I. INTRODUCTION

A diffuser is a device that increases the pressure of a fluid at the expense of its kinetic energy [1]. The cross section area of diffuser increases in the direction of flow, therefore fluid is decelerated as it flows through it, causing a rise in static pressure along the stream. Annular diffusers are extensively used in axial flow compressors and turbines to convert the kinetic energy of the exhaust flow into pressure. This makes the diffuser a critical element in the performance of the turbine, which is often neglected. The exhaust diffuser of an industrial gas turbine recovers the static pressure by decelerating the turbine discharge flow. This allows an exhaust pressure lower than the atmospheric pressure, thus increasing the turbine efficiency. In the modern turbine the Mach number at the exhaust is around 0.4-0.45 and the total energy produced by the turbine is approximately 350KJ/kg, in consequence, a 10% of the total energy of the turbine which is 35KJ/Kg is being wasted or lost at the exhaust by entering into atmosphere[10]. Only a very few studies on experimental and numerical investigations on simple diffusers are available [2], [3], [4] and the factors influencing their performance are predominantly the area ratio and the length of the flow path over which diffusion occurs. At diffusers inlet, the intensity of the turbulence is usually very high due to the swirl nature of the flow. It is a well known fact that, within diffusers the flows are characterized by strong adverse pressure gradients which tends to flow to separate from the walls. To avoid unacceptable weight penalties the diffusion of the flow must occur in the shortest possible length and to prevent flow separation smaller divergence angles are suitable. In the literature a very few researches are available on experimental analysis concerning on annular diffusers downstream to the turbine [5], [6] or a compressor [7], [8]. The detailed three dimensional investigations of a scaled down model of an annular exhaust diffuser, with turbulent flow field and the separation of the boundary layer were shown in [9], [10].

II. GEOMETRICAL DETAILS AND MATHEMATICAL FORMULATION

Diffuser Geometry Numerical investigations have been carried out in a gas turbine exhaust diffuser (Annular Diffuser) to study the effect of the divergence angle and the Reynolds number. A series of 24 guide vanes were used in the upstream, which provides a means of introducing swirl and aerodynamic blockage into the test section. The Diffuser assembly of the 35% scaled PGT 10 gas turbine exhaust diffuser is as shown in Fig.1. To keep the geometric similarity of the model with the GT diffuser, the area ratio is maintained same along the axis in both the model and the GT diffuser. In long diffusers of low diverging angle, the pressure loss is high owing to skin friction along the walls. With increase in divergence angle both the diffuser length and friction losses are reduced, but the stall losses become more significant. For any area ratio there must be an optimal angle of divergence at which the pressure loss is minimum and it lies between 7° to 12°.

Fig. 1: Diffuser Assembly
The geometrical details of 35% scaled PGT 10 gas turbine exhaust diffuser is shown in Fig.2, where the diffusing length is 450mm, inlet and outlet diameters are 190mm and 320mm, respectively and half cone angle is 7°. Further, the half cone angle is increased in the increments of
2° i.e 9°, 11° and 13°, by keeping the diffusion length constant.

Fig. 2: Geometry Of 35% PGT 10 Gas Turbine Exhaust Diffuser [9, 10]

III. GRID GENERATION

A commercially available meshing tool is used to generate good quality hexa grids. Generating high quality structured (Hexa) grids over the exhaust diffuser is a challenging task (Fig. 2). The overall quality of the grids on the diffuser is maintained ranging from 0.90 to 0.98. Where digit 1 represents 100% quality and digit 0 represents 0% quality, in order to select the suitable grid size, a through grid independent studies have been carried out by varying the node spacing on the edges and by using uniform mesh law method. In the present analysis grid independence study has been done for 9° divergence angle (Fig. 2) for four set of grids, which are coarse (5 mm), Medium (4 mm), Fine (3 mm) and super fine (2 mm). The variation of static pressure at the hub and casing for all set of grids are shown in Fig. 3 & Fig. 4. From the results it can be observed that there is no considerable amount of change in the static pressure values for all set of grids. The computational time taken for fine grid is less compared to super fine grid. A fine grid of 3 mm spacing is chosen for further analysis.

