Design and analysis of annular combustion chamber of a low bypass turbofan engine in a jet trainer aircraft

C. Priyant Mark\textsuperscript{a,*}, A. Selwyn\textsuperscript{b}

\textsuperscript{a}Department of Aerospace Engineering, Karunya University, Coimbatore, T.N., India
\textsuperscript{b}A.E.R.D.C., Hindustan Aeronautics Ltd., Bangalore, KA, India

Received 19 May 2014; accepted 4 March 2016
Available online 26 May 2016

\textbf{Abstract} The design of an annular combustion chamber in a gas turbine engine is the backbone of this paper. It is specifically designed for a low bypass turbofan engine in a jet trainer aircraft. The combustion chamber is positioned in between the compressor and turbine. It has to be designed based on the constant pressure, enthalpy addition process. The present methodology deals with the computation of the initial design parameters from benchmarking of real-time industry standards and arriving at optimized values. It is then studied for feasibility and finalized. Then the various dimensions of the combustor are calculated based on different empirical formulas. The air mass flow is then distributed across the zones of the combustor. The cooling requirement is met using the cooling holes. Finally the variations of parameters at different points are calculated. The whole combustion chamber is modeled using Siemens NX 8.0, a modeling software and presented. The model is then analyzed using various parameters at various stages and levels to determine the optimized design. The aerodynamic flow characteristics is simulated numerically by means of ANSYS 14.5 software suite. The air-fuel mixture, combustion-turbulence, thermal and cooling analysis is carried out. The analysis is performed at various scenarios and compared. The results are then presented in image outputs and graphs.

\textcopyright{} 2016 National Laboratory for Aeronautics and Astronautics. Production and hosting by Elsevier B.V.

This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).
1. Introduction

Gas turbine engine evolved as a critical part and the most efficient propulsion unit for aircrafts. It is now used in almost all of the passenger aircrafts worldwide with different variations. Military aircraft made the debut in using the turbojet engine. As the technology progressed, high performance military aircraft began using low bypass turbofan engines due to its advanced capabilities, efficiency and reliability, even at supersonic speeds.

Low bypass turbofans have a bypass ratio of around 1:1 or less [1]. A high specific thrust/low bypass ratio turbofan normally has a multi-stage fan, developing a relatively high pressure ratio and, thus, yielding a high (mixed or cold) exhaust velocity. The core airflow needs to be large enough to give sufficient core power to drive the fan. A smaller core flow/higher bypass ratio cycle (for the fan operation) can be achieved by raising the high pressure (HP) turbine rotor inlet temperature. The temperature rise of the airflow from the intake to the nozzle of the engine is also less, which results in a reduced fuel flow leading to a better specific fuel consumption (SFC) for the same pressure ratio. Thus, a low bypass turbofan would add to the efficiency of the engine.

Jet fighters as well as trainers are high performance aircraft that use the most powerful engines for producing thrust. The process of upgrading military hardware has initiated the race to develop even more powerful engines. By increasing power, the engines require more fuel input, thereby resulting in fuel guzzling engines. This directly points to an inefficient engine in terms of fuel consumption. Fuel consumption efficiency is required even in military aircraft as it can aid in increasing the range. For improving efficiency, the very fundamentals lie in the combustion chamber. An efficient combustion chamber is the answer for better performances.

The most commonly used type of combustor is the fully annular combustor, the others being tubular and tuboannular combustor. Annular combustors [2] do away with the separate combustion zones and simply have a continuous liner and casing in a ring (the annulus). There are many advantages to annular combustors, including more uniform combustion, shorter size (therefore lighter), and less surface area. Additionally, annular combustors tend to have very uniform exit temperatures. They also have the lowest pressure drop of the three designs (on the order of 5%). The annular design is also simpler, although testing generally requires a full size test rig. Most modern engines use annular combustors; likewise, most combustor research and development focuses on improving this type. This paper deals with designing an efficient annular combustion chamber for use in jet trainer aircrafts.

