

ANALYSIS OF FLOW ON ANNULAR CURVED DIFFUSER BY USING CFD

Sangappa Hadapad¹, Basavaraj Dotihal²

¹Asst. Professor, Mechanical Engineering, Balekundri Institute of Technology Belgaum, Karnataka, India

²Asst. Professor, Mechanical Engineering, Tontadarya College of Engineering, Gadag, Karnataka, India

Abstract

The integral component of the gas turbine engines are curved annular diffusers at high speed air craft. By reducing the total pressure loss the diffuser facilitate effective operation of the combustor. The performance of these diffusers depends on their geometry and inlet conditions. In the present investigation the distribution of static pressure and total pressure are studied with the help of CFD on a curved annular diffuser of 30° , 35° and 40° angle of turn and also by varying the circular hub of diameter to 5mm, 7.5mm, 10mm, 12.5mm, 15mm and 20mm and allowing the air as fluid to pass through the diffuser. ANSYS FLUENT 14.5 is used for this assessment and Turbulence model as standard $k-\epsilon$ also used. In this investigation it was observed that for a small angle of turn the static pressure recovery is more and as the angle of turn increases static pressure recovery reduces and also the total pressure loss is more. And it was also observed that for an angle of 30° with 20mm hollow diameter the static pressure is better and total pressure is less.

Keywords: Annular Curved Diffuser, CFD, Ansys Fluent

1. INTRODUCTION

Diffusers are reducing the velocity of incoming air. These are installed under the wings, base of vertical stabilizer; root of the wings and under the fuselage. The main function of diffuser is to convert dynamic pressure into static pressure without loss of total pressure. Diffuser operation and design mainly depend on the subsonic and supersonic entry flow. Diffuser is annular adiabatic duct, no heat and work transfer, total temperature remains constant in this study and the performance depends in terms of temperature and pressure rise (as well as geometry of curved annular duct and inlet conditions).

Curved annular diffuser can be of two classes (1) diffusers whose axis is straight but whose walls are curved in a bell or trumpet shape or (2) diffusers whose axis is curved so that diffuser is the sum of the bend and straight axis diffuser. Tested by Calsons et al [1] indicate that there is little advantage in utilizing a trumpet or bell wall shape over a straight wall in straight in a straight axis, two dimensional ; diffuser moreover the straight wall design is for easier to construct.

The losses in a curved axis diffuser (a diffusing bend) exceed those in a comparable straight axis diffuser. The pressure recovery is lower and the bending of the flow results in an exit misdistribution with higher flow velocity on the outside of the bend. However a curved axis diffuser can be more efficient than the combination of straight axis diffuser and bend.

2. LITERATURE REVIEW

Annular curved diffusers are common devices, however, their performance and flow characteristics depends on their geometry and inlet conditions. Annular curved diffusers have been investigated by many researchers due to their wide spread usage some of which are listed in brief. **Prasanta K Sinha et al** [2] investigated flow through annular diffusing duct numerically and experimented mean velocity distribution. They changed the static pressure and total pressure on an annular curved diffuser of 30° angle and ratio 1.28 and centre line length was chosen three times of inlet diameter. Compare the results for validation, by varying different area ratios from 1.15 to 3.75 with same centre line length and inlet diameter. By this investigation they found increase in static pressure recovery up to 2.86 and between areas 2.86 to 3.75. Recovery pressure decreases steadily. Change in area ratio for same inlet condition, the coefficient of total pressure loss remains constant. **A. Sheeba et al** [3] Studied on annular pre diffuser for marine turbine combustor on the effect of strut configuration analyzed by CFD, also investigating numerically and experimentally of straight cone annular diffuser normally used in marine gas turbine combustor and flow in annular diffuser. The investigation completely focused on the distortion of the flow produced by the strut. **S.N Singh et al** [4] studied the performance of S shaped rectangular diffuser by momentum injection by CFD. During analysis they found flow distribution and pressure recovery improved significantly because they changed the diffuser having aspect ratio of 6 at inlet. And 3cm diameter placed inflexion plane across width of diffuser and different speeds to inject momentum to the decelerating flow as a result of boundary layer separation.

