Parametric analysis of axial annular diffuser with diverging hub and casing using CFD tool ANSYS

Vishesh Shukla[1], Devesh Kumar[2]
M. Tech Student[1], Assistant Professor[2]
MECHANICAL ENGINEERING DEPARTMENT
MMMUT, Gorakhpur, UP, India

Abstract **:** Diffuser is a engineering device used for converting kinetic energy of fluid into pressure. It has many applications in field of engineering like gas turbine for increasing outlet pressure, multistep expansion of gas in gas turbine power plant for matching the flow condition of low pressure turbine, it is also used in under floor air distribution system for air distribution in hall or room for better air circulation and in cooling of composite materials after manufacturing. In this work annular diffuser with diverging hub and casing is modelled in CFD tool- ANSYS Fluent for studying the pressure and velocity variation along the length of annular diffuser. The study of pressure and velocity variation is done at different area ratio and different divergence angles of hub. The inlet flow conditions is given as velocity-10 m/s, hub divergence angle is 3° and area ratio is 2, 2.1 and 2.4.The Reynolds number is calculated as 28752.65, so turbulent k- ε model is used for modelling and working fluid is taken as air. The results shows that there is better pressure recovery is obtained by using diverging hub instead of simple cylindrical hub.

Key Words : CFD, ANSYS, annular diffuser, and hub angle.

1.Introduction

The diffuser design have attracted the attention of researchers from many years and the geometry of the passage through which fluid flows has drawn the attention of researchers. Sovran and klomp have derived the formula for shape parameter (ω) and entropy change ($C_f * \omega$) for annular diffuser and drawn the graph between shape parameter and entropy change. They have calculated the values for shape parameter and entropy change using FORTRAN code. When they ploted graph between shape parameter and entropy change, they found a trend in curve that a minima point at shape parameter = 10, which shows that at that shape parameter value of entropy change is minimum and avalibility will be maximum. So, the shape parameter at which the entropy is minimum can be used for designing diffuser.

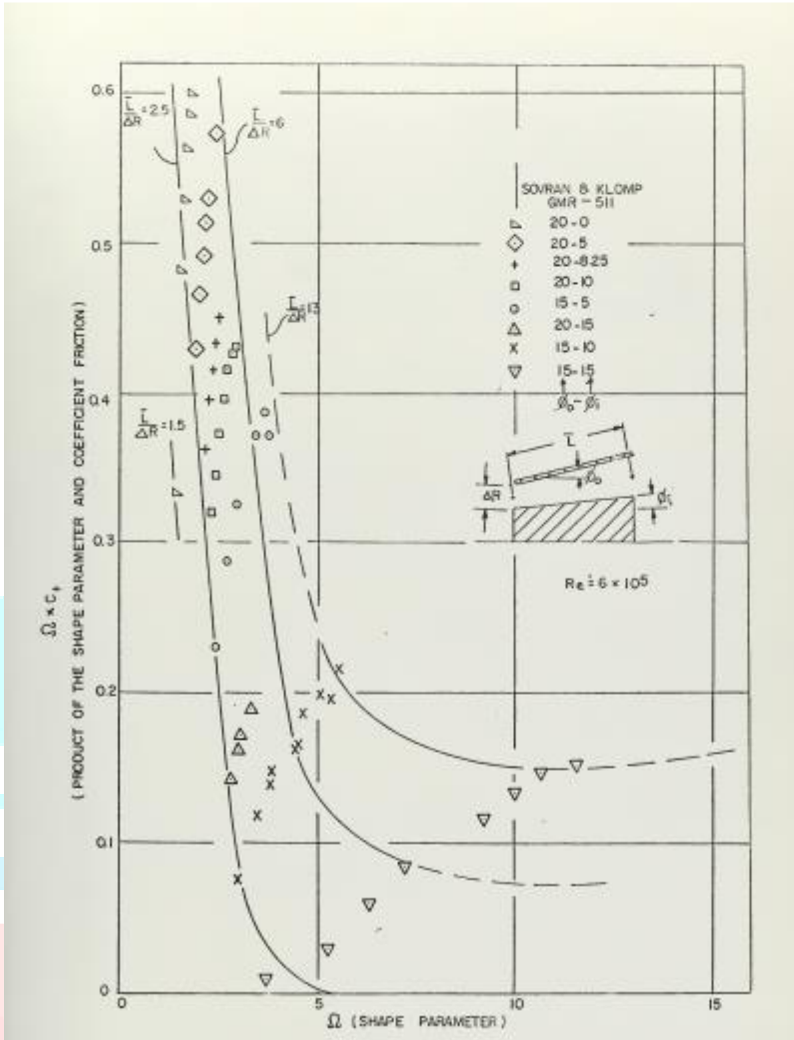


Fig.1- Graph between shape parameter (ω) and entropy change ($C_f * \omega$) for annular diffuser. (Reference- A book on" A design criteria for arbitrary diffuser by Donal Robert Gapp")

Gordon Holloway provided the writer recent example, where an improved diffuser reduced the power requirement of a wind tunnel by 40 percent (100 hp or 75kW decreases!).

