Experimental and Computational Investigations of Flow Through Diffuser with Perforated Plates

P. Karthikeyan
Assistant Professor
Dept of AERO-
Apollo Eng College.

Ajith Kumar
Dept.of AERO-
Apollo Eng College.

A. Aakash
Dept.of AERO-
Apollo Eng College.

A. Abhilash Meshak
Dept.of AERO-
Apollo Eng College.

V. Vimalprakash
Dept.of AERO-
Apollo Eng College.

***Abstract:-* Experimental and computational investigation made in an symmetric 7° conical diffuser model. The flow through the diffuser is studied with perforated plate . A simple rectangular perforated plate is placed at the inlet of the diffuser. The velocity at the near wall regions of the diffuser are well captured with the probe. The measurements along the entire length of the model at different stations, allow the determination of axial mean velocity component and pressure field parameters, which provide comprehensive information to aid and understand such complex flows. Performance and flow characteristics along the centreline of diffuser are presented. The computational analysis of the flow through the diffuser shows an considerable increase in the pressure by the use of plates. .The predicted axial variations ofstatic pressure along the diffuser are analyzed usingANSYS FLUENT and the modeled was carried out usingCATIA V5.**

NOMENCLATURE

M_a=Flight Mach number

P_0 = Stagnation pressure

P = Static pressure

A_a = Capturing area

$\dot{m}a$ = mass flow rate

M_i =inlet mach number

T_o = Stagnation temperature

D_i = Diffuser inlet diameter

D_o =Diffuser outlet diameter

η_d =Isentropic efficiency

C_p =Coefficient of pressure

π_d =Total pressure ratio

P_{02} =Outlet stagnation pressure

P_{0a} =Inlet stagnation pressure

h =Enthalpy

γ =Heat capacity ratio

1INTRODUCTION

1.1 DIFFUSER

The diffuser is the divergent section of the engine after the compressor and before the combustion section . It has the all-important function of reducing high-velocity compressor discharge air to increased pressure at a slowervelocity.This prepares the air for entry into the flame burning area of the combustion section at a lower velocity so that the flame of combustion can burn continuously. If the air passed through the flame area at high velocity, it could extinguish the flame.Diffusers are widely used in fans, pumps, turbines, compressors and many other fluid machines. In its simplest form, a diffuser is a diverging passage in the flow direction, in which the kinetic energy is converted to pressure energy by decelerating the flow. This energy conversion process of the diffuser decides the fluid machine performance. Types of diffusers are conical, subsonic Supersonic. Other aerodynamic design considerations important in diffuser section arise from the need for a short flow path, uniform flow distribution and low drag loss. In addition to critical aerodynamic functions, the diffuser also provides: Engine structural support, including nacelle, Support for the rear compressor bearings and seals, Bleed air ports, pressure and scavenge oil passages for the rear compressor and front turbine bearings, mounting for the fuel nozzles.

1.2 PERFORATED PLATES

Modern architecture leverages a broad range of building materials to improve both functionality and appearance. Perforated metal plates are increasingly seen as an excellent option for both. Accurate Perforating, a leader in perforated plate design, has the skills, expertise and resources to help you get the most out of your design. Perforated plated are panels crafted with rows of perforations, highly versatile building materials, visually appealing materials.

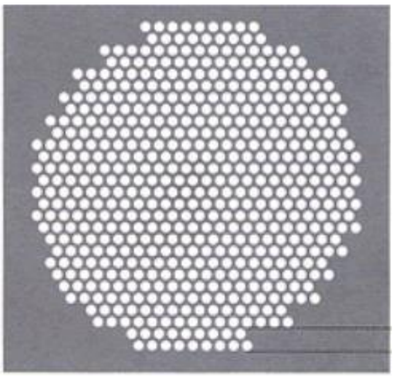

FIG : 1.2.1 PERFORATED PLATES

The wide applications of the perforated plates are: building design, architecture, machinery ventilation, industrial filtration, acoustical applications and balancing natural light and shade, providing privacy by limiting visibility. These plates can reduce the velocity and increase the pressure of the air entering the compressor up to a considerable extent.