Conrado [3] has discussed a design methodology which follows a similar approach for designing a micro gas turbine combustor. It also showcases an example and further automating the same approach using a computer program for ease of use. Silva [4] has discussed a consolidated design methodology for an automotive turbocharger utilizing a micro gas turbine combustor. It gives a brief report along with heat-transfer analysis. Generally, the computational fluid dynamics (CFD) analysis of a combustor is carried out based on different combustion models [5]. Few models such as Westbroor-Dryer one step model and Westbroor-Dryer two step model hold good for laminar combustion simulation. Likewise K-epsilon model, K-omega model and K-omega shear stress transport (SST) model hold good for turbulent combustion simulation. The present paper discusses mainly about designing a gas turbine combustor at a scale of a jet trainer aircraft engine using the most straightforward and transparent approach. It also focuses on reducing the development time and gives ample support for refining the design at every phase. The paper also presents a computer aided design (CAD) model designed using the same principles to show the practicality in using the design. For an accurate CFD analysis result of a gas turbine combustion chamber, it needs to simulate combustion and turbulence simultaneously. This paper gives a detailed CFD analysis report of the designed combustor based on the combustion-turbulence interaction model.

2. Aerodynamic design

2.1. Preliminary design procedure

The procedure purposed by Melconian and Modak (1985) [6] to design a combustor is described in Figure 1. The equations utilized in the design procedure is presented, which is sufficient for the reader to understand the design methodology idea.

2.2. Initial design parameters

The initial design parameters are mostly the compressor exit and turbine inlet constraints, which is usually absorbed for any combustion chamber design. Others include customer specifications, constants, experimental values and limits. Table 1 shows the initial parameters used for the design, which were obtained from real-time data.

3. Dimensions

3.1. Casing area

Eq. (1) calculates the reference area [7].

\[
A_{ref} = \left[ \frac{R}{2} \left( \frac{m_3 T_3^{0.5}}{P_3} \right)^2 \frac{\Delta P_{3-4}}{q_{ref}} \left( \frac{\Delta P_{3-4}}{P_3} \right)^{-1} \right]^{0.5}
\]  

(1)
3.2. Liner area

The combustor sectional area ($A_L$) can be calculated by Eq. (2) [6].

$$A_L = 0.66A_{ref}$$  \hspace{1cm} (2)
Aan = 0.02794 m²

3.4. Casing and liner diameter

Figure 2 presents the reference length $D_{ref}$ for annular combustor configuration. The value of $D_{ref}$ is calculated from $A_{ref}$ and $D_L$ is calculated from $A_L$ and it must be chosen such that it accommodates the aerodynamic considerations in every operating condition.

$D_{ref} = 0.061$ m

$D_L = 0.04026$ m

3.5. Pattern factor

The pattern factor or temperature traverse quality gives the temperature distribution of the efflux gases across the radial and circumferential direction at the exit of the combustor. It is an important factor for the turbine inlet blades. It also influences the liner length. It is defined in Eq. (4).

\[
PF = \frac{T_{max} - T_4}{T_4 - T_3}
\]  

$(4)$

$PF = 0.25$

3.6. Liner length

The liner length [7] provides the total length of the zones. It can be calculated from Eq. (5).

\[
L_L = \frac{-D_L}{0.05 \Delta P_{ref} \ln(1 - PF)}
\]  

$(5)$

$L_L = 0.15719$ m

3.7. Primary zone length

The length of the primary zone can be calculated from Eq. (6) [6].

\[
L_{PZ} = \frac{3}{4} D_L
\]  

$(6)$

$L_{PZ} = 0.03020$ m

3.8. Secondary zone length

The length of the secondary zone can be calculated from Eq. (7) [6].

\[
L_{SZ} = \frac{1}{2} D_L
\]  

$(7)$

$L_{SZ} = 0.02013$ m

3.9. Dilution zone length

The length of the dilution zone can be calculated from Eq. (8) [6].

\[
L_{DZ} = D_L \left(3.83 - 11.83PF + 13.4PF^2\right)
\]  

$(8)$

$L_{DZ} = 0.06933$ m

4. Air flow distribution

For conventional design [2], about half of primary zone air mass flow rate would be admitted through the swirler

