Effect of Dilution Hole Geometry on Gas Turbine Combustor Pattern Factors - A CFD Study

Yathisharadhya¹ Dr.H.R. Purushothama²
¹M.Tech Student (T.P.E) ²Associate Professor
¹,²S.I.T, Tumakuru

Abstract— The quality of hot gases originating from gas turbine combustor is measured using two parameters viz., circumferential pattern factor and radial pattern factor. These parameters have a direct effect on the life of turbine vanes and blades and ideally they should be as small as possible. A higher total pressure loss in the combustor liner reduces the pattern factors but from the engine overall performance perspective, it has to be a minimum, as every one percentage increase in total pressure loss results in a half percent reduction in thrust and around a quarter of a percent increase in specific fuel consumption. Among all the variables that influence the pattern factors, the dilution hole area significantly alters the pattern factors. In the present work, CFD analyses have been carried out to estimate the effect of dilution hole geometry on gas turbine combustor pattern factors. During this process, it has been ensured that the total pressure remains unaltered. Analyses have been carried on 20 degree sector of combustor geometry using CFD code “Fluent”. The modifications carried out to the dilution hole geometry were categorized into three groups. Turbulent reacting flow analyses have been carried out using steady state RANS model. The liquid fuel has been modelled as a discrete phase. A reduced two step reaction is used to model the chemistry; turbulence has been modelled using realizable k-epsilon model and combustion by the laminar flamelet model (non premixed combustion model). The flow field characteristics within the combustor, pressure loss, mass flow distribution, maximum temperature and pattern factors at the combustor exit were extracted and reported. It is found that even through the dilution hole remains unaltered, the disposition of dilution holes effects the pattern factors significantly.

Key words: Gas Turbine Combustor, Dilution Hole, Pattern Factors, CFD

I. INTRODUCTION

Gas turbine combustor designs for aero applications are becoming increasingly challenging in order to meet the stringent requirements such as lower emissions, higher durability, lower maximum exit temperatures, lower fabrication and maintenance costs and reduced design and Time to market cycle times. These requirements necessitate more emphasis on Computational Fluid Dynamics (CFD) simulation of the combustion flow field to reduce testing and improve performance, which, for practical aero-engine applications is a complex problem. Improvements in engine performance come in the form of increasing thrust production while increasing the working life of the individual engine components. Increasing the thrust can be accomplished by increasing the gas working temperature of the turbine section. The fact that the combustor exit temperature, especially when it is nonuniform, has a drastic effect on the life of turbine blades, and hence the maintenance costs, makes it a critical design requirement.

Temperature non-uniformity at the exit of the combustor is often referred to as hot streaks. The existence of hot streaks causes local hot spots on the blade surfaces, leads to heat fatigue of blade and reduces blade life. Pattern factor and profile factor of the combustor indicate the non-uniformity of the temperature at the exit of combustor which dictates the life of the turbine blades. A lower value of the same is always desirable. Gas turbine combustor should possess a minimum value of total pressure loss as everyone percentage increase in pressure loss can result in either a half percent reduction in thrust or around quarter of a percent increase in specific fuel consumption. It comprises of cold loss that include loss due to friction, turbulence and hot loss, which is due to heat addition. In a combustor, hot losses are unavoidable, whereas, cold losses can be reduced by suitably modifying the combustor geometry. However, some pressure loss is beneficial to the combustion and dilution processes, because it gives high injection air velocities and steep penetration and high levels of turbulence, which promotes good mixing and can result in a shorter liner. Equations 1.1, 1.2, 1.3 define the pressure loss, Circumferential Pattern Factor (CPF) and Radial Pattern Factor(RPF) respectively.

\[
\text{Pressure loss} = \frac{P_{\text{avg}} - P_{\text{inf}}}{P_{\text{inf}}} \quad (1.1)
\]

\[
\text{CPF} = \frac{T_{\text{avg}} - T_{\text{avg}}} {T_{\text{avg}}} \quad (1.2)
\]

\[
\text{RPF} = \frac{T_{\text{avg}} - T_{\text{avg}}}{T_{\text{avg}} - T_{\text{avg}}} \quad (1.3)
\]

