Geometrical optimization of diffuser under swirl flow inlet condition using CFD analysis

[1]Alok Ranjan Jha, [2]Ashish Muchrikar
[1]Student, [2]Assistant Professo
Corporate Institute of Science and Technology, Bhopal, India

Abstract - **In the present study a 3 dimensional diffuser model is simulated by using CFD technique. The objective of the present work is to analyse the effect of the rotating domain in the diffuser which makes the flow through diffuser complicated to analyse analytically. In the study three cone angle 100, 200 and 300 is considered and at each cone angle 5 to 15% length is changed in diffuser section to observe the effect of change in length in diffuser section. After analysing the pressure drop curve it is found that increasing the length of the diffuser section pressure at constant diffuser angle pressure drop increases but negligibly while at constant diffuser length increasing the diffuser angle results in considerable amount of pressure drop. The maximum change in pressure is found at 20 degree diffuser angle and 0.561 m length of diffuser section which is our optimized value in terms of pressure drop. After analysing the velocity data it is found out that both increasing the angle and length results in increased peripheral velocity while vice versa for axial velocity. Also at each cone angle the change in pressure and velocity is given for percentage change in diffuser section length**

keywords - **computational fluid dynamics, Swirl motion, Diffuser section, Cone angle, Boundary conditions**

I. INTRODUCTION

A diffuser is applied in the mechanical systems comprising internal flow. Such systems are faced in turbo-machines between a compressor and a combustor or at the exit of a turbine, duct flows, flow meters, aircraft engine inlets, closed circuit wind tunnels, and pumps. The prime resolution of employing a diffuser in various sectors of the aforesaid applications is to upsurge static pressure retrieval while minimizing total pressure damage along the direction of the stream. In additional words, a diffuser functions as a transformation device of the stream dynamic head to static pressure. This parameter is concerned by the varying cross sectional area of the divergent segment of the diffuser.

Due to enhancement in the cross sectional area in forward direction of stream an adverse pressure gradient is formed which comprises different fluid flow phenomenon of interest. These singularities spans from flow separation and tremulousness to transitory stall and violent flow-excited static pressure vacillations. These types of stream conditions can be predicted qualitatively but very complex to quantify. Or in other words we can simply say that a diffuser is a device that increases the pressure of a fluid at the expense of its kinetic energy. As seen simply in the flow region the stream gets decelerated in forward motion through the divergent section the deceleration causes an upsurge in pressure in the respective motion at the expense of momentum. Such a process is known as diffusion. The stream flowing near the vicinity of the wall is obstructed by the wall. A study of the parameters governing the development of the boundary layer and their relationship with diffuser performance is, therefore vital in optimizing the design of a diffuser.

The energy transferal in these turbomachinery encompasses the altercation of substantial levels of kinetic energy in order to undertake the envisioned resolution. Consequently, prodigious levels of lingering kinetic energy recurrently go with the work input and work mining procedures, erstwhile as much as 50% of the whole energy shifted. A minor alteration in pressure recovery leads to an efficient diffuser system which ultimately smoothens the system performance. Therefore due to above mentioned benefits the diffuser systems are very useful for various applications.

II. LITERATURE REVIEW

Anders Olsson et al. [1] has accompanied CFD scrutiny on nozzle diffuser essentials of valve-less diffuser pump employed in dynamic micro pumps. The quantitative outcomes acquired are in close promise with analytical outcomes.

Sovran and Klomp et al. [2] examined various divergent section of conical profile to test the behavior of flow. The inlet section radius ratio were 0.55 to 0.7 for all models. They suggested after reading that pressure recovery is function of boundary layer formation.

Shuja and Habib et al. [3] has steered CFD simulations for swirl flow number stretching from 0.0 to 0.9 in axisymmetric annular diffuser. Heat and corresponding convection to stream is predicted using k-epsilon turbulence algorithm. The outcomes in the study is concluded as heat is conveyed in the stream. The outcomes also validated with analytical data.

Barker and Carrotte et al. [4,5] has investigated annular diffuser of combustor. Their conclusions have revealed that blade stirs produced causes left-over kinetic energy and boosted pressure upsurge can be engendered by involvement of wakes inside the downstream.