2.PRINICPLE AND CURENT WORKS

The purpose of the diffuser is to reduce the kinetic energy and increase the pressure energy. The increase in the pressure can produce, an increase in the pressure of the air entering the compressor. The maximum recovery of static pressure without substantial loss of total pressure must be achieved in order to obtain good performance. This increase in pressure causes the efficiency of the engine to increase. So we need to find a way to increase the pressure at the exit of the diffuser. The flow separation can get initiated at three possible locations: external intake on the nacelle, diffuser internal surface, central body(hub). The task of the air intake is to channel the flow at low velocity through the compressor (or to the combustor in the case of the ramjet) without causing the detachment of the boundary layer. The air intake must be designed to provide the engine the required flow rate and also so that the output of the dynamic intake flow entering the compressor is uniform, stable, and with "good quality". So the goals of the Inlet are: increase the pressure, uniform flow at upstream of the compressor, minimal loss of total pressure and aerodynamic disorder. The diffuser performance depends on isentropic efficiency, stagnation pressure ratio, distortion coefficient. The efficiency of the diffuser depends on the pressure of air entering the engine, the capture area of the inlet and the speed of air. The aim of the Inlet is to slowing down the flow, therefore its shape is divergent. The aim is to reduce the length of the inlet as much as possible to reduce the weight and to minimize the losses of total pressure.Because the loss of total pressure is proportional to the length.. The ideal condition is that with an undisturbed air flow in entrance to the Inlet, thus M1=Ma, in this case the thermodynamic variables are equal to the environmentalones.Generally this condition is not satisfied.

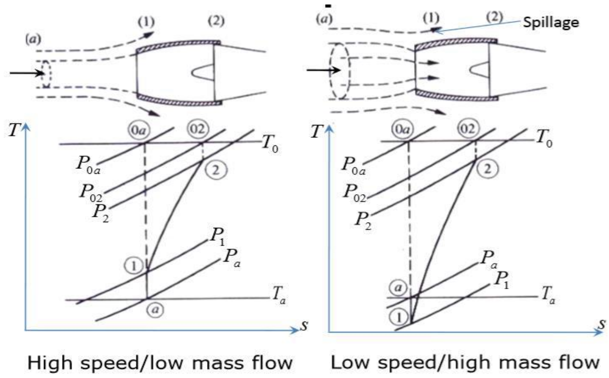

FIG: 2.1 DIFFUSER WITH DIFFERENT AREA RATIO

By knowing the altitude and the speed (for the hypothesis of isentropic flux, T0 and p0 are constant) and the flow rate ma requested by the engine, can be used this relation to calculate M1 by replacing A with A_1 and M_2 by replacing A with A_2:

$$\dot{m}_a = \frac{P_0 AM}{\sqrt{\gamma R T_0}} \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{\gamma+1}{2(\gamma-1)}}$$

From the flight Mach number Ma is possible to calculate the section of tube at infinite (at upstream), the so called "catching area" Aa. M_1 can be different with respect to M_a.$M_1 < M_a$ the flux tube at upstream of the Inlet is also divergent. At low speed, $M_1 > M_a$, that means the flux at upstream is convergent. Aa >A1 implies that the flux accelerates externally, thus slow down the flux, while Aa<A1 it has to be avoided because causes an increment of the drag. The ideal condition is Aa=A1.

