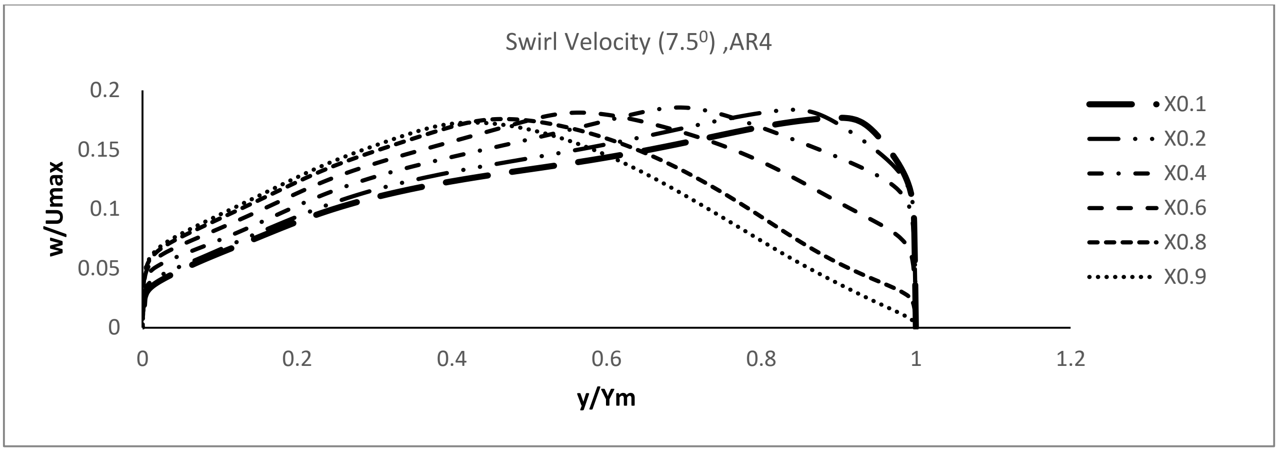

Figure-4.1:- Swirl Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle α= 7.5^0 at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

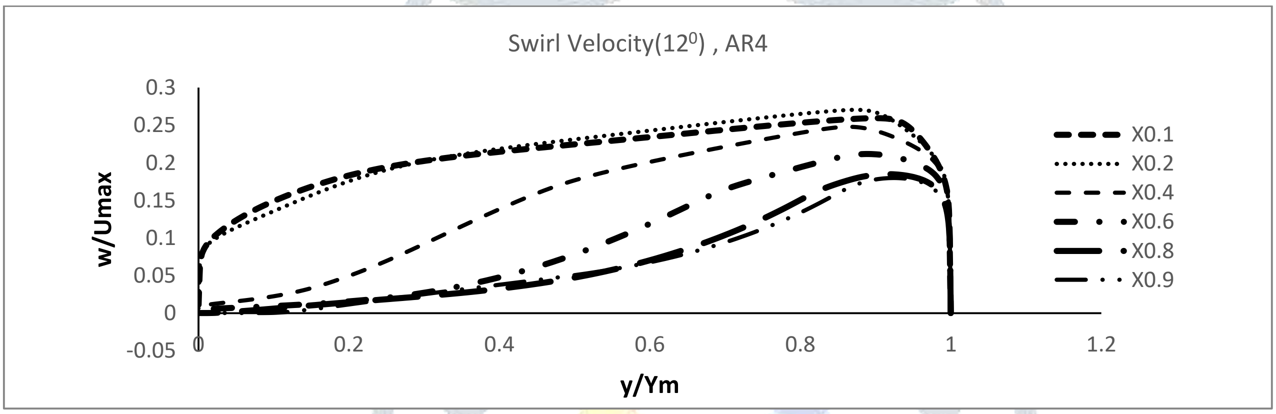

Figure-4.2:- Swirl Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle α= 12^0 at Various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