1.1 Gradual Expansion- The Diffuser

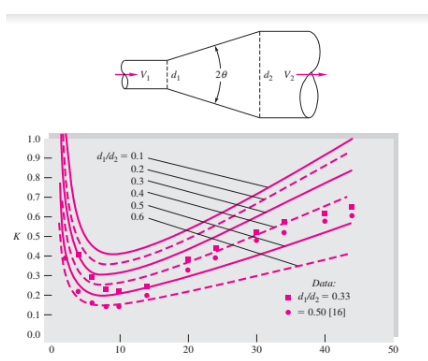


Fig. 2 –Flow losses in a gradual conical expansion region, as calculated from Gibson suggestion Eq. - 1, for a smooth wall.

As flow enters a gradual expansion or diffuser, such as the conical geometry of Fig. 2 , the velocity drops and the pressure rises. Head loss can be large, due to flow separation on the walls, if the cone angle is too grate. A thinner enterance boundary layer, causes a slightly loss than a fully developed inlet flow. The flow loss is a combination of non-ideal pressure recovery plus wall friction. Some correlating curves are shown in Fig. 2. The loss coefficient K is based on the velocity head in the inlet (small) pipe and depends upon cone angle 2θ and the diffuser diameter ratio d_1/d_2.

$$K_{diffuser} = \frac{h_m}{V_1^2/(2g)} = 2.61\sin\theta\left(1 - \frac{d^2}{D^2}\right) + f_{ave}\frac{L}{d_{ave}}\ for\ 2\theta \leq 45° \qquad Eq.-1$$

For large angles, $2\theta > 45$, the loss coefficient (2.61 sinθ) is high as in case of sudden expansion but for minimum loss lies in region $5° < 2\theta < 15°$, which is best geometry for efficient diffuser. For diffuser $\theta < 5°$ the diffuser is too long and prone to much friction losses.

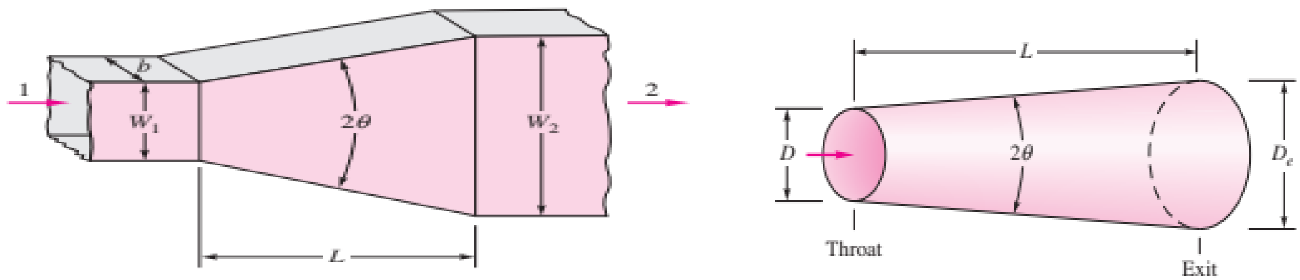


Fig. 3 – Diffuser geometry

A diffuser, shown in Fig. 3, is an expansion or increase intended to reduce velocity in order to recover the pressure head of the flow. Rouse and Ince relate that it may have been invented by customers of the early Roman (about 100 A.D.) water supply system, where water flowed continuously and was billed according to pipe size. The ingenious customers discovered that they could increase the flow rate at no extra cost by flaring the outlet the outlet section of pipe.

Engineers have always designed diffusers to increase pressure and reduce kinetic energy of ducted flows, but until about 1950, diffuser design was a combination of art, luck, and vast amounts of empiricism. Small changes in design parameter caused large change in performance. The Bernoulli equation seemed highly suspect as a useful tool.

Neglecting losses and gravity effects, the incompressible Bernoulli equation predict that

$$p + \frac{1}{2}\rho V^2 = p_0 = constant$$

Where p_0 is stagnation pressure the fluid would achieve if the fluid were slowed to rest (V = 0) without losses.

