ABSTRACT

Time-accurate CFD analysis is used to model combustion instability in a premixed axisymmetric combustor typical of industrial gas turbine engines. The experiment of Richards and Janus (1997) is modeled; the hardware consists of a fuel injector similar to industrial premix fuel nozzles, a water-cooled can combustor, an uncooled refractory plug that reduces flow area, and a long exhaust duct. The CFD calculation domain extends from the air swirler within the fuel nozzle to the exhaust duct exit. Two cases are modeled using 2D time-accurate axisymmetric CFD analysis: a low nozzle air velocity (u=30 m/s) case that exhibits combustion instability and a high nozzle air velocity (u=60 m/s) case that does not. The CFD analysis agrees well with the experimental measurements, including peak-to-peak pressure variation and instability frequency for the unstable case. For the unstable case, the airflow through the swirler actually flows upstream part of the time, and hot combustion products are forced into the premix annulus. The potential of using a time-accurate CFD approach for modeling combustion instability in complex 3D combustors is discussed.

INTRODUCTION

Combustion instability is a serious challenge for combustor designers of modern gas turbine engines. Requirements of low emissions, high combustion efficiency, and/or low pattern factor have necessitated some amount of fuel-air premixing before combustion. Fuel is injected upstream of the flame zone, producing a time lag between airflow perturbation at the fuel injector and the resulting flamefront disturbance. The airflow perturbation thus produces a periodic heat release that, if in-phase with resonant acoustic frequencies, can drive the combustor into instability. If the heat release is not in-phase with acoustic resonant frequencies, the combustion remains stable.

Combustion instability is not new. It is a common challenge in rocket combustion chambers, ramjets, gas turbine afterburners, and industrial combustors (e.g. Rayleigh (1878), Putnam (1971), Candel (1992), Sivasegaram and Whitelaw (1991), Schadow and Gutmark (1991), Keller (1995), and Yang and Anderson (1995)). When an industrial combustor has excessive dynamics, there are many practical consequences, such as heavy wear at the liner mounting points, fuel nozzle interfaces and cross-fire tubes; liner cracking from high cycle fatigue, and flashback in premix tubes. Various mechanisms can drive the instability, including hydrodynamic instabilities, vortex roll-up and burning, vortex interactions, flame front pulsations, flame acceleration, extinctions and ignitions. In industrial gas turbine combustors, the typical driving mechanism appears to be heat release variations caused by airflow and/or fuel flow fluctuations.

To try and understand the mechanisms of combustion instability in premixed gas turbine engines, Richards and Janus (1997) have performed experimental tests of a combustor typical of industrial gas turbine combustors. The fuel nozzle employed was similar to premixed (natural gas) gas turbine fuel nozzles, including 45 degree swirl vanes and multi-orifice fuel spokes. The orifices were sized for near sonic velocities to eliminate fuel flow variations as the driving mechanism for combustion instability. Figure 1 (taken from Richards and Janus, 1997) presents the overall features of the experimental combustor. Experimental tests were performed by Richards and Janus (1997) at the following conditions:

- Pressure = 5.75 and 10 atmospheres
- Air Temperature = 533 and 589 K
- Nozzle velocity = 30, 40, 50, and 60 m/s
- Equivalence ratio = 0.59, 0.63, 0.67, 0.71, and 0.77

Over this range of parameters, both unstable combustion and
stable combustion were recorded. Generally speaking, a 5% peak-to-peak pressure instability was recorded at the low fuel nozzle velocity, and stable combustion was recorded at the high fuel nozzle velocity. Overall NOx and COx emissions were measured far downstream, and the levels were typical of premixed, low emission combustors.

Previous Modeling Activities

Modeling of combustion instability has been performed in the past. One-dimensional time lag models [e.g. Crocco (1965)], have been used in rocket engine instability predictions, based on the argument fuel injected into the combustor would burn after a certain time delay. Marble and Candel (1979) first indicated that the nonsteady response of the flamefront was a possible source of low frequency oscillations in afterburners, and they developed a one-dimensional model describing the motion of the flame sheet and the dynamics of fresh and burnt gases. Culick and coworkers [Culick (1990), Yang et al. (1989)] have developed a unified model that describes the nonlinear acoustics of combustion instability. They solve a set of coupled ordinary differential equations representing the dynamics of the system. Numerical calculations of the unsteady combustion flow dynamics were developed by Baum et al. (1984) using a one-dimensional model. Menon and Jou (1991) have used Large Eddy Simulation of the flow with a model of the flame front dynamics. Liu and McGuirk (1995) showed that a 1D CFD method based on a pressure-correction formulation could accurately capture reheat-buzz type combustion induced oscillations.

