Computational Modeling and Velocity Prediction of a Diffuser Using CFD and Regression Analysis

Gaurav Chandra

Assistant Engineer, Public Works Department Uttrakhand, Govt. Of Uttrakhand, Bazpur,Udham Singh Nagar, India

Shuvam Mohanty

Research Scholar, Department of Mechanical Engineering, Amity University, Gurgaon, Haryana

Abstract – The present study involves the regression analysis for the prediction of velocity in the diffuser by using the experimental analysis. The annular diffuser considered in the present case has both the hub and casings are diverging with equal angles and hub angle keeping constant as 10°. The characteristic quantities such as static pressure distribution at hub and casing walls, velocity profiles at various sections and flow patterns have been presented for studying. The effect of change in swirl angle and Y/Ym in diffuser is studied in the present study. The cases that were studied are 0°, 7.5°, 12°, 17°, and 25° at various sections of casing.

Index Terms - Diffuser, Regression Analysis, Velocity Profiles, Annular Diffuser, Hub.

1. INTRODUCTION

Diffusers are one of the standard challenges in fluid mechanics. The task of a diffuser is to decelerate the flow and to regain total pressure. It is more difficult to arrange for an efficient deceleration of flow than it is to obtain an efficient acceleration. Diffusers play a vital role in many fluid machines to convert kinetic energy into pressure energy. The efficiency of this conversion process is important as it affects the overall performance of the machine. The pressure recovery, which is the measure of performance of diffusers, depends on many geometrical and dynamical parameters. Some geometrical parameters that govern the performance of a diffuser are inlet length and size of the duct, area ratio of the diffuser, angle of expansion, length of the diffuser, shape of the exit duct with free or submerged discharge conditions, etc. The dynamical parameters are inlet velocity profile, boundary layer parameters, Reynolds number, Mach number and so on. In the present work, two parameters namely inlet velocity profile and the geometry of the diffuser were selected in order to study their effects on the flow structure and performance of annular diffusers. Swirling flow in the diffuser plays an important role in controlling the flow separation and therefore the performance of the diffuser. Diffusers are extensively used in centrifugal compressors, axial flow compressors, ram jets, combustion chambers, inlet portions of jet engines etc. The

energy transfer in these turbo machineries involves the exchange of significant levels of kinetic energy in order to accomplish the intended purpose. They reported that performance of annular diffuser having parallel diverging hub and casing was improved on the introduction of swirl. The optimum swirl angle was between 20°and 30°. They found that further increase in swirl degraded the performance of diffusers. Moreover, the present Thesis addresses the aforementioned issues and makes a systematic study of the influence of the inlet velocity component profiles (addressing the effect of the shape and the ratio of (averaged) radial to axial velocity) and outlet boundary conditions on the flow characteristics in the Annular Diffuser. Diffusers are extensively used in centrifugal compressors, axial flow compressors, ram jets, combustion chambers, inlet portions of jet engines etc. The energy transfer in these turbo machineries involves the exchange of significant levels of kinetic energy in order to accomplish the intended purpose. As a consequence, very large levels of residual kinetic energy frequently accompany the work input and work extraction processes, sometime as much as 50% of the total energy transferred. A small change in pressure recovery can increases the efficiency significantly. Therefore diffusers are absolutely essential for good turbo machinery performance.

2. RELATED WORK

Sovran and Klomp (1967) tested over one hundred different geometries, nearly all of which had conically diverging center bodies with an inlet radius ratio [Ri/Ro] of 0.55 to 0.70. The tests were carried out with a thin inlet boundary layer and the diffusers have free discharge. The tests were present as Contours of pressure recovery plotted against area ratio and non-dimensional length Howard et al (1967) also tested symmetrical annular diffusers with center bodies of uniform diameter, using fully developed flow at inlet. The limits of the various flow regimes and the optimum performance lines were established. Besides it, some other researchers also contributed in the field of annular

