NUMERICAL MODELING OF GAS TURBINE COMBUSTOR INTEGRATED WITH DIFFUSER

K. Su and C.Q. Zhou
Purdue University Calumet, Department of Engineering
Hammond, IN 46323

Keywords: CFD, Combustion Modeling, Gas Turbine Engine, Diffuser, KIVA

ABSTRACT
Numerical study of combustor-diffuser flow interactions was conducted using KIVA-3V code. The simulation was based on the solution of Navier-Stokes with phenomenological models of turbulence, sprays, and chemical reactions. A sector of a typical annular combustor, including diffuser, outer and inner annuli, swirler, and liner, was employed. The swirler was simplified in terms of the conservation of mass and momentum. Operating conditions were chosen with a range of \( P_3 \) from 0.1 to 1.2 MPa, \( T_3 \) from 450 to 730 K and FAR from 0.007 to 0.024. Static pressure recovery coefficients along the inner and outer walls of the pre-diffuser and combustor casing were obtained from the numerical solution and agreed well with published measured data. Combustion efficiency and overall temperature distribution factor (OTDF) were predicted in a reasonable agreement with those from semi-empirical correlation. Effects of non-uniform profiles of velocities at the inlet of diffuser on combustion performance were investigated. Flow fields and temperature distributions in axial and circumferential directions were analyzed. It is concluded that the non-uniform velocity at the diffuser inlet has an influence on temperature distributions. The transient behavior of flows in compressor-combustor system will be investigated based on this study.

NOMENCLATURE

- \( L_s \): eddy characteristic size
- \( m_d \): single droplet mass
- \( OTDF \): overall temperature distribution factor
- \( P_3 \): combustor inlet pressure
- \( R \): universal gas constant
- \( r_d \): droplet radius
- \( Re_d \): droplet Reynolds number
- \( S \): source term
- \( SMD \): Sauter mean diameter
- \( T \): temperature
- \( T_i \): combustor inlet temperature
- \( T_e \): combustor exit temperature
- \( t_e \): eddy lifetime
- \( T_{\text{max}} \): combustor maximum exit temperature
- \( t_i \): droplet transit time
- \( U \): gas phase velocity
- \( U_3 \): average inlet velocity
- \( U_d \): droplet velocity
- \( \alpha_n \): log normal rule distribution constant
- \( \phi \): variables
- \( \eta \): combustion efficiency
- \( \Gamma \): effective diffusivity
- \( \rho_d \): droplet density
- \( \sigma \): non-uniform factor of velocity
- \( \tau_r \): droplet relaxation time

INTRODUCTION
The gas turbine engine combustor increases enthalpy of the working fluid by combustion and produces temperature distributions acceptable to the turbine by the subsequent dilution of resulting products with additional air. Some operating conditions, such as...
ignition, acceleration, idle, etc., have transient behavior with non-uniform distribution of quantities on transverse sections. During these procedures, physical processes in individual components of the gas turbine are strongly coupled. Interactions between connected components, such as diffuser and liner are important to gas turbine performance. A CFD calculation for a complete annular combustor - from compressor exit to turbine inlet - is needed for investigation of the interaction between the diffuser and combustion flows.

In recent years, numerical methods have been extensively developed for the simulation of gas turbine combustor flows. Most of such numerical studies solve the time-averaged Navier-Stokes equations combined with turbulence, chemical reaction and spray dynamics models. For a long time, the diffuser and liner were simulated separately due to limitations in computers capacity (Correa and Shyy, 1987; Karki et al., 1992; McGuirk and Palma, 1993; Carrotte et al., 1994; Tolpadi, 1995; Correa et al., 1996; Honami et al., 1996; Tolpadi et al., 1997; Agrawal et al., 1998; Su and Zhou, 1999). Some predictions have shown reasonable agreements with measurements and have been applied in industry (Karki et al., 1992; Tolpadi, 1995; Tolpadi et al., 1995; Anand, et al., 1999). As the accuracy of physical models and the capacity of computers increase, efforts on numerical simulation of the entire combustor, including both diffuser and liner, are being made. Scott (1998) employed a mesh of 370,000 cells for numerical solution of combustor flows from the exit of compressor to the turbine inlet. In his work, swirler passages were modeled with approximately 35,000 cells and swirler vanes could be added with additional 100,000 cells. The execution time was about 48 hours on a 500 MHz DEC Alpha 500. For most researchers, this computation is a huge work and not affordable. Some simplifications and new technologies such as parallel processing are should be accepted.