For gas turbine combustion, Mehta et al. (1990) describes a one-dimensional model that combines linear acoustics and a heat release model. More recently, Bohn and Deuker (1993) and Bohn, Deutsch and Kruger (1996) have developed a model that utilizes 2D steady-state CFD analysis to determine resonant acoustics and a time-accurate step-change CFD analysis to determine heat release rates as a function of time.

In this paper, time-accurate CFD analysis is used to model combustion instability. Most instability models simplify the system as one-dimensional, and the heat release models are very simplistic. Although such models have their place, the flamefront and flowfields in practical combustors are much more complicated. As a first step to modeling complex 3D combustors, a 2D axisymmetric case was selected for analysis. This paper describes the analysis that was performed, the comparison with experimental data, and the potential of extending the analysis to complex 3D combustors. CFD-ACE, a state-of-the-art CFD code, was used to perform the calculations.

STEADY-STATE CALCULATIONS

To provide a starting point for the time-accurate cases, steady-state CFD computations were first performed. The test conditions were:

- Pressure = 5 atmospheres
- Inlet Air Temperature = 533 K
- Equivalence Ratio = 0.77
- Fuel Nozzle Velocity = 30 m/s and 60 m/s

The calculation domain started at the discharge plane of the air swirler. The discharge plane of the swirler was located 0.07 m (2.75 inches) upstream of the combustor inlet plane. The dimensions of the premix passage, combustor, neckdown plug, and exhaust duct are shown in Figure 2. A total pressure and swirl vane flow direction (45 degrees) were specified at the swirler discharge plane, and a fixed pressure boundary was specified at the exhaust duct exit. For case 1, the inlet total pressure was 0.05 atmospheres (relative to an exhaust exit pressure of 5 atmospheres), and for case 2, the inlet total pressure was 0.20 atmospheres. These total pressures produced inlet axial velocities of 30 and 60 m/s, respectively. Fuel was injected through four rings evenly distributed in the premix annulus. The rings were located 0.044 m (1.75 inches) upstream of the combustor inlet plane. The water-cooled walls of the combustor section and the exhaust duct were assumed to be cold (300 K) while the ceramic neckdown plug wall was assumed to be adiabatic.

The Favre-averaged Navier Stokes equations were solved. This involved solving for u, v, and w momentum; continuity, turbulence (k and E), energy (h), mixture fraction (f1) and progress variable (f2). The following numerics and physical models were used:

1. Second order upwind for spatial differencing on all variables (u, v, w, pp, k, E, h, f1, f2);
2. Renormalization Group (RNG) turbulence model [Yakhot et al. (1992)] and standard wall functions; and
3. One-step reaction rate equation (methane reacting with air to equilibrium products). The Westbrook and Dryer (1981) reaction rates were implemented.

A four block grid (see Figure 3) was generated using the grid generation code CFD-GEOM. Block 1 was the premix passage (15x28), block 2 was the combustion chamber (40x52), block 3 was the neckdown plug (40x30), and block 4 was the exhaust duct (50x48). The many-to-one feature was utilized between block 1 and block 2. 28 grid cells were used in the radial direction in the premix passage, and these 28 cells were converted to seven cells in the combustion chamber. This allowed the fuel rings to be modeled in the premix passage, but didn't burden the calculation with extra (unneeded) cells in block 2. The many-to-one feature will be even more useful when modeling 3D geometries that require detailed gridding in the fuel nozzle to the flamefront.

