

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

## Experimental and Numerical Analysis of Complex Diffusers for Ventilation Strategy

Gaurav Chandra<sup>1</sup>, Shuvam Mohanty<sup>2</sup>

Assistant Engineer, Public Works Department Uttarakhand, Govt. of Uttarakhand, Bazpur, Udham Singh Nagar, India<sup>1</sup>

M.Tech Student, Department of Mechanical Engineering, Amity University, Gurgaon, Haryana, India<sup>2</sup>

**ABSTRACT:** This paper presents a simplified method for description of complex diffusers in computational fluid dynamics simulation of displacement ventilation at high computational efficiency. It may be to assist the design and analysis of air distribution for displacement ventilation as well as other types of ventilation with the complex diffusers. Correct description of the flow and thermal information for an air supply diffuser is important for reliably predicting room air distribution by using computational fluid dynamics. Simplified methods that can be used to describe flow and thermal information from eight commonly used diffusers were developed in this study. The investigation used the box and momentum methods. The momentum method performs well for five diffusers. Because the momentum method is simpler than the box method, the momentum method should be used whenever it is applicable. 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  $10^\circ$ . The effect of change in swirl angle and  $Y/Y_m$  in diffuser is studied in the present study. The cases that were studied are  $0^\circ$ ,  $7.5^\circ$ ,  $12^\circ$ ,  $17^\circ$ , and  $25^\circ$  at various sections of casing.

**KEYWORDS:** Diffuser, Computational Fluid Dynamics, Swirl angel, Annular section.

### I. INTRODUCTION

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. But, the annular diffusers have a very strong industrial significance and have received attention in recent years. These types of diffuser are very much used in aircraft applications. With the help of annular diffuser the maximum presser recovery is achieved within the shortest possible length. With annular diffuser, good performance is possible with large wall angles since an inner surface is present to guide the flow radially outward.

### II. RELATED WORK

Many persons have done a lot of tests on geometric parameter of diffuser Sovran and Klomp [1967] who tested over one hundred different geometries, nearly all of which had conically diverging centre bodies with an inlet radius ratio  $[R_i/R_o]$  of 0.55 to 0.70. The tests were carried out with a thin inlet boundary layer and the diffusers have free discharge.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

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 centre bodies of uniform diameter, using fully developed flow at inlet. The limits of the various flow regimes and the optimum performance lines were established. In these types of annular diffusers both hub and casing are diverging outward with same angle of divergence. A survey of diffuser research has revealed that considerably more investigations have been carried out on two dimensional and conical diffusers. Much of the extent data covering annular diffusers was done in the experimental laboratory Stafford and James [1957]. It is more difficult to define the essential geometric parameters for annular diffusers since the numbers of independent variables are large Goebel and Japikse [1981]. The essential variables to define the geometry of annular diffuser are two wall angles, area ratio, non-dimensional length and inlet radius ratio. As the number of variables increases, geometry becomes more complex. This has not been economically possible by experiments and hence led to the development of computational fluid dynamic methods to analyse the performance characteristics of annular diffuser Arora, Pathak and Singh [2005]. 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 HoadleyD,et.al, [1969]. Shaalan, et al [1975] studies reported in this field suggest that the Reynolds number is a comparatively weak parameter as long as the flow is fully turbulent regime. 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 is studied by Thayer E B, [1971]. Many persons have done a lot of tests on geometric parameter of diffuser Sovran and Klomp [1967] who tested over one hundred different geometries, nearly all of which had conically diverging centre bodies with an inlet radius ratio  $[R_i/R_o]$  of 0.55 to 0.70. The tests were carried out with a thin inlet boundary layer and the diffusers have free discharge. Area ratio and non-dimension length prescribes the overall diffusion and pressure-gradient respectively, which is the principle factor in boundary layer development. 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  $5^\circ$  and  $10^\circ$  were tested at various Mach number. It found by this study that most of data clusters around a line of constant Effectiveness. Johnston [1959] and Johnston [1953] reported a study of four different annular diffusers. Three of the four agree tolerably well with the basics Sovran and Klomp [1967] map, one of them disagree substantially; the case a strong disagreement is probably in stall. Srinath [1968] studied four equiangular annular diffuser with  $2\theta = 7^\circ, 10^\circ, 15^\circ$  and  $20^\circ$  respectively. Tests were reported with a variety of  $L/\Delta r$  values. An extensive study of diffusers which, although annular, begin with a circular cross section was reported by Ishikawa and Nakamura [1989]. 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.

### III. METHODOLOGY AND FLOW GEOMETRIC PARAMETERS

In an annular diffuser, a number of different geometric variables can influence the variation of pressure recovery and inlet condition of flow. The basic equations of motion reveal the importance of both geometric and aerodynamic parameters on the ultimate performance of annular diffuser. 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.