Fig. 3: Variation of Static Pressure at Hub for different Grid spacing for 9° Casing Angle

IV. GOVERNING EQUATIONS

The calculation procedure is based on the solution of the equations governing the conservation of mass and momentum in the time averaged form for a steady incompressible flow. These equations can be written in tensor notation as:

\[
\frac{\partial}{\partial x_i} (\rho U_i) = 0
\]

\[
\frac{\partial}{\partial x_i} (\rho p U_i U_j) = -r \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu_j \frac{\partial U_i}{\partial x_j} - \rho U_i U_j \right]
\]

The quantities \((-\rho U_i U_j)\) represent the turbulent Reynolds stresses. The turbulence model which is used in the analysis is k-ε model.

k-ε Model:

The Reynolds stresses are linearly related to the mean rate strain as

\[
(-\rho U_i U_j) = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho \epsilon
\]

The turbulent viscosity \(\mu_t\) is expressed as

\[
\mu_t = C_p \rho \frac{k^2}{\epsilon}
\]

Where \(k\) and \(\epsilon\) are the turbulence kinetic energy and dissipation rate of turbulence, the values of which are obtained from the solution of the following transport equations:

\[
\frac{\partial}{\partial x_i} (\rho U_i U_k) = \frac{\partial}{\partial x_i} \left( \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + (G - \rho \epsilon) r
\]

\[
\frac{\partial}{\partial x_i} (\rho U_i \epsilon) = \frac{\partial}{\partial x_i} \left( \frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_i} \right) + \left( C_{t1} \frac{\epsilon}{k} G - C_{t2} \rho \frac{\epsilon^2}{k} \right) r
\]

The turbulence generation term \(G\) is written as:

\[
G = \mu_{eff} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)
\]
The effective viscosity $\mu_{eff}$ is calculated from

$$\mu_{eff} = \mu + \mu_t$$

The five empirical model constant are assigned the following values:

$$C_k = 0.09, C_{s1} = 1.44, C_{s2} = 1.92, \sigma_k = 1.0, \beta = 1.3$$ [13]

V. BOUNDARY CONDITIONS

There are four types of boundary conditions to specify for the computation in annular exhaust diffuser, they are Inlet, Outlet, Wall and Symmetry boundary conditions.

Inlet: The present analysis involves the velocity as the input boundary condition. The incorporation of velocity can be specified by any one of the velocity specification methods described in ANSYS CFX. Turbulence intensity is specified as Medium Turbulence i.e 5% with respect to the equivalent flow diameter. In the present analysis the inlet boundary condition is varied from $3.36537 \times 10^3$ to $6.73075 \times 10^3$ to evaluate the effect of Reynolds number on the flow development in the passage.

Outlet: Atmospheric pressure condition is applied at the outlet boundary where in the pressure at the exit of the diffuser is set to Atmospheric.

Wall: The no slip condition and smooth surface conditions was used for all walls.

Symmetry: Symmetry boundary conditions are also applied to the model at the SYMM_BOTTOM and SYMM_TOP, because we considered only the quarter part of geometry for the analysis.

Higher order schemes are chosen for the better accuracy compared to lower order schemes. No single turbulence model is universally accepted for all kinds of flows. An industrial standard k-ε turbulence model shows good agreement for internal flows and shear bounded flows. In the present analysis numerical simulations were carried out by choosing k-ε turbulence model. The residual target of $1 \times 10^{-12}$ has been set to ensure the convergence of the equations.

VI. RESULTS & DISCUSSIONS INTRODUCTION TO NUMERICAL INVESTIGATIONS

This paper presents the performance of PGT10 gas turbine exhaust diffuser, in terms of Static pressure development and pressure recovery measured in the scaled down model. In the next graphs and contour plots, it is referred that the position of measuring point in terms of axial position, radial position, and angle or circumferential position considering that:

- Axial position of 0mm is the leading edge of inlet of diffuser.
- Angular position is measured from hub to casing in Angle of 0°, 3.5°, and 7.5°.
- Results were produced in the diffuser model without struts for 7.5° sector with one of the inlet guidevane located at 0°, the reason for considering only 7.5° sector is, in the model there are 24 inlet guide vanes which are equally spaced. For every 7.5° the flow repeats. The results are extracted for three angular positions at 0°, 3.5° and 7.5°.

