ABSTRACT

Computational fluid dynamics and experiments have been used to study the mixing of multiple jets in a can combustor. An existing configuration having a poor exit temperature profile was chosen as the baseline case. In the computations, the air split and axial location of the primary, secondary and dilution jets were held constant while the number of jets at each location were varied parametrically to determine their effect on the exit temperature profile. As a result of these studies, two configurations were selected for experimental verification of the anticipated improved performance. The modified design was found to have a more uniform exit temperature profile than the baseline case. Thus the experimental results generally confirmed the predictions and demonstrated the potential utility of CFD as a design tool.

INTRODUCTION

In a conventional gas turbine combustor the combustion products exiting the primary zone are diluted by cold air injected through a pattern of holes. These jets should mix efficiently with the combustion gases to provide the required radial temperature profile at the exit. At the same time the jets should minimise the circumferential variation in temperature at any radius so that hot spots are not generated.

Both cyclic and non-cyclic circumferential temperature distributions are a concern. Cyclic temperature variations usually result from uneven mixing between fuel and air and incomplete mixing between the dilution jets and the combustion products and can be reduced by design improvements. Non-cyclic variations are caused by non-uniformities in the fuel and air supply and are more difficult to control.

In the past, extensive numerical and experimental work has been carried out to investigate the behaviour of single or multiple jets projected into a cross-flow, as summarised by Holdeman (1993). These investigations have demonstrated that the primary factors defining jet penetration and the rate of mixing are the ratio between the jet and cross-flow momentum fluxes and the hole spacing-to-duct height ratio. Algebraic correlations for the cross-flow temperature distributions of a number of geometries have been determined and the degree of mixing predicted by these relationships has been shown to be strongly sensitive to the cross-flow temperature distribution (Holdeman et al., 1987). It is therefore necessary to be able to accurately define the temperature distribution before the jets. This is difficult to predict for ‘real’ combustion systems which involve reverse flows without recourse to numerical models.

Computational fluid dynamics (CFD) is therefore the only practical tool available for analysis of complex cross-flow problems, and in the current work CFD is used to analyse the mixing of three rows of jets arranged in different patterns in a can combustor and to predict the resulting temperature profiles.

Emphasis has been placed on the use of simple computational models in this parametric analysis of jet distribution, using an existing can as the baseline design, see Figure 1. To open the use of computationally-based combustion modelling techniques to the broadest spectrum of users, models must be sufficiently simple to allow their solution with minimal computational overheads using standard personal computers. Coarse numerical grids of approximately 10,000 cells and a single step reaction mechanism were therefore utilised in this study.

The aim was to achieve a more uniform temperature profile, both radially and circumferentially. In a parametric study the number of jets were varied between six, twelve and, in the case of the dilution holes, twenty-four. The restriction to multiples of six was chosen to agree with the cyclic symmetry allowing the modelling of only a 60° sector. The basic design of the
combustor, the axial location of the holes and the percentage of air introduced at each stage remained identical.

The results obtained from these analyses were then used to design two cans with jet distributions which differ from the baseline. The original and the two modified designs were tested at atmospheric conditions burning propane and the experimental results were compared with the numerical predictions.

NUMERICAL MODEL

The can investigated is based on a research combustor by Lycoming used in previous studies (Felton et al., 1978, Boysan et al., 1982 and Wild et al., 1988). Air enters the combustor through a swirler and three rows of jets (primary, secondary and dilution jets) which are equally spaced around the circumference. The air flow split is: swirl: 9.7%, primary jets: 24.7%, secondary jets: 29.4% and dilution jets: 36.2%. The can has no film cooling slots. The propane fuel in this study is admitted through a fuel injector with 6 holes.

The CFD code FLUENT Version 3.03 was used to perform the computations. Cyclic symmetry allows the user to limit the computations to a 60° sector, which is modelled using a cylindrical polar co-ordinate system with a grid of 38 x 19 x 14 (x, y, Φ) cells, see Figure 2. As the grid is very coarse (~10,000 cells), grid independence is not expected, although doubling the grid in all directions did not result in significantly different predictions. The modelled hole area is equal to the geometric area multiplied by the discharge coefficient.

