Design And Optimization Of Flow Performance For Annular Curved Diffusing Duct Using CFD

Mohammed Maqsood[1], Kailash B A [2], Gowreesh Subramanya [3]

[1] P.G Student, Dept. of Thermal Power Engineering, VTU PG Centre Kalaburagi, Karnataka, India
[2] Asst. Professor, Dept. of Thermal Power Engineering, VTU PG Centre Kalaburagi, Karnataka, India
[3] Associate Professor, Dept. of Mechanical Engineering. JSSATE Bangalore, Karnataka, India

---***---

Abstract - *Curved annular diffusers are the main component of the gas turbine engine for high speed aircraft. The performance of such diffuser depends on inlet conditions and geometry. In the present study different arbitrary shapes are tried and results were compared for pressure recovery. Four different arbitrary shapes of diffuser are tried and results are presented here.*

Experimental set-up being very costly and time consuming, CFD techniques are considered here which is a best replacement for experimental. CFD Commercial tool used for this project. From the CFD results it is found that the third model with an arbitrary shape of the diffuser has given the best pressure recovery compared to the other models and compared to literature. Hence the third model can be considered as the optimized model.

Key Words: : *Diffuser, CFD, Pressure recovery, diffuser shapes, k-epsilon turbulence model.*

1. INTRODUCTION

It is today's need to extract as much energy as possible from given input volume of petroleum; as the standards of carbon emission are very strict and it's our job to design a most common components which are deals with this problem, one of the most common used components in Gas engine particularly is diffuser, diffuser used to increase static pressure and maintain high outlet area as required. I was interested to enhance the performance of given standard curved diffuser.

In biological sense diffusion is the process of moment of molecules from high concentration to low concentration high pressure to low pressure it's not a flow, it's not convection, it's not pumped, not slid, it is just like a soaking. In Aerospace engineering diffusion means conversion of kinetic pressure energy in to static pressure Energy. Consider nozzle, this works as flow velocity is less at inlet and velocity is increased at outlet so work is not extracted nor inputted so inlet KE should get conserved in something this is the increase in acceleration at outlet and keeping the total energy constant.

Part turn or curved diffusers are used in wind tunnels, compressor crossover, air conditioning and ventilation ducting systems, plumes, draft tubes, etc. The objective of the present study is to investigate the flow characteristics within a circular cross sectioned annular curved diffuser and enhance its performance. The performance of an annular curved diffuser is characterizes by static pressure recovery and total pressure loss coefficient. As mentioned diffuser are used to recover static pressure especially in high speed gas turbine engine of aircraft. And performance is highly depends on geometrical shapes of the diffuser and inlet conditions. They are shaped as requirement ultimately   for effective operation of the combustor by reducing the total pressure loss. There are lot of Research already made on the straight and curved shape performance but different geometries are not tried to enhance the performance.

1.1 DIFFUSER WORKING

Diffuser are works on exactly opposite of venturi effect which means when high velocity air is passed from smaller area to larger area then its velocity decreases and static pressure increases this processes on which diffusers are works. The velocity is increasing as we go towards outlet (larger area) from inlet (smaller area)this indicates that pressure is decreasing from inlet to outlet in Nozzle that is exactly opposite to that of diffuser means diffuser are reverse of nozzle.

For example diffusers are widely used in formula 1 car design to create more down force by providing diffuser at the rear end of the car, as car speed is high the air is flowing under the car is at very high speed then at the rear of the car sudden expansion duct is provided this causes decease in velocity and produces very high static pressure at rear end the high speed region is considered as vacuum which pull the car towards road producing high down force. The effective diffuser should create less turbulence and more increase in static pressure by keeping the geometry as simple and compact as possible.

1.2 Literature Review

P.K. Sinha and Mullick [1] Studied the parametric investigation of curved annular diffuser the increase in area ratio from1.15 to 2.15, static pressure recovery increases sharply and after this the pressure recovery increases slowly up to 2.8 and beyond the area ratio 2.8 the pressure recovery decreases steadily, after this they concluded that coefficient of pressure recovery for the computational investigation was obtained as 34% compared to the experimental investigation as 31%. The measurements were taken at Reynolds number 2.45×10^5 based on inlet diameter and mass average inlet velocity. Predicted results of coefficient of mass averaged static pressure recovery and coefficient of mass averaged total pressure loss are in good agreement with the experimental results. The Geometry was used is look like follows.

Inlet Diameter: 78mm
Outlet OD: 88mm
Outlet ID: 15mm
Center line radius: 430mm
Inclination Angle: 28°

R.Prakash, D.Christopher and K.Kumarrathinam [2] published a paper entitled "cfd analaysis of flow through a conical exhaust diffuser" in which different half cone angles for the diffuser was tried and the model with best pressure recovery was considered as the optimized model. The diffuser with the optimized geometry was then fabricated and tested. The experimental value of the diffuser pressure recovery coefficient was calculated and compared with the theoretical value. It was found that the diffuser with a half cone angle of 70 provided the maximum recovery of static pressure, therefore proving to be the most efficient. A diffuser with this geometry was fabricated and was tested. Using the data collected, the experimental value of pressure recovery coefficient was calculated. The values of pressure recovery coefficient were compared and it was found that the experimental value was in close agreement to the theoretical value.

2. Problem Definition

To perform the steady state CFD analysis for diffuser with different geometric consideration. The simulation is performed in ANSYS 15.0 and is based on the use of governing equation of Mass. Momentum, energy and turbulence. K-Ɛ turbulence model is used for this assessment due to wide applications in internal flows.
To construct a suitable geometry for the diffuser this helps in increase in static pressure at the outlet.

To compare different cases of diffuser with different shapes and conclude on the best shape of the diffuser with highest pressure recovery.

3. Specification of working Models

Geometric dimension of diffuser is shown below.

Unit of dimension is in millimeter.

Steps used for the generation of 3-D model are as follows:-

➢ Create 3-D geometry using model option in toolbar. There are different types of tool available for the creation of geometry such as sketching, extrude, revolve etc.
➢ The file format in Design Modeller is .agdb.

There are different designs which have been considered for the analysis.

Dimensions of the diffuser

The test diffuser was designed with increase in area from inlet to exit and it distributed normal to the centerline as suggested by Prasanta Sinha and Anantdas [1]. Centerline was turned at 30° from inlet to exit with inlet diameter of 78 mm and exit diameter of 92.
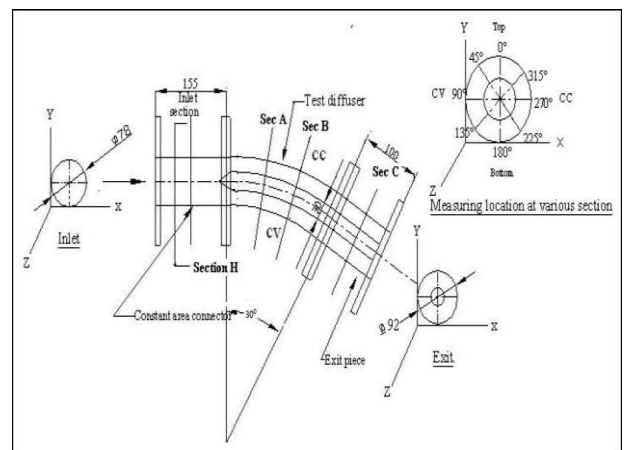

Fig -1: Basic working model

3.1 Methodology and CFD Simulation

This chapter focuses on the methodology adopted to achieve the objectives set in chapter one. The main objective is to get the values of pressure and velocity at the inlet and outlet of the diffuser. To achieve this requirement CFD analysis has been done for flow in an annular curved diffuser at Reynolds number 2.45×10^5.

The numerical results extracted from CFD simulations are compared and presented. A grid-independence study is performed to confirm the results are independent of the grid size. Time independent calculations for the flow have

been performed in three dimensions for turbulent flow and k-ε turbulence model has been used.

Table -1: properties of fluid materials

Fluid	Air
Density	1.25 kg/m^3
Viscosity	0.001002 kg/ms

Table -2: Boundary conditions

Inlet	35 m/s
Outlet	0 Pa
Cylinder	No-Slip Wall

4. Results and Discussions
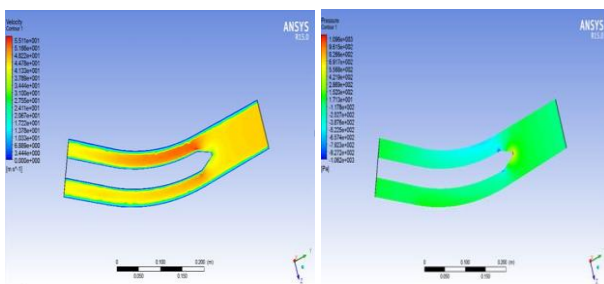

Fig -2:Velocity contour   **Fig -3**:Pressure contour
for model-1   for model-1
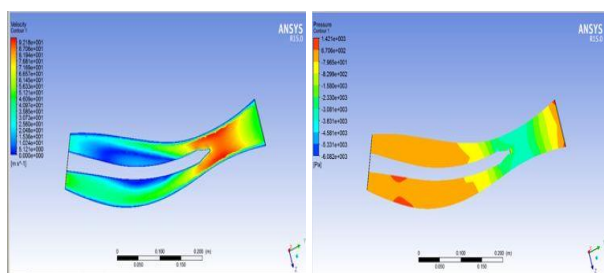

Fig -4:Velocity contour   **Fig -5**: Pressure contour
for model-2   for model-2
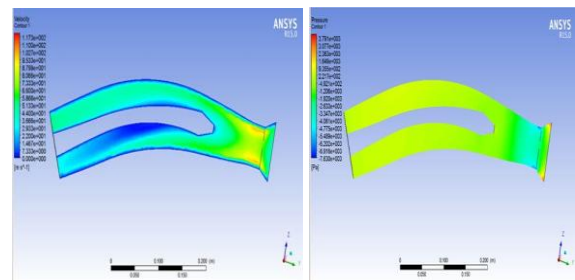

Fig -6:Velocity contour   **Fig -7**:Pressure contour
for model-3   for model-3
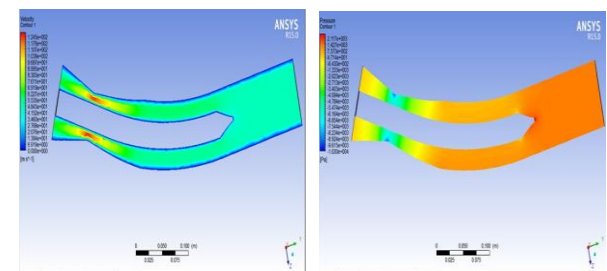

Fig -8:Velocity contour   **Fig -9**:Pressure contour
for model-4   for model-4

Figure above shows the results for the modified case of diffuser. Velocity streamlines, pressure contour and velocity contours are shown above. From the streamlines figure it is found that the flow of air is uniform without reversals and irregularity. The increase in pressure is uniform from inlet to outlet. In this model the pressure at the inlet is 16 Pa and at the outlet the pressure is 935 Pa and hence the increase in pressure is 920 Pa.. So model 3 is preferred for practical applications.

Table -3: Pressure Reading For All 4 Cases in (Pa)

Models	Average Inlet Gauge Pressure (Pa)	Average Outlet Pressure (Pa)	Pressure Rise (Pa)
Model 1	17	152	135
Model 2	20	680	660
Model 3	16	935	919
Model 4	15	45	30

Table -4: Velocity Reading For All 4 Cases in (m/s)

Models	Average Inlet Velocity (m/s)	Average Outlet Velocity (m/s)
Model 1	35	31
Model 2	35	27
Model 3	35	24
Model 4	35	84

5. CONCLUSION

From this CFD simulation we tried to find the best shape and size of the diffuser. We can observe that as the shape of the diffuser was changed in second and third case there was significant increase in the value of the pressure but in fourth case when there was change in shape of the diffuser in such a way that the area near the outlet was made smaller there was decrease in the value of static gauge pressure as due to contraction of the area of the diffuser there was increase in the value of velocity which led to the decrease in the static pressure..

➤ The velocity in the first three cases was decreasing gradually from inlet to outlet due to expansion in the model.
➤ For fourth case there was an increase in the value of velocity and hence decrease in the pressure and hence this model is not preferred for diffusers.
➤ In all the first three cases, the first model gives a pressure increase of 120 Pa. The area average gauge pressure at the inlet is 17 Pa and at outlet the area average gage pressure is 152 Pa
➤ In second model the area average gauge pressure at the inlet is 20 Pa and at the outlet the area average gauge pressure is 680 Pa and hence there is a pressure rise of 660 Pa.
➤ In third model the area average gauge pressure at the inlet is 16 Pa and at the outlet the area average gauge pressure is 935 Pa.
➤ In the fourth model with narrow area near the outlet the pressure at the inlet is 200 Pa and at the outlet the pressure is 47 Pa and hence there is a decrease in the value of the pressure and hence is not a good model for diffuser..

REFERENCES

[1] P.K. Sinha and Mullick's Analysis of Flow Performances through Annular Curved Diffusing Duct Using CFD

[2] R.Prakash, D.Christopher and K.Kumarrathinam published a paper entitled "cfd analaysis of flow through a conical exhaust diffuser"

[3] R.KEERTHANA and G.JAMUNA RANI's "FLOW ANALYSIS OF ANNULAR DIFFUSERS" Vol. 2, Issue 3, May-Jun 2012, pp.2348-2351 IJERA.

[4] Prasanta K Sinha, 2Ananta K Das, 3Bireswar Majumdar's "Numerical Investigation of flow through Annular diffusing duct"

[5] P. K. Sinha1, A.N. Mullick, B. Halderand B. Majumdar "Analysis of Flow Performances through Annular Curved Diffusing Duct Using CFD"

[6] Applied fluid dynamics handbook "theoretical diffuser performance", General considerations in diffuser Analysis.

[7] Design, fabrication of subsonic, supersonic diffuser by Air force institution of Technology.

[8] R.W. Fox, and S.J. Kline, "Flow Regimes in Curves Subsonic Diffuser," Trans ASME, Journal of Basic Engineering, Vol. 84, 1962, pp. 303 – 316.

[9] J. Seddon, "Understanding and Countering the Swirl in S- ducts;" Test on the Sensitivity of swirl to Fences, Aeronautical journal, 1984, pp. 117-127.

[10] Vakili, J.M. Wu, P. Liver, and M.K. Bhat, Experimental Investigation of Secondary Flow in a Diffusing S- Duct, University of Tennessee Space Inst, Report No. TRUTSI 86/14, University of Tennessee, Tullahoma, TN. 1984.