A numerical study of pre-diffuser optimization of an aero gas turbine combustion chamber

G Ananda Reddy & V Ganesan
Internal Combustion Engine Laboratory, Department of Mechanical Engineering,
Indian Institute of Technology Madras, Chennai 600 036, India

Received 4 November 2003; accepted 20 May 2005

This paper describes the attempts made in the optimization of pre-diffuser geometry of an aero gas turbine combustion chamber for obtaining minimum total pressure losses and for good static pressure recovery. Optimization of pre-diffuser has been attempted by changing the geometric parameters like length, divergence angle and wall contour. A 30° sector has been considered for the present study because of the periodicity with respect to the air blast atomizer. The geometry is modeled using GAMBIT and CATIA software. Meshing is done by GAMBIT. The governing equations are solved using the SIMPLE algorithm with iterative multigrid acceleration. Turbulence effects are simulated using the (RNG) k-ε model. Flow field characteristics have been analyzed. Optimization is attempted from the point of view of achieving minimum pressure loss. From the present study it is concluded that a 6° straight walled pre-diffuser with area ratio of 1.6574 and a dump gap ratio (D/H) of 0.08 is found to be most optimum, considering the flow development and minimum total pressure loss.

IPC Code: F23M

Gas turbine engines employ a diffuser between the compressor discharge and combustor(s). The primary function of a diffuser is to decelerate the compressor discharge flow so as to distribute the air evenly to various inlets on the combustor liner. The diffusion process must be accomplished with minimum total pressure loss because these parasitic losses have an adverse effect on the thermal efficiency. The flow uniformity around the liner is important to achieve efficient combustion, to prevent liner hot spots, and to provide an acceptable combustor exit pattern factor. Basic configuration and design considerations for the combustor diffusers are discussed by Lefebvre. A recent review by Klein elaborates on flow characteristics of the combustor diffusers.

A commonly used configuration of combustor-diffuser in aircraft engines is the dump diffuser, wherein the compressor discharge-air is decelerated in a short, conventional pre-diffuser and then "dumped" into large chamber, which divides flow to the flame tube, to the customer bleed and inner and outer annuli around the combustor liner. The sudden expansion at the pre-diffuser exit causes flow recirculation in the dump, which helps to maintain a stable flow pattern rather insensitive to the engine operating conditions. The sudden expansion causes pressure losses. However, the required diffusion is accomplished in a short length. Most of the losses occur in the dump chamber, while nearly all of the static pressure recovery takes place in the pre-diffuser.

The airflow path in the present diffuser system is described as follows: The air exiting the compressor is decelerated in an annular pre-diffuser and it is discharged into a dump chamber. Then, a portion of discharge-air flows through customer bleed holes for the cooling. Remaining air will split and flow through the outer annulus, inner annulus and cowl. Finally, air will reach flame tube through cowl, primary, dilution, and cooling ring holes on the outer and inner liners. The design requirements for the diffuser system are: (i) maximum static pressure recovery in the pre-diffuser and minimum total pressure losses in the pre-diffuser and dump chamber to improve turbine efficiency, (ii) uniform flow distribution in the combustor, (iii) uniform and stable flow conditions at outlet, and (iv) required velocity reduction within shortest possible distance.

Literature Review

Jeyachandran and Ganesan investigated the turbulent boundary layer under adverse pressure gradient and found that for mild pressure gradient the inlet velocity distortion affects the boundary layer growth but for strong pressure gradient flows, there is little effect. Jeyachandran and Ganesan have investigated diffuser flow numerically and it was seen that the
performance, effectiveness and pressure gradient of a diffuser could also be predicted successfully using parabolic equations. The prediction method was used for 2.5° and 6° diffusers without and with inlet velocity distortion at the inlet. It is seen from the results that the performance and effectiveness of the diffuser increases slightly due to the presence of velocity distortion at inlet for 2.5° diffuser and it is the other way for the 6° diffuser. It is further seen that the pressure gradient increases with increase in the half cone angle and also the inlet velocity distortion increases the pressure gradient. It was concluded in both of these investigations that the plausible results obtained by the prediction need to be validated with experiments.