This paper presents numerical simulation of combustor-diffuser flow interactions using the KIVA-3V code, which solves the Navier-Stokes equations with and phenomenological models of turbulence, sprays, and chemical reactions. Turbulent dispersion, collision and breakup of fuel droplets were considered in spray models. Combustion processes distinguished by slow reactions that proceed kinetically, and fast reactions that are assumed to be in equilibrium. The fuel was Jet-A. Its chemical formula is C\(_{12}\)H\(_{25}\). The combustion is divided into two classes: those that proceed kinetically (Reactions a to d) and those that are assumed to be in equilibrium (Reactions e to j). Assuming a simplified single-step oxidation model, the four kinetic reactions are

\[
\begin{align*}
4C\text{H}_\text{12}&+7O\text{2} \rightarrow 48C\text{O}_\text{2} + 46H\text{2}O \\
O\text{2} + 2N\text{2} & \rightarrow 2O\text{2} + 2N\text{2}O \\
2O\text{2} + N\text{2} & \rightarrow 2O + 2N\text{2}O \\
N\text{2} + 2OH & \rightarrow 2H + 2N\text{2}O
\end{align*}
\]

and the six equilibrium equations are

\[
\begin{align*}
H\text{2} & \leftrightarrow 2H \\
O\text{2} & \leftrightarrow 2O \\
N\text{2} & \leftrightarrow 2N \\
O\text{2} + H\text{2} & \leftrightarrow 2OH \\
O\text{2} + 2H\text{2}O & \leftrightarrow 4OH \\
O\text{2} + 2CO & \leftrightarrow 2CO\text{2}
\end{align*}
\]

Chemical reaction rate expressions are evaluated by a partially implicit procedure. The reaction rate constants are in the Arrhenius form as shown in Eq. 2.

\[
k = Ae^{-\frac{E}{RT}}
\]
where $E$ is the activation energy, and $A$ is the constant. With the reactions rates determined for the above ten reactions, the chemical source terms in the species equations were obtained. The mixing-controlled turbulent combustion model based on the eddy-dissipation was used for the turbulence combustion. Details of the combustion model and the related constants can be found in the literature (Amsden et al., 1989; Amsden, 1993, 1997).

**Liquid Phase Equations**

The Lagrangian method was used in the liquid phase modeling. The fuel was assumed to inject into the combustor as a fully atomized spray, which consists of spherical droplets. Liquid sprays are represented by a discrete-particle technique, in which each computational particle represents a number of droplets of identical size, velocity, and temperature. In the KIVA code droplet properties are determined by using the Monte Carlo sampling method.

The log-normal rule was accepted to describe the droplet size distribution at injection. It is given by

\[ \frac{dm}{dy} = \alpha_s \frac{1}{\sqrt{2\pi}} \exp\left(-\frac{y^2}{2}\right) \]

where $y = \ln(\frac{d_d}{d_m})$, $d_d$ is the droplet diameter and $d_m$ is the average droplet middle diameter, and $\alpha_s$ is the log normal rule distribution constant. In this study, $\alpha_s = 1.5$ was used.