3. METHODOLOGY AND OBSERVATIONS

In this study the annular curved diffuser is modeled by using CATIA V5R20, and is meshed in ICEM CFD and appropriate boundary conditions used for annular curved diffuser. Standard k- ϵ method is used in this analysis and analysis is done by using ANSYS FLUENT 14.5 software and static pressure recovery and total pressure recorded in this investigation.

3.1 Methodology

- The basic geometry of annular curved diffuser has been taken from the literature Survey.
- A 3-D model of the basic geometry was created by using software CATIA V5R20.
- Modification was done in the annular curved diffuser by inserting the circular hub with different diameter of 5mm, 7.5mm, 10mm, 12.5mm, 15mm and 20mm and also angle of turn changes like 30° , 35° , and 40° .
- In the modification circular hub is placed in the hollow section of the basic annular curved diffuser.
- The created 3-D model was imported to ANSYS 14.5 work bench in the form of stp format.
- Meshing was done by using ICEM CFD.
- The standard k- ϵ model used.
- Analysis was done by using ANSYS FLUENT 14.5 software.
- Finally obtaining the pressure and velocity contours. Also plot the graph of static pressure recovery.

3.2 Model

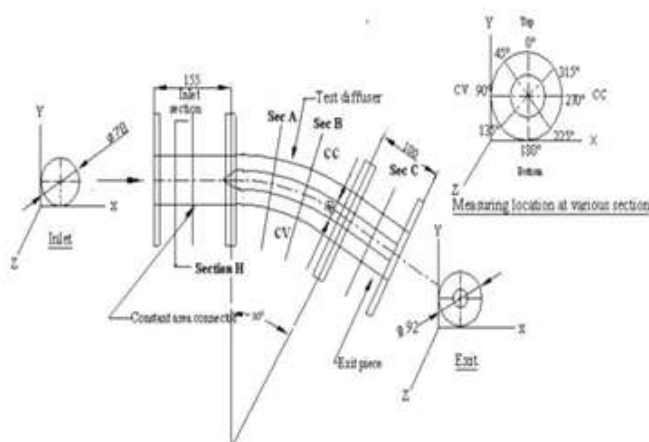


Fig 3.1 Basic geometry of annular curved diffuser.

Figure 3.1 shows the basic geometry of annular curved diffuser. The basic geometry was taken according to the literature survey and a 3-D model was created by using CATIA V5R20.

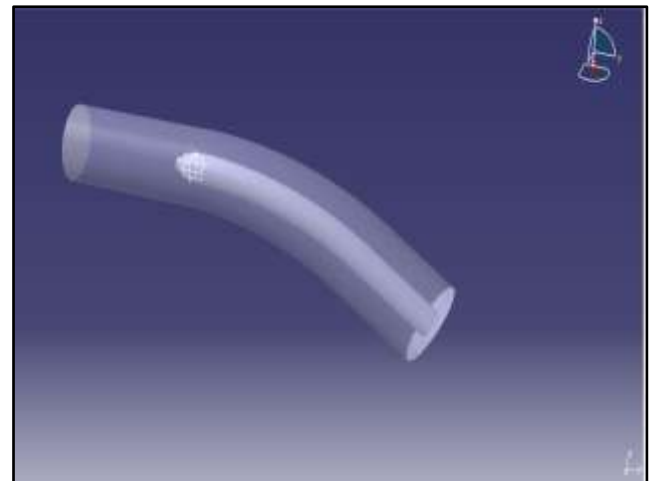


Fig 3.2 CATIA model of basic geometry of annular curved geometry.

Reynolds number 2.45×10^5 based on inlet diameter [4].