Diffuser and concluded various important results. Much of the extent data covering the annular diffusers was done in the experimental laboratory to uncover some of the unusual performance characteristics of annular diffusers. But there are still some important unresolved questions. The reason for it is that the numbers of independent variables are large for annular diffusers. In the annular diffuser the flow take place between two boundary surfaces which can varies independently This chapter involves a systematic study of different geometric and flow parameters which influence the overall and internal performance of annular diffusers. In this regard the available literature has been examined with a view to make comments on the state of the art and to recognize the scope of further research on the subject. Hoadley D,et.al, (1969) reported thin inlet boundary layers tends to be beneficial to high diffuser recovery and those longer diffusers necessary to achieve high levels of recovery as the inlet boundary thickness increases as stated by. Thayer E B, (1971) studied the Mach number at the inlet to the diffuser was thought to be important at values as low as approximately 0.7 and performance to fall off past this point. No significance on Mach number develops at throat for Mach numbers of less than 1.0. Arora B.B., Pathak B.D. (2005) describes the specification of a wide variety of geometric parameters is essential before the performance of diffuser is given. In this section, the various geometric parameters and there influence on diffuser performance is reviewed. The study by Henry and Wood (1958]) is useful to understand the subsonic annular diffuser. Two diffusers with area ratio 2.1 and divergence of 5° and 10° were tested at various Mach number. It found by this study that most of data clusters around a line of constant Effectiveness. It is also observed that the inner wall is being starved of fluid. If a higher divergence had been used, then one might anticipate stall on the inner surface. An extensive study is carried out by Kmonicek and Hibs (1974) in which, the pressure loss coefficient is found out on the basis of the work of compression required to meet the static pressure rise, the results are very interesting but difficult to understand due to use of unconventional terminology. Howard et al. (1967) produced the first widely used annular diffuser maps for channel diffusers. Sovran and Klomp (1967) conducted a large number of performance measurements which spanned a broad selection of geometric types of diffusers. The map is only a broad representation of the bulk of configurations tested in the vicinity of their best performance areas. The poorer diffusers are not well defined by the map. These maps also show optimal diffuser geometrics under different conditions and two optimum lines are established. Ishikawa and Nakamura (1989) reported an extensive study of diffusers which, although annular, begin with a circular cross section.

The author found that the performance of the diffuser differed significantly depending on whether it is parallel or diverging for L/r_1 greater than about 2. When both types have the same non dimensional length and area ratio, the parallel diffuser has the higher CP.

3. PORPOSED CFD MODELLING

FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes. FLUENT also refine or coarsen grid based on the flow solution. The FLUENT solver has the following modeling capabilities:

- 2D planar, 2D axisymmetric, 2D axisymmetric with swirl (rotationally symmetric), and 3D flows
- Quadrilateral, triangular, hexahedral (brick), tetrahedral, prism (wedge), pyramid, and mixed element meshes
- Steady-state or transient flows
- Incompressible or compressible flows, including all speed regimes (low subsonic, transonic, supersonic, and hypersonic flows)
- Inviscid, laminar, and turbulent flows
- Newtonian or non-Newtonian flows
- Heat transfer, including forced, natural, and mixed convection, conjugate (solid/fluid) heat transfer, and radiation
- Lumped parameter models for fans, pumps, radiators, and heat exchangers
- Inertial (stationary) or non-inertial (rotating or accelerating) reference frames
- Multiple reference frame (MRF) and sliding mesh options for modeling multiple moving frames
- Mixing-plane model for modeling rotor-stator interactions, torque converters, and similar turbo machinery applications with options for mass conservation.

3.1. Linearization: Implicit vs. Explicit

In both the segregated and coupled solution methods the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution. The manner in which the governing equations are linearized may take an "implicit" or "explicit" form with respect to the dependent variable (or set of variables) of interest. By implicit or explicit

We mean the following:

- Implicit: For a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring

cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

- Explicit: For a given variable, the unknown value in each cell is computed using a relation that includes only existing values. Therefore each unknown will appear in only one equation in the system and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities.

3.2. Discretization

The governing equations are converted into algebraic equations with the help of the finite volume technique that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis.