Limin Wang et al. [6] examined a diffuser with rotating domain. The rotating domain of 52.646 rad/s is smeared using EMMS built and k-epsilon algorithm turbulence model as shown in Fig. 2.1. The mentioned algorithm predicted the promising result in the core but deterioration also present neat the vicinity of the wall.

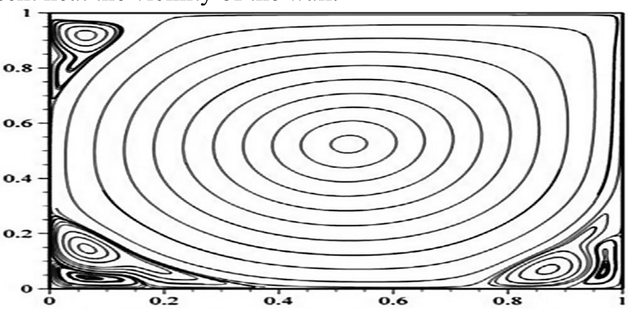

Figure 01 : Streamlines using EMMS turbulence model.

For assessing the different parameters for the diffuser system for multipurpose application various experimental and numerical analysis have been carried out.

Patterson [7], from the eworks of Gibson [8,9] and others, assesed for the standard value for the specific model divergence angles: 6–80 for a conical profile divergent segment diffuser, 60 for a square are diffuser, and 110 for a rectangular two-dimensional diffuser.

Lyon [10] found an 80 conical draft tube to be most efficient if a thin boundary layer is fed at its inlet.

Robertson and Ross [11], Uram [12], and Robertson and Hall [13] examined different model of diffuser to asses the boundary flow separation in diffuser system. He tried 80 models of diffusers and predicted the boundary separation effects.

Diffusers are one of the standard challenges in fluid mechanics. The diffuser is intended to upsurge the total pressure on the expense of fluid retardation. Since the positive pressure gradient exists in the stream direction it is mpre likely to flow back in reverse direction, making the diffusing process difficult to attain. There is a usual propensity in a diffusing progression for the stream to halt away from the walls of the diverging passage, converse its route, and drift back in path of the pressure gradient. Too rapid divergent angle causes eddy formation ehich ultimately consumes the kinetic energy of flow and less kinetic energy is available for being converted into pressure upsurge in the diffuser system. Also if we construct a diffuser of low conical angle the diffuser segment will increase for and skin friction loss will be increased. Flow separation also causes the loss in pressure upsurge so it is also non desirable for diffuser and design should made accordingly. In a vast number of case where strong flow separation occurs the reversal of flow can not be predicted by numerical analysis easily. Thus making the diffuser analysis very complicated and sensitive. As diffuser is a very common device so their study have been done for a long time. Not only because of its application but also to analyze the flow separation, have the diffuser been analyzed frequently by the researchers.

Historically, annular diffusers have ranked after channel and conical diffusers in terms of interest for research and hence fewer works is available upon which to establish the technology base for design and performance evaluation. Although annular diffusers are used in gas turbines and turbomachinery installations, it is only in the last decade that there have been any systematic investigations of there performance characteristics. The most notable contribution is that due to Sovran and Klomp [14] who tested over one hundred different geometries, nearly all of which had conically diverging centre 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 [15] 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. 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.

III. OBJECTIVE

The objective of the present work is to analyse the effect of the rotating domain in the diffuser which makes the flow through diffuser complicated to analyse analytically.

1. The main aim of the study is to analyse a three dimensional diffuser domain with swirl motion by performing a computational fluid dynamic analysis using a CFD code.
2. To validate the data of the present study with the existing work.
3. To observe the change in pressure across the diffuser by changing the length of the diffuser system.
4. To observe the effect of change in length of the diffuser section in peripheral velocity at outlet and axial velocity at outlet.
5. To visualize the pressure and velocity flow field associated with the flow.
6. To compare the results obtained for optimizing the length of the diffuser section.