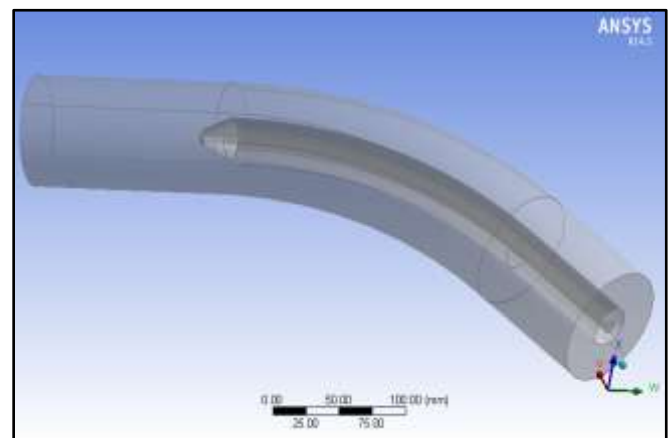


Fig 3.3 Modification model of basic annular curved diffuser

Figure 3.3 shows the modification model of the annular curved diffuser. Modification was done by inserting the different diameters of circular hub like 5mm, 7.5mm, 10mm, 12.5mm, 15mm and 20mm inside the hollow section of the basic geometry and also angle of turn changes like 30° , 35° and 40° .

3.3 Meshing

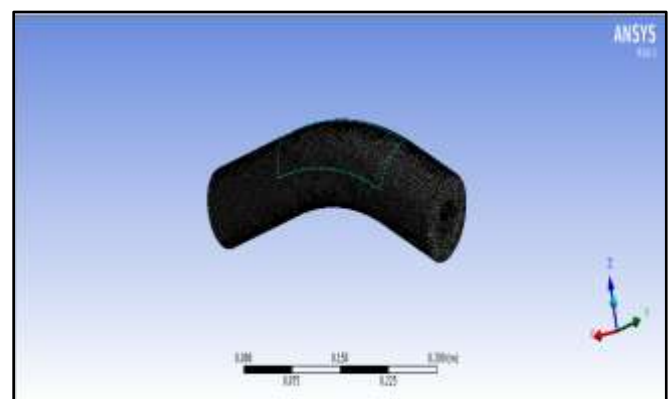


Fig 3.4 Meshing model of annular curved diffuser

Figure 3.4 shows the meshing model of annular curved diffuser. Meshing was done by ICFM CFD. In meshing inflation method was used for good meshing.

3.4 Boundary Conditions

Mesh Model

Solver type	: Pressure based
Problem model	: Standard $k-\epsilon$
Fluid	: Air
Velocity	: 47m/sec
Outlet gauge pressure	: 0 Pa
Wall condition	: stationary and no slip condition
Reynolds number	: 2.45×10^5

Standard k-ε model is used for all model analysis. Air is used as the fluid. At the inlet, constant velocity condition is given for all model analysis and ambient pressure is used as the exit boundary condition. Stationary and No-slip conditions are assumed on the wall of all models.

3.5 Observations

- In the present investigation observed that the static pressure recovery, total pressure loss and percentage of head loss on basic curved annular diffuser and also on modification curved annular diffuser.
- Flow characteristics were observed in pressure and velocity contour.
- Here observed where the pressure recovered more in the annular curved diffuser.
- In this investigation on percentage of head loss and total pressure observed.

4. RESULTS AND DISCUSSIONS

In this study flow characteristics is observed by plotting the pressure and velocity contours for all models. Static pressure recovery and total pressure loss observed by plotting the graphs for all models.

4.1 Stream Lines of Annular Curved Diffuser

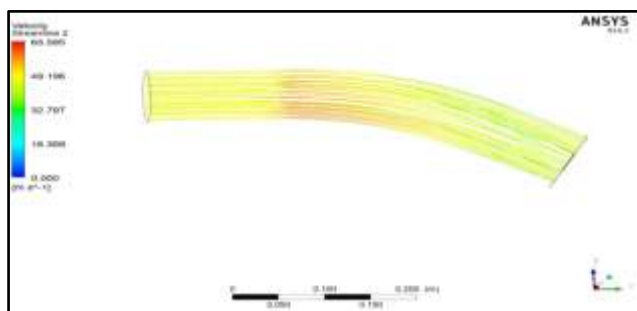


Fig 4.1 Stream lines of annular curved diffuser.