The basic output of a diffuser is the pressure-recovery coefficient C_p defined as

$$C_p = \frac{P_e - P_t}{P_{0t} - P_t}$$

Where subscripts e and t means the exit and the throat (or inlet), respectively. Higher C_p means better performance.

Consider the flat walled diffuser where section 1 is the inlet and section 2 the exit. Application of Bernoulli's equation to this diffuser predicts that

$$p_{01} = p_1 + \frac{1}{2}\rho V_1^2 = p_2 + \frac{1}{2}\rho V_2^2 = p_{02}$$

Or $$C_{p,frictionless} = 1 - \left(\frac{V_1}{V_2}\right)^2 \qquad Eq. - 2$$

Meanwhile, steady one – dimensional continuity would require that

$$Q = V_1 A_1 = V_2 A_2 \qquad Eq. - 3$$

Combining 2 and 3, we can write the performance in terms of the area ration AR = A_2/A_1 , which is a basic parameter in diffuser design:

$$C_{p,frictionless} = 1 - (AR)^{-2} \qquad Eq. - 4$$

A typical design would have AR = 5:1, for which Eq. – 4 predicts $C_p = 0.96$, or nearly full recovery. But, in fact, measured values of C_p for this area ratio are only as high as 0.86 and can be as low as 0.24.

The basic reason for discrepancy is flow separation, as sketched in Fig. 4. The increasing pressure in the diffuser is an unfavourable gradient, which causes the viscous boundary layers to break away from the walls and greately reduces the performance. Computational fluid dynamics (CFD) can now predict this behaviour.

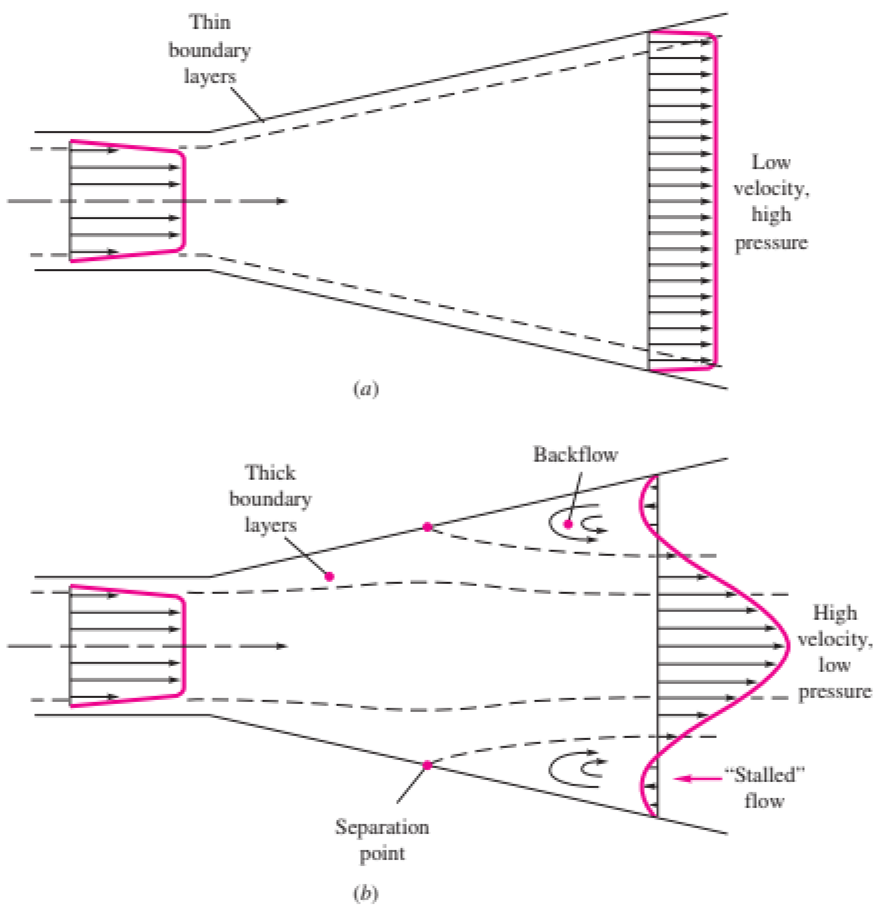


Fig. 4 – Diffuser performance: (a) ideal pattern with good performance; (b) actual measured pattern with boundary layer separation and resultant poor performance.

As an addition complication to boundary layer separation, the flow pattern in a diffuser are highly variable and were considered mysterious and erratic until 1955, when Kline revealed the structure of these pattern with flow visualization techniques in a simple water channel.