IV. METHODOLOGY

Diffuser are used to maximize static pressure recovery in flow systems such as turbo machines, turbine exit, duct flows, aircraft engine inlets, closed circuit wind tunnels etc. Many of turbo-machines has swirl flow at inlet which requires thorough investigation to predict fluid flow parameters like velocity, pressure and mass flow. In this chapter we are going to observe the setup that are done for reaching the end simulation. We had changed the length of diffuser by taking three different constant angle and observe what we are getting its variation on output parameters. Computational fluid dynamic analysis have been performed for all 12 models are described below:

Table 01: List of cases

Diffuser Angle (degree)	Diffuser Length(m)	Percentage change in length (%)
10	0.51	0
10	0.4845	+5
10	0.5355	-5
10	0.361	+10
20	0.51	0
20	0.4845	+5
20	0.5355	-5
20	0.361	+10
30	0.51	0
30	0.4845	+5
30	0.5355	-5
30	0.361	+10

Computational fluid dynamics analysis of diffuser for different diffuser length:

CAD geometry of diffuser: Three dimensional CAD model of diffuser is created using design modular software with dimension from base paper, A three dimensional views for all cases of diffuser is shown in figure below:

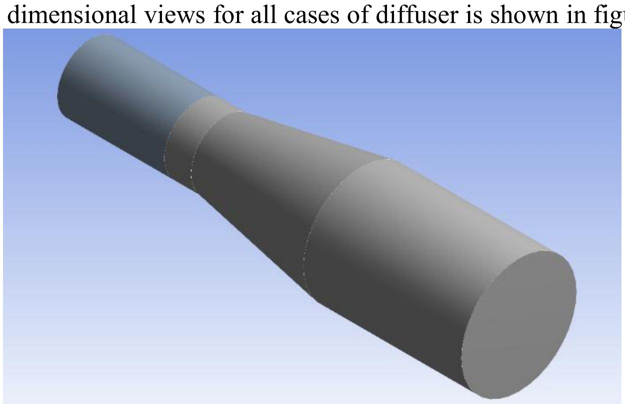

Figure 02: Three dimensional CAD geometry of model (Isometric view 1)

Studied diffuser : A diffuser with simple geometry is used in the present study to clarify the effect of changing the length of diffuser. The diffuser main specifications are taken from the base paper and is presented below with dimension

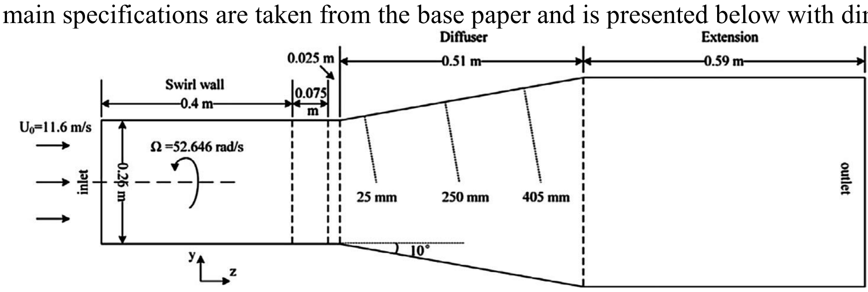

Figure 03: View of the selected pump with main dimensions.

Meshing: After completing the CAD geometry of model of diffuser is imported in ANSYS mechanical for further computational fluid dynamics analysis where next step is meshing. Meshing is a critical operation in finite element analysis in this process CAD geometry is divided into large numbers of small pieces called mesh. The total number of nodes generated for this model is 467746 and total number of Elements is 454950. Types of elements used are tetrahedral which is triangular in shape with four nodes on each element.