A one-step global reaction mechanism for propane is assumed. The chemical reaction is modelled using the Magnussen mixing controlled reaction model and an Arrhenius rate mechanism, the smaller of the two rates defining the reaction rate for each cell. The physical properties of the combustion reactants and products for each cell are assumed to be dependent on species concentration. The specific heat capacity is also a function of temperature.

The combustion walls are assumed to be adiabatic. The standard k-ε model of turbulence, the Power Law differencing scheme and the SIMPLE solution algorithm were used.

A solution is assumed to be converged if the normalised residuals are less than 10^{-3}, with the exception of the enthalpy residual which should be less than 10^{-6} (FLUENT, 1990). It was found that to evaluate a jet configuration quantitatively, an enthalpy residual of 10^{-5} was sufficient, e.g. about 500 iterations were necessary if a solution of a case with similar geometry was used as an initial guess. All numerical results presented are converged.

In the first phase of the work the boundary conditions for the parametric study were approximated using values from a study by Felton et al. (1978 previous), where kerosene was burned. These assumed boundary conditions proved to be unobtainable in the subsequent experimental study and a comparison between the different inlet conditions can be found in Table 1. It was decided to repeat the calculations for the cans investigated experimentally with matching boundary conditions to facilitate a direct comparison between the numerical prediction and experimental results without using a normalisation procedure.

Nevertheless, it is stressed that the decisions concerning the improved design were made from the first set of calculations.

NUMERICAL RESULTS

In the parametric study fifteen different geometries were investigated. In addition to varying the number of jets per row, the circumferential position of the jets of one row in relation to the jets of the following row was also changed.

The baseline can combustor design with six injection holes per row, has a temperature profile characterised by a cold central area throughout the combustor and circumferential temperature peaks downstream of the primary and secondary jets at 30°, as shown in Figures 4 and 5. These temperature peaks are the result of a recirculation zone caused by the jet momentum which entrains the hot air from near the rim and pushes colder air in the centre region out towards the rim.

The temperature in the core of the combustor can rise if the jet penetration is reduced by increasing the number of jets. It was found that an improvement in the uniformity of the circumferential temperature distribution was achieved if the number of dilution holes was equal to or larger than the number of secondary holes. The primary jets have no discernible influence on the temperature distribution at the exit plane, as they are masked by the secondary and dilution jet flows.

The two designs chosen for comparison with experiments, were selected because they provided a more uniform exit temperature distribution in comparison with the baseline design, the geometric outline of the three cans are shown in Figure 3. The first design, CAN1, has six primary, twelve secondary and twelve dilution holes, where the dilution holes are angularly staggered in relation to the secondary holes. This design still exhibits a cooler centre, but the temperature is almost constant until midway between the can centre and wall, then increases rapidly, forming two cyclic peaks behind the secondary and dilution jets, as seen in Figures 6 and 7.

CAN2 has twelve primary, six secondary and twenty-four dilution jets, which are all staggered in relation to each other. Increasing the number of primary jets to twelve lowers the maximum temperature by 11% at x = 135mm in comparison to the baseline design. The overall shape of the temperature profile is maintained as in both cases the secondary jets are crucial, Figure 8. The increase of the number of dilution jets to twenty-four concentrates the mixing near the rim of the can, the smaller distances between jets lead to a more homogeneous temperature distribution about the circumference. The result is a very flat temperature profile, both radially and circumferentially. Circumferentially only a small peak can be
FIGURE 1 SCHEMATIC DIAGRAM OF THE CAN COMBUSTOR (BASELINE DESIGN)

FIGURE 2 COMPUTATIONAL GRID

FIGURE 3 JET LOCATIONS a) BASELINE DESIGN, b) CAN1 AND c) CAN2
FIGURE 4 TEMPERATURE PROFILE AT X = 135 mm - BASELINE NUMERICAL RESULT