A complete stability map of diffuser flow pattern was published in 1962 by Fox and Kline, as shown in Fig. 5. There are four basic regions. Below line aa there is steady viscous flow, no separation, and moderately good performance. Note that even a very short diffuser will separate, or stall, if its half-angle is greater than 10°.

Between lines aa and bb is a transitory stall pattern with strongly unsteady flow. Best performance (highest C_p) occurs in this region. The third pattern, between bb and cc, is steady bistable stall from one wall only. The stall pattern may flip-flop from one wall to the other, and performance is poor.

The fourth pattern, above line cc, is jet flow, where the wall separation is so gross and pervasive that the mainstream ignores the walls and simply passes on through at nearly constant area. Performance is extremely poor in this region.

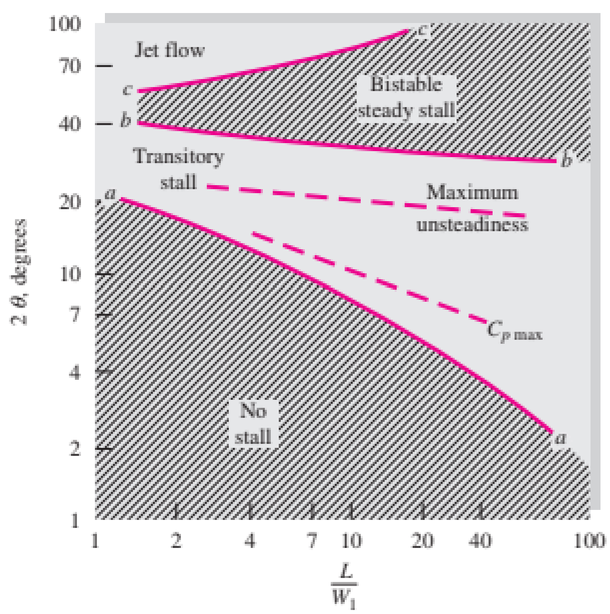


Fig. 5 – Diffuser stability map

1.2 Dimensional analysis of a flat-walled or conical diffuser shows that C_p should depend on the following parameters:

1. Any two of the following geometric parameters:
a. Area ratio AR = A_2/A_1 or $(D_e/D)^2$
b. Casing Divergence angle 2θ
c. Hub divergence angle
d. Slenderness L/W_1 or L/D
2. Inlet Reynolds number $Re_t = \rho V_1 W_1/\mu$ or $Re_t = \rho V_1 D/\mu$
3. Inlet Mach number $Ma_t = V_1/a_1$
4. Inlet boundary layer blockage factor $B_t = A_{BL}/A_1$, where A_{BL} is the wall area blocked, or displaced, by retarded boundary layer flow in the inlet (typically B_t varies from 0.03 to 0.12)

2. Geometrical Description

Three different hub geometries in an annular diffuser are modelled in this paper. One is at hub divergence angle 6, second is at hub divergence angle 5 and third is at hub divergence angle 4. They are modelled with different hub angles to study the effect on the pressure and velocity through the annular diffuser. Figure 6(a) show the annular diffuser fitted with hub of divergence angle 6.All dimensions are given in the Figure 6(a).