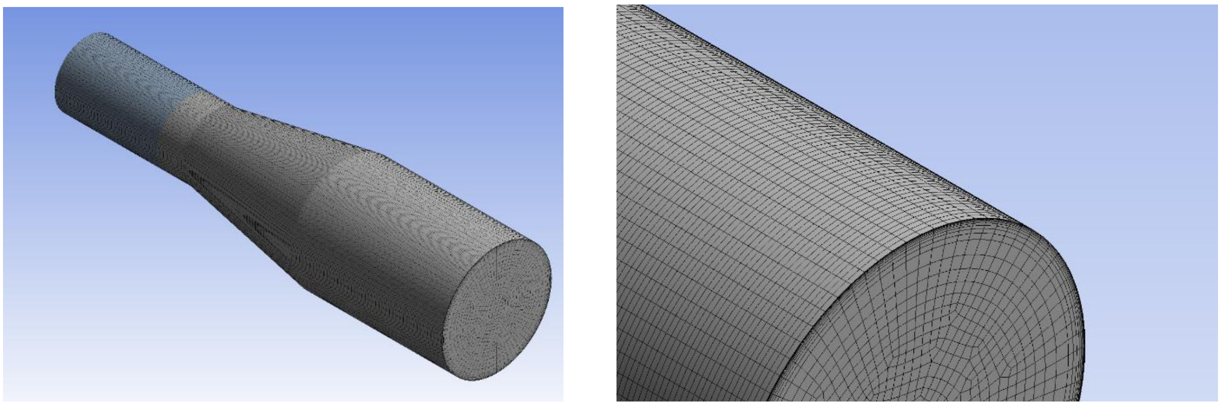

Figure 04: Meshing of CAD model isometric view and it's zoomed view.

Boundary conditions are assigned to create a virtual environment of the real life working of the system. The boundary conditions for simulation of centrifugal pump are explained below :

- Define the solver settings as pressure based and enable gravity option in y direction with the value of -9.81 m/s^2.
- Working fluid is air with density 1.22 kg/m^{3}and viscosity 1.789 e-05 kg/m-s for air.
- Set viscose model K-epsilon RNG model with enhanced wall function
- In cell zone condition set the domain as rotating by using reference frame motion.
- Air inlet velocity is 2 m/sec and for turbulence specification method intensity and viscosity ratio where turbulent intensity is 5% and turbulent viscosity ratio 10.
- Set the heat flux at wall as per cases in boundary conditions.
- For the operating condition the operating pressure needs to be set as 101325pa.
- Under Discretization, select standard for COUPLED, and second order for Momentum and Energy equation
- The Fluent solver is used for CFD analysis.

V. RESULT AND DISCUSSION

In the current work computational fluid dynamics analyses have been performed for diffuser with constant inlet velocity and diffuser is of standard size selected from base paper and we are using ANSYS fluent to investigate the effects of better performance by changing of length of diffuser at three constant angles 10o , 20o and 30o. For that CAD model of diffuser is created using design modeler software with dimensions available in base paper. There are following validation and results have been discussed using contours diagram and graphical representations.

The main objective of the present work is to perform computational Fluid Dynamics analysis to observe the variation in diffuser after changing the length of diffuser at different angles . For the validation of this work the CAD dimensions of the diffuser is taken from a research paper of A.Agrawal "CFD analysis of conical diffuser under swirl flow inlet conditions using turbulence models" Materials today (2020) 2214-7853 contents available at science direct. The geometrical parameters for diffuser is considered from base paper . the working fluid used in simulation is air having properties such as density 1.225 Kg/m3and viscosity 1.7894e-5 kg/ms. Types of elements used are hexahedron which is square prism in shape with eight nodes on each element

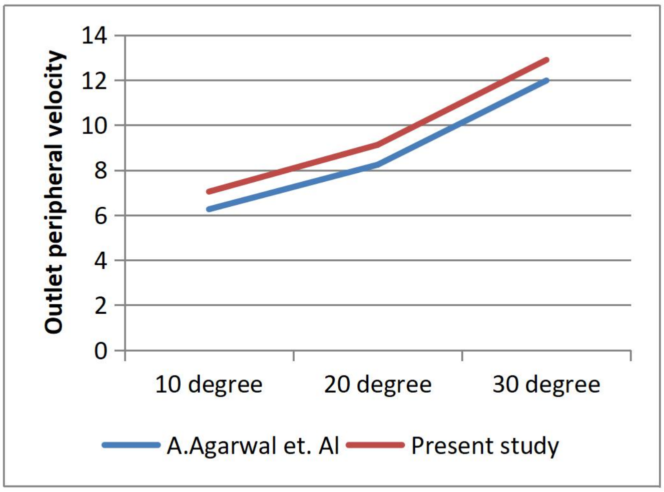

Figure 05 : Outlet peripheral velocity at constant angles 0.51m and 0.51m diffuser length