They have made attempts to compare the predicted results with experimental results. It was found that for the 6° diffuser, the velocity profile showed satisfactory agreement near the wall and only fair near the axis. Further, it was found that the agreement between the predicted values and the experimental values of boundary layer characteristics such as displacement thickness, shape factor and blockage factor are quite satisfactory as in the case of velocity profiles.

Klein in his work showed that the radial velocity, which is more than the circumferential velocity is the one which determines the performance characteristics of combustor diffuser.

Su and Zhou carried out the numerical study of combustor-diffuser flow interaction using the KIVA-3V code. The simulation was based on the solution of Navier-Stokes equations with phenomenological models of turbulence, sprays, and chemical reactions. The swirl was simplified in terms of the conservation of mass and momentum. Static pressure recovery coefficients along the inner and outer walls of the pre-diffuser and combustor casing were obtained from the numerical solutions, which agreed well with published/measured data. Flow fields in axial and circumferential direction are analyzed.

Relation et al. carried out numerical simulation of non-reacting flows for industrial gas turbine combustor geometries. They evaluated the application of the computational fluid dynamics (CFD) to calculate the flow fields in industrial combustors. Comparison was made between the standard k-ε turbulence model and a modified version of the standard k-ε turbulence model in which a second time scale was added to the turbulent dissipation equation. Results obtained from the CFD calculation were compared with experimental data. For the case under study, the standard k-ε model was found to diffuse the swirl and axial momentum, which resulted in the inconsistent prediction of the re-circulation zone for the test cases. However, the modified k-ε model showed an improved prediction regarding the location, shape and size of the centerline re-circulation zone. The large swirl and axial velocity gradients, which are diffused by the standard k-ε model, are preserved by the modified model, and good agreement was obtained between the calculated and the measured axial and swirl velocities. The over-prediction of the turbulent eddy viscosity in regions of high shear, which is characteristic of the standard k-ε model, could be taken care of by the modified turbulence model.

Agarwal et al. carried out an experimental/computational study of the cold flow in the combustor-diffuser system of the industrial gas turbine combustors and impingement-cooled transition pieces. The primary objectives were to determine flow interactions between the pre-diffuser and the dump chamber, to evaluate circumferential flow non-uniformities around the transition pieces and combustors, and to identify the pressure loss mechanism. They conducted flow experiments in an approximately one-third scale. Wall static pressure and velocity profiles were measured at selected locations in the test model. A three-dimensional computational fluid dynamics analysis employing a multi-domain procedure was performed to supplement the flow measurements. The complex geometric features of the test model were included in the analysis. The measured data correlated well with the simulations. The results revealed strong interactions between the pre-diffuser and dump chamber flows. The pre-diffuser exit flow was distorted indicating that the uniform exit conditions assumed in the diffuser design were violated. The pressure varied circumferentially around the combustor casing and impingement sleeve. The circumferential flow non-uniformities increased toward the inlet of the turbine expander. A venturi effect causing flow to accelerate and decelerate in the dump chamber was identified. The dump chamber contained several re-circulation regions contributing to the losses. A realistic test model and three-dimensional analysis used in this study provided new insight into the flow characteristics of practical combustor systems.

Karki et al. analyzed the dump diffuser by using computational procedure. The procedure is based on the's fluid using mod. pr ac. fuel be al inc of simi simi loss e data cont flow pre c t re dime and sym that throu thrrough.