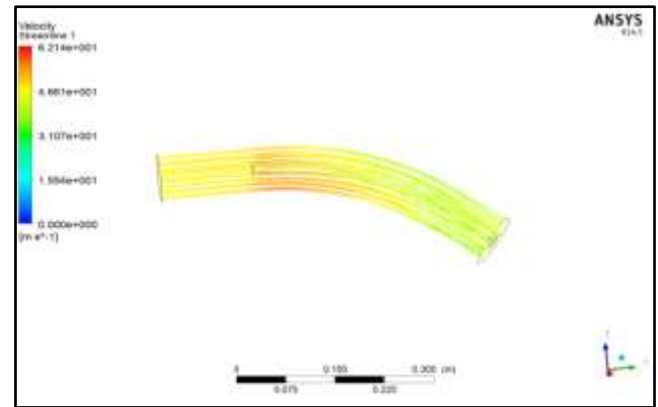


Fig 4.2 Stream lines of annular curved diffuser by inserting circular hub.

Figure 4.1 and 4.2 show the velocity stream lines of annular curved diffuser and curved diffuser by inserting circular hub respectively. From the figure 4.1 the stream line is uniformly distributed at inlet of the diffuser, when the hollow section starts stream lines suddenly changes and stream line decreases towards the outlet of the diffuser. From the figure 4.2 the velocity stream line uniformly flows at inlet, when the inserting circular hub inside the hollow section starts the stream line flow gradually increases but towards the outlet of the diffuser stream line decreases.

4.2 Pressure Contours

4.2.1 Pressure Contour of Annular Curved Diffuser

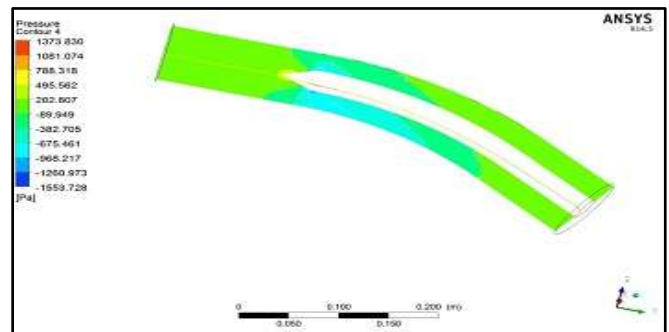


Fig 4.3 Pressure contour of basic annular curved diffuser

4.2.2 Pressure Contour of 30° Angle with Diameter 12.5mm Circular Hub of Annular Curved Diffuser

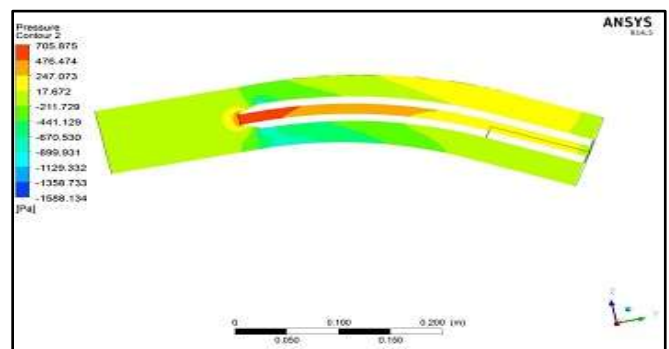


Fig 4.4 Pressure contour of annular curved diffuser with diameter 12.5 mm circular hub