Isentropic efficiency, ηd, of the diffuser is,

$$\eta_d = \frac{h_{02s} - h_a}{h_{0a} - h_a} \cong \frac{T_{02s} - T_a}{T_{0a} - T_a}$$

Stagnation pressure ratio or pressure recovery is the ratio of the outlet stagnation pressure to the inlet stagnation pressure.

$$\pi_d = \frac{P_{02}}{P_{0a}}$$

Isentropic efficiency can be related to the total pressure ratio (πd) and Mach number,

$$\eta_d = \frac{\left(1 + \frac{\gamma-1}{2} M^2\right) \pi_d^{\frac{\gamma-1}{\gamma}} - 1}{\left[\frac{\gamma-1}{2}\right] M^2}$$

The relationship between the dimensionless coefficient and the dimensional numbers is,

$$C_p = \frac{P - P_\infty}{\frac{1}{2}\rho_\infty {V_\infty}^2}$$

3. DESIGN AND EXPERIMENTAL SETUP

The inlet diameter of the diffuser is 100mm and the outlet diameter is 170mm. The diverging portion of the diffuser is at an angle of 7°. There are two plates placed at a distance of 450 mm from the inlet.

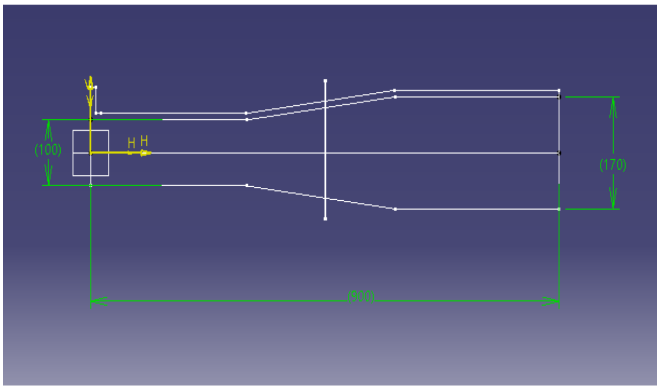

FIG:3.1.1 DIFFUSER WITH PLATES

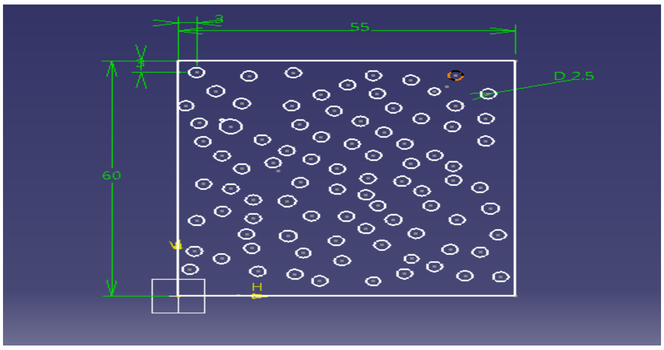

Fig:3.1.2 Perforated plate with dimensions.

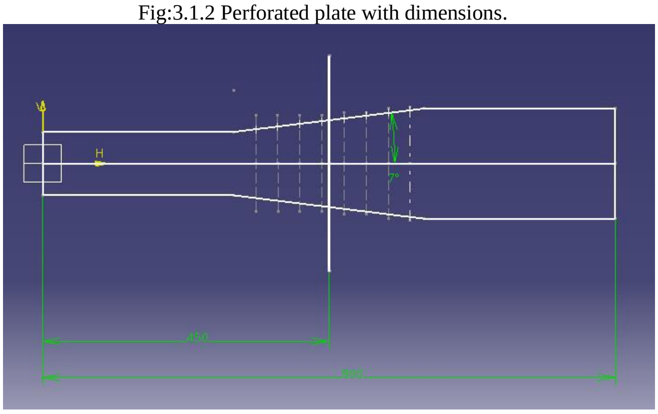

FIG: 3.1.3 DIFFUSER WITH PRESSURE PORTS

EXPERIMENTAL SETUP:

The experiments were conducted in a blower driven, supersonic wind tunnel setup.