#### CFD Analysis

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.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

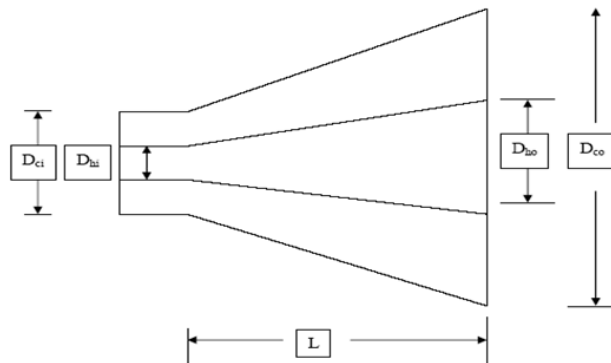


Fig.1. Annular Diffuser

## Boundary Layer Parameter

The flow in diffuser is governed by the behaviour of the boundary layers at the diffuser walls. The deceleration of the flow through the diffuser produces a pressure rise in the stream wise direction. The wall shear layers are therefore subjected to a positive or adverse pressure gradient. As is well known, an adverse pressure gradients cause the wall boundary layers to thicken and possibly separate from the diffuser walls, forming areas of backflow in the diffuser. The net result of thinking of the wall boundary layers or the formation of regions of backflow is the blockage of flow area which reduces the effective area available to the flow. Reduction in effective flow area in turn results in a reduced pressure rise through the diffuser.

## Convergence criteria

Finally, one needs to set the convergence criteria for the iterative method. Usually, there are two levels of iterations, within which the linear equations are solved and outer iteration that deal with the non-linearity and coupling of the equations. Deciding when to stop the iterative process on each level is important, from both the efficiency and accuracy point of view. A numerical is said to be convergent if the solution of the discretized equations tend to exact the solution of the differential as the grid spacing tends to be zero. For convergence criteria around  $10^{-6}$  for X velocity variable, the results are stable in the present problem.

## Implementation of boundary conditions

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.

## 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

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

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

Where,  $\text{Re}_{\text{DH}}$  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.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

## 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.

## Data Obtained from CFD Analysis

**Table.1. In the straight core annular diffuser for various swirl angle data are (at section x=0.1)**

Y/Ym	U/Um	S
0	0	0
0	3.00E-05	0
0.00248	0.42806	0
0.00499	0.47973	0
0.00752	0.49544	0
0.01008	0.5066	0
0.01266	0.51602	0
0.01526	0.52459	0
0.0179	0.53272	0
0.02056	0.5406	0
0.02325	0.5484	0
0.02596	0.55591	0
0.03829	0.58692	0
0.04117	0.59428	0
0.05637	0.63101	0
0.06516	0.64881	0
0.07157	0.66199	0

**Table.2. In the straight core annular diffuser for various swirl angle data are (at section x=0.9)**

Y/Ym	U/Um	S
0	0	0
0.00289	0.54166	0
0.00581	0.6487	0
0.00877	0.70089	0
0.01175	0.73383	0
0.01476	0.75786	0
0.0178	0.7767	0
0.02087	0.79216	0
0.02397	0.80527	0
0.0271	0.81666	0
0.03027	0.82623	0
0.03377	0.83408	0
0.04799	0.86598	0
0.06572	0.89322	0
0.06586	0.89338	0

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

## Formulation of Equation using MINITAB

In order to carry on with my project, I was directed to make use of CFD related software, a kind of equation solver, which forms graphs for the data given and forms the different equations. This software is called **MINITAB**. The software is basically centred on the regression plots of the various data, in numerous forms and ways. It can also form the various equations which may either be linear or nonlinear, depending upon the type of the data in question.

## Regression in MINITAB

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.

## IV. EXPERIMENTAL RESULTS

In the present paper subsonic turbulent flows inside annular diffusers is carried out in GAMBIT and FLUENT. Different type of diffuser also having varying area ratio gives the effect of geometry on the pressure recovery coefficient. The following conclusions can be drawn from the results

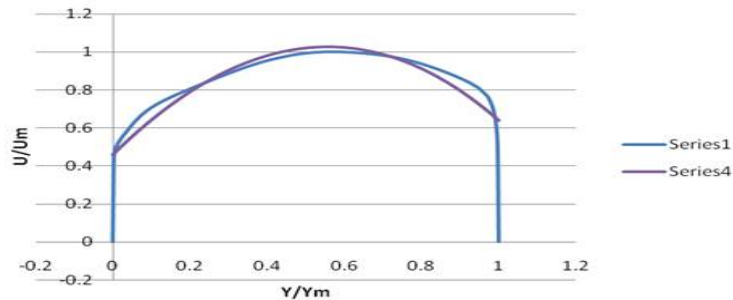


Fig.2. Straight core annular diffuser for 0 Degree swirl angle graphs are at section  $x=0.1$