4.2.3 Pressure Contour of 30° Angle with Diameter 15mm Circular Hub of Annular Curved Diffuser

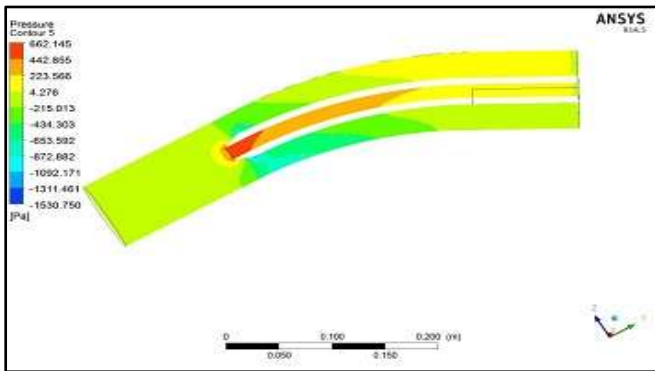


Fig 4.5 Pressure contour of annular curved diffuser with diameter 15 mm circular hub

4.2.4 Pressure Contour of 35° angle with Diameter 10mm Circular Hub of Annular Curved Diffuser

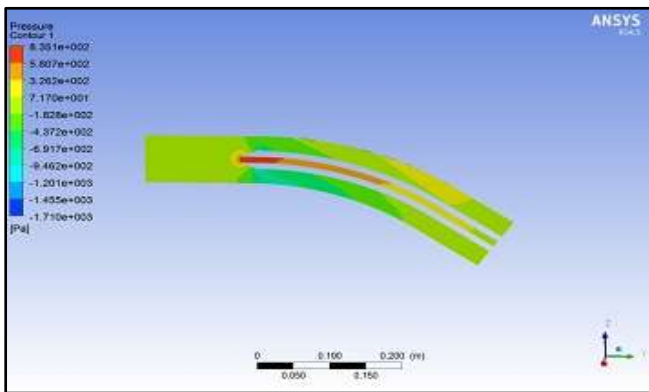


Fig 4.6 Pressure contour of annular curved diffuser with diameter 10mm circular hub

4.2.5 Pressure contour of 40° angle with Diameter 20mm Circular Hub of Annular Curved Diffuser

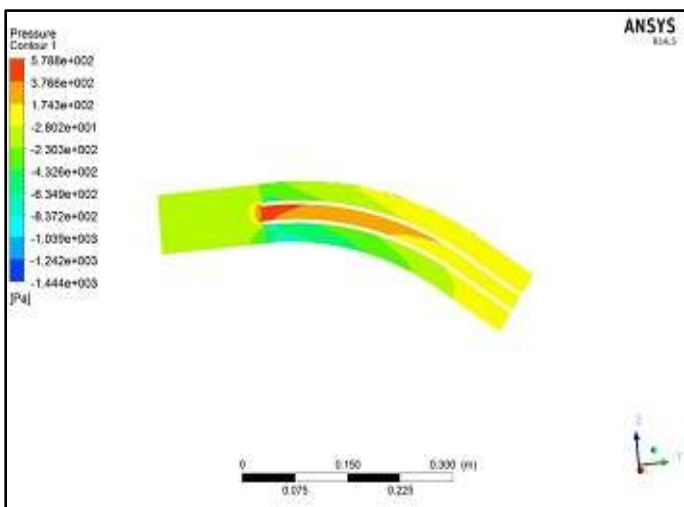


Fig 4.7 Pressure contour of annular curved diffuser with diameter 20mm circular hub

Figures 4.3 to 4.7 show the pressure contours of all geometric models for different circular hub with angles 30°, 35° and 40°. In the pressure contour the color code indicates the pressure variation from inlet of the diffuser to outlet. From all the pressure contours initially the pressure is less due to the high velocity at inlet. As the fluid flows along the diffuser the pressure gradually increases due to decrease in velocity. Inserting the circular hub with different diameter inside the hollow section of the diffuser it gives more pressure recovery than the basic annular curved diffuser. From all the pressure contours it is observed that when the hollow section starts the pressure drops and then it recovers as the geometry increases.