Table 1  Initial design parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$m_3$</td>
<td>28.7103</td>
<td>kg/s</td>
</tr>
<tr>
<td>$T_3$</td>
<td>743.352</td>
<td>K</td>
</tr>
<tr>
<td>$P_3$</td>
<td>2083450</td>
<td>Pa</td>
</tr>
<tr>
<td>$m_{mf}$</td>
<td>0.25818</td>
<td>kg/s</td>
</tr>
</tbody>
</table>
and as dome cooling. The mass flow rate \(m_{RZ}\) corresponds to the sum of the air admitted in primary zone through the swirler and the air admitted through dome cooling slots. The swirler mass flow rate \(m_{SW}\), for flame ignition and stability, should have an equivalence ratio above 1. The rest of the air flow goes through the annulus \(m_{AN}\). Then it is distributed to the primary, secondary and dilution zones as per the requirement. For cooling, based on the formula [5]:

\[
\text{Cooling air} \% = 0.1T_1 - 30, 40\% \text{ of the total mass flow rate is taken and distributed along the zones based on the temperature. Table 2 and Figure 3 show the air mass flow rate distribution in the combustor.}
\]

The primary/recirculation zone equivalence ratio should never be richer than 1.5 in order to minimize smoke, carbon monoxide (CO) and unburned hydrocarbons (UHC) in the exhaust gases. To prevent nitrous oxides (NO\(_x\)) and other pollutants due to thermal dissociation, the equivalence ratio is capped at a maximum value of 0.6. The equivalence ratio of secondary zone shall not be higher than 0.8. The fuel/air ratio and the equivalence ratio [8] for the respective zones was calculated and was found to be within the above limits. It is presented in Table 3.

### Table 2 Air mass flow rate distribution.

<table>
<thead>
<tr>
<th>Air mass flow rates</th>
<th>Symbol</th>
<th>Percentage/%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>(m_1)</td>
<td>100</td>
</tr>
<tr>
<td>Recirculation zone/snout</td>
<td>(m_{RZ})</td>
<td>20</td>
</tr>
<tr>
<td>Swirler</td>
<td>(m_{SW})</td>
<td>12</td>
</tr>
<tr>
<td>Dome cooling</td>
<td>(m_{Dcool})</td>
<td>8</td>
</tr>
<tr>
<td>Annulus</td>
<td>(m_{AN})</td>
<td>80</td>
</tr>
<tr>
<td>Primary zone</td>
<td>(m_{PZ})</td>
<td>20</td>
</tr>
<tr>
<td>Secondary zone</td>
<td>(m_{SZ})</td>
<td>10</td>
</tr>
<tr>
<td>Dilution zone</td>
<td>(m_{DZ})</td>
<td>10</td>
</tr>
<tr>
<td>Cooling air</td>
<td>(m_{cool})</td>
<td>40</td>
</tr>
</tbody>
</table>

### Table 3 Fuel/air ratio and equivalence ratio for the zones.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Recirculation zone</th>
<th>Primary zone</th>
<th>Secondary zone</th>
<th>Dilution zone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel/air ratio</td>
<td>0.0450</td>
<td>0.0150</td>
<td>0.0118</td>
<td>0.0090</td>
</tr>
<tr>
<td>Equivalence ratio</td>
<td>0.6701</td>
<td>0.2234</td>
<td>0.1763</td>
<td>0.1340</td>
</tr>
</tbody>
</table>

5. Diffuser dimensions

The diffuser, swirler and the recirculation zone geometry is presented in Figure 4.

#### 5.1. Snout outer area

The snout outer area \(A_o\) is calculated assuming the air velocity in this sectional area is equal to \(A_{an}\) air velocity, then use Eq. (9) [3].

\[
A_o = \frac{m_3}{m_{an}} A_{an}
\]

\(A_o = 0.03492\ m^2\)

#### 5.2. Snout outer diameter

The snout outer diameter is obtained from \(A_o\) with the calculations similar to liner diameter.

\(D_o = 0.02593\ m\)

#### 5.3. Diffuser angle

The diffuser angle \(\phi\) can be obtained from Eq. (10) [7].

\[
\phi = \tan^{-1} \left[ \frac{\Delta P_{dif} A_3^2 P_3^2}{502.4 \left(1 - \frac{A_3}{A_1}\right)^2 m_3^2 T_3} \right]^{1/1.22}
\]

\(\phi = 60.14^\circ\)

#### 5.4. Diffuser length

The diffuser length can be obtained with the help of Eq. (11) [3], where \(R_o\) and \(R_3\) are \(D_o/2\) and \(D_3/2\), respectively.

\[
L_{dif} = \frac{(R_o - R_3)}{\tan \phi}
\]
\[ L_{\text{diff}} = 0.00170 \text{ m} \]

6. Swirler dimensions

6.1. Snout area

The snout area can be calculated from Eq. (12) [6].