FIGURE 5 TEMPERATURE PROFILE AT X = 205 mm - BASELINE NUMERICAL RESULT

FIGURE 6 TEMPERATURE PROFILE AT X = 135 mm - CAN1 NUMERICAL RESULT

FIGURE 7 TEMPERATURE PROFILE AT X = 205 mm - CAN1 NUMERICAL RESULT

FIGURE 8 TEMPERATURE PROFILE AT X = 135 mm - CAN2 NUMERICAL RESULT

FIGURE 9 TEMPERATURE PROFILE AT X = 205 mm - CAN2 NUMERICAL RESULT
to judge the behaviour of the different jet patterns. Comparing the secondary and the dilution holes (x = 135 mm). For ease of comparison in the figures, all results are plotted with secondary compared at the exit plane (x = 205 mm) and a plane between numerical results.

Temperatures are presented as a function of radius and angle and were obtained by linear interpolation between adjacent grid points.

It was found that all experimental results exhibit a non-cyclic temperature peak near 270°, as seen in Figure 10. As this study addresses the cyclic variation, mean temperature profiles for a 60° sector were calculated and used for comparison with the numerical results.

EXPERIMENTAL TECHNIQUE
To limit the difference between the cans to the jet distribution, the baseline and the improved cans were manufactured at the Department workshop using the same way and one swirler and fuel nozzle were used. The cans were mounted inside a large plenum chamber, hence the jets entering the combustion chamber possess no axial velocity components.

Temperatures were measured using bare wire K-type and R-type thermocouples connected to a digital sampling system. The mean temperature for each point was calculated from 600 measurements taken in 20 seconds and corrected for radiation.

The thermocouple was mounted on a support which provided movement on three translational axes and it was aligned parallel to the axis of the can to minimise the flow perturbations. Measurements were taken on a regular grid at a number of axial planes, the horizontal and vertical grid spacing was 3.175 mm. Temperatures are presented as a function of radius and angle and were obtained by linear interpolation between adjacent grid points.

At the exit plane the main difference between the predicted profiles and the measurements lies again in the absence of significant cyclic variations in the experiment.

For the baseline design the expected rise of temperature from the centre of the can to the rim is validated, see Figure 13. In general the temperature is lower than predicted, about 200K at the centre and the maximum temperature at r/R = 0.8 is 1040K in comparison to the predicted 1160K.

RESULTS AND DISCUSSION
The predicted and experimental temperature profiles are compared at the exit plane (x = 205 mm) and a plane between the secondary and the dilution holes (x = 135 mm). For ease of comparison in the figures, all results are plotted with secondary holes in the middle of the 60° sector.

The initial numerical study was based on approximated boundary conditions, this approximation proved to be sufficient to judge the behaviour of the different jet patterns. Comparing the predictions of the calculation using the initial boundary conditions, Figure 11, with the calculations employing the revised boundary conditions, Figure 5, the main difference lies in the range of temperature predicted, this is to be expected as in the second case the air flow is about halved. With the boundary conditions matched to the experimental input data, the range of temperatures predicted (Figure 5) are close to the measured ones, Figure 13, although the predicted temperatures tend to be higher in general.

The experiments show that the temperature distributions of all cans at the plane between secondary and dilution holes (x = 135 mm) are characterised by a colder central area with a steady increase in temperature towards the combustor rim, Figures 12, 14 and 16. Cyclic temperature variations corresponding to the position of the jets can be discerned for CAN2, where the temperature rises by about 7% behind the jet at r/R = 0.7. In contrast numerical predictions for the three designs show significant cyclic variations at x = 135 mm. The predicted increase of temperature behind the secondary jets is greatest near the rim, where for r/R = 0.8 increases by about 38% for the baseline design, 35% for CAN2 and 15% for CAN1 are predicted. The predicted radial temperature range also varies between designs. Maximum temperatures of almost 1800K are predicted for the baseline design and CAN1, whilst for CAN2 a lower maximum temperature of 1600K was estimated. The experimental maximum radial temperatures are located at the non-cyclic hot spot where temperatures up to 1850 K were measured, the mean temperatures of a 60° sector are below 1600K in all cases.