4.3 Velocity contours

4.3.1 Velocity Contour of Annular Curved Diffuser

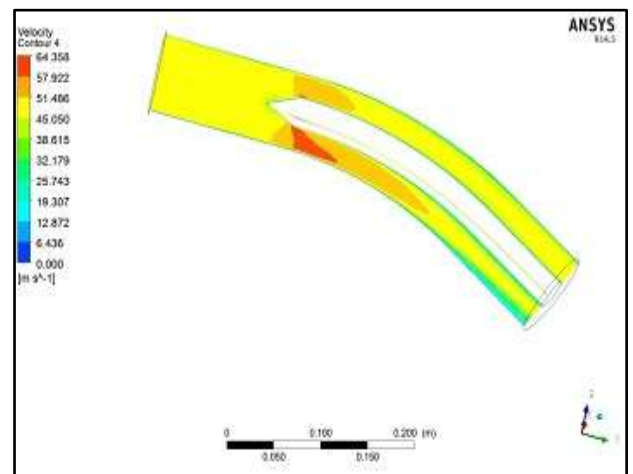


Fig 4.8 Velocity contour of basic annular curved

4.3.2 Velocity Contour of 30° Angle with Diameter 5mm Circular Hub of Annular Curved Diffuser

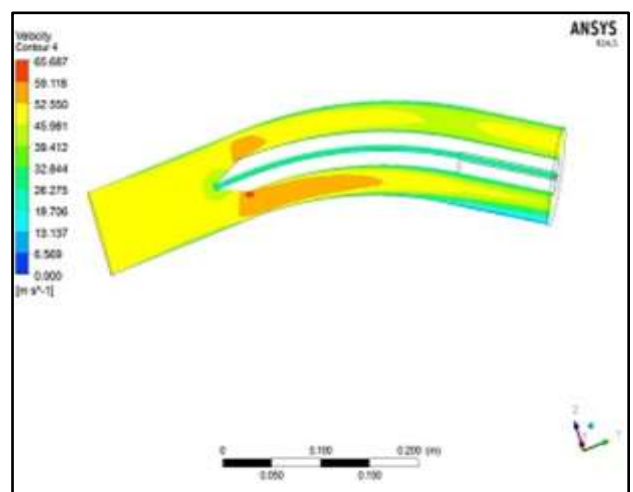


Fig 4.9 Velocity contour of annular curved diffuser with diameter 5mm circular hub.

4.3.3 Velocity Contour of 35° Angle with Diameter 5mm Circular Hub of Annular Curved Diffuser

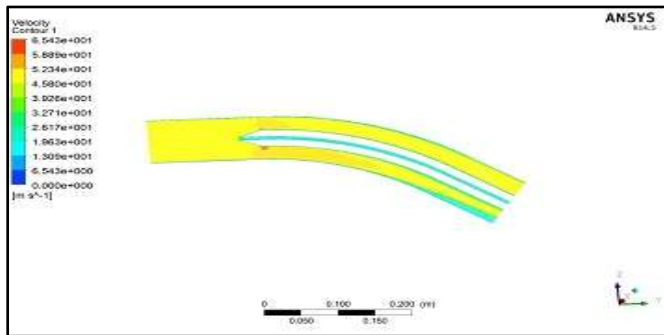


Fig 4.10 Velocity contour of annular curved diffuser with diameter 5mm circular hub

4.3.4 Velocity Contour of 40° Angle with Diameter 5mm Circular Hub of Annular Curved Diffuser

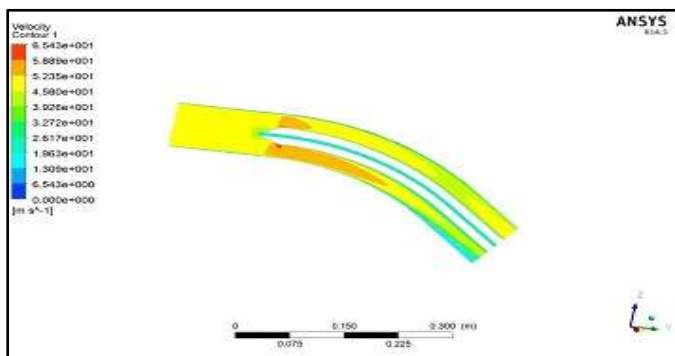


Fig 4.11 Velocity contour of annular curved diffuser with diameter 5mm circular hub.

Figures 4.8 to 4.11 Shows the velocity contours of all annular curved diffuser models. In the velocity contours the color code indicates the variation of velocity in the diffuser. From the above figures it is observed that velocity uniformly varies at the inlet of the diffuser, when the hollow hub starts velocity increases gradually but at the outlet it again decreases due to increase in area towards the outlet of the diffuser.

4.4 Graphs

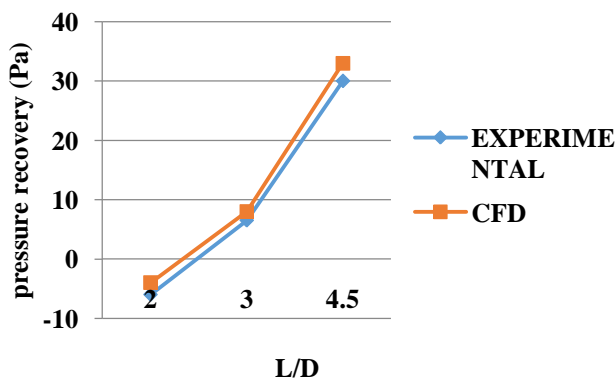


Fig 4.12 Comparison of CFD and experimental values for basic annular curved diffuser

Figure 4.12 shows the comparison between CFD values and experimental results for a basic annular curved diffuser model. The graph predicts that results of CFD values almost match with experimental results for basic annular curved diffuser model.

4.5 Comparison of Static Pressure Recovery for 300, 350 and 400 Angle

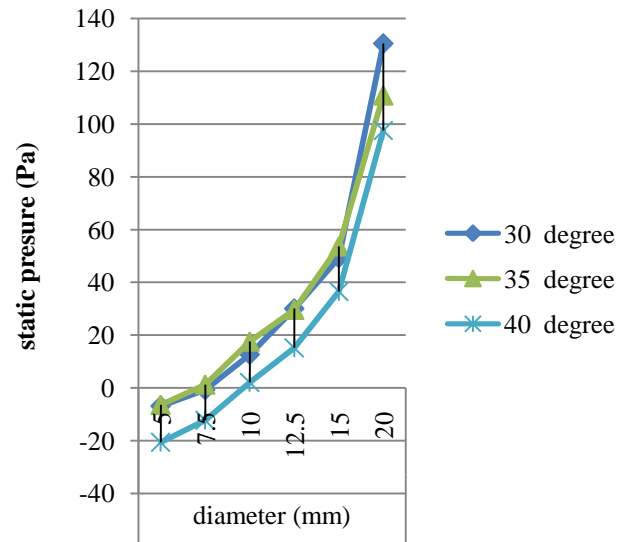


Fig 4.13 Pressure recoveries for 30°, 35° and 40° angle with different hub diameter

Figure 4.13 shows the comparison between static pressure recovery for 30°, 35° and 40° angle with different hub diameter 5mm,7.5mm,10mm,12.5mm,15mm and 20mm. The graph predicts that 30° angle of 20mm diameter annular curved diffuser with circular hub gives the more pressure recovery than the other models.

4.6 Calculations

Annular curved diffuser with circular hub diameter 5mm for 30° angle.

$$\text{Total inlet pressure} = 1358.42 \text{ Pa}$$

$$\text{Total outlet pressure} = 1096.79 \text{ Pa}$$

$$\text{Total pressure loss} = \text{Total inlet pressure} - \text{Total outlet pressure}$$

$$= 1358.42 - 1096.79$$

$$= 261.63 \text{ Pa}$$

Percentage of head loss

$$= \frac{\text{Total inlet pressure} - \text{Total outlet pressure}}{\text{Total inlet pressure}} \times 100$$

$$= \frac{1358.42 - 1096.79}{1358.42} \times 10 = 19.25\%$$

The total pressure loss and percentage of head loss for different circular hub diameter with angle of annular curved diffuser as shown in below table.

Table 1: Result sheet for annular curved diffuser

Angle (Degree)	Diameter (mm)	Velocity (m/s)	Static pressure (pa)	Total pressure (Pa)		Total pressure loss (Pa)	% of Head loss
				Inlet	Outlet		
30°	5	47	-6.8379	1358.42	1096.79	261.63	19.25%
	7.5		-0.7038	1338.92	1074.23	264.69	19.76%
	10		12.6849	1321.53	1057.4	264.13	19.98%
	12.5		30.0817	1352.33	1087.41	264.92	19.58%
	15		49.3248	1302.27	1037.14	265.13	20.35%
	20		130.529	1221.37	977.925	243.445	19.93%
35°	5	47	-6.4170	1358.31	1087.69	270.62	19.92%
	7.5		1.2857	1350.61	1077.84	272.77	20.19%
	10		17.5849	1334.31	1064.45	269.86	20.22%
	12.5		29.6295	1322.27	1047.92	274.35	20.74%
	15		53.657	1298.25	1027.85	270.4	20.82%
	20		110.87	1241.03	979.207	261.823	21.09%
40°	5	47	-20.5597	1372.44	1084.07	288.37	21.01%
	7.5		-12.2272	1364.13	1079.46	284.67	20.86%
	10		2.1101	1349.78	1066.25	283.53	21.47%
	12.5		15.1964	1336.69	1049.66	287.03	21.47%
	15		36.567	1315.33	1029.17	286.16	21.75%
	20		97.525	1254.39	981.074	273.316	21.78%

5. CONCLUSION

Based on the present investigation the following conclusions have been drawn.

In the present investigation the CFD values almost matches with experimental results for basic curved annular diffuser model.

- The modified model gives more static pressure than the basic annular curved diffuser.
- By inserting circular hub in the hollow section of the annular curved diffuser also gives more static pressure recovery.
- The diameter 20mm of circular hub of 30° angle gives the more static pressure (130.529Pa) recovery than the other annular curved diffuser.
- In this analysis observed that as the angle of turn increases the static pressure recovery goes on decreases and total pressure loss is also decreases.
- The percentage of head loss increases as the angle of turn increases from 30° to 40° angle.
- The percentage of head loss is minimum for the diameter 5mm of circular hub for 30° angle of annular curved diffuser.
- From the present investigation it is concluded that inserting the circular hub inside hollow section of the annular curved diffuser gives more satisfactory values of static pressure recovery than the basic annular curved diffuser model.

REFERENCES

- [1]. Carlson J J et al “effects of wall shape on flow regimes and performance in straight two dimensional diffusers” J basic eng. 89, 151-161(1967)
- [2]. Prasanta K Sinha, Ananta K Das, and Bireswar Majumdar. “Numerical investigation of flow through Annular diffusing duct” International Journal of Mechanical & Industrial Engineering, Volume-1 Issue-1, 2011.
- [3]. A. Sheeba and V. Ganesa. on “Modeling of annular pre diffuser for marine gas turbine combustor using cfd-a study on the effect of strut configuration” Journal of Naval Architecture and Marine Engineering December, 2005.
- [4]. S.N Singh, V. Seshadri, Sunil Chandel, Mahendra Gaikwad. “Analysis of the Improvement in Performance Characteristics of S-Shaped Rectangular Diffuser by Momentum Injection using CFD” Engineering Applications of Computational Fluid Mechanics Vol 3, No.1, pp. 109-122 (2009).