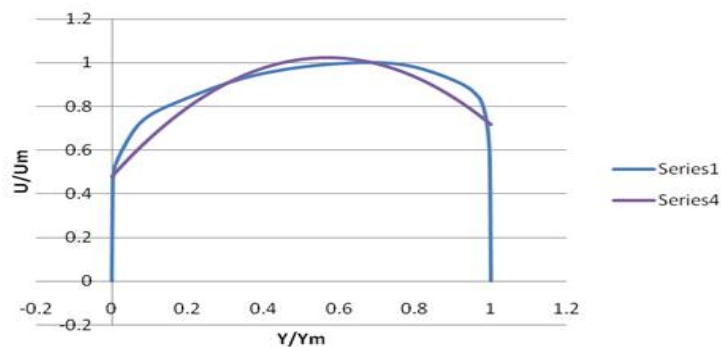


Fig.3. Straight core annular diffuser for 7.5 Degree swirl angle graphs are at section  $x=0.1$

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

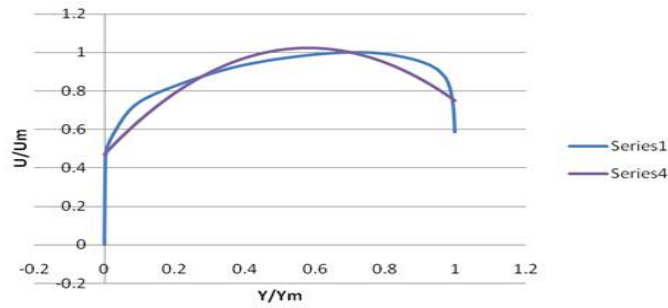


Fig.4. Straight core annular diffuser for 12 Degree swirl angle graphs are at section  $x=0.1$

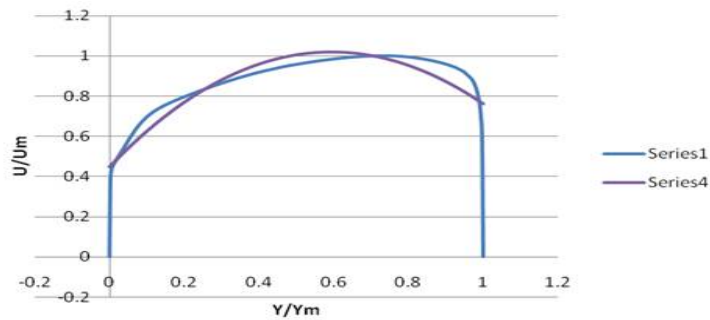


Fig.5. Straight core annular diffuser for 17 Degree swirl angle graphs are at section  $x=0.1$

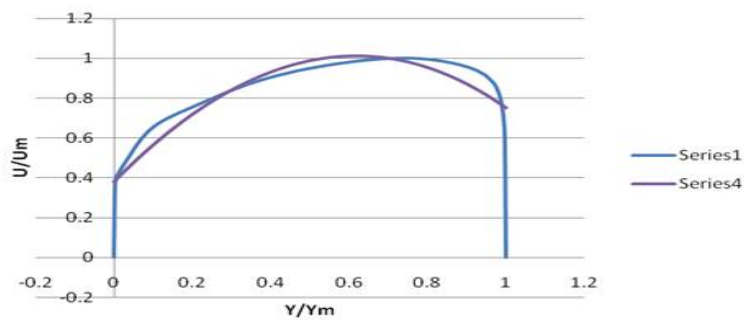


Fig.6. Straight core annular diffuser for 25 Degree swirl angle graphs are at section  $x=0.1$

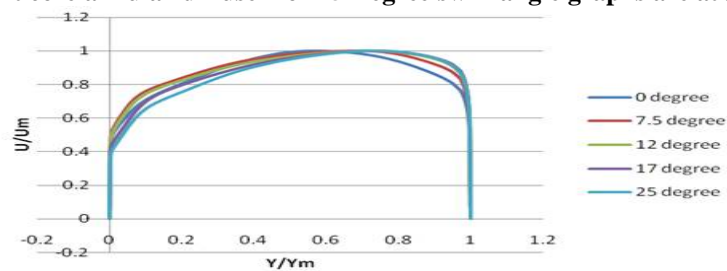


Fig.7. Observed values plotted on the graph give the following type of chart distribution (at section  $0.1x$ )