\[
A_S = A_o \frac{m_{RZ}}{m_c} C_{dc} \tag{12}
\]

\[ A_S = 0.01075 \text{ m}^2 \]

6.2. Snout diameter

The snout diameter is calculated from \( A_S \) with similar calculations to liner diameter.

\[ D_S = 0.00798 \text{ m} \]

6.3. Swirler flow area

The swirler flow area can be calculated from Eq. (13) [3].

\[
A_{SW} = \left( \frac{A_{ref}^2}{\frac{\nu_{SW}}{n_{SW}} \left( \frac{m_L}{m_{SW}} \right)^2 + \left( \frac{A_{ref}}{\beta_{ref}} \right)^2} \right) \cos^2 \beta_{SW} \cos \beta_{SW} \tag{13}
\]

\[ A_{SW} = 0.00464 \text{ m}^2 \]

6.4. Swirler diameter

The diameter of swirler is calculated using Eq. (14).

\[
D_{SW} = \sqrt{\left( \frac{A_{SW}}{n_B} + \left( \frac{\pi}{4} D_{hub}^2 \right) \right) \frac{4}{\pi}} \tag{14}
\]

\[ D_{SW} = 0.02290 \text{ m} \]

7. Recirculation zone dimensions

7.1. Recirculation zone length

The length of the recirculation zone approximates two swirler diameters and can be obtained from Eq. (15) [3].

\[
L_{RZ} = 2D_{SW} \tag{15}
\]

\[ L_{RZ} = 0.04579 \text{ m} \]

7.2. Recirculation zone angle

The recirculation zone angle can be obtained from Eq. (16) [3].

\[
\theta_{RZ} = \cos^{-1} \left( \frac{-D_L(D_L - 2D_{SW}) - (D_L - 4L_{RZ}) \sqrt{D_L^2 - 4D_LD_{SW} + 4D_{SW}^2 - 8D_LL_{RZ} + 16L_{RZ}^2}}{2D_L^2 - 4D_LD_{SW} + 4D_{SW}^2 - 8D_LL_{RZ} + 16L_{RZ}^2} \right) \tag{16}
\]

\[ \theta_{RZ} = 14.4^\circ \]

7.3. Dome length

The dome length can be calculated from Eq. (17) [3].

\[
L_{dome} = \frac{D_L - D_{SW}}{2 \tan \theta_{RZ}} \tag{17}
\]

\[ L_{dome} = 0.03448 \text{ m} \]

8. Holes

Firstly, it is necessary to verify if there is enough air to enter through every combustion chamber zone holes. Then, it is determined how much air enters through the primary zone, the secondary zone and the dilution zone from the difference between the total amount of air to enter each zone and the amount of air that enters through other means (for example, through the swirler or slots). After the determination of the mass flow rate in each zone and the main hole type – plain and cooling hole type – ‘stacked ring’, then it is possible obtain the hole area for each zone. This is an iterative process that follows a structured sequence [3,9]. Table 4 shows the calculated holes.

<table>
<thead>
<tr>
<th>Zones</th>
<th>Main holes</th>
<th>Cooling holes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>No. of holes</td>
<td>Hole diameter/m</td>
</tr>
<tr>
<td>Primary zone</td>
<td>40</td>
<td>0.01516</td>
</tr>
<tr>
<td>Secondary zone</td>
<td>20</td>
<td>0.01502</td>
</tr>
<tr>
<td>Dilution zone</td>
<td>20</td>
<td>0.01502</td>
</tr>
</tbody>
</table>

Table 4: The holes for the zones.
9. Gas temperature profile

9.1. The adiabatic flame temperature

The gas temperature profile is predicted theoretically using numerical calculations to obtain design point values. The profile is more associated with the core temperature of the gas mixture due to the flame concentration at the core where the combustion is at the maximum. It is connected with the adiabatic flame temperature as this temperature determines the core temperature of the gas mixture inside the liner. This is the temperature that the flame would attain if the net energy liberated by the chemical reaction that converts the fresh mixture into combustion products were fully utilized in heating those products. In practice, heat is lost from the flame by radiation and convection, so the adiabatic flame temperature is rarely achieved. It means that, the theoretical calculations give only an approximate value of the gas temperature. Nevertheless, it plays an important role in the determination of combustion efficiency and in heat-transfer calculations.

9.2. Calculation

For the average gas temperature calculations [3] inside the liner at different zones, only the core air mass flow rate is considered. The core air mass flow excludes cooling air and other combusted byproducts.

Table 5 CFD modeling data.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discretization</td>
<td>Finite volume method</td>
</tr>
<tr>
<td>Domain</td>
<td>Combustion-eddy dissipation</td>
</tr>
<tr>
<td>Meshing method</td>
<td>Octree/advancing front</td>
</tr>
<tr>
<td>Total elements</td>
<td>2906742</td>
</tr>
<tr>
<td>Total nodes</td>
<td>592157</td>
</tr>
</tbody>
</table>

The combustor is divided into four zones: recirculation zone, primary zone, secondary zone and dilution zone. For each zone, the local temperature is assumed to vary linearly between the zone inlet temperature ($T_{in}$) and zone outlet...
temperature ($T_{\text{out}}$). For every zone, the outlet temperature is calculated by Eq. (18).

$$T_{\text{out}} = T_3 + \eta \Delta T$$

(18)

For recirculation zone, $T_{\text{in}}$ is assumed to be equal to $T_3$ and the inlet temperature for every other zone is the outlet temperature of the preceding zone. $\Delta T$ is the temperature rise from $T_3$ to adiabatic flame temperature which is
calculated from the chart [7] on Figure 5 for the fuel JP-5 using the corresponding FAR values of each zone.

9.3. **Theoretical results**

The calculated zone outlet temperatures [4] for each zone is presented as graph in Figure 6.

10. **Modeling**

10.1. **CAD model**

The design was modeled using Siemens NX 8.0. The views are presented in Figures 7 and 8.

10.2. **Analyzable model**

To analyze the created model and obtain quicker results, due to computing limitations, the model was simplified into a 20° cut section for a single burner. The view is presented in Figure 9.

11. **Aerodynamic analysis**

The computational aerodynamic analysis is carried out to validate the theoretical results and to obtain a detailed preview of the outcome of the design in real-time working conditions. It was done using the commercially available CFD code ANSYS 14.5 CFX to get a quick report of the computed data. The analysis was performed using the design parameters from Table 1 as inlet and turbine inlet data as outlet conditions. The initial setup data is given in Table 5. The eddy dissipation combustion model which uses Eq. (19) and (20) [10], in combination with the finite rate chemistry model which uses Eq. (21) [10] was used in the analysis, which allows accurate simulation of the heat release and the distribution of the main chemical species. This is a combustion-turbulence interaction model, which significantly improves accuracy of analysis results. Figures 10–14 give the results.

\[
R_{i,r} = \nu_i/M_w \rho A P e \min \left( \frac{Y_{\kappa}}{\nu_{R_i} M_w \kappa} \right)
\]  
(19)
The comparison [11] of radial and circumferential pattern factor is given in Figure 16.

The comparison of target and achieved pressure loss across the combustor is given in Figure 17.

12. Results and discussion

The complete annular combustor design using just the initial design parameters has been clearly discussed in this paper. This is a more refined design methodology which can be used for the preliminary design. The transparent and detailed approach is focused on reducing design time and complexity. This gives an overall advantage in total design time and prototype building. Using the methodology, a practical design is presented. It follows the optimum values for a preliminary setup. The obtained values are used for modeling and further simplified for analysis. The analysis was also carried out with higher accuracy using the combustion-turbulence interaction model and the results show that the optimum (higher) gas exit temperature was obtained for the present design. This gives a very promising outcome with respect to the exit temperature. The pattern factor was obtained as desired. The SFC was also reduced by regulating the temperature rise across the combustor. The required efficiency and pressure loss was achieved by a thin margin. The designed combustor was found to be shorter than the other combustors of its class, which gives it an edge over others in the space constraint category. Based on theoretical calculations and obtained results, the design point combustor exit temperature (enthalpy addition) was achieved within 96% efficiency. Thus, the design is capable of reaching higher exit temperatures.

13. Conclusions

The design was successfully calculated and modeled. The required simpler model for analysis was also created. Then the model was aerodynamically analyzed at design point and the geometry was optimized based on the results. This has delivered one of the most efficient combustion chamber design that can be used in the Jet Trainer Aircraft.

Acknowledgments

The authors are grateful to HAL, Bangalore for the contribution of resources to complete the design and analysis successfully.

References


