A detailed experimental and numerical investigation of a single can combustor has been carried out. Using thermocouples, inner and exit gas temperatures as well as temperatures along the liner wall were recorded. These temperatures are then compared to temperatures obtained using a numerical model for the same combustor. The numerical model constitutes a CFD model developed using a commercial CFD code and implementing the presumed-PDF model of unpremixed turbulent reaction. The flow rate at each air jet is the actual flow rate measured at that hole, while the angle at which the air enters is calculated using an empirical formulation of the discharge coefficient. Despite the assumptions made during development of the model, trends and distributions obtained experimentally were predicted correctly, pointing out the potential of numerical models and the need for further development. Evaluation of the numerical model showed that the temperatures predicted compared reasonably with that obtained experimentally.

Introduction

Gas turbine engineering faces many challenges in the constant strive to increase the efficiency of engines during the various stages of development. Current developments in gas turbine combustors are dominated by the search for computational fluid dynamics (CFD) models that can simulate combustion accurately. Such numerical models must be able to accurately predict the combustion process in terms of temperatures, species concentration and velocities. However, despite the obvious advantages of CFD in reducing costs and development time, current models developed with the aid of general codes
are not capable of accurately simulating the combustion process. Ebbinghaus and Swithenbank [1] applied CFD to obtain a more uniform temperature profile in a can combustor. Temperature measurements in the secondary zone and at the exit plane of the combustor, showed that the numerical model underestimates the mixing behind the jets which results in an overprediction of the cyclic temperature variations immediately downstream of the jets. Similarly Kurreck et. al. [2] validated a two-phase flow simulation code against temperature and velocity values, concluding that the numerical model correctly predicted the distribution of properties, but that the temperatures were predicted much higher. Koopman [3] investigated the concept of rich quench lean combustion numerically and experimentally. It was concluded that the calculated temperatures of the central region decrease slower axially than the comparative measured temperatures. In this study a single can research combustor is investigated experimentally and numerically, with the purpose of obtaining a numerical model that can accurately predict the combustion properties that were obtained experimentally.

Research combustor

The single can combustor is divided into a primary, secondary and dilution zone as shown in Figure 1. Six air jets are located in the primary zone, with twelve air jets in the secondary and ten air jets in the dilution zone. An axial swirler centered on a single fuel nozzle enhances the mixing process and the formation of recirculating zones. Film-cooling air is injected into the secondary and dilution zone using stacked rings with 40 entry holes. The liner is made from AISI 304 stainless steel with the inside coated with a thermal barrier coating. The diameter ratio of the combustor casing relative to the combustor liner was equal to 2.42. This is large compared to normal practice and was done in an effort to reduce the influence that the measuring equipment has on the flowfield. The combustor reference length (L) is defined from the swirller inlet to the exit of the combustor and is equal to 174.8 mm. The reference radius (R) is equal to 41.2 mm.

![Figure 1: Sectored view of the can combustor with different operating zones](image-url)
Experimental measurements

The tests were carried out at atmospheric conditions with the test parameters presented in Table 1. Measurements consisted of liner wall, exit gas and inner gas temperatures. The outer liner temperatures were measured using k-type thermocouples protected by a 1.5mm diameter 300mm long stainless steel sheath. A total of twenty measurement points were taken with the geometrical locations of the different points shown in Figure 2. The combustor is divided into four circumferential positions, with each position at an angle of forty-degrees from the horizontal symmetry line. At position one, eight thermocouples were attached axially along the combustor. At position two, three and four, four thermocouples were attached axially that coincided with locations at position one. Measurements therefore give both the axial temperature distribution from the primary to the dilution zone, as well as the circumferential temperature distribution.

Exit gas temperatures are also measured using the same type of thermocouples. The thermocouples are positively ventilated and fitted with radiation shields in an effort to render true convective temperature measurements. Five thermocouples, radially spaced are mounted on a traverse mechanism at the exit of the combustor and measurements are taken at 10-degree intervals.

<table>
<thead>
<tr>
<th>Test parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet air pressure</td>
<td>94 [kPa]</td>
</tr>
<tr>
<td>Inlet air temperature</td>
<td>300 [K]</td>
</tr>
<tr>
<td>Combustion chamber pressure drop</td>
<td>2.9 [kPa]</td>
</tr>
<tr>
<td>Air mass flow-rate</td>
<td>0.1 [kg/s]</td>
</tr>
<tr>
<td>Fuel mass flow-rate</td>
<td>0.769 [g/s]</td>
</tr>
<tr>
<td>Air-Fuel ratio</td>
<td>130</td>
</tr>
</tbody>
</table>

Figure 2: Liner wall temperature measurement positions
Gas temperatures inside the combustor were measured using an R-type thermocouple fitted with a ceramic sheath of 4.6mm in diameter. Figure 3 shows the five cross-sections in the axial direction where measurements were taken. Measurements were restricted to 30-degree intervals for a total of 84 points.