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 5, May 2018

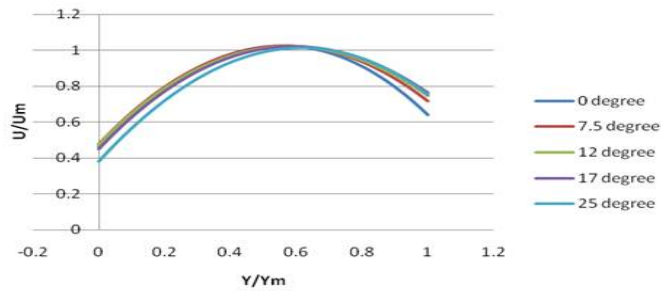


Fig.8. Obtained from the cubic equations solved were as given below (at section 0.1x)

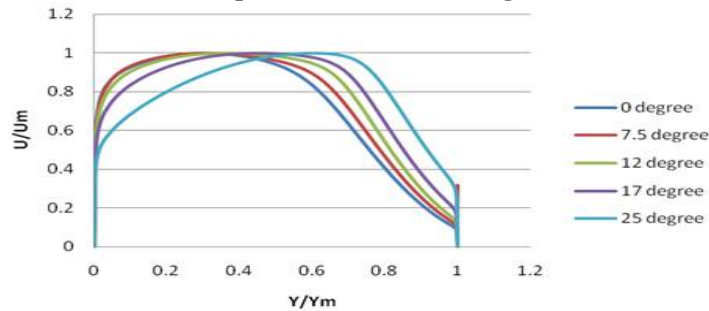


Fig.9. Observed values plotted on the graph give the following type of chart distribution (at section 0.9x)

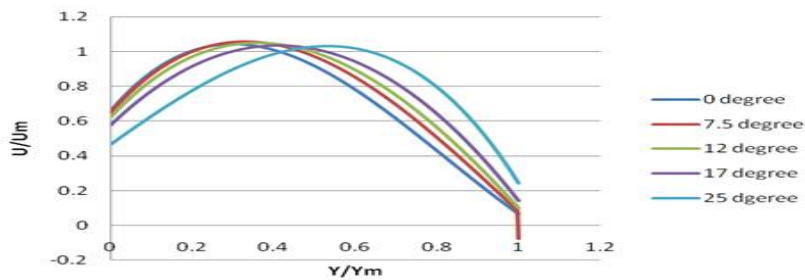


Fig.9. Obtained from the cubic equations solved were as given below (at section 0.9x)

## V. CONCLUSION

According to experimental and simulation results, the new ventilation strategy does have an ability to improve local fresh air percentage in the occupied zones without causing excessive draught rating. Long protruding distance of fresh air jet, small blade angle of recirculated air and high fresh air inlet velocity are observed to make this new method more efficient. Different angles of the protruding fresh air jet are observed to make this new method more flexible for different occupied zones.

## REFERENCES

- [1] B B Arora, B D Pathak , International Journal Of Dynamics Of Fluid (IJDF), Vol.7, No.1. (2011) p.p. 109-120, ISSN 0973-1784.
- [2] B.B.Arora, Manoj Kumar, SubhashisMaji, " 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., Elsevier Amsterdam, pp1.

## International Journal of Innovative Research in Science, Engineering and Technology

*(A High Impact Factor, Monthly, Peer Reviewed Journal)*

Visit: [www.ijirset.com](http://www.ijirset.com)

**Vol. 7, Issue 5, May 2018**

- [5] Adkins R.C ,Jacobsen OH , Chevealier P 1983 A Preliminary 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.
- [10] Goebel, J. H., Japikse, D., "The Performance of an Annular Diffuser Subject to Various Inlet Blockage and Rotor Discharge Effects," Consortium Final Report, Creare TN-325, March 1981.
- [11] Hesterman R, Kim S ,Ban Khalid A, Wittigs 1995. Flow Field And Performances Characteristics Of Combustor Diffusers: A Basic Study. Trans. ASME Journal Engineering for Gas Turbine and Power 117: pp 686-694.
- [12] Hoadley D,1970. Three Dimensional Turbulent Boundary Layers in an Annular Diffuser. PhD Thesis University of Cambridge.
- [13] Hoadley, D., Hughes, D.W.,1969. "Swirling Flow in an Annular Diffuser," University of Cambridge, Department of Engineering, Report CUED/A-Turbo/TR5.
- [14] Howard, J. H. G., Thorton –Trump A. B., Henseler H. J. 1967" Performance And Flow Regime For Annular Diffusers".ASME paper no . 67-WA/FE-21.
- [15] IshkawaK,Nakamura I 1989"An Experimental Study on The Performance of Mixed Flow Type Conical Wall Annular Diffuser " ASME FED-69.
- [16] Japikse, D., 1986. "A New Diffuser Mapping Technique – Studies in Component Performance: Part 1," ASME Paper No. 84-GT-237, Amsterdam, June 1984; also, Journal of Fluids Engineering, Vol. 108, No. 2. pp. 148-156.