Fi: in vee diffu static but n re gion press atten sh ap min sym ther surro get d CC dump cond meas ratio dump has i pr ed with st ng diff 0.48 was;
the solution of the Navier-Stokes equations on a body
conforming grid. The turbulence effects are simulated
using the high Reynolds number form of the k- model. The calculation method has been applied to a
practical configuration that includes support strut and
fuel nozzles. In the first case, the flow is assumed to be axisymmetric. The effects of various blockages are
incorporated in the simplified geometry. This is followed by results of the three dimensional
simulation. The estimates provided by the simplified
model for static pressure recovery and total pressure
losses are within 10% to 15% of the experimental
data. The author says that the axisymmetric
combustor analysis is adequate for most applications.
However, full simulation is necessary for accurate
prediction of three-dimensional process including hot
streaks and pattern factor. The result of the full
simulation provides an insight into the three-
dimensional nature of the flow. The presence of struts
and fuel nozzles results in significant non-
symmetrical effects in the flow. The authors observed
that the model under investigation indicates the flow
through annulus effectively. But flow distribution
through the dome swirler is found to be non-uniform.

Fishenden and Stevans conducted experimental
investigations on the performance of an annular dump
diffuser for combustor and concluded that the major
static pressure rise occurs in the pre-diffuser portion,
but most of the total pressure loss occurs in the dump
region. The principal determinants of the total
pressure loss are the amount of diffusion being
attempted-downstream of the pre-diffuser and size and
shape of the flame tube and also the dump gap. The
minimum overall pressure loss was obtained with a
symmetrical velocity profile at pre-diffuser outlet.
The geometry of the pre-diffuser, flame tube and
surrounding annuli need to be carefully matched to
get desired performance.

Corrente et al. investigated the performance of the
dump diffuser by experimental method with the inlet
conditions being generated by axial flow compressor
measurements. Studies were made for the dump gap
ratio of 1.0 (datum configuration) and three other
dump gaps (0.8, 0.65, 0.5). The following conclusion
has been drawn from the experiments: The overall
pre-diffuser performance does not vary significantly
with dump gap. For the datum configuration, the
stagnation pressure loss coefficients between pre-
diffuser inlets and outlets and inner feed annuli were
0.483 and 0.43 respectively. The majority of this loss
was generated around the flame tube head while most
of the static pressure rise occurred within the pre-
diffuser. Also, the stagnation pressure losses
increased by 32% (outer) and 48% (inner) as the dump
gap reduced from 1.0 to 0.5. This high value was
associated with the relatively deep flame tube and
short system length. This increases the amount of
flow curvature around the flow tube head, which
becomes more severe as the dump gap is reduced.

Objective
Based on the above literature survey it is seen that
there is considerable interest regarding flow
development in pre-diffuser for the combustion
system. Hence, for the present study there are three
main objectives: (i) Non-reacting flow analysis for the
given design configuration, (ii) optimization of pre-
diffuser geometric parameters like length, divergence
angle and wall contours for minimum total pressure
loss, and (iii) to investigate the effect of increasing the
inlet velocity on total pressure loss.

Computational Domain Considered
The combustor configuration under study is
illustrated in Fig. 1. The airflow leaving the
compressor enters the pre-diffuser section, where
large portion of the velocity head will be recovered,
and then the flow enters into dump region of diffuser.
In this region, the flow will split into three streams.
One stream enters the flame tube and the other two
into the surrounding feed annuli. The dump diffuser
combustor design parameters are governed by
geometry and flow properties. These include the
length and area ratio (ratio between outlet to inlet area
of the pre-diffuser) of pre-diffuser, dump gap velocity
distribution and turbulence level at the compressor
exit. Additional considerations are locations of
support struts and atomizer. These sub-components
should be designed and placed such that they have

![Fig. 1.—Two-dimensional geometry of gas turbine combustor](image-url)
minimal effect on pressure loss and combustor exit temperature quality. The diffuser system should be designed such that there are no unwanted total pressure losses. Further, it should deliver the desired mass flow into various regions. In this study for optimization of pre-diffuser, the length, divergence angle and wall contour have been changed. Fig. 2 shows the three-dimensional view of a 20° sector of the gas turbine combustor under investigation. A 20° sector is taken for the prediction because of the periodicity with respect to air blast atomizer. Therefore, by applying periodic boundary conditions, predictions can be carried out without loss of accuracy. Further, it will save the computer time to a great extent.