The diffusers performances have been also determined by the following parameters:

- Pressure Recovery coefficient
  $$C_F = \frac{1}{2} \rho_{\infty} \frac{v^2}{\theta}$$
  Where,
  - $P_{\infty}$ is the pressure in the free stream.
  - $P$ is the pressure at the point at which pressure coefficient is being evaluated.
- Ideal Pressure Recovery coefficient
  $$C_{Fp} = 1 - \left(\frac{A_2}{A_1}\right)^2$$
- Diffuser Efficiency
  $$\eta = C_{Fp} / C_p$$
- Pressure loss coefficient
  $$K = C_{p1} - C_p$$

VII. ANALYSIS WITH GUIDEVANES

The static pressure distribution along the hub and the casing are shown in Fig. 5 (a, b, c, d) at different values of Reynolds No. (Different velocity) and also for different angular positions. Expect at the entrance region, there is continuous growth of pressure along the hub and casing wall, this indicates that there is no separation nearby the casing wall or recirculation zone nearby the hub wall. From the figures, it is observed that pressure at hub wall is less than the casing wall. As the flow passes through the diffuser it glides along the casing wall therefore the casing wall behaves as pressure side and hub wall behaves as the suction side, so pressure at casing side is more. As the value of Reynolds number increases, the difference in the pressure at hub and casing wall at the inlet section increases, therefore values at entrance losses also increases. But the value of overall losses reduces with increase in Reynolds number.

Fig. 5(a): Static Pressure distribution along the hub and the casing for 7 Degree casing angle and for different Reynolds number.
Fig. 5(b): Static Pressure distribution along the hub and the casing for 9 Degree casing angle and for different Reynolds number.

Fig. 5(c): Static Pressure distribution along the hub and the casing for 11 Degree casing angle and for different Reynolds number.

Fig. 5(d): Static Pressure distribution along the hub and the casing for 13 Degree casing angle and for different Reynolds number.

The velocity distribution along the hub and the casing are shown in Fig. 6 (a, b, c, d) at different values of Reynolds number and also at different angular positions. From the inlet of the diffuser the velocity goes on reducing towards the outlet because of the diffusion. From the figures it is evident that for the results of velocity for 0° angular position, the velocity at the inlet will increases and after some time the velocity again starts reducing this is because of the aerodynamic blockage, i.e there are inlet guide vanes, these inlet guide vanes blocks the flow, due to this blockage the velocity will increases and simultaneously swirling motion will be introduced in the test section. After attaining the flow, again due to the diffusion the velocity starts reducing. Similarly, the results for 3.5° and 7.5° angular position, there is continuously retardation of the flow, there is no increase in the velocity like in the results for 0° angular position. This is because, there is no any blockage at the 3.5° and 7.5°, there will be free flow at these positions and due to this reason there will be no increase in velocity at the inlet of diffuser. As the Reynolds number increases, the velocity for the different also goes on decreases giving rise to the increase in the static pressure.

Fig. 6(a): Velocity distribution along the hub and the casing for 7 Degree casing angle and for different Reynolds number.

Fig. 6(b): Velocity distribution along the hub and the casing for 9 Degree casing angle and for different Reynolds number.

Fig. 6(c): Velocity distribution along the hub and the casing for 11 Degree casing angle and for different Reynolds number.

Fig. 6(d): Velocity distribution along the hub and the casing for 13 Degree casing angle and for different Reynolds number.
In Fig. 7(a, b, c), the graphs are plotted for the values of $C_p$ along the hub and casing wall. These graphs follow the same trend as the static pressure distribution graphs. Near the entrance region, the value of $C_p$ is negative for the casing and hub wall due to entrance static pressure disturbance. As the value of divergence angle increases, this negative value of $C_p$ at hub and casing increases.

The Fig. 8 (a to l) shows the pressure distribution within the diffuser at different location with the help of different color. Due to the change of kinetic energy into pressure energy there is continuous increase in the magnitude of pressure from inlet to outlet.

Fig. 7(a): $C_p$ distribution along the hub and the casing for Different casing angles for 80 m/sec.

Fig. 7(b): $C_p$ distribution along the hub and the casing for Different casing angles for 120 m/sec.

Fig. 7(c): $C_p$ distribution along the hub and the casing for Different casing angles for 160 m/sec.

Fig. 8(a): Contour plot of static pressure for 7$^\circ$ casing angle and Re. = 3.36537 $\times 10^5$.

Fig. 8(b): Contour plot of static pressure for 7$^\circ$ casing angle and Re. = 5.04806 $\times 10^5$.

Fig. 8(c): Contour plot of static pressure for 7$^\circ$ casing angle and Re. = 6.73075 $\times 10^5$.

Fig. 8(d): Contour plot of static pressure for 9$^\circ$ casing angle and Re. = 3.36537 $\times 10^5$.

Fig. 8(e): Contour plot of static pressure for 9$^\circ$ casing angle and Re. = 5.04806 $\times 10^5$. 