The equation of motion of a spherical droplet is given by the following equation.

\[ \frac{d\hat{U}_d}{dt} = \frac{\hat{U} - \hat{U}_d}{\tau_d} + \frac{\hat{F}_d}{m_d} \]

where $\hat{U}_d$ is the droplet velocity, $m_d$ is the single droplet mass, $\tau$ is the dynamic relaxation time of the droplet, and $\hat{F}_d$ is the body force, which is the gravitational force in this study. The droplet relaxation time is defined as

\[ \tau_d = \frac{3}{8} \frac{\rho_g}{\rho_d} C_D \left| \hat{U} - \hat{U}_d \right| \]

where $r_d$ is the droplet radius, $\rho_g$ and $\rho_d$ are the density of gas and droplets, respectively, $C_D$ is the viscous drag coefficient, which is described as

\[ C_D = \begin{cases} \frac{24}{\text{Re}_d} \left( 1 + \frac{1}{6} \text{Re}_d^{\frac{2}{3}} \right) & \text{if } \text{Re}_d \leq 1000 \\
0.424 & \text{if } \text{Re}_d > 1000 \end{cases} \]

where $\text{Re}_d$ is the Reynolds number based on the droplet diameter and the relative velocity between the gas and droplet. The physical properties of the gas phase are calculated based on the temperature averaged between droplets and the surrounding gas.

The changes of mass, momentum and energy of droplets from conservation equations are added into the source terms of the governing equations. The momentum exchange is treated by implicit coupling procedures to avoid the prohibitively small time steps that would otherwise be necessary. The accurate calculation of mass and energy exchange is ensured by automatic reductions in the time step when the exchange rates become large.

Turbulence effects on the droplet motion are accounted for in the following way. When the time step exceeds the turbulence correlation time, turbulent changes in droplet position and velocity are chosen randomly from analytically derived probability distributions for these changes. The magnitude of the fluctuating velocity component is proportional to $\sqrt{2k/3}$ by assuming that the turbulence is isotropic. The interaction time between the droplet and the eddy is taken as the minimum of the eddy lifetime and the transit time required for the droplet to cross the eddy

\[ t_e = \min(t_e, t_k) \]

where $t_e$ is the eddy lifetime and $t_k$ is the droplet transit time. The eddy lifetime $t_e = L_e / \sqrt{2k/3}$, where $L_e$ is the eddy characteristic size and $k$ the turbulent kinetic energy. The droplet transit time is calculated by $t_k = -\tau, \ln(1.0 - L_e / \tau, |\vec{u} - \vec{u}_d|)$ (Zhou and Yao, 1992).

**NUMERICAL SCHEME**

The gas-phase solution procedure is based on a finite volume method called the ALE (arbitrary Lagrangian-Eulerian) method. The basic equations are differenced in integral form with the volume of a typical cell used as the control volume and with divergence terms transformed to surface integrals. Spatial differences are formed on a finite-difference mesh that subdivides the computational region into a number of small cells that are hexahedrons. The transient solution is marched out in a sequence of time steps. The block-structured boundary-fitted coordinate mesh is employed for the complex geometry of gas turbine combustor. The governing equations together with their discretization and the numerical algorithm have been described in detail in references (Amsden et al., 1989; Amsden, 1993, 1997).

A model of a typical combustor was analyzed in this study. The combustor is annular with 16 domes equally spaced along the circumferential direction. A single dome sector of 22.5° span of the combustor, which includes a swirler and a fuel nozzle (at the inlet of liner) and a set of primary and secondary holes (on the outer and inner walls of liner), was simulated with periodic boundary conditions on two sides. The swirler was simplified according to the conservation of mass and momentum in the swirler. It means the flow leaves the swirler with the same mass flow rate and momentum as it enters the swirler, but with different flow pattern, i.e., swirling.

A grid of 80,000 cells for the entire combustor including the diffusor was used, as shown in Fig.1. For sensitivity study of grids, three cases with different grids for the liner were run. Because of the complicated flow conditions in the liner, the grid for the liner has been changed to validate the sensitivity. The baseline grid has 24×24×42 cells in radial, circumferential and axial directions for the liner. The other two grids have 24×24×84 and 36×24×42 for the liner respectively. Velocity profiles at the exit of the combustor for the three cases were compared. There is no obvious difference among them. It then is decided that the baseline grid is sufficient for the simulation of combustor flows.
In this study, a steady state flow is considered. Since the equations were solved using the time-marching scheme in the KIVA code, the steady state solution was obtained after many steps of marching. Calculations were performed at medium and high