Discretization of the governing equations can be illustrated most easily by considering the steady-state conservation equation for transport of a scalar quantity φ. This is demonstrated by the following equation written in integral form for an arbitrary control volume V as follows:

$$\oint \rho\phi\,\vec{v}\cdot d\vec{A} = \oint \Gamma_\phi \nabla\phi\cdot d\vec{A} + \int_V S_\phi\,dV$$

Where;

ρ = density
v = velocity vector A = surface area vector
Γ_φ = diffusion coefficient for φ
∇_φ = gradient of φ
S_φ = source of φ per unit volume

Above equation is applied to each control volume, or cell, in the computational domain. Discretization of Equation on a given cell yields

$$\sum_f^{N_{faces}} \rho_f \vec{v}_f \phi_f \cdot \vec{A}_f = \sum_f^{N_{faces}} \Gamma_\phi (\nabla\phi)_n \cdot \vec{A}_f + S_\phi V$$

Where;

N_{faces} = number of faces enclosing cell
φ_f = value of φ convection through face f

$\rho_f\,v_f\,A_f$ = mass flux through the face
A_f = area of face f, $_|A$
$(\nabla\varphi)_n$ = magnitude of $\nabla\varphi$ normal to face f
V = cell volume
 The equations take the same general form as the one given

above and apply readily to multi-dimensional, unstructured meshes composed of arbitrary polyhedral, the discrete values of the scalar φ at the cell centers. However, face values φ_f is required for the convection terms in Equation and must be interpolated from the cell center values. This is accomplished using an upwind scheme. Up winding means that the face value φf is derived from quantities in the cell upstream, or "upwind," relative to the direction of the normal velocity v_n.

4. IMPLEMENTATION OF BOUNDARY CONDITION

Each CV provides one algebraic equation. Volume integrals are calculated for every control volume, but flux through CV faces coinciding with the domain boundary requires special treatment. These boundary fluxes must be known, or be expressed as a combination of interior values and boundary data. Two types of boundary conditions need to be specified.

Dirchlet conditions where variable values are given at boundary nodes.

Neuman conditions where the boundary fluxes are incorporated at the boundary.

4.1. Inlet boundary condition

The present analysis involves the velocity with and without swirl. The incorporation of velocity without swirl can be specified by any one of the velocity specification methods described in FLUENT. Turbulence intensity is specified as

$$I = 0.16(Re_{DH})^{-1/8} \times 100$$

The inlet based on the Reynolds number with respect to equivalent flow diameter.

Where, Re_{DH} is the Reynolds number based on the hydraulic diameter.

For specifying the velocity in case of flow with swirl, tangential component of velocity will also have to be defined along with axial component. Velocity components are calculated on the basis of inlet swirl angle. In the present case swirl angle of 5, 10, 15, 20, 25 degrees are considered. Inlet velocity of 50 m/s with flat profile is considered for both the cases.

4.2. Outlet boundary condition

Atmospheric pressure condition is applied at the outlet boundary condition and set a "back flow" conditions is also specified if the flow reverses direction at the pressure outlet boundary during the solution process. In the "back flow" condition turbulence intensity is specified based on the equivalent flow diameter.

4.3. Wall boundary condition

Wall boundary conditions are used to bind fluid and solid regions. In viscous flows the no slip boundary condition is enforced at the walls. Wall roughness affects the drag (resistance) and heat and mass transfer on the walls. Hence roughness effects were considered for the present analysis and

a specified roughness based on law of wall modified for roughness is considered. Two inputs to be specified are the physical roughness height and the roughness constant. And the default roughness constant (0.5) is assigned which indicates the uniform sand grain roughness.

5. RESEARCH METHODOLOGY

Regression analysis is used to investigate and model the relationship between a response variable and one or more predictors. MINITAB provides various least-squares and logistic regression procedures.

- Use least squares regression when your response variable is continuous.
- Use logistic regression when your response variable is categorical.

Both least squares and logistic regression methods estimate parameters in the model so that the fit of the model is optimized. Least squares minimizes the sum of squared errors to obtain parameter estimates, whereas MINITAB's logistic regression commands obtain maximum likelihood estimates of the parameters.

