ABSTRACT

Numerical analyses of two existing gas turbine combustors gave predictions of idle power emissions. The calculated exit emissions of unburned hydrocarbons (UHC) and carbon monoxide (CO) are compared to engine test data. For the first combustor, the effects of varying fuel flow on the UHC and CO emissions were investigated while liner cooling flow changes were examined in the second combustor. A fully elliptic three-dimensional computational fluid dynamics code based on pressure correction techniques was employed to model the flow field inside the combustor. Fuel injection was handled using a Lagrangian liquid droplet spray model coupled to the gas phase equations. The combustion model consists of a two-step global reaction mechanism with reaction rates computed using a modified eddy-breakup technique. The numerical algorithm employs non-orthogonal curvilinear coordinates and the standard k-e turbulence model. The results for the first combustor agree well with the test measurements. The baseline result for the second combustor shows good agreement with test data. Predicted effects of cooling flow changes agree with trends from past experience of idle power emissions.

PRESENTED AT

The Society shall not be responsible for statements or opinions advanced in papers or discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. Discussion is printed only if the paper is published in an ASME Journal. Papers are available from ASME for 15 months after the meeting.

Printed in U.S.A. Copyright © 1993 by ASME

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>D (_{32})</td>
<td>Sauter Mean Diameter (SMD) of liquid fuel spray, (\mu m)</td>
</tr>
<tr>
<td>EI</td>
<td>Emission Index, g/kg fuel</td>
</tr>
<tr>
<td>FAR</td>
<td>Fuel to air mass flow rate ratio</td>
</tr>
<tr>
<td>R</td>
<td>Reaction rate, kg/m(^3)-s</td>
</tr>
<tr>
<td>T</td>
<td>Temperature, K</td>
</tr>
<tr>
<td>W</td>
<td>Mass flow, kg/s</td>
</tr>
<tr>
<td>k</td>
<td>Turbulent kinetic energy, m(^2)/s(^2)</td>
</tr>
<tr>
<td>m</td>
<td>Mass fraction</td>
</tr>
<tr>
<td>q</td>
<td>Drop size spread parameter</td>
</tr>
<tr>
<td>r</td>
<td>Specie mass ratio</td>
</tr>
</tbody>
</table>

Greek:

\(\Gamma()\) Gamma function
\(\epsilon\) Turbulence dissipation rate, m\(^2\)/s\(^3\)
\(\rho\) Density, kg/m\(^3\)

Subscripts:

arr Arrhenius
c Combustor
c\(_{0}\) Carbon monoxide
ebu Eddy-breakup
fu Fuel
in inlet
ox Oxygen
x Moles of carbon
y Moles of hydrogen

INTRODUCTION

Responding to the environmental legislation of the early 1970's, aircraft gas turbine engine manufacturers greatly reduced pollutant emission levels of unburned hydrocarbons (UHC) and carbon monoxide (CO) at ground taxi/idle power. In the mid 1980's global warming and ozone depletion became issues of concern. Reduction of emissions, especially of oxides of nitrogen (NO\(_x\)) at aircraft cruising altitudes, became a driving force in combustor design. Meeting both goals of low CO at idle power and low NO\(_x\) at cruise has proven to be quite a challenge. Reductions in UHC emissions at idle power have been accomplished primarily through improving the atomization ability of the fuel injectors, but the reduction of CO emissions involved setting a near stoichiometric equivalence ratio in the primary combustion zone, thus raising the flame temperature in order to expedite CO oxidation. These high temperatures unfortunately produce the undesired effect of accelerating the production of NO\(_x\). One possible solution to
this conflict is a dual staged combustor where one stage is
designed for idle operation and low CO emissions while the
other stage is used for higher power operation with low NOx
formation. If near-term modifications are to be made for
existing single stage combustors, some form of compromise
must be made between minimization of both CO and NOx.
Typically, the designer attempts to reduce NOx levels while
maintaining CO emissions at the current level of the baseline
combustor.

To achieve an optimum low emissions combustor design, a
good understanding of the flow field and fuel-air mixing
characteristics present within the combustor is essential. The
harsh environment inside an operating combustor makes
measurement of flow field characteristics and chemical species
concentrations a very imposing task. Use of standard
measuring probes is not a viable option due to their bulky and
obtrusive nature. State-of-the-art non-obtrusive laser diagnos-
tics require optical access which more often than not cannot be
accommodated in full scale combustor component test rigs.
Computational methods are one possibility for visualizing and
analyzing the physical phenomena inside a combustor.