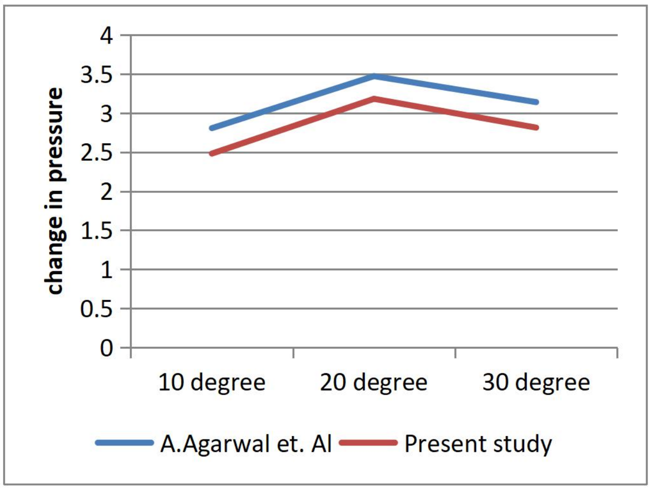

Figure 06 : Change in pressure at constant angles and diffuser length

In above two graph for the same cases of base paper is simulated in ANSYS and the resultant chart is generated for validation purpose so that we can get what we are getting the results are same correct or not because ANSYS tool used for simulation is numerical approach and we know numerical approach needs to be verified either by experimental data or previous verified data.

After the validation of base model some other cases of diffuser have been used for computational fluid dynamics analysis to investigate effect such as velocity, pressure, etc. In the present work it has been tried to investigate diffuser at different length of diffuser with constant angles.

Results for CFD analysis of Diffuser

Computational fluid dynamics analysis of diffuser at inlet velocity 2 m/s with swirl motion of 52.646 rad/sec is done. Results for all cases are presented in form of streamlines and contours of pressure and velocity on plane and user defined surface which has been generated by CFD-Post which is a post processing software for CFD simulations in ANSYS

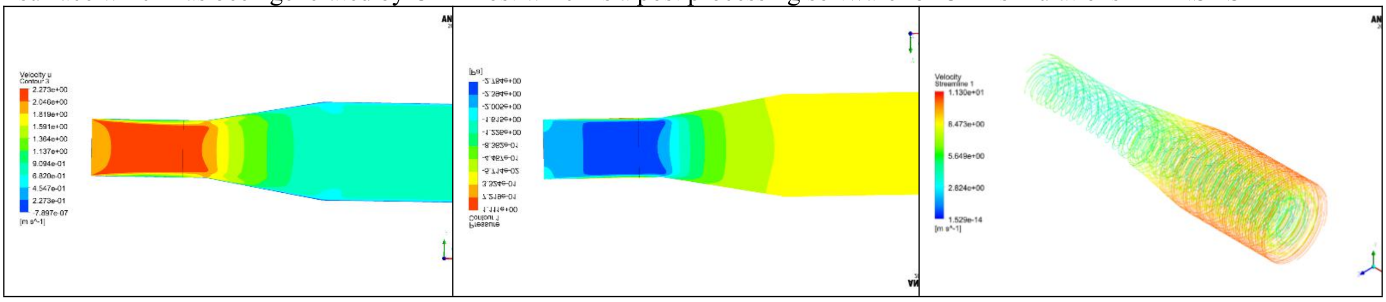

Figure 07: Velocity contour, Pressure contour and streamlines for diffuser with 10º angle and length 0.51m