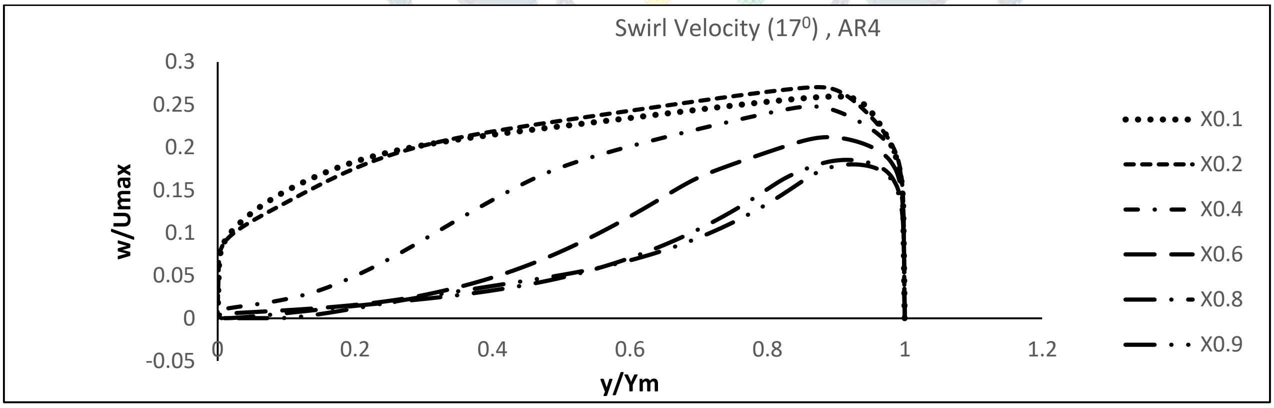

Figure-4.3:- Swirl Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle α= 17^0 at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

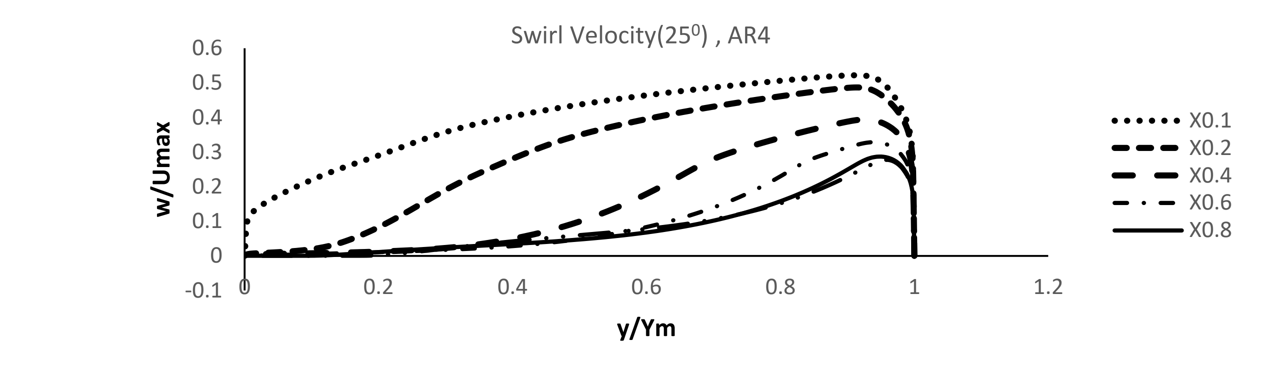

Figure-4.4:- Swirl Velocity variation with diffuser passage height for AR 4 and inlet Swirl angle α= 25^0 at various traverses x/L = 0.1, 0.2, 04, 0.6, 0.8 and 0.9.

CONCLUSIONS

Here Validated CFD RNG k-ε model was used to show the performance analysis of the annular axial diffuser. Following conclusions are drawn for area ratio of 4 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.
- For flow without swirl, there was no flow reversal and separation is found on the hub wall.
- By the application of swirling flow, the flow is pushed towards casing wall thus making the flow stronger towards casing than hub wall.
- 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.

(20).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.