Computational fluid dynamics analysis of turbulent, react-
ing flows has progressed in recent years to a point where a
three-dimensional, periodic section of a combustor can be
modeled with a certain degree of success (Burrus, 1989).
Predicting combustor exit temperature distributions and pro-
files has been the primary role of the existing computational
tools. This capability allows for the selection of a dilution air
injection arrangement that will provide an acceptable exit
temperature pattern. These predictions have typically involved
a simple combustion model using infinitely fast, equilibrium
chemistry with an assumed shaped PDF for scalar fluctuations
(Shyy et al., 1988). These models do not lend themselves to
calculation of chemical specie emissions. The rigorous
description of the system of chemical reactions present in a
combustor in conjunction with three-dimensional CFD is not
feasible due to current computer capability constraints. Some
form of reduced chemical mechanism is necessary to attempt
combustor emissions predictions.

In the present paper, both the GE/SNECMA CFM56 (Figure
1) and the GE CF34 (Figure 2) single annular combustors were
modeled at ground idle power conditions. The numerical
analysis was conducted using the proprietary CFD package
CONCERT developed jointly by GE Aircraft Engines and GE
Corporate Research and Development (Shyy et al.,1986). A
reduced mechanism two-reaction combustion model coupled
with a Lagrangian based liquid droplet model for fuel injection
was used to give predictions of UHC and CO emissions. The
simple two-reaction scheme used in this work was implement-
ed into CONCERT as a first attempt at modeling combustion
systems with dependence on chemical reaction rates where its
viability would be assessed before moving onto more compi-
lcated reduced mechanism schemes. The purpose of this work
was to determine the predictive capability of the model for low
power emissions, then apply this modeling capability to
investigate the impact of liner cooling flow modifications on
UHC and CO emissions for the CF34 combustor.

**FIGURE 1. SCHEMATIC DRAWING OF GE/SNEC-
MA CFM56 SINGLE ANNULAR TURBO-
FAN ENGINE COMBUSTOR.**

**FIGURE 2. SCHEMATIC DRAWING OF GE CF34
SINGLE ANNULAR TURBOFAN ENGINE
COMBUSTOR.**

**MATHEMATICAL MODEL DESCRIPTION**

The CONCERT CFD code is based on a fully elliptic,
three-dimensional, body-fitted control volume formulation
with pressure correction techniques. The governing equations
for the gas phase are those representing the conservation of
mass and momentum in the three coordinate directions.
Turbulence is modeled using the standard k-ε model (Lauder
and Spalding, 1974) along with wall function treatment for
near-wall regions. The combustion model employed for these
calculations uses a global two-step oxidation scheme (Mongia
et al., 1979) made up of the following reactions:

\[
\begin{align*}
C_xH_y + (x/2+y/4)O_2 + 3.76N_2 & \rightarrow xCO + \frac{y}{2}H_2O + \frac{x}{2+y}N_2 = 3.76N_2 \\
\end{align*}
\]

\[
\begin{align*}
xCO + (x/2)O_2 + 3.76N_2 & \rightarrow xCO_2 + \frac{x}{2}N_2 = 3.76N_2 \\
\end{align*}
\]

Solution of the conservation equations for unburned fuel
\((C_xH_y)\), CO and the mixture fraction (mass fraction of total
fuel) are necessary. Concentrations of other species are
obtained from linear functions of the amount of fuel consumed. Reaction rates of the fuel and CO are determined from a modified eddy-breakup model (Spalding, 1971; Magnussen and Hjertager, 1976) which uses the minimum of three rate expressions taken from Mongia et al. (1979) as

\[ R_{fu,ebu,1} = 3 \varphi \frac{m_{fu}}{r_1} \]  
\[ R_{fu,ebu,2} = 3 \varphi \frac{m_{ox}}{r_1} \]  
\[ R_{fu} = \min (R_{fu,ebu,1}, R_{fu,ebu,2}) \]  
\[ R_{co,ebu,1} = 6.0 \times 10^8 \varphi^2 \frac{m_{co}}{m_{ox}} e^{-12500/T} \]  
\[ R_{co,ebu,2} = 4 \varphi \frac{m_{co}}{r_2} \]  
\[ R_{co} = \min (R_{co,ebu,1}, R_{co,ebu,2}) \]