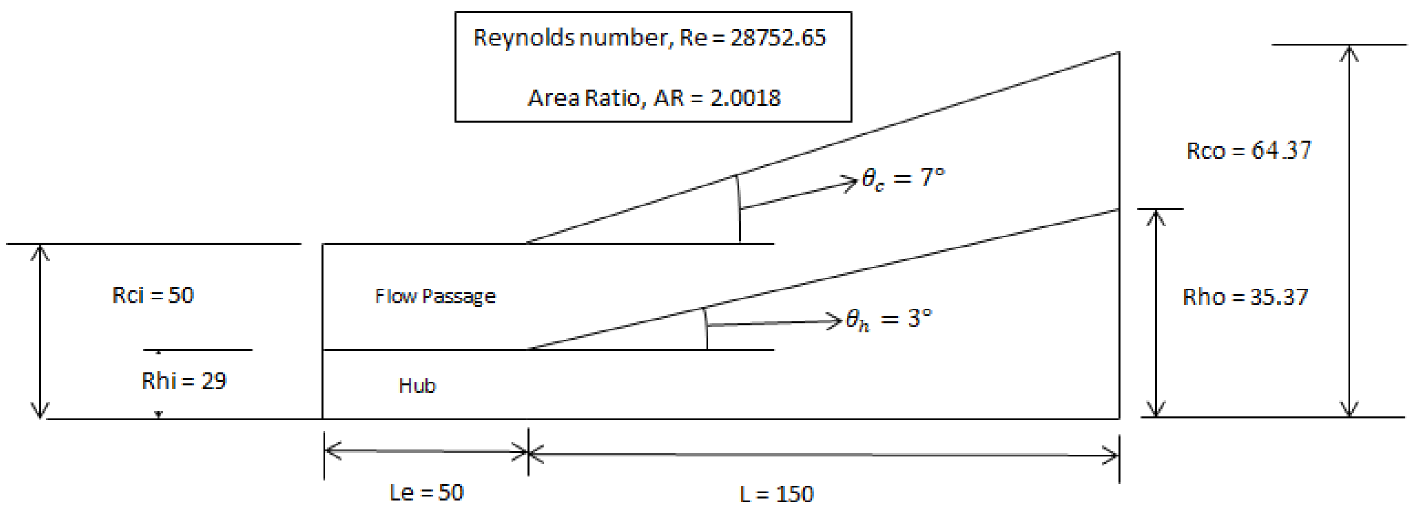


Figure 6(a) – Geometry of annular diffuser with diverging hub (hub divergence angle = 6)

Table of Symbols –

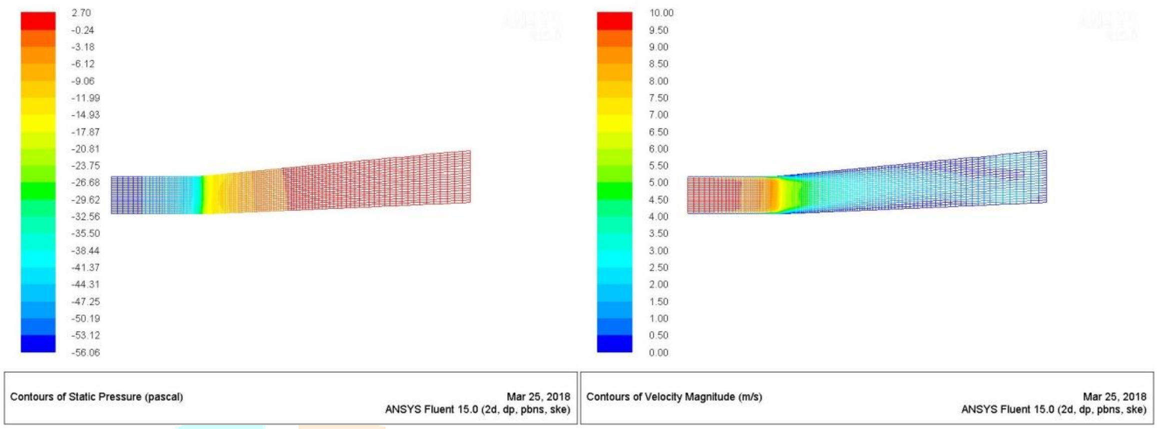
Symbols	Meaning
Rci	Radius of casing at inlet
Rco	Radius of casing at outlet
Rhi	Radius of hub at inlet
Rho	Radius of hub at outlet
θ_c	Angle of divergence of casing
θ_h	Angle of divergence of hub

3. Numerical model

3.1. Computational modelling - The commercial software ANSYS 16.1 was employed to solve the governing differential equation. It was chosen as the CFD tool for this work. Some simplified assumptions were applied to model the pressure distribution in an annular diffuser with diverging hub and casing. The major assumptions are; (1) the flow is steady and incompressible, (2) the flow through the annular diffuser is turbulent, and (3) the thermos-physical properties of the fluid at the annular diffuser inlet are remain constant. The numerical analysis was performed in tow dimensional domains applying standard k- ε as turbulent model.