Grid size used in this study is 8.7 lakhs, which is of tetrahedral type. In order to test the adequacy of grid, the grid density was reduced by 50% and then increased by 50%. The variation of predicted velocity at the critical sections, namely the primary zone and secondary zone with respect to 50% increased number of grids were less than 4%. Hence for the present study 8.7 lakhs cells were used. The minimum grid size was 0.1 mm and maximum was 2 mm.

**Governing Equations**

The time-averaged governing equations for fluid flow in the Cartesian tensor form are given below:

**Continuity equation**

\[
\frac{\partial}{\partial X_i} (\rho u_i) = S_m
\]  

(1)

The source term \( S_m \) is the mass added to the continuous phase from the dispersed second phase (due to vaporization of liquid droplets). For the non-reacting flow \( S_m \) takes a value of zero.

**Momentum equation**

\[
\frac{\partial}{\partial X_i} \left( \rho u_i u_j \right) = - \frac{\partial p}{\partial X_i} \quad \text{for } \quad i = j
\]

\[
\sum_{j=1}^{3} \left( \frac{\partial}{\partial X_j} \rho \left( u_i u_j \right) \right) + \frac{\partial}{\partial X_i} \left( \rho k \right) + \frac{\partial}{\partial X_i} \left( \rho \mu \left( \epsilon \right) \right) + F_i = 0
\]

\[
\text{where } p \text{ is the static pressure, } \mu \text{ is the molecular viscosity, } \delta_{ij} \text{ is Kronecker-delta function and } F_i \text{ is the external body force that arises from interaction with the dispersed phase in the } i \text{ direction. The second term in the right hand side represents the stress tensor denoted by } \tau_{ij} \text{ and the third term represents Reynolds stresses. These are modeled using Boussinesq hypothesis. According to this hypothesis the Reynolds stresses are related to the mean velocity gradients by:}
\]

\[
-\rho u_i u_i = \mu \left( \frac{\partial u_i}{\partial X_j} + \frac{\partial u_j}{\partial X_i} \right) - \frac{2}{3} \left( \rho k + \mu \frac{\partial u_i}{\partial X_j} \right) \delta_{ij}
\]

(3)

**Energy equation**

\[
\frac{\partial}{\partial X_i} \left[ \mu_i (\rho E + p) \right] = - \frac{\partial}{\partial X_i} \left( k_{\text{eff}} \right) + \frac{\partial}{\partial X_i} \left( \rho k \right) - \frac{\partial}{\partial X_i} \left( \rho \left( u_i \epsilon \right) \right) + S_{\text{h}}
\]

\[
\frac{\partial}{\partial X_i} \left[ \mu_i (\rho E + p) \right] = - \frac{\partial}{\partial X_i} \left( k_{\text{eff}} \right) + \frac{\partial}{\partial X_i} \left( \rho k \right) - \frac{\partial}{\partial X_i} \left( \rho \left( u_i \epsilon \right) \right) + S_{\text{h}}
\]

(4)

where \( k_{\text{eff}} \) is the effective thermal conductivity \((k + k_{\text{th}})\), \( k_{\text{th}} \) is the turbulent thermal conductivity, and \( J \) is the diffusion flux of the species \( j \). \( S_h \) includes the heat of the chemical reaction. In the above equation:

\[
E = h - \frac{p}{\rho} + \frac{\epsilon^2}{2}
\]

(5)

where sensible enthalpy \( h \) is defined for ideal gases as

\[
h = \sum_{j} \mu_j h_j
\]

(6)
The mass fraction of the species $f'$ and
\[ m_f' = \frac{1}{T} \int C_{r,v} dT \]
where $T_{in}$ is 298.15 K.