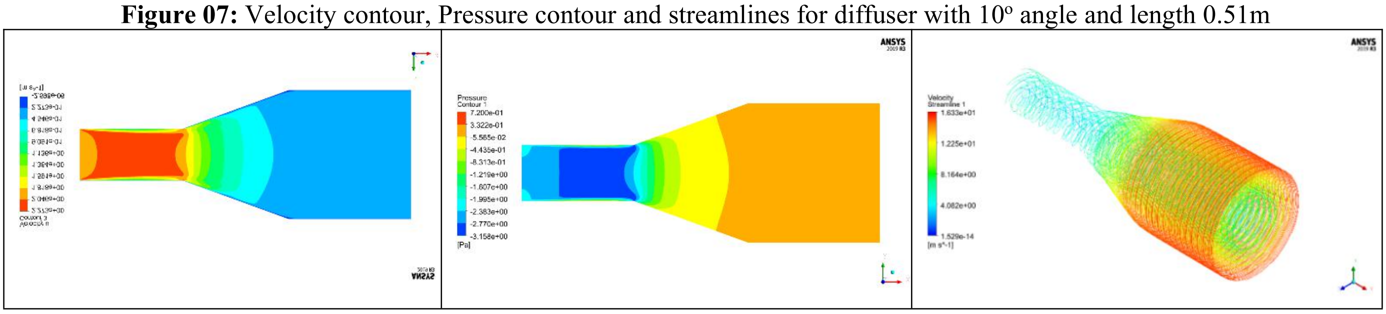

Figure 08: Velocity contour, Pressure contour and streamlines for diffuser with 20º angle and length 0.51m

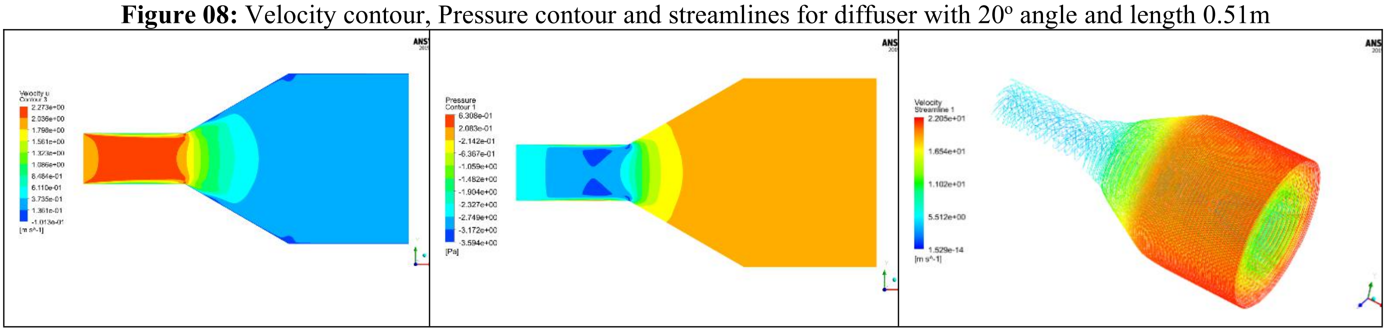

Figure 09: Velocity contour, Pressure contour and streamlines for diffuser with 30º angle and length 0.51m

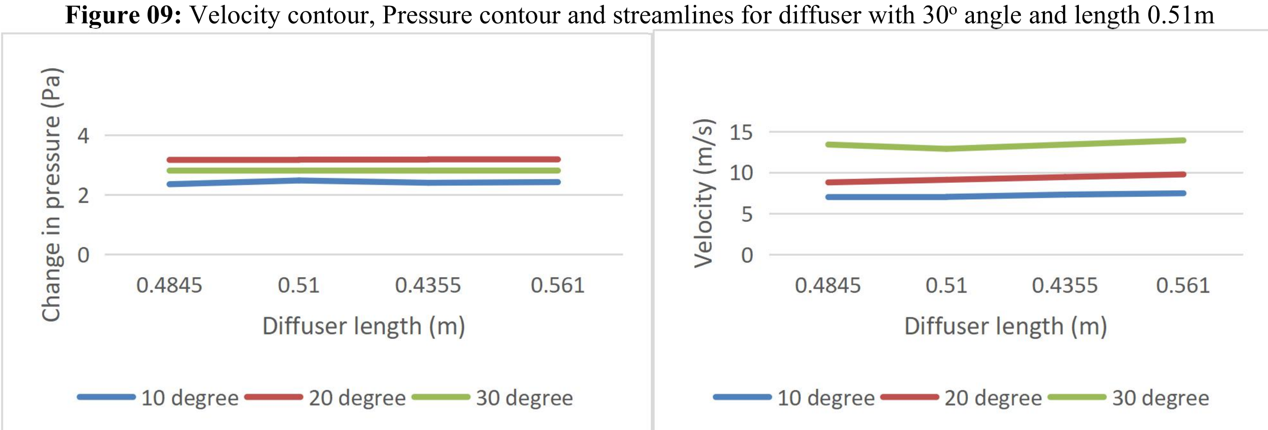

Figure 10: Chart of velocity and change in pressure w.r.t diffuser length for all cases