At the exit plane the main difference between the predicted profiles and the measurements lies again in the absence of significant cyclic variations in the experiment.

For the baseline design the expected rise of temperature from the centre of the can to the rim is validated, see Figure 13. In general the temperature is lower than predicted, about 200K at the centre and the maximum temperature at r/R = 0.8 is 1040K in comparison to the predicted 1160K.
FIGURE 12 TEMPERATURE PROFILE AT X = 135 mm
- BASELINE EXPERIMENTAL MEAN

FIGURE 13 TEMPERATURE PROFILE AT X = 205 mm
- BASELINE EXPERIMENTAL MEAN

FIGURE 14 TEMPERATURE PROFILE AT X = 135 mm
- CAN1 EXPERIMENTAL MEAN

FIGURE 15 TEMPERATURE PROFILE AT X = 205 mm
- CAN1 EXPERIMENTAL MEAN

FIGURE 16 TEMPERATURE PROFILE AT X = 135 mm
- CAN2 EXPERIMENTAL MEAN

FIGURE 17 TEMPERATURE PROFILE AT X = 205 mm
- CAN2 EXPERIMENTAL MEAN
The predicted peak temperature for r/R = 0.8 is located angularly behind the jets, but the peak is located increasingly towards higher angles for lower r/R values. Since the displacement of the temperature peaks is increasing with distance from the swirler, it cannot be caused by the swirling flow, but could be the result of numerical errors. To check this possibility, additional calculations showed that increasing the number of cells used for the model alleviated the problem.

For CAN1 the experiments agree well with the prediction of a temperature variation of about 8% across the central area of the can up to r/R = 0.5, and then a steep rise in temperature to the rim, Figure 15. The experiments show that the maximum temperature reached at the rim is of the same magnitude as seen for the baseline design. The cyclic variations prove to be small in both the baseline design and CAN1, the only improvement of the exit profile achieved by CAN1 is therefore the more uniform temperature in the central area.

With CAN2 the predicted reduction of the mean radial temperature has been achieved as shown in Figure 17 (c.f. Figure 9). A slightly higher temperature than predicted is measured. As the cyclic temperature variations predicted were a result of the continuing influence of the secondary jet, which has already been shown not to exist in the expected magnitude at x = 135mm, no cyclic temperature variations were expected to be seen at the exit.

Additional measurements taken at axial planes closer to the secondary and dilution jets, e.g. at x = 122mm and x = 180mm, displayed temperature variations which confirmed the predicted development of temperature peaks behind the jets. The CFD model under-predicts the mixing behind the jets which flattens the temperature profile circumferentially. This underestimation of the mixing has been reported previously (Boersma 1993, Crocker and Smith 1993). On the other hand the radial temperature distributions have been predicted well and a more uniform temperature profile has been achieved with the aid of simple CFD models.

It was evident from the experiments that temperature hot spots are due to non-cyclic behaviour, which has previously been shown to be the result of uneven fuel or air distribution (Crocker et al. 1994). If those variations in the boundary conditions were known, this could be incorporated in the numerical model, but as flow symmetry will no longer be present, the numerical effort would increase accordingly.

CONCLUSION
Using a basic numerical model the influence of air jet pattern on the temperature profile of a can combustor was investigated and the comparison of three different designs with experiments in general validated the predictions.

As would be expected, only qualitative agreement is achieved if the fuel and air flow boundary conditions are not equal to the experimental conditions. To quantitatively compare temperatures, the inlet conditions of the model must coincide with the experiment.

A weakness of the numerical model is the underestimation of mixing behind jets which results in an over-prediction of the cyclic temperature variations immediately downstream of the jets.

The numerical model has demonstrated CFD's usefulness as an engineering design tool in predicting trends in the temperature distribution of a can combustor. The major advantage is the compactness of the model used which makes it feasible to run on a PC. A large number of variations can be processed effectively and possible improvements identified.

ACKNOWLEDGEMENTS
The authors wish to thank the Commission of the European Communities for funding this work under the JOULE programme. The authors also thank Dr. Ian Greaves for assistance during preparation of the paper.

REFERENCES