Turbulence model equations

In the present study, (RNG) $k$-$\varepsilon$ turbulence model is used for turbulence modeling. Navier-Stokes equations are solved, using a mathematical technique called "renormalization group" (RNG) method to take care of physical modeling. These features make the RNG $k$-$\varepsilon$ model more accurate and reliable especially for swirling flows compared to the standard $k$-$\varepsilon$ model. The transport equation for turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$ are obtained from the following transport equations

\[
\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i} \left[ \alpha \mu_{ef} \frac{\partial k}{\partial x_i} \right] + G_k - \rho \varepsilon - Y_{st} \tag{8}
\]
and

\[
\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i} \left[ \alpha \mu_{ef} \frac{\partial \varepsilon}{\partial x_i} \right] + C_{\mu} \frac{\varepsilon}{k} G_k - C_{\varepsilon} \rho \frac{\varepsilon^2}{k} - R \tag{9}
\]

Here, $G_k$ represents the generation of turbulent kinetic energy due to the mean velocity gradients calculated as

\[
G_k = -\rho \mu' \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} \tag{10}
\]

where $C_{ed}$ is a constant.

The $R$ term in the epsilon equation marks the difference in $\varepsilon$ equation between the RNG and standard $k$-$\varepsilon$ model and it accounts for the effects of rapid strain. It is given by:

\[
R = \frac{C_{\varepsilon} \rho \varepsilon^2 (1 - \beta \varepsilon)}{1 + \beta \varepsilon^2} \tag{12}
\]

where $\eta = Sk/\varepsilon$, $\eta_0 = 1.38$, $\beta = 0.012$. The values of model constants used are given in Table 1.

Assumptions

The following assumptions pertaining to the flow in the combustor have been made in the present study: (i) flow is considered as steady and incompressible, (ii) buoyancy effects are negligible, and (iii) radiation effects are negligible.

Boundary Conditions

The present problem has three types of boundaries. They are inlet, outlet and wall. The way these boundary conditions are prescribed is described below.

Inlet

Velocity boundary condition is used to define the flow velocity along with all relevant scalar properties of the flow at the flow inlet. Velocity magnitude and direction, the velocity components or the velocity magnitude normal to the boundary specifies inlet velocity boundary conditions. Inlet velocity used in the present study is 198.5 m/s.

Outlet

Outflow boundary condition is used to model the flow at exit where the details of the flow velocity and pressure are not known prior to solution of the problem. As these variables are not known for the case under study, we have adopted this boundary condition for the combustor exit. When this condition is specified, the code extrapolates the required information from the interior. In the present study there are five outlets, the flow rate weighting has been set to indicate what fraction of total flow rate takes place through the given boundary. Table 2 gives the mean mass flow rate through bleed at different locations. These inputs are predetermined and cannot be changed. The different outlets are shown in Fig. 3.

\[
\begin{array}{cccc}
S. No. & Location & \% of bleed of total mass flow \\
1 & Outer & 6.97\% (of total mass flow) \\
2 & Inner front (rotor) & 3.94\% (of total mass flow) \\
3 & Inner rear & 4.01\% (of total mass flow) \\
4 & Customer bleed & 0.73\% (of total mass flow) \\
5 & Flame tube & 84.35\% (of total mass flow) \\
\end{array}
\]
Wall

In any flow, Reynolds number of the flow becomes quite low and turbulent fluctuations are damped considerably near the walls and the laminar viscosity starts to play a significant role. In the present case, walls are assumed to be adiabatic with no slip condition. Standard wall functions are used to calculate the variables at the near wall-cells and the corresponding quantities on the wall.

Results and Discussion

The velocity magnitude contours at 0, 10, and 20° planes are shown in Fig. 4. As expected, velocity magnitude contours at 0 and 20° are almost same due to the periodic boundary condition used. Due to the velocity reduction there will be static pressure recovery due to energy transformation in the pre-diffuser. This static pressure recovery is the one which go to determine the downstream flow condition and therefore the details of the downstream flow field also presented. This can be clearly seen in Fig. 4. Figure 5 shows the details of the velocity vectors inside the flame tube, just behind the swirler.