Component	Specification
Blower motor	11kW
Blower speed	2500 rpm
Settling chamber	1mx1mx2m
Screens	No.80 four layer screens
Pipe inlet diameter	100mm
Length	900mm
Pressure manometer	PVM-620

TABLE 3.2.1 SPECIFICATION

FIG:3.2.1 EXPERIMENTAL SETUP OF DIFFUSER.

The measurement stations were designated as A, B, C, D, E, F and G. The distance between the measuring stations from the inlet of the diffuser is given in the table . The reference station for flow measurements is located in the inlet pipe at X = 35 mm ,Mean Velocity = 35m/s

Stations	A	B	C	D	E	F	G	H
X	35	70	105	140	175	210	245	280
X/L	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8

TABLE:3.2.2 DISTANCE BETWEEN THE MEASURING STATIONS FROM INLET

The velocity calibration is carried out for the known velocities of range from 20 m/s to 35 m/s. The chamber pressure ranges from 3 bar to 10 bar. The graphs are plotted for the pressure ratios and the length of the diffuser section.The graphs are obtained for the various velocities and corresponding graphs for these values are obtained.

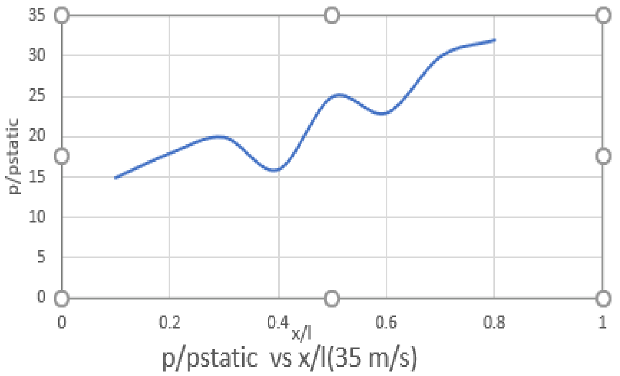

Fig3.2.2: Velocity distribution(35m/s)

4.NUMERICAL CALCULATIONS

CONTINUITY EQUATION:

When a fluid is in motion, it must move in such a way that mass is conserved. The inflow and out flow are one-dimensional, so that the velocity V and density ρ are constant over the area A. This is a statement of the principle of mass conservation for a steady, one-dimensional flow, with one inlet and one outlet. This equation is called the continuity equation for steady one-dimensional flow. For acontrol volume with many inlets and outlets, the net mass flow must be zero, where inflows are negative and outflows are positive.

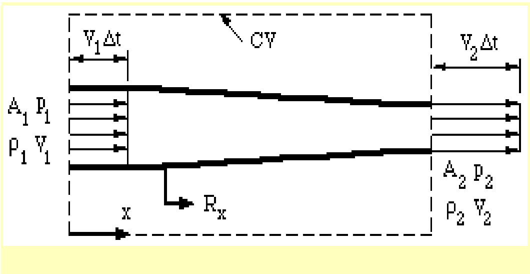

FIG :4.1.1 DIFFUSER SECTION

$$A_1V_1 = A_2V_2$$

CALCULATION

Diffuserinletdiameter D_1 = 100mm

Diffuseroutletdiameter D_2 =170mm

Area for inlet $=\frac{\pi}{4} D_1^2$

= 7853.98mm

=78.53cm

Area for outlet = $\frac{\pi}{4} D_2^2$

= 22698.00 mm

= 229.98 cm

Velocity, V_1= 35 m/s

$$V_2 = \frac{78.53\times35}{226.98}$$

V_2 = 12.10 m/sec

Storage pressure P = 10 bar

Dynamic pressure = $\frac{1}{2}\rho V_2^2$

Velocity V_2 = 12.1m/sec;dynamic pressure=88.24

To find :