Temperature isotherms of the premix passage and combustion chamber for cases 1 and 2 are shown in Figure 4. The flame has the appearance of a premixed flame; the flame is slightly stretched for the higher velocity (u = 60 m/s) case. Otherwise, little difference is seen between these two cases. Figure 5 presents the equivalence ratio for case 1, showing that the fuel and air are very well premixed at the inlet plane to the combustor. Figure 6 shows the time lag of three streamlines from the swirler inlet to the flamefront. The time lag from swirller exit to flamefront varies between 3.0 msec to 4.0 msec, depending on the streamline. The fallacy of assuming a single time lag (a common assumption) is evident.
TIME-ACCURATE SIMULATIONS

Starting from the steady-state solutions, time-accurate CFD analyses were performed for both cases. The second-order Crank-Nicholson temporal scheme was used, and a time step of 4E-06 sec was used. The time step was selected based on the acoustic velocity in the combustion chamber and the requirement that the acoustic wave travel only one cell per time step. In reality, for the low instability frequencies in this combustor, the time step probably could have been increased fourfold without decreasing the accuracy of the predictions. For case 1, the initial numerical disturbance caused by starting from a steady-state solution continues to grow into a predicted combustion instability mode. Various parameters at three monitor points were repeatedly recorded and saved in files; the time histories of the parameters are shown in Figure 7. All of the parameters reach a periodic motion after about six cycles. One of the more interesting observations is the mass-averaged swirl velocity in the premix passage at location 2 actually increases as the instability progresses. This is caused by the higher swirl being introduced at the time of highest mass flow.

Figure 8 shows the history of the pressure in the combustion chamber at mid-length; the pressure grows into a periodic peak-to-peak variation of about 5 percent. The missing data in Figure 8 were lost during the computations. Figure 8 also presents the experimental measurements of Richards and Janus (1997). The predicted pressure compares well with the measurements, especially the peak-to-peak pressure peaks. The predicted frequency is 294 hertz, while the measured frequency is 220 hertz. This difference between predicted and measured frequency is discussed below.

Time snapshots of the predicted combustion instability are shown in Figure 9. A transient vortex is formed that is close to stoichiometric fuel-air ratio; when this vortex burns, a higher heat release is produced. As the maximum pressure node arrives at the swirler discharge, the airflow through the swirler reduces until the flow actually wants to travel upstream through the swirler. In the CFD analysis, the axial velocity is set to zero until the static pressure reduces below the specified inlet total pressure. This is shown by a time history plot of the axial velocity at the swirler inlet plane (see Figure 7). Although the flow is restrained from flowing out the swirler inlet plane, hot gases are still predicted to enter the premix passage (e.g. see snapshots 2-5 in Figure 9).

The longitudinal resonant frequency for the simulation is determined by the temperature field and total axial length of the calculation domain (from swirler discharge plane to exhaust exit plane). This frequency turns out to be 294 hertz. In the experiment, the acoustic frequency was probably determined by the length from the inlet (choked) air plenum feeding the fuel nozzle to the exhaust exit plane; also, quench water was injected in the exhaust duct that reduced gas temperature. Since the total acoustic length and injection of quench water were not modeled, it was probably fortuitous that the predicted acoustic resonant frequency was about the same as the experiment (294 hertz for the computations versus 220 hertz for the experiment). However, once a resonant frequency was established in the computations, the prediction of combustion instability was correctly calculated.

For the high velocity case, the time-accurate CFD analysis predicted stable combustion, just like the experiment. Figure 10 presents both numerical and experimental pressure histories at the combustor section mid-length point. The predicted frequency is 453 hertz, corresponding to the time delay from the fuel injection location to the flamefront. This frequency is obviously out-of-phase with the acoustic resonant frequency; hence stable combustion is predicted.

POTENTIAL OF COMBUSTION INSTABILITY ANALYSIS FOR COMPLEX 3D COMBUSTOR GEOMETRIES

In this 2D axisymmetric CFD analysis, the capability of modeling combustion instability using a time-accurate solution of the Favre-averaged Navier-Stokes equations has been demonstrated. In particular, the need to accurately capture resonant acoustic frequencies and the time lag of fuel injection to the flamefront has been shown. This approach can be extended to more complex 3D gas turbine combustor geometries if 1) the acoustic resonant frequency can be predicted and 2) some improvement (about tenfold over a single SGI R8000 workstation) in computational speed is realized.

The usual instability mode in premixed (or partially premixed) gas turbine combustors is longitudinal instability. The length associated with the resonant frequency extends from the compressor discharge to the turbine inlet. It is impractical to perform time-accurate 3D CFD analysis on a calculation domain that extends from the compressor discharge to the turbine inlet, especially for aircraft or aeroderivative gas turbine combustors. However, it is rather easy to perform an acoustic wave analysis on the temperature field (determined from steady-state CFD analysis in the combustor and compressor discharge temperature elsewhere). Once the resonant acoustic frequency is predicted, time-accurate CFD analysis can be used on just the combustor. The combustor exit pressure can be oscillated at the resonant acoustic frequency, and the combustor response can be determined from the CFD analysis. The total pressure (and flow direction) boundary conditions would be appropriate at the swirler(s), like that used in the 2D CFD analysis described earlier. If the pressure oscillation grows in time, the combustor is unstable at the specified conditions. This approach can also be adapted to liquid spray combustion by modifying the atomization based on the relative velocity between air and fuel.