Fig. 8, Fig. 9 and Fig. 10 shows the contour plots and vector plots generated by CFX. In these figures fluid characteristics like velocity, pressure is shown by different color. A particular color does not give single value of these characteristics, but show the range of these values. If the value of a characteristic at a particular point falls in this range, there will be color of that range.
Fig. 8(f): Contour plot of static pressure for 9° casing angle and Re. = $6.73075 \times 10^5$.

Fig. 8(k): Contour plot of static pressure for 13° casing angle and Re. = $5.04806 \times 10^5$.

Fig. 8(g): Contour plot of static pressure for 11° casing angle and Re. = $3.36537 \times 10^5$.

Fig. 8(l): Contour plot of static pressure for 13° casing angle and Re. = $6.73075 \times 10^5$.

Fig. 8(h): Contour plot of static pressure for 11° casing angle and Re. = $5.04806 \times 10^5$.

Fig. 8(i): Contour plot of static pressure for 11° casing angle and Re. = $6.73075 \times 10^5$.

The Fig. 9(a to l) shows the velocity variation of the fluid at different points in diffuser, the velocity at these points is shown by different color. In the early part of the diffuser section, the velocity at a particular cross section is almost uniform, but as the flow proceeds, the boundary layers at the hub and casing wall grow in size, so at the exit cross section of the diffuser there is large change in velocity. In the velocity diagram there is a region, which is shown by red color. It is the region where velocity is greater than the velocity applied at the inlet section. This increase in velocity is due to the aerodynamic blockage effect imposed by the guide vanes.

Fig. 9(a): Contour plot of Velocity for 7° casing angle and Re. = $3.36537 \times 10^5$.

Fig. 9(b): Contour plot of Velocity for 7° casing angle and Re. = $5.04806 \times 10^5$. 

397
The Fig. 10 (a to l) shows the vector plot of velocity, the direction and magnitude of velocity of fluid particles is shown at different points. It is observed that there is no negative value of fluid velocity at any point, which indicates that there is no reverse flow or the separation at casing wall.
or recirculation zone near the hub wall for both divergence angles.

Fig. 10(a): Vector plot of Velocity for 7° casing angle and \( \text{Re.} = 3.36537 \times 10^5 \).

Fig. 10(b): Vector plot of Velocity for 7° casing angle and \( \text{Re.} = 5.04806 \times 10^5 \).

Fig. 10(c): Vector plot of Velocity for 7° casing angle and \( \text{Re.} = 6.73075 \times 10^5 \).

Fig. 10(d): Vector plot of Velocity for 9° casing angle and \( \text{Re.} = 3.36537 \times 10^5 \).

Fig. 10(e): Vector plot of Velocity for 9° casing angle and \( \text{Re.} = 5.04806 \times 10^5 \).

Fig. 10(f): Vector plot of Velocity for 9° casing angle and \( \text{Re.} = 6.73075 \times 10^5 \).

Fig. 10(g): Vector plot of Velocity for 11° casing angle and \( \text{Re.} = 3.36537 \times 10^5 \).

Fig. 10(h): Vector plot of Velocity for 11° casing angle and \( \text{Re.} = 5.04806 \times 10^5 \).

Fig. 10(i): Vector plot of Velocity for 11° casing angle and \( \text{Re.} = 6.73075 \times 10^5 \).

Fig. 10(j): Vector plot of Velocity for 13° casing angle and \( \text{Re.} = 3.36537 \times 10^5 \).
VIII. CONCLUSION

Numerical investigations have been carried out to investigate the static pressure development and pressure recovery coefficient through an industrial gas turbine exhaust diffuser without struts.

From the above discussion the following conclusions can be made:

1. The static pressure distribution increases uniformly along the length of diffuser for both hub and casing walls, since there is no separation or recirculation on the walls.
2. The pressure recovery within the diffuser increases as the flow proceeds, consequently the pressure also increases with the decrease in velocity level.
3. There is no flow separation is observed at the casing wall even at the divergence angle of 11°.
4. With increase in area ratio pressure recovery increases due to higher rate of diffusion but pressure recovery loss also increases.
5. With increases in inlet velocity, there is increase in pressure recovery, since the entrance losses increases marginal with velocity.
6. The Pressure recovery at the casing is higher than that calculated at the hub and this happens probably because of the hub diameter is constant along the duct and then in the upper part of the duct a higher diffusion occurs.

REFERENCES