As is evident there is clear recirculation zone (Fig. 5) that has developed just downstream of the swirler. This recirculation zone is formed due to the effect of the radial pressure gradient created by the swirler and the interaction of opposing primary jets, which have been modeled carefully. This recirculation in the primary zone is very important for flame stabilization so that the flame remains anchored to the atomizer and effects complete burning of the fuel in the primary zone to provide a good exit pattern factor.

For the development of proper recirculation zone the primary jets should penetrate fully into the flame tube. As already mentioned the penetration of primary jets into the flame tube can be seen clearly in Fig. 6. The interactions of primary opposing jets will create recirculation that leads better mixing of air and fuel. These jets act as flame stabilizers and helps to stabilize the flame. Figure 7 shows the flow through the dilution holes. The function of the dilution air jet is to mix with the products of combustion with increasing air from dilution holes in order to cool them to a value acceptable by the turbine and also to reduce the pattern factor at the exit of combustor.

The total pressure drop across the combustor is due to the various blockages like the atomizer mount, swirler and swirl cone. The pressure drop is due to the flow across the primary, dilution and the cooling holes and also due to the skin friction along the walls of the combustor casing, the flame tube, swirler and swirl cone. The total pressure drop across the combustor for the case under consideration is 6.85% (Fig. 8).

Figure 9 shows the static pressure contours at the mid-plane of combustor. In pre-diffuser the velocity reduces so that the dynamic head is converted into the static head. It could be seen in Fig. 4 that as the flow passes through the pre-diffuser there is a velocity reduction and the corresponding static pressure increases (Fig. 9).

The actual static pressure recovery coefficient ($C_p\text{ actual} = \Delta P/(0.5 \rho U^2)$) is 0.612 for the case under consideration. The ideal pressure recovery coefficient for the diffuser ($C_p\text{ ideal} = 1 - (A_1/A_2)^2$) is equal to 0.63809. Pressure recovery effectiveness, which represents the ability of the diffuser to achieve ideal recovery characteristics, is the ratio of the $C_p\text{ actual}$ to the $C_p\text{ ideal}$. Thus, for the case under consideration pressure recovery effectiveness is about 96% which can be considered as a very good value and shows that the diffuser design as well as the performance of the pre-diffuser are quite reasonable.
In order to study the effect of the inlet velocity on the present design a parametric study has been carried out. It is a known fact that the total pressure losses are influenced by the inlet velocity. The inlet velocity of the design-combustor is 198.5 m/s. A parametric study has been carried out with ±10 and 20% of the design velocity to establish the pressure recovery coefficient as well as total pressure losses. The inlet velocity from 158.8 to 238.2 m/s in steps of 19.85 m/s. Table 3 shows how the total pressure losses, velocity outlet of flame tube and the static pressure recovery coefficient vary with inlet velocity.

It should be recognized that the pressure loss is mainly influenced by turbulence rather than heat addition. In the present study cold losses due to friction and turbulence come into play. As expected that increase in inlet velocity increases the total pressure losses, which can be clearly seen in Fig. 10. The variation of static pressure recovery coefficient with inlet velocity is shown in Fig. 11. As the velocity increases the static pressure recovery coefficient increases marginally. To put this quantitatively a 50% increase in the inlet velocity (from 160-240 m/s) produces an increase of about 1% (0.609 to 0.616) in static pressure recovery coefficient.

Optimization of pre-diffuser geometry

The pre-diffuser geometry has been optimized by changing the geometric parameters such as divergence angle, length and wall contour. The
following steps have been considered for optimization of pre-diffuser.

1. Divergence angle has been optimized by changing the angle from 6.5° to 5° in steps of 0.5° with same length and wall contour as original configuration. It has concluded from the numerical studies that the geometry with 6° divergence angle found to be the best possible configuration.