VI. CONCLUSION

After analyzing the various data observed in the study following conclusive points were drawn:

1. An increase in length of diffuser section by 15% results 11.67% decrease in axial velocity while 6.58% increase in peripheral velocity at constant conical angle of diffuser at 10 degree.
2. An increase in length of diffuser section by 15% results 16.27% decrease in axial velocity while 11.02% increase in peripheral velocity at constant conical angle of diffuser at 20 degree.
3. An increase in length of diffuser section by 15% results 18.72% decrease in axial velocity while 3.80% increase in peripheral velocity at constant conical angle of diffuser at 30 degree.

4. After analysing the pressure drop curve it is found that increasing the length of the diffuser section pressure at constant diffuser angle pressure drop increases but negligibly while at constant diffuser length increasing the diffuser angle results in considerable amount of pressure drop. The maximum change in pressure is found at 20 degree diffuser angle and 0.561 m length of diffuser section which is our optimized value in terms of pressure drop.
5. After analysing the velocity data it is found out that both increasing the angle and length results in increased peripheral velocity while vice versa for axial velocity.

REFERENCES

[1] A. Olsson, G. Stemme, E. Stemme, Simulation Studies of Diffuser and Nozzle Elements for Valve-less Micro pumps, the 10th International Conference on Solid-State Sensors and Actuators (Transducers 97), Chicago, USA, 1997, pp.1039– 1042.
[2] G. Sovran E.D. Klomp, Experimentally determined optimum geometries for rectilinear diffusers with rectangular, conical or annular cross-section. Fluid mechanics of internal flow (Ed. G. Sovran), 1967, pp. 270–312
[3]] L.R. Reneau, J.P. Johnston, S.J. Kline, Performance and design of straight, two imensional diffusers, ASME J. Basic Eng. 95 (1967) 141–150.
[4]] S.O. Adenubi, Performance and flow regime of annular diffusers with axial turbo machine discharge inlet conditions, J. Fluid Eng. Trans. ASME 98 (1976) 236–242.
[5] S.J. Stevens, G.J. Williams, The influence of inlet conditions on the performance of annular diffusers, Trans. ASME J. Fluids Eng. 102 (1980) 357–363.
[6] Limin Wang, Turbulence originating from compromise in competition between viscosity and inertia, Chem. Eng. J. 300 (2016) 89–97.
[7] G.N. Patterson, Modern diffuser design, Aircraft Eng. 10 (1938) 267–273.
[8] A.H. Gibson, On the flow of water through pipes having converging or diverging boundaries, Proc. Roy. Soc. A 83 (1910) 366–378.
[9] A.H. Gibson, On the Resistance to Flow of Water through Pipes or Passages Having Divergent Boundaries, Trans. Roy. Soc. A. Edinburgh 48(1), Part 5, 1911.
[10] G.E. Lyon, Flow in conical draft tubes of varying angles, Mech. Eng. 44 (1922) 177–180.
[11] J.M. Robertson, D. Ross, Effect of entrance conditions on diffuser flow, ASCE Trans. (1952) 1068–1097.
[12] E.M. Uram, The Growth of an Axisymmetric Turbulent Bound- ary Layer in an Adverse Pressure Gradient, in Proc. Second U.S. Natl. Congr. Appl. Mech., Univ. Michigan, Ann Arbor, Michigan, ASME, 1954, pp. 687-695.
[13] J.M. Robertson, J.W. Hall, Effect of adverse pressure gradients on turbulent boundary layers in axisymmetric con- duits, ASME J. Appl Mech. 79 (1957) 191–196.
[14] Sovran G., Klomp E.D., 1967. "Experimentally Determined Optimum Geometries for Rectilinear Diffusers with Rectangular, Conical or Annular Cross-Section". Fluid Dynamics of Internal Flow, Elsevier Publishing Company.
[15] Howard J. H. G., Thornton –Trump A. B., Henseler H. J., 1967. "Performance and Flow Regime for Annular Diffusers". ASME Paper No. 67-WA/FE-21.