6.   RESULTS AND DISCUSSION

In different type of diffuser also varying area ratio is shown. Analysis gives the effect of geometry on the velocity in diffuser. The following conclusions can be drawn from the results.
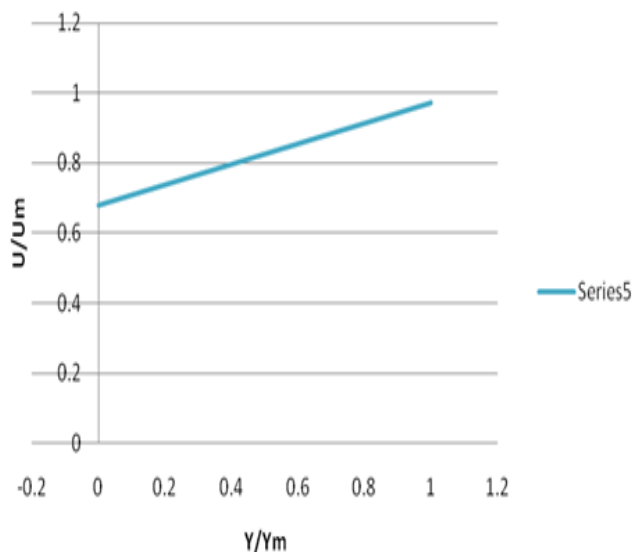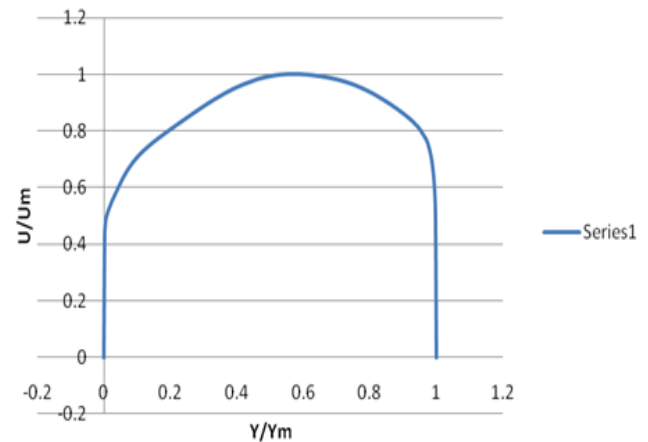

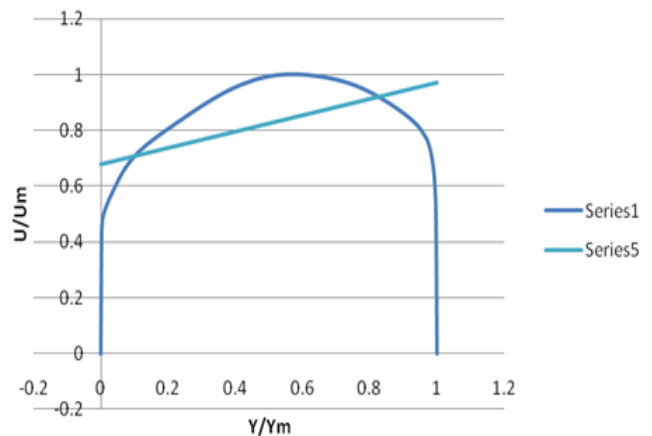Figure 2 Graph for CFD data given (0 degree swirl angle, at section 0.1x)

Figure 3 Comparison between linear regression graph and CFD data graph

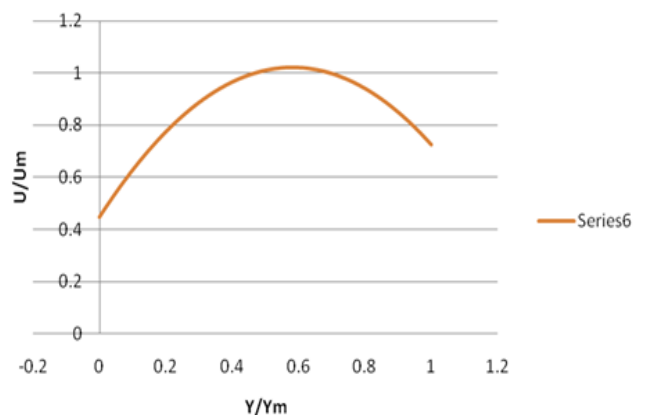Figure 1 Graph for the linear regression equation (0 degree swirl angle, at section 0.1x)