2. Keeping the same divergence angle (6°) the length of the diffuser has been varied from 96 to 165 mm in steps of 8 mm. The pre-diffuser with 88 mm length was found to be good.

3. Keeping the same divergence angle of 6° and length 88 mm, the wall contours have been optimized by changing the wall shape, which can be seen from Fig. 12.

### Analysis for nine different geometries

Basically nine geometries were created for optimization of pre-diffuser by varying the divergence angle, length and wall contours. The performance of the nine different pre-diffusers is shown in Tables 4 to 6. It can be seen from the tables that total pressure losses, static pressure recovery coefficient and overall effectiveness vary with the change of parameters like divergence angle, length and wall contours of pre-diffuser geometry.

Table 4 shows the variation of the diffuser performance with the divergence angle for the four different cases studied. As the divergence angle is increased from 6° to 6.5°, the total pressure loss decreases marginally and the effectiveness of the diffuser is almost same. Actual static pressure recovery coefficient increases with increase in divergence angle. And also the ideal recovery coefficient increases with increase in divergence angle because of the increased outlet area of the pre-diffuser. However, the overall decrement in the total pressure losses is almost negligible. As the divergence angle decreases from 6° to 5°, the total pressure loss increases. But the actual static pressure recovery coefficient showed reasonable decrement. The overall effectiveness of the diffuser also decreases for
REDDY & GANESAN: AERO GAS TURBINE COMBUSTION CHAMBER

Fig. 11—Variation of static pressure recovery coefficient with inlet velocity

![Graph showing variation of static pressure recovery coefficient with inlet velocity.](image)

The variation of performance parameters with respect to the length of the pre-diffuser for fixed divergence angle (6°) is shown in Table 5. The total pressure losses decrease marginally with increasing length of the diffuser. If the diffuser length increases the flame tube length decreases. For good combustion there should be sufficient length of flame tube for mixing, flame stabilizing and complete combustion of fuel. Because of the above reasons the length of the diffuser is not possible to increase and also the length between the compressor exit to the turbine inlet is

---

<table>
<thead>
<tr>
<th>Table 4—Performance of pre-diffusers for different angles</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle</td>
</tr>
<tr>
<td>-------</td>
</tr>
<tr>
<td>6.5</td>
</tr>
<tr>
<td>6.0</td>
</tr>
<tr>
<td>5.5</td>
</tr>
<tr>
<td>5.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 5—Performance of pre-diffusers for different Length</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
</tr>
<tr>
<td>--------</td>
</tr>
<tr>
<td>72</td>
</tr>
<tr>
<td>80</td>
</tr>
<tr>
<td>88</td>
</tr>
<tr>
<td>96</td>
</tr>
</tbody>
</table>

---

Fig. 12—Different Pre-Diffuser Geometries

![Diagram showing different pre-diffuser geometries.](image)

divergence angles from 6° to 5°. Hence, the original divergence angle of 6° can be considered to be comparatively good with respect to overall effectiveness.
Table 6—Performance of pre-diffusers for different wall contours

<table>
<thead>
<tr>
<th>Shape</th>
<th>Actual static pressure recovery coefficient</th>
<th>Ideal static pressure recovery coefficient</th>
<th>Total pressure loss</th>
<th>Overall effectiveness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>0.6119</td>
<td>0.638</td>
<td>6.85</td>
<td>96.335</td>
</tr>
<tr>
<td>Case 1</td>
<td>0.5949</td>
<td>0.648</td>
<td>7.14</td>
<td>92.067</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.6086</td>
<td>0.648</td>
<td>7.00</td>
<td>94.046</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.6145</td>
<td>0.648</td>
<td>6.90</td>
<td>95.022</td>
</tr>
</tbody>
</table>

fixed. Hence the original length of the diffuser is considered to be good for better performance and minimum total pressure loss.

The effect of wall contour is studied for four different cases and is shown in Table 6. From the results it is seen that straight wall diffuser gives better performance and less total pressure losses than others. From the present investigation it may be concluded that the original geometry of the pre-diffuser gives optimum performance and therefore no change is required. The mass flow split and the pressure losses for various geometries are given below.