with all values expressed in S.I. units. The constant \( K_0 \) has a value of \( 3.3 \times 10^4 \) for Jet-A type fuel which is represented as \( C_{12}H_{22} \). \( r_1 \) and \( r_2 \) are the mass ratios of \( O_2 \) to fuel and CO to fuel, respectively. An equation is also solved for specific enthalpy through equating the total specific enthalpy to the sum of the species enthalpies. The density of the gas is updated as a function of pressure and temperature using the ideal gas law. The liquid droplet model used to simulate fuel injection (Tolpadi, 1992) computes the motion of fully atomized spherical droplets in a Lagrangian frame of reference. Droplet heating as well as vaporization are included with source terms for the gas phase equations being computed as the droplets evaporate. The droplet size distribution is given by the function developed by Rosin and Rammler (1933) and is given by the relation

\[ Q = 1 - \exp (-\Gamma (1-q)^{-4} (D/D_{32})^q) \]

where \( Q \) is fraction of total mass contained in drops of diameter less than \( D \) and \( q \) is the drop size spread parameter and its value is 2.5 (Lefebvre, 1989).

Using a coordinate transformation, the governing conservation equations are transformed from an arbitrary shaped physical domain to a rectangular parallelepiped. A staggered grid system as described by Patankar (1980) is employed where scalar variables are located at the center of the control volumes, while the velocity components are located at the control volume faces. After discretization of the equations, they are solved by a SIMPLE-like algorithm (Patankar, 1980) extended to the curvilinear coordinate system. The governing equations together with their discretization and the numerical algorithm have been described in detail elsewhere (Tolpadi, 1991; Braaten and Shyy, 1986) and will not be repeated here.

RESULTS AND DISCUSSION

The GE/SNECMA CFM56 turbofan engine combustor is the first of two combustors to be analyzed in this study. The annular combustor comprises 20 swirl cup/fuel injector ports equally spaced along the circumferential direction. Calculations were conducted within an 18-degree one-cup periodic sector of the combustor. This geometry was represented by a body-fitted mesh (Figure 3) of 57x25x33 grid points with accurate representation of liner cooling features and dilution holes. This grid resolution, while somewhat coarse, has been found from modeling application experience to be adequate for providing reasonably accurate predictions of combustor exit gas temperature average radial profile and harmonic pattern.

Three cases from an available test database representing overall fuel-air ratios of 0.008, 0.0109, and 0.0149 were calculated at ground idle power pressure and temperature conditions. Air flow amount and distribution were based on available data and are constant for the three cases. Inlet swirl cup discharge velocities were determined from calculated 2D/axisymmetric modeling results verified by available laser Doppler anemometry measurements. Since the liquid droplet model was not yet coupled to the gas phase flow solver at the time this calculation was performed, the Jet-A fuel was assumed to be prevaporized upon entering the combustor from the swirl cup. Examples of results for flow velocity and temperature are shown in a side plane view in Figures 4a and b. By integrating the mass flows of UHC and CO at the exit of the combustor model, values for emissions index (EI), defined as grams of pollutant per kilogram of fuel, can be computed.

A comparison between predicted and measured emission levels is presented in Figure 5. The predicted emissions levels not only follow the trend of the test data but also closely agree with absolute values, especially at the higher fuel-air ratios.

The GE CF34 turbofan engine combustor is also an annular type with 18 swirl cup/fuel injector ports. Again, a one-cup periodic sector of the combustor was represented by a mesh, this time having 73x25x25 grid points. A side plane cross section of the grid is shown in Figure 6. Notice the extra refinement employed along the top and bottom liner wall boundaries to enhance the calculation of the mixing of the injected slot liner cooling flow with the main stream combustor flow.

FIGURE 3. THREE-DIMENSIONAL VIEW OF COMPUTATIONAL GRID FOR A ONE CUP SECTOR OF AN ANNULAR COMBUSTOR.
Calculations for the CF34 combustor were performed at ground idle power conditions for three different air flow distributions involving variations in first panel cooling and second and third panel dilution flows. The baseline case used 3.5% of the total combustor air flow for the first panel outer liner cooling. For the Mod I case, the first panel outer liner cooling was increased to 4.2% of combustor air flow taking the additional air away from the second panel primary dilution flow. For the Mod II case, the first panel outer liner cooling was again increased to 4.2% of combustor air flow, but the additional air was instead taken from the third panel secondary dilution flow. The overall fuel-air ratio was constant for all three cases. For these combustor analysis cases, the liquid droplet model was employed for fuel injection. Data taken at the exit of the CF34 swirler using a Malvern drop size distribution analyzer dictated that an SMD of 11.1 μm be used for the calculation of the spray drop size distribution. Using this value of SMD along with the Rosin-Rammler distribution function led to droplets ranging in size from 3.1 to 27.9 microns. The droplets were assumed to enter the combustor with the same velocities as the swirler discharge air velocities as determined again through 2D/axisymmetric modeling calculations. The fuel was assumed to enter the combustor only in the liquid phase since the amount of prevaporization was unknown. Examples of results for flow velocity and temperature are shown in a side plane view in Figures 7a and b. A comparison of measured versus predicted combustor exit gas temperature profile is shown in Figure 8 as a first step in evaluation of the results. The measured data closely agrees with the predicted profile.