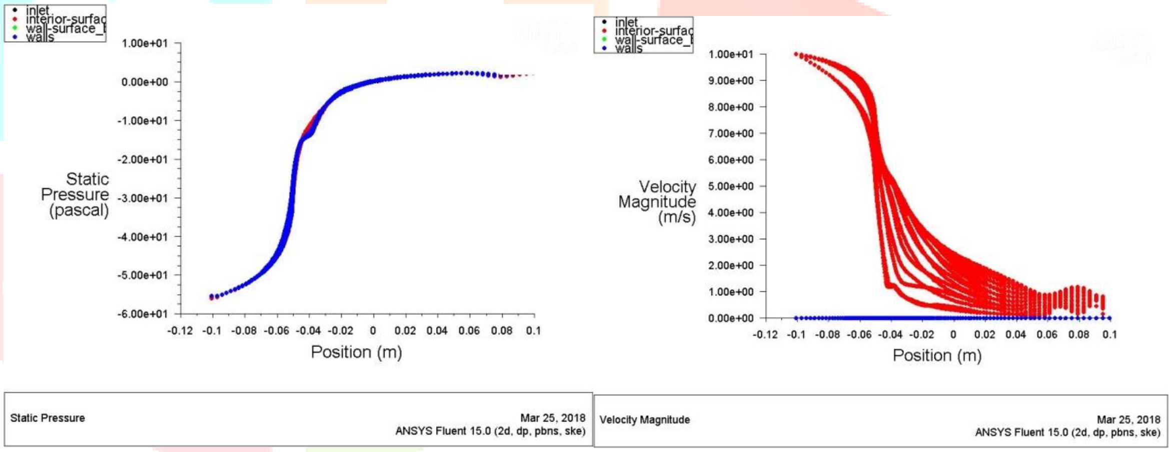
4. Result And Discussion

4.1 Results of annular diffuser with casing divergence angle = 7, hub divergence angle = 3, area ratio = 2.0018 and inlet velocity = 10 m/s. Calculated Reynolds number = 28752.65.



(a) (b)

Figure 7(a) and (b) – Contour plot of Pressure and velocity distribution along the length of annular diffuser.

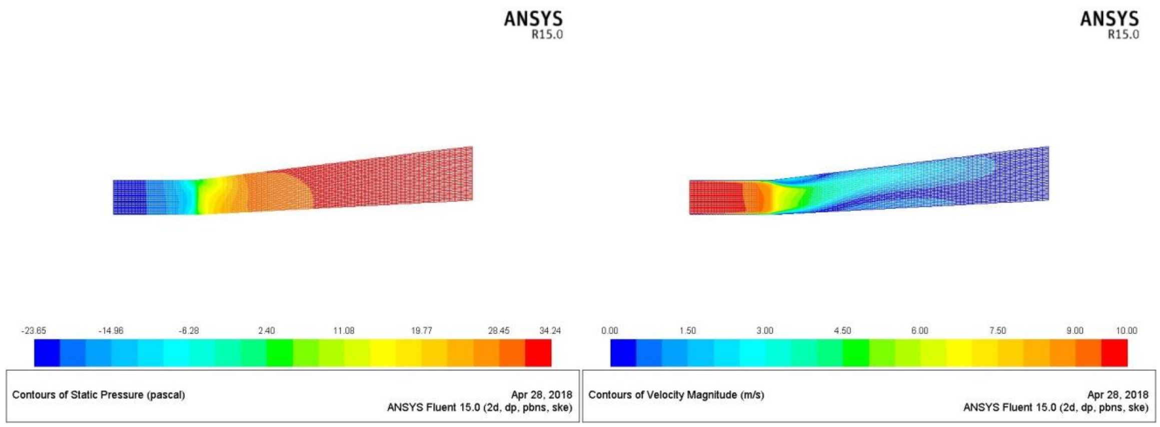


(a) (b)

Figure 8(a) and (b) – Contour plot of Pressure and velocity distribution along the length of annular diffuser.

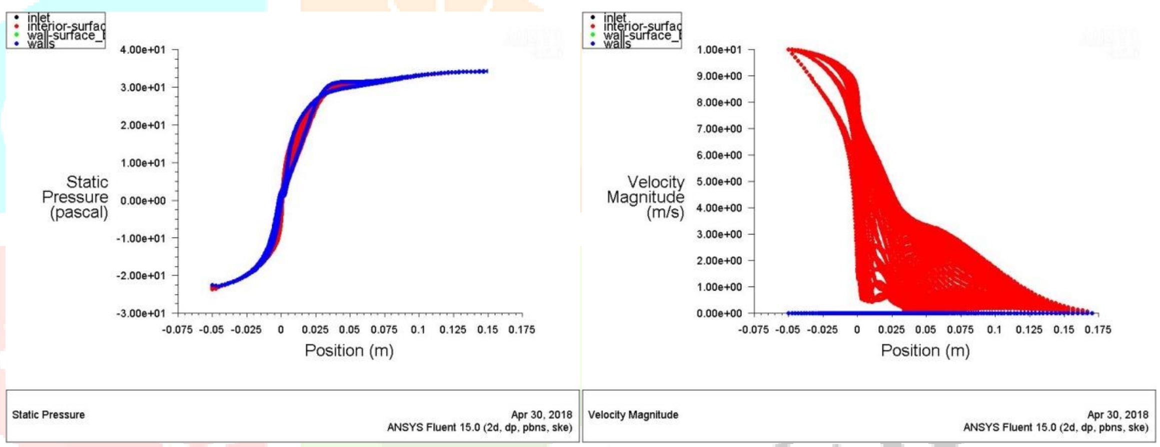
It is seen from the figure 7(a) and 8(a) that the pressure increases along the length of diffuser from -57 Pa at inlet to 2.7 Pa at outlet which happens in diffuser. Also, it is seen from Figure 7(b) and 8(b) that velocity decreases from 10 m/s at inlet to 0 m/s at outlet. The velocity is converted into pressure which comes out to be true using Bernoulli's theorem.