Validation

Validation of total pressure losses for the present combustor configuration could not be carried out in the view of proposed combustor geometry is under development and no experimental data is available. However, Balasubramaniam has satisfactorily validated the FLUENT code for the baseline combustor configuration, which is similar and has close resemblance to the present configuration. It may be noted that the main aim of the combustor design is to obtain minimum pressure loss, which depends on the aerodynamics design. Further, to the best of authors’ knowledge there are no satisfactory velocity measurements inside the combustion chamber either for the cold or hot flow. Hence, at present one has to be satisfied with such validation of pressure loss only. A typical comparison between experimental and predicted results of total pressure is shown in Fig. 13.

From the graph it is clear that the experimental values are slightly higher than the predicted values. However the trend of predictions is quite satisfactory. This is mainly attributed to the presence of appendages and struts and other obstructions in the flow path in the experimental model and accuracy of experiments. It is to be noted that the present predicted results are quite reasonable and therefore the results can be used with confidence. It should also be remembered that experimental data for a similar combustor system for any other parameter shown in Fig. 1 is almost non-existent because of the complexity of geometry and the flow.

Conclusions

A numerical study has been performed to calculate the nonreacting flows in a typical aero gas turbine combustor geometry, which contains a combination of swirling and recirculating flows. The major conclusions are:

1. Based on the static pressure recovery coefficient and diffuser effectiveness it can be concluded that the original diffuser design is quite satisfactory for wide rage of inlet velocities. Increasing the inlet velocity causes only marginal changes in static pressure recovery.
2. With the increase in inlet velocity the total pressure loss also increases.
3. Based on the optimisation studies on divergence angle, length and wall contours, the original configuration of 6° with 88 mm length and straight wall pre-diffuser is found to be the best configuration.

Greek symbols

\( \Omega \) = angular velocity
\( \alpha \) = angle
\( \beta \) = angle
\( \epsilon \) = angle
\( \mu \) = angle
\( \rho \) = mass density
\( \beta_{r} \) = angle
\( \tau \) = time

Subscripts

Eff = effective
D = divergent
K = constant
Ref = reference

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( A )</td>
<td>area of cross section, m²</td>
</tr>
<tr>
<td>( V )</td>
<td>sonic velocity, m/s</td>
</tr>
<tr>
<td>( C_D )</td>
<td>drag coefficient</td>
</tr>
<tr>
<td>( C_p )</td>
<td>specific heat, J/kg-K</td>
</tr>
<tr>
<td>( E )</td>
<td>energy, J</td>
</tr>
<tr>
<td>( F )</td>
<td>external body force, N</td>
</tr>
</tbody>
</table>
Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h$</td>
<td>sensible enthalpy, J/kg</td>
</tr>
<tr>
<td>$f$</td>
<td>diffusion flux, kg/m$^2$s</td>
</tr>
<tr>
<td>$K$</td>
<td>turbulent kinetic energy, m$^2$/s $^2$</td>
</tr>
<tr>
<td>$l$</td>
<td>mass transfer co-efficient, m/s</td>
</tr>
<tr>
<td>$k_e$</td>
<td>effective thermal conductivity, W/m-K</td>
</tr>
<tr>
<td>$P$</td>
<td>static pressure, Pa</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature, K</td>
</tr>
<tr>
<td>$U$</td>
<td>mean velocity component, m/s</td>
</tr>
<tr>
<td>$X$</td>
<td>longitudinal distance, m</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{Eff}$</td>
<td>effective value (laminar + turbulent)</td>
</tr>
<tr>
<td>$i,j,k$</td>
<td>tensorial notation</td>
</tr>
<tr>
<td>$K$</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>$\text{Ref}$</td>
<td>reference value</td>
</tr>
<tr>
<td>$T$</td>
<td>turbulent</td>
</tr>
</tbody>
</table>

References