The CPU time for the CFD combustion instability analysis discussed earlier was approximately two days for a 6,000 cell calculation. If a many-to-one grid is employed in 3D, it is feasible that only 100,000 cells would be needed in 3D. This is because only the distance from the swirler(s) to the flamefront needs to be accurately resolved, and the rest of the combustor can be rather coarse. Hence, only a tenfold improvement in computer speed is needed. As parallel computing is becoming more popular, such an improvement in computer speed is certainly possible in the next few years. Further improvement in computer workstation speeds is also expected. The time-step could also be reduced for low frequency instability calculations.

Thus, the time is right for exploring the approach discussed in this paper to predict combustion instability in complex 3D combustor geometries using time-accurate CFD analysis.
CONCLUSIONS

Time-accurate 2D CFD analysis has been shown to predict combustion instability in a premixed axisymmetric combustor typical of gas-fired gas turbine combustors. The experiment of Richards and Janus (1997) was modeled. Relevant conclusions of this study are:

1. Use of total pressure and flow direction boundary conditions at the swirler discharge plane are adequate for capturing the variation of airflow versus time;
2. The driving mechanism for the combustion instability experiment of Richards and Janus (1997) (i.e. the variation of heat release caused by airflow variations) was captured numerically. The time lag associated with the time from the fuel injection location to the flamefront was enough in-phase with the acoustic resonant frequency that combustion instability was predicted. The flamefront is not one-dimensional and the time lag from the fuel injector to the flamefront is not uniform.
3. For the case of combustion instability (air velocity through fuel nozzle of 30 m/s), the numerical simulation predicted the correct level of peak-to-peak pressure variation within the combustor. The predicted resonating frequency was slightly in error (294 hertz for the computations versus 220 for the experiment). This difference could be attributed to problem specification errors.
4. When the air velocity through the fuel nozzle was doubled (i.e from 30 m/s to 60 m/s), the time lag between the swirler to the flamefront became out-of-phase with the acoustic resonant frequency. The simulation thus predicted stable combustion, as observed in the experiment.
5. There is good potential that the time-accurate CFD methodology developed in this study can be used to predict combustion instability in complex 3D gas turbine combustors.

ACKNOWLEDGEMENTS

The authors express their appreciation to George Richards, DoE METSI, for providing the experimental data and for his helpful discussions. The authors also appreciate the helpful discussions of modeling combustion instability with D. Scott Crocker, Peter A. Liever, and Maciej Z. Pindera of CFD Research Corporation. The help of Mark J. Rist of CFDRC in implementing the total pressure boundary condition was most appreciated. We are also thankful for the effort of Marni S. Kent in accurately preparing the typescript.

REFERENCES

Yang, V. and Anderson, W., eds., 1995, Liquid Rocket Combustion Instability, Published by the American Institute of Aeronautics and Astronautics, Cambridge, MA.
Figure 1. Features of Richards and Janus (1997) Experimental Rig

Figure 2. Dimensions of Calculation Domain (all dimensions in meters)
Figure 3. Many-to-one Grid Used in the Calculations
Figure 4. Steady-State Calculations; Little Difference in Temperature Contours is Seen Between Cases
Figure 5. Equivalence Ratio Contours Show the Fuel and Air are Well Mixed Leaving the Premix Passage for Steady-State Cases

Figure 6. Time Lag from Swirler to Flamefront for Three Streamlines; Each Streamline was stopped at 4.0 msec
Figure 7. Time Histories of Various Parameters for Unstable Case
(See Figure 3 for locations)
Richard's Experiment

CFD-ACE Modeling

Figure 8. Time History of Wall Pressure at Combustor Mid-Length for Unstable Case (Case I)
Figure 9. Time Snapshots of Temperature Contours for Unstable Case (Case 1)
Figure 10. Time History of Wall Pressure at Combustor Mid-Length for Stable Case (Case 2)