RPF and CPF indicate the non-uniformity of temperature at the combustor exit and it normalizes the difference between the maximum and mean exit temperatures [1].The quality of combustor exit temperature is influenced by many factors. Dilution and combustion chamber cooling air, the geometry of the combustion chamber, fuel spray characteristics and operating conditions all contribute to the pattern factor in varying degrees. These factors, expanded in Fig. 1, have a high level of interdependency, and it is therefore difficult to separate fully the results of each of their influences.
Fig. 1: Contributors to pattern factor

According to Momtchiloff [2] the two parameters that strongly influence the combustion chamber outlet temperature profile are the combustion chamber length and the pressure loss across the combustion chamber. Sjöblom [3] highlighted the effects of dilution air on pattern factor. He found that as the turbine inlet temperature increased, there was a reduction in the amount of air available for dilution purposes. This reduction in dilution air degraded the dilution zone mixing and resulted in an increase in pattern factor. That study showed that with a 15% increase in dilution air, the pattern factor was reduced from 0.35 to 0.2. This author also describes the three leading contributors to pattern factor. The first was the temperature profile which was created in the primary zone. The second was the behaviour of the combustion chamber cooling air, which created peaks in pattern factor. Finally, the design of the dilution air holes. George and Cox [4] performed analytical and experimental research into pattern factor. Using an annular combustion chamber, they investigated the effects of changing dilution and cooling air, dilution zone geometry and operating conditions. Their conclusions included the observation that pattern factor was mainly affected by the mixing processes within the combustion chamber and the temperature profile created in the primary zone. Clayton Kotzer et al.[5] have experimentally investigated the effect of combustor geometry on the exit temperature fields using an ambient pressure test rig. They concluded that relatively small geometric changes in dilution zone can lead to dramatic changes in the exit temperature field. Lefebvre [6] carried out significant research into the parameters that effect pattern factor. He presented that pattern factor was mainly influenced by a combination of the number of dilution jets and their penetration depth into the combustion chamber. The overall combustion chamber geometry, pressure loss, discharge coefficient of all holes, and airflow distribution still played a role in the creation of the exit temperature profile, but to a lesser degree. He also commented that the temperature profile which enters into the dilution zone contributes to the pattern factor. This entering temperature profile is a function of the fuel spray characteristics of droplet size, evaporation constant and spray angle. Kishore kumar et al.[7], narrated the application of CFD tools in the design and analysis of gas turbine combustors. Sivaramakrishna et al.[8], have carried out 3-D cold flow CFD analysis of an aero gas turbine combustor using the experimental data obtained through customized lab scale tests on an annular aero gas turbine combustor. Srinivasa Rao et al.[9,10], have carried out 3-D reacting flow analysis of an aero gas turbine combustor using „Fluent“. They have also reduced the combustor total pressure loss through CFD analysis by modifying the shape of pre-diffuser struts [11], Motsamai et al.[12] used CFD and mathematical optimization to minimize the combustor exit temperature distribution.

A. Objective

In this work, the CFD analyses has been made to know the effect of dilution hole geometry on gas turbine pattern factors. During this process, it has been ensured that the total pressure remains almost the same. Analyses has been carried out in Fluent for three groups first two groups having three variants, Last group having two variants. The classification of variants is based on their inner and outer dilution hole dimensions. The eight variants having different inner and outer dilution hole diameters (There is only disposition in the holes, But total area of dilution holes are same in all variants) and also few variants are having dilution holes alignment in inner liner walls while few are having alignment in the outer liner wall and also few are having alignment in both inner and outer liner walls respectively. Results obtained through computations were compared with baseline combustor and how dilution holes disposition affects the combustor pattern factors has been studied.

B. Geometrical Modeling

The combustor considered in the present analysis is an annular combustor with 18 swirlers and 18 air blast atomizers equally spaced along the circumferential direction. Owing to the symmetry of the combustor, the calculations are conducted for the flow in a 20° sector with an atomiser and a swirler. The solid model of combustor has been generated using Uni-graphics NX-6, a commercial CAD tool. Figure 2 shows a 3-D 20° sector CAD Model of the combustor which has been used for CFD analyses.

Fig. 2: Geometric model for flow

The baseline and various modified configurations of inner and outer dilution hole geometry that were considered for the analyses have been shown in figure 3.
C. Grid Generation

In the present configuration, unstructured grids fine mesh has been generated using GAMBIT 2.4 software. The 3-d hybrid grid with a cell count of approximately 2.6 million (26,32,261) has been generated for baseline and three different groups. It was found that 76.64% of the grids generated are hexahedral elements, 19.9% tetrahedral elements (0.53% pyramid elements) and remaining 3.37% are wedge elements. The pre-processor used is fluent. Figure 4 shows 3-D view of overall grid generated.