Test data of UHC and CO emissions were available for comparison only with the baseline case. Qualitative evaluation of the emissions results for the two modified cases were based upon past experience of measured trends in other aircraft gas...
turbine engine combustors. The measured and predicted emissions levels of UHC and CO are shown in Table 1. For the baseline case, the predicted value for UHC is in good agreement with the test data while the CO value is 11% lower than measured. An additional baseline calculation with the SMD increased to 30 μm was made to check the sensitivity of the results to the SMD input. The EIHC and EICO increased from 3.5 to 7.8 and from 38.0 to 67.3, respectively, indicating a strong sensitivity to SMD. Plots of droplet trajectories for both the 11.1 μm and 30 μm calculations (Figure 9) indicate a drastic difference in fuel droplet evaporation characteristics within the combustor. The evaporation delay for the larger drop sizes hinders the oxidation process for both the hydrocarbons and CO.

Past experience with two experimental low emissions engines designed in the mid to late 1970’s supports the predicted emissions results for the two modified cases. Both the Mod I and Mod II results show an increase in both UHC and CO with the increased first panel liner cooling. Testing of design changes made during development of the NASA Energy Efficient Engine (E3) combustor (Burrus et al., 1982) and the NASA Experimental Clean Combustor Program (ECCP) (Gleason et al., 1976) indicated the same trend of increased UHC and CO emissions with increased first panel liner cooling at ground idle conditions. Figures 10a and b display the stated trend in emissions for both the E3 and ECCP combustors. This trend has been attributed to additional quenching of chemical reaction near the liners caused by the increase in the cool air flow which inhibits overall fuel and CO burnout. The small difference in predicted emissions values between the Mod I and Mod II cases suggests that wall quenching may not account for all of the observed increased emissions. For the Mod I case, the slight enrichment of primary zone due to the decrease in primary dilution air may have

---

**Table 1. Measured and Predicted Emission Levels of UHC and CO for the CF34 Combustor.**

<table>
<thead>
<tr>
<th>CASE</th>
<th>EIHC</th>
<th>EICO</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASELINE: TEST DATA</td>
<td>3.5</td>
<td>42.5</td>
</tr>
<tr>
<td>BASELINE: CALCULATIONS</td>
<td>3.6</td>
<td>38.0</td>
</tr>
<tr>
<td>MOD I: CALCULATIONS</td>
<td>4.3</td>
<td>41.5</td>
</tr>
<tr>
<td>MOD II: CALCULATIONS</td>
<td>4.0</td>
<td>30.0</td>
</tr>
</tbody>
</table>
caused the slight increased emissions levels over the Mod II case where the the secondary dilution was decreased.

CONCLUSIONS

Two aircraft gas turbine combustors have been modeled for idle power emissions using a two-step combustion scheme coupled with a liquid droplet spray model within the three-dimensional, body-fitted CONCERT CFD code. The results of these analyses suggest that the modeling technique employed has practical application for prediction of UHC and CO levels in $10^{-1}$ to $100$ range for EI. The calculations will probably yield the proper trends, if not reliable absolute emissions values. Additional study of this model will be performed against other data bases to further assess its usefulness for making predictions of UHC and CO emissions in typical aircraft gas turbine engine combustors.

ACKNOWLEDGMENTS

The author wishes to thank GE Aircraft Engines for granting permission to publish this work. Also, the author thanks J.D. Monty of GE Aircraft Engines for furnishing necessary information about the CF34 combustor for modeling purposes.

REFERENCES


Shyy, W., Correa, S.M. and Braaten, M.E., 1986, Calculation of Turbulent Reactive Flows (R.M.C. So et al., Eds.), p.141, ASME.