4.2 Results of annular diffuser with casing divergence angle = 7, hub divergence angle = 3, area ratio = 2.1 and inlet velocity = 10 m/s. Calculated Reynolds number = 28752.65.



(a) (b)

Figure 9(a) and (b) – Contour plot of Pressure and velocity distribution along the length of annular diffuser.

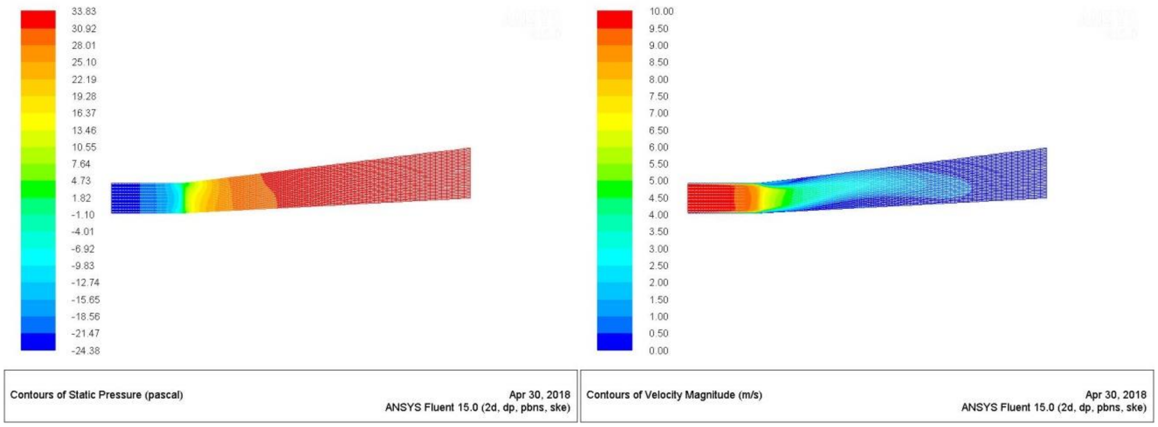


(a) (b)

Figure 10(a) and (b) – Graph of Pressure and velocity distribution along the length of annular diffuser.

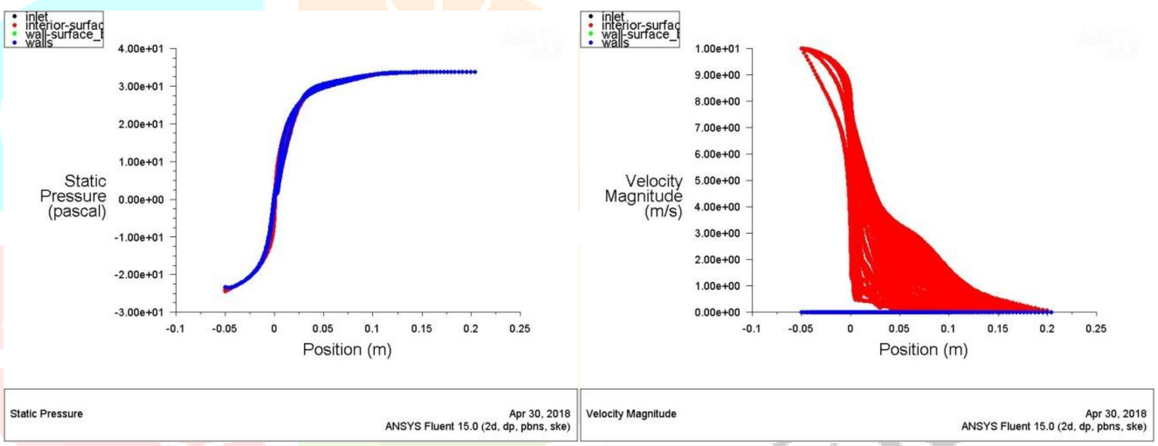
It is seen from the figure 9(a) and 10(a) that the pressure increases along the length of diffuser from -23.65 Pa at inlet to 34.24 Pa at outlet which happens in diffuser. Also, it is seen from Figure 9(b) and 10(b) that velocity decreases from 10 m/s at inlet to 0 m/s at outlet. The velocity is converted into pressure which comes out to be true using Bernoulli's theorem.