D. Grid Quality

While generating the grid, care has been taken to maintain the quality of mesh with regard to the aspect ratio, equi-angle skew edge ratio and other parameters like equisize skew angle skew are also within acceptable range.

E. Boundary Conditions

Total pressure and total temperature have been specified at the combustor inlet, static pressure along with the target mass flow rate has been specified at the combustor core exit. Turbulent intensity and hydraulic diameter have been specified as the initial conditions for the inlet turbulence. At the combustor bleed where no combustion takes place, mass flow rate boundary condition has been specified in such a way that the prescribed quantity of flow goes out of the domain. All the combustor walls have been treated to be adiabatic. Periodic boundary conditions have been imposed on both the sides of the sector in the circumferential direction.

F. Governing Equations

In the present study, flow is treated to be steady, turbulent, compressible and reacting. The governing Navier-Stokes equations (RANS) for the conservation of mass, momentum, energy and species concentration for the gas, together with an equation of state are approximated for each mesh cell. The resulting sets of equations are solved numerically to obtain the flow field, mixing and combustion data.

G. Injection Model

The aviation turbine fuel (C12H23) has been injected as discrete phase using cone injection model at the exit of the injector fuel passage. Fuel injection parameters used for analyses have been listed in table 1.

<table>
<thead>
<tr>
<th>Type of injection</th>
<th>Conical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injection velocity</td>
<td>40m/s</td>
</tr>
<tr>
<td>Spray cone angle</td>
<td>40°</td>
</tr>
<tr>
<td>Sauter Mean Diameter (SMD)</td>
<td>20 microns</td>
</tr>
</tbody>
</table>

Table 1: Fuel Injection Parameter

H. Chemical Reaction Scheme

A key component of the reacting CFD analysis is a mathematical description of the combustion process and its interaction with the turbulent flow field. In general, numerical modeling of the combustion process requires detailed or reduced chemical reaction mechanisms for adequate accuracy. In the present study, Non premixed combustion model has been used for the simulation of chemical reaction for fuel-air mixture. C12H23 fuel chemistry is modeled by using a simplified two step chemical reaction scheme given below.

\[ C_{12}H_{23} + 11.75O_2 \rightarrow 12CO + 11.5H_2O \quad (2) \]
\[ CO + 0.5O_2 \rightarrow CO_2 \quad (3) \]

The reaction rate is calculated using a combined Arrhenius and Eddy Breakup model. The minimum of these two rates is taken into consideration.

I. Turbulence model

Turbulence has been modeled using Realizable k-ε two equation model[13]. The values of turbulence intensity and hydraulic diameter have been specified as the turbulence initial conditions.

J. Numerical Integration Scheme

The partial differential equations for conservation of mass, momentum, energy, chemical species, turbulent kinetic energy and its dissipation rate are integrated over individual finite control volumes and the resulting volume integrals are transformed into their surface counterparts. The pressure-velocity coupling is achieved using SIMPLE (Semi Implicit Method for Pressure Linked Equations) algorithm.

II. RESULTS & DISCUSSIONS

Table 2 shows the pressure loss predicted for different cases in reacting flow. These pressure loss values have been obtained at an inlet Mach number of 0.34.

<table>
<thead>
<tr>
<th>Regions</th>
<th>Overall Loss (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet</td>
<td></td>
</tr>
<tr>
<td>Baseline</td>
<td>7.50</td>
</tr>
<tr>
<td>Variant 1</td>
<td>7.76</td>
</tr>
<tr>
<td>Variant 2</td>
<td>7.67</td>
</tr>
<tr>
<td>Variant 3</td>
<td>7.58</td>
</tr>
<tr>
<td>Variant 4</td>
<td>7.54</td>
</tr>
<tr>
<td>Variant 5</td>
<td>7.62</td>
</tr>
<tr>
<td>Variant 6</td>
<td>7.53</td>
</tr>
<tr>
<td>Variant 7</td>
<td>7.57</td>
</tr>
<tr>
<td>Variant 8</td>
<td>7.58</td>
</tr>
</tbody>
</table>

Table 2 Pressure Loss - Reacting flow

It is found that pressure loss for reacting flow analyses is about 0.3 to 0.5% higher than that of non-reacting flow analyses.