Coefficient of pressure, $C_p = \frac{p-p_\infty}{\frac{1}{2}\rho V_2^2}$

Storage pressure p = 10 bar

At velocity = 35 m/s ,8 pressure ports,

$C_{p1} = \frac{10-0.99}{88.24} = 0.1021$

$C_{p2} = \frac{10-1.02}{88.24} = 0.1017$

$C_{p3} = \frac{10-0.99}{88.24} = 0.1021$

$C_{p4} = \frac{10-1.01}{88.24} = 0.1018$

$C_{p5} = \frac{10-0.99}{88.24} = 0.1021$

$C_{p6} = \frac{10-0.99}{88.24} = 0.1021$

$C_{p7} = \frac{10-1.00}{88.24} = 0.1019$

$C_{p8} = \frac{10-0.99}{88.24} = 0.1021$

5.RESULTS AND DISCUSSION

The analysis of the diffuser using computational fluid dynamics approach to investigate the mean and turbulence characteristics of fluid flow through the model, and to study the effect of pressure distribution through the diverging portion of the diffuser.

ANALYSIS:

An appropriate turbulence model must be selected to simulate the flow.However, the internal nozzle flow exhibits features which are quite different from the jet and therefore one cannot assume that the k–ε standard model will accurately predict the nozzle flow. To assess the capability of different turbulence models to accurately capture the main turbulence features of the flow in the nozzle, we have implemented a one-equation model The flow analysis in the diffuser section is done and the necessary graphs and contours are obtained for the velocities given. The pressure variation graphs are obtained for the velocity values. The velocity variation is done between 25m/s to 40 m/s. The corresponding values of pressure are obtained in contours. The analysis for 35 m/s is shown.

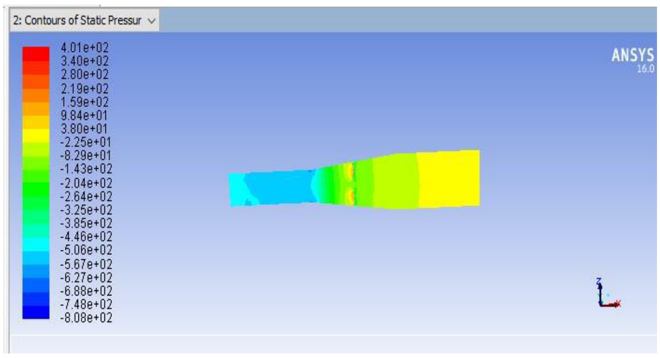

FIG:5.1 FLOW ANALYSIS CONTOUR OF STATIC PRESSURE (35 M/S)

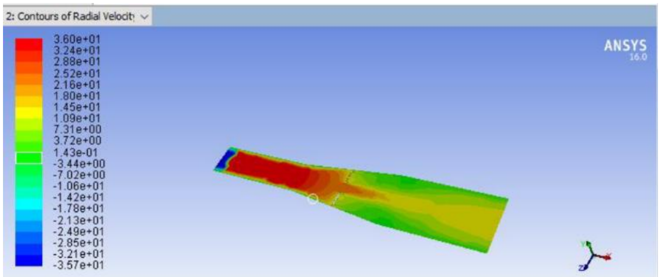

FIG:5.2 FLOW ANALYSIS CONTOUR OF RADIAL VELOCITY(35M/S)

6.CONCLUSION

The analysis of the diffuser is done with perforated plates.The analysis was done in ANSYS FLUENT by varying the velocity at the inlet of the diffuser.The flow characteristics ahead an behind the plate is found for varying values of velocity. The analysis was done by varying the position of the plate and the results were interpreted for obtaining the position at which the maximum value of pressure value was obtained. These results were compared with the anlaysis done on the diffuser without any perforated plates . The diffuser with perforated plate showed an considerable increase in pressure when compared to the diffuser without plates . ANSYS results showed that the use of plates at the diverging portion of the diffuser section showed an increase in the pressure value. So the same geometry was tested for varying velocity and the results showed an increase in the pressure by the use of these plates.