Figure 4 Graph for the quadratic regression equation (0 degree swirl angle, at section 0.1x)
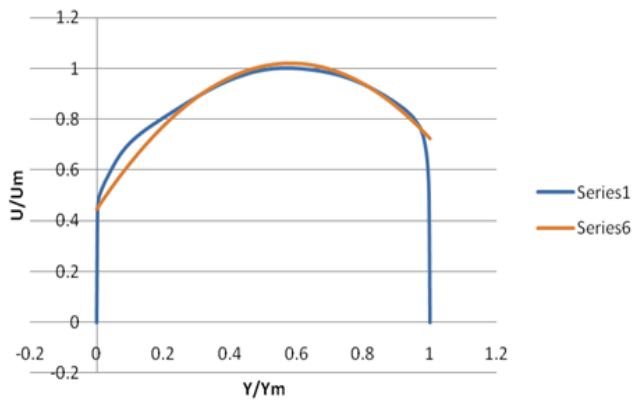
Figure 5 Comparison between quadratic regression graph and CFD data graph

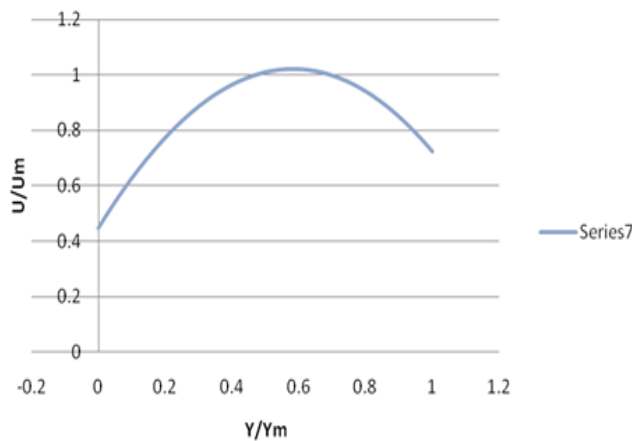(0 degree swirl angle, at section 0.1x)

Figure 6 Graph for the cubic regression equation (0 degree swirl angle, at section 0.1x)
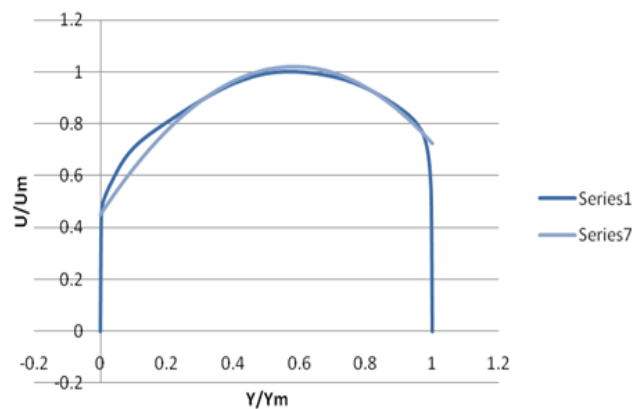

Figure 7 Comparison between cubic regression graph and CFD data graph

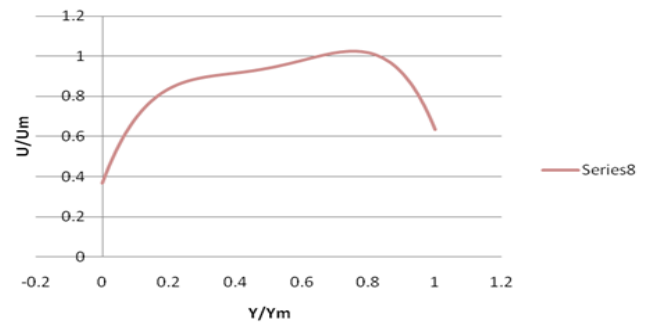(0 degree swirl angle, at section 0.1x)