**Numerical model**

An investigation of the different combustion models available led to the implementation of the presumed-PDF model of unpremixed turbulent reaction. In order to reduce the computational effort, it was decided to split the combustor along the symmetry plane and model only one half of the combustor. The passages of the swirler were not modelled, but only the outlets of the passages through the liner. The holes in the cooling rings were modelled as rectangular holes with the same area as the effective round holes. The annulus together with an outlet section present in the experimental rig was included in the computational domain. A view of the computational grid without the external flowfield is shown in Figure 4.

The boundary conditions prescribed consist of the conditions at the various inlets, outlet, the symmetry plane and inside and outside surfaces of the liner. The radial $v_r$; tangential $v_\theta$; and axial $v_z$ velocities; the turbulence intensity $I$; mixing length $L_\text{mix}$; temperature $T$ and density $\rho$ which were prescribed at the various inlets are summarized in Table 2. The velocity boundary values prescribed at the various inlets are obtained from the experimental jet angles ($\theta$) and discharge coefficients ($C_d$) for isothermal conditions [4]. The discharge coefficient is obtained from the mass rate of flow ($\dot{m}$), pressure change ($\Delta P$) and density ($\rho$) at each set of holes:

$$\dot{m} = C_d A \sqrt{2\Delta P \rho}$$  \hspace{1cm} (1)
The hole pressure-drop coefficient is defined from the jet and annulus dynamic pressure:

\[ K_c = 1 + \frac{\Delta P_b}{q_{in}} \]  

(2)

The jet angles at the various holes is then calculated as:

\[ \theta = \sin^{-1}\left(\frac{(K_c - 1)}{(1.2C_sK_s)}\right) \]  

(3)

The effective hole area is defined using the discharge coefficient and geometrical area (\(A_g\)):

\[ A_{eff} = A_e \times C_d \]  

(4)

The effective velocity at each inlet is then calculated from the mass rate of flow and density:

\[ V_{eff} = \frac{m}{\rho A_{eff}} \]  

(5)

The radial, tangential and axial velocity component is lastly obtained from the effective velocity and jet angles:

\[ v_{r,t,a} = V_{eff} \cos \theta \]  

(6)

Table 2: Boundary conditions at the various inlets

<table>
<thead>
<tr>
<th>Inlet</th>
<th>(v_r) [m.s(^{-1})]</th>
<th>(v_t) [m.s(^{-1})]</th>
<th>(v_a) [m.s(^{-1})]</th>
<th>(I)</th>
<th>(L_c) [m]</th>
<th>(T) [K]</th>
<th>(\rho) [kg.m(^{-3})]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel</td>
<td>23.792</td>
<td>0</td>
<td>23.792</td>
<td>0</td>
<td>2.66\times10^{-4}</td>
<td>293</td>
<td>1.679</td>
</tr>
<tr>
<td>Swirl</td>
<td>0</td>
<td>53.88</td>
<td>53.88</td>
<td>0.1</td>
<td>1.25\times10^{-4}</td>
<td>300</td>
<td>1.011</td>
</tr>
<tr>
<td>Cooling</td>
<td>0</td>
<td>0</td>
<td>78.415</td>
<td>0.1</td>
<td>2.82\times10^{-5}</td>
<td>300</td>
<td>1.011</td>
</tr>
<tr>
<td>Primary</td>
<td>-67.756</td>
<td>0</td>
<td>39.338</td>
<td>0.1</td>
<td>1.97\times10^{-4}</td>
<td>300</td>
<td>1.011</td>
</tr>
<tr>
<td>Secondary</td>
<td>-65.54</td>
<td>0</td>
<td>42.837</td>
<td>0.1</td>
<td>1.53\times10^{-4}</td>
<td>300</td>
<td>1.011</td>
</tr>
<tr>
<td>Cooling II</td>
<td>0</td>
<td>0</td>
<td>78.868</td>
<td>0.1</td>
<td>2.82\times10^{-5}</td>
<td>300</td>
<td>1.011</td>
</tr>
<tr>
<td>Dilution</td>
<td>-68.906</td>
<td>0</td>
<td>30.636</td>
<td>0.1</td>
<td>3.39\times10^{-4}</td>
<td>300</td>
<td>1.011</td>
</tr>
</tbody>
</table>
It was found that the turbulent character of the flow is dominated by the vigorous mixing which takes place inside the combustor, and as a result the turbulence intensities prescribed at the inlets are not critical. The values for both the fuel mass fraction and the mixture fraction at the fuel were prescribed as 1.0, whilst the oxygen mass fraction and nitrogen mass fraction were respectively prescribed as 0.233 and 0.767 at all the other inlets. The liner was not included in the simulation and except for the various inlets was modelled as a solid boundary. The upper and lower halves of the symmetry plane were defined as cyclic boundaries and all corresponding faces were integrally matched.