4.3 Results of annular diffuser with casing divergence angle = 7, hub divergence angle = 3, area ratio = 2.4 and inlet velocity = 10 m/s. Calculated Reynolds number = 28752.65.



(a) (b)

Figure 11(a) and (b) – Contour plot of Pressure and velocity distribution along the length of annular diffuser.



(a) (b)

Figure 12(a) and (b) – Graph of Pressure and velocity distribution along the length of annular diffuser.

It is seen from the figure 11(a) and 12(a) that the pressure increases along the length of diffuser from -24.38 Pa at inlet to 33.83 Pa at outlet which happens in diffuser. Also, it is seen from Figure 11(b) and 12(b) that velocity decreases from 10 m/s at inlet to 0 m/s at outlet. The velocity is converted into pressure which comes out to be true using Bernoulli's theorem.

5. Conclusion

5.1 Various conclusions can be drawn from this study. They are –

1. As area ratio increases the pressure at outlet decreases. So, it is beneficial to take area ratio of 2 for 10 m/s velocity inlet for given geometry for maximum pressure outlet.

6. References

[1] A Book on Fluid Mechanics by FM White.

[2] A thesis on Arbitary design of diffuser by Robert Donald Gapp.

[3] A thesis on experimental and numerical investigations of flow through an annular diffuser with and without strut by PRAKASH R (2013).

[4] Ehan Sabah Shukri, Basil Hassan Abdullah and Omer Muwafaq Mohammed Ali., 2017, "Velocity Distribution in an annular a diffuser Fitted with Twisted Hub for Different Area Ratios", 4th International Conference on Power and Energy System Engineering, Energy Procedia 141 , pp. 639-643.

[5] Ehan Sahab Shukri., "Numerical Comparison of Temperature Distribution Fitted with Helical Screw-Tape Hub and Pimpled Hub", 4th International Conference on Power and Energy System Engineering, Energy Procedia 141 , pp. 625-629.

[6] Y.H. Yau, K.S. Poh, A. Badarudin., "A numerical airflow pattern study of a floor swirl diffuser for UFAD system", Energy and Buildings , Energy and Building 158, pp. 525-535.

[7] Cherry, E. M., Padilla, A. M., Elkins, C. J., and Eaton, J. K., 2010, "Three-Dimensional Velocity Measurements in Annular Segments Including the Effects of Upstream Strut Wakes", International Journal of Heat and Fluid Flow, Vol. 31, pp. 569-575.

[8] Jakirlic´, S., Kadavelil, G., Sirbubalo, E., von Terzi, D., Breuer, M., and Borello., D., 2010, "14th ERCOFTAC SIG15 Workshop on Turbulence Modelling: Turbulent Flow Separation in a 3-D diffuser", ERCOFTAC Bulletin, Vol. 82. Khalid, S. A., Khalsa, A. S., Waitz, I. A.,Tan, C. S., Greitzer, [7]E. M., Cumpsty, N. A., Adamczyk, J. J., and Marble, F. E., 1999, "Endwall Blockage in Axial Compressors", ASME Journal of Turbomachinery, Vol. 121, pp. 499-509.

[9] Cherry, E. M., Elkins, C. J., and Eaton, J. K., 2008, "Geometric Sensitivity of Three-Dimensional Separated Flows", International Journal of Heat and Fluid Flow, Vol. 29, pp. 803-811.

[10] Sieker, O. and Seume, J. R., 2008, "Effects of Rotating Blade Wakes on Separation and Pressure Recovery in Turbine Exhaust Diffusers", Proceedings of ASME Turbo Expo 2008: Power for Land, Sea and Air, GT2008-50788.

[11] Stevens, S. J., and Williams, G. J., 1980, "The Influence of Inlet Conditions on the Performance of Annular Diffusers", ASME Journal of Fluids Engineering, Vol. 102, pp. 357-363.

[12] Wolf, S., and Johnston, J. P., 1969, "Effects of Non uniform Inlet Velocity Profiles on Flow Regimes and Performance in Two-Dimensional Diffusers", Transactions of the ASME, Journal of Basic Engineering, Vol. 91, pp. 462- 474.

[13] Sovran, G. and Klomp, E. D., 1967, "Experimentally Determined Optimum Geometries for Rectilinear Diffusers with Rectangular, Conical or Annular Cross-Section", Fluid Mechanics of Internal Flows, Elsevier Publishing, New York, pp. 270-319.