BASE LINE | Variant 1 | Variant 2 | Variant 3 |
Effect of Dilution Hole Geometry on Gas Turbine Combustor Pattern Factors - A CFD Study

Table 3: Comparison of change in temperature distribution at exit plane with respect to baseline combustor

<table>
<thead>
<tr>
<th>Variant</th>
<th>Max temp</th>
<th>Mean temp</th>
<th>CPF</th>
<th>RPF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Variant-5</td>
<td>+34</td>
<td>+4</td>
<td>+0.03</td>
<td>+0.01</td>
</tr>
<tr>
<td>Variant-6</td>
<td>+8</td>
<td>+3</td>
<td>0</td>
<td>+0.02</td>
</tr>
<tr>
<td>Variant-7</td>
<td>+66</td>
<td>+4</td>
<td>+0.08</td>
<td>+0.07</td>
</tr>
<tr>
<td>Variant-8</td>
<td>+68</td>
<td>+2</td>
<td>+0.10</td>
<td>+0.07</td>
</tr>
</tbody>
</table>

III. DISCUSSION

In this section the results obtained by the CFD analyses for the different combustor configurations have been compared with the baseline combustor.

1) Group 1: The Variant-1, Variant-2& Variant-3 comes under group-1. In Variant-1 both CPF and RPF values are increased by 0.02 and 0.06 respectively with respect to baseline case. In Variant-2 both CPF and RPF values are increased by 0.02 and 0.08 respectively with respect to baseline case. In Variant-3 also both CPF and RPF values are increased by 0.02 and 0.04 respectively with respect to baseline pattern factors. In this group none of the variants giving better CPF and RPF values than baseline case.

2) Group 2: There are 3 variants i.e. Variant-4, Variant 5& Variant 6 in group-2. In Variant-4 both CPF and RPF values are increased by 0.02 and 0.06 respectively with respect to baseline case. In Variant-5 also both CPF and RPF values are increased by 0.03 and 0.01 respectively with respect to baseline case. But in Variant-6 only RPF value is increased by 0.02 and CPF value remain same as baseline pattern factors. In this group none of the variants giving better CPF and RPF values than baseline case. But Variant-5 and variant-6 CPF and RPF values are almost nearer to base case. So group-2 variants are better than group-1 variants.

3) Group 3: The Variant-7 and Variant-8 comes under group-3. In Variant-7 both CPF and RPF values are increased by 0.08 and 0.07 respectively with respect to baseline case. In Variant-8 both CPF and RPF values are increased by 0.10 and 0.07 respectively with respect to baseline case. These group variants are giving worst CPF and RPF values than all other variants. So group-3 variants are worst group among all groups.
The RPF curves along the annulus height for all the studies have been plotted in figure 7. It can be seen from the figure that the peak of RPF profile occurs near 82% of the annulus height for all configurations and variant 5&6 RPF profiles are nearer to base line profile when compare to all other variants.

IV. CONCLUSION
Numerical prediction of flow is carried out inside an aero gas turbine combustor under non-reacting and reacting conditions. The inner and outer dilution hole diameter and also alignment has been varied (without changing the area of dilution holes) to improve the pattern factors. Overall total pressure loss, percentage mass flow split and pattern factors have been estimated for all the configurations. The following conclusions are drawn from this study.
- The flow decelerates smoothly along the diffuser passage from inlet to exit. There is flow separation at the end of diffuser wall leading to a small recirculation zone.
- Mass flow distribution through all the liner holes has been estimated and it is found that the mass flow rate on dilution holes in the inner annulus region reduces as hole diameter is reduced.
- The overall pressure loss for reacting flow conditions is about 0.3 to 0.5% higher than that of non-reacting flow conditions and overall pressure loss increases as the dilution diameter is reduced.
- It is found that even through the dilution holes area remains unaltered; the disposition of dilution holes affects the pattern factors significantly.
- From all the geometrical configurations we have studied, none of the configurations gave better CPF and RPF values than base line. But CPF and RPF values of variant-5 and variant-6 are good when comparing to other variants. And also CPF and RPF values are almost nearer to base case. So further investigation on group-3 will give better pattern factors and increases the life of engine.

REFERENCES