Results

One of the major advantages of CFD is the ability to provide detailed insight and comprehensive information that is not always possible to obtain from experimental measurements. The distribution of velocity magnitudes on the geometrical symmetry plane obtained from the numerical simulation is shown in Figure 5. The highest velocities are present at the swirler and various air jet inlets. A recirculating, low velocity area is formed in the primary zone by the interaction of the inlet air from the swirler and primary jets. These low-pressure areas provide a stable-burning region and the conical fuel spray from the burner must intersect the recirculation vortex at its center. This action, together with the general turbulence in the primary zone, greatly assists in breaking up the fuel and mixing it with the incoming air [5].

The air entering the primary jets has an increased penetration depth compared to the secondary and dilution air. The reason is twofold, firstly, the primary hole area is smaller in relation to the dilution holes and, secondly, the inlet air from the secondary and dilution holes are diverted by the combustion products flowing along the core. A larger recirculating region is also formed between the primary and secondary air jets.

Figure 5: Velocity magnitudes depicting the flowfield inside the combustor
The temperature distribution on the geometrical symmetry plane of the combustor obtained numerically and experimentally is shown in Figure 6 and Figure 7 respectively. A maximum temperature of 2160 Kelvin is predicted numerically in a small area close to the combustor liner in the primary and secondary zone. Experimentally the highest temperatures are also recorded close to the liner. Comparing the results, it is seen that the numerical model overpredicts the maximum temperatures that were measured. However, along the core and in the dilution zone, the temperatures are in close proximity. The hot sections between the primary and secondary jets, as well as after the secondary jets indicate that the combustion products penetrate between the jets and that unburnt fuel is transported into the secondary zone where combustion is completed. Significant from the results is the inability of the film-cooling air to provide a protective cooling layer between hot combustion products and the inner liner wall. From the velocity field it is noted that the film cooling air proceeds a short distance from the inlets before being halted by the hot gases flowing downstream.

In both cases, results indicate that the temperatures increase from the center to the liner wall. The lowest temperatures are predicted in the dilution zone and along the center of the combustor. The dilution air overpenetrates and results in the formation of a cold core and consequently unsatisfactory mixing with the hot combustion products is obtained. Despite the differences in maximum values, the results illustrate the ability of the CFD model to predict the correct distributions and trends.

Figure 6: Temperature field inside the combustor obtained numerically
Figure 7: Temperature field inside the combustor obtained experimentally

The temperature distribution at the exit plane of the combustor obtained numerically and experimentally is presented in Figure 8 and Figure 9 respectively. In both cases the distribution of temperatures and range of values are in close proximity. The numerical model accentuates the penetration of hot combustion products between the air entering the dilution jets, compared to the experimental measurements.

Figure 8: One-half of the symmetrical temperature distribution at the exit plane obtained numerically
Figure 9: Temperature distribution at the exit plane obtained experimentally.

Although no comparison is made between liner temperatures, Figure 10 displays the liner wall temperatures that were measured. These results confirm the numerical and experimental results and show the increase in liner temperatures in the secondary zone as a result of incomplete combustion in the primary zone.

Figure 10: Combustor liner wall temperatures obtained experimentally.
Conclusion

An experimental and numerical investigation was done that captured the thermal field of a can-type combustor. Comparison of the experimental and numerical results showed that the numerical model overpredicts the gas temperatures inside the combustor. This is directly related to the shortcomings and assumptions made in the development of the computational model. Despite these shortcomings, trends and distributions obtained experimentally were predicted correctly. The numerical model is capable of providing detailed insight and comprehensive information that is not always possible to obtain from experimental measurements. Further development of the CFD model is necessary to utilize the available resources and potential to accurately predict combustion properties.

References