Figure 8 Graph for the biquadrate regression equation (0 degree swirl angle, at section 0.1x)
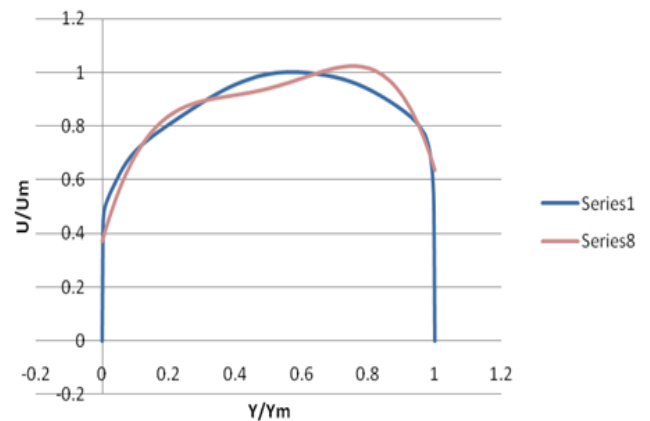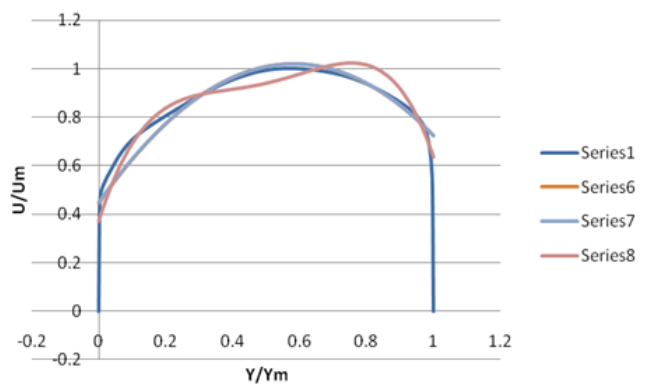

Figure 9 Comparison between biquadrate regression graph and CFD data graph

(0 degree swirl angle, at section 0.1x)

Series 1- CFD data plot   Series 2-Biquadratic Plot

Series 3 – Cubic Plot   Series 4- Quadratic Plot

Figure 10 Superimposed graphs on one chart

From these charts, we find out that;

- The cubic equation most closely follows the observed data.
- The quadratic equation closes on the observed values of graphs but does not follow the graph as closely as possible.
- The equation formed from the fourth power is not in tandem with the observed values.
- Higher power equations, higher degrees, are not needed as the fourth degree equation deviates from the main values.

For the linear variable:

We use MINITAB wherein the observed values are charted in the table and a regression equation obtained, as shown earlier.

- This equation is then used to obtain the calculated values from putting in the said values of Y/Ym.

- The calculated values are then re-tabulated in the MS Excel and graphs plotted accordingly.

- The equation obtained is given by;

    $U/Um = 0.679896050 + 0.291406112*(Y/Ym)$

A) For quadratic equation:

The equation obtained from linear regression is then used for squaring the Right Hand Side of above equation and obtaining the squared version of U/Um (0 degree).However, on plotting, we see that the graph obtained is not near to the original observations. Hence, we use the fitted line plot function of the MINITAB and then use the values to plot the graph as well as obtain the equation.

The equation obtained on squaring linear regression equation is;

$U/Um = 0.488601 + 0.39624572*(Y/Ym) + 0.08491396*(Y/Ym)^2$

The corrected equation from fitted line plot is;

$U/Um = 0.447269149 + 1.96947453*(Y/Ym) - 1.69222387*(Y/Ym)^2$

B) For Cubic Equation:
- The graph obtained is basically an extension of the regression equation cubed to obtain the U/Um values from the regression equation obtained earlier.
- However the quadratic plot tells that using fitted line plot is a better way of obtaining the equation.

- So we use the fitted line plot function to obtain calculated values of the function.
- These values are then used to plot a graph in MS Excel.
- The graph obtained most closely follows the observed values.

The equation obtained is;
$U/Um = 0.448087601 + 1.95580298*(Y/Ym) - 1.65536232*(Y/Ym)^2 - 0.0249087914*(Y/Ym)^3$

C) For Fourth degree:

For the fourth degree of equation we need regression function as well as the values of the squares, cubes and fourth powers of (Y/Ym).After obtaining these values, we use the response of U/Um (0 degree) against the predictors of (Y/Ym) and its squares, cubes, and fourth powers. The equation obtained is then used to calculate the values of the U/Um and hence used in MS Excel to plot the graphs. The graph obtained is deviating more now from the real graph of observed values. Hence we need to stop our analysis at this stage. The equation obtained in MINITAB is;

$U/Um = 0.369050979 + 4.42691651*(Y/Ym) - 13.9869796*(Y/Ym)^2 + 19.8533024*(Y/Ym)^3 - 10.0271374*(Y/Ym)^4$

This equation is not closer to the observed graphs. Hence we discard this equation and observe that higher equations are more deviant. So we stop at the cubic equation.

What we need now is to obtain cubic equations through fitted line plot of MINITAB for other degrees also.

For this, we formulate the following equations (at section x=0.1)

1) For 0 Degree swirl:
   $U/Um = 0.452317269 + 2.00753657*(Y/Ym) - 1.68530447*(Y/Ym)^2 - 0.134163177*(Y/Ym)^3$

2) For 7.5 Degree swirl:
   $U/Um = 0.495183620 + 1.90208197*(Y/Ym) - 1.70145320*(Y/Ym)^2 + 0.164208764*(Y/Ym)^3$

3) For 12 Degree swirl:
   $U/Um = 0.481386605 + 1.87487665*(Y/Ym) - 1.65072511*(Y/Ym)^2 + 0.0458006999*(Y/Ym)^3$

4) For 17 degree swirl:
   $U/Um = 0.423097198 + 2.02411771*(Y/Ym) - 1.78157300*(Y/Ym)^2 + 0.0970189234*(Y/Ym)^3$

5) For 25 degree swirl:

$$U/Um = 0.388453313 + 1.97040203*(Y/Ym) - 1.45775581*(Y/Ym)^2 - 1.49621280*(Y/Ym)^3$$

7. CONCLUSION

The velocity contour plot diagram of the redesigned concept shows that the high velocity contours are actually occupying the mid area of the wind turbine blades. There is an agreement that the splitter and shroud concepts might prove valuable if the turbine fan design is integrated into the power augmentation analysis. It is realized that inlet shroud can provide a better flow direction and streamlines with the flanges at the end of the diffuser. Also the back flow which produces vortices inside the diffuser is also reduced. As suggestion, further work is necessary to optimize the diffuser design in terms of power coefficients by considering momentum transfer to the turbine blade.

REFERENCES

[1]   Frankovoc, B.; and Vrsalovic, I, "New high profitable wind turbines," Renewable Energy, Vol.24, No.3 (2001), pp.491-499.

[2]   B.B.Arora, Manoj Kumar, Subhashis Maji, " Analysis of flow separation in wide angle annular diffusers," International Journal of Applied Engineering Research, Vol.5, No.20. (2010) p.p. 3419-3428, ISSN 0973-4562.

[3]   B. B. Arora, B. D. Pathak, "Effect of Geometry on the Performance of Annular Diffuser," International Journal of Applied Engineering Research, Vol.5, No.20. (2009) p.p. 3419-3428, ISSN 0973-4562.

[4]   Ackert ,J. 1967. Aspect Of Internal Flow. Fluid Mechanics Of Internal Flow ,Ed. Sovaran G., Elsvier Amsterdum,pp1.

[5]   Adkins R.C ,Jacobsen OH , Chevealier P 1983 A Premilary Study of Annular Diffuser With Constant Diameter Outer Wall. ASME paper no. 83-GT-218

[6]   Adkins R.C.,1983. A simple Method For Design Optimum Annular Diffusers. ASME Paper No. 83-GT-42.

[7]   Cockrell, D.J., Markland, E., 1963 .A Review of Incompressible Diffuser Flow. Aircraft Engg. Volume 35 , pp 286.

[8]   Coladipietro, R., Schneider, J.M., Sridhar, K.1974. "Effects of Inlet Flow Conditions on the Performance of Equiangular Annular Diffusers," Trans. CSME 3 (2): pp. 75-82.

[9]   Dovzhik, S.A., Kartavenko, V.M.,1975. "Measurement of the Effect of Flow Swirl on the Efficiency of Annular Ducts and exhaust Nozzles of Axial Turbomachines," Fluid Mechanics/Soviet Research 4(4): 156-172.