REFERENCES

[1] McDonald, A. T., Fox, R. W., An Experimental Investigation of Incompressible Flow in Conical Diffusers, 13International Journal of Mechanical Sciences, 8 (1966), 2, pp. 125IN5131 130IN6139

[2] Okwuobi, P. A. C., Azad, R. S., Turbulence in a Conical Diffuser with Fully Developed Flow at Entry, Journal of Fluid Mechanics, 57 (1973), 3, pp. 603-622

[3] Klein, A., Effects of Inlet Conditions on Conical-Diffuser Performance, Journal of Fluids Engineering, 103 (1981), 2, pp. 250-25

[4] Azad, R. S., Turbulent Flow in a Conical Diffuser: A Review, Experimental Thermal and Fluid Science,13 (1996), 4, pp. 318-337

[5] Mahalakshmi, N. V., et al.., Experimental Investigations of Flow through Conical Diffusers with and without Wake Type Velocity Distortions at Inlet, Experimental Thermal and Fluid Science, 32 (2007), 1, pp. 133-157

[6] Van Dewoestine, R. V., et al.., Effects of Swirling Inlet Flow on Pressure Recovery in Conical Diffusers AIAA Journal, 9 (1971), 10, pp. 2014-2018

[7] Senoo, Y., et al., Swirl Flow in Conical Diffusers, Bulletin of JSME, 21 (1978), Jan., pp.

[8] Okhio, C. B., et al., Effects of Swirl on Flow Separation and Performance of Wide Angle Diffusers, International Journal of Heat and Fluid Flow, 4 (1983), 4, pp. 199-206

[9] Clausen, P. D., et al., Measurements of a Swirling Turbulent Boundary Layer Developing in a ConicalDiffuser, Experimental Thermal and Fluid Science, 6 (1993), 1, pp. 39-48

[10] Lai, Y. G., et al., Calculation of Planar and Conical Diffuser Flows, AIAA J., 27 (1989), 5, pp. 542-548

[11] Jiang, G., et al., Numerical Prediction of Inner Turbulent Flow in Conical Diffuser by Using a New Five-Point Scheme and DLR k-e Turbulence Model, Journal of Central South University of Technology, 15(2008), Suppl. 1, pp. 181-186

[12] Armfield, S. W., Fletcher, C. A. J., Numerical Simulation of Swirling Flow in Diffusers, International Journal for Numerical Methods in Fluids, 6 (1986), 8, pp. 541-556

[13] Chou, N. H., Fletcher, C. A. J., Computation of Turbulent Conical Diffuser Flows Using a Non-Orthogonal Grid System, Computers & Fluids, 19 (1991), 3-4, pp. 347-361

[14] Okhio, C. B., et al., The Calculation of Turbulent Swirling Flow through Wide Angle Conical Diffuserand the Associated Dissipative Losses, International Journal of Heat and Fluid Flow, 7 (1986), 1, pp.37-48

[15] From, C. S., et al., Turbulent Dense Gas Flow Characteristics in Swirling Conical Diffuser, Computers & Fluids, 149 (2017), June, pp. 100-118

[16] Jorgenson, F., How to Measure Turbulence with Hot Wire Anemometers, Dantec Dynamics, Skovlunde, Denmark, 2004

[17] Bilen, K., et al., Heat Transfer from a Plate Impinging Swirl Jet, International Journal of Energy Research,26 (2002), 4, pp. 305-320

[18] Lefebvre, A. H., Gas Turbine Combustion, CRC Press, Boca Raton, Fla., USA, 1998

[19] Jeyachandran, K., Ganesan, V., Numerical Modelling of Turbulent Flow through Conical Diffusers with the Uniform and Wake Velocity Profiles at the Inlet, Mathematical and Computer Modelling, 10 (1988), 2, pp. 87

[20] Ganesan, V., Flow and Boundary Layer Development in Straight Core Annular Diffusers, International Journal of Engineering Science, 18 (1980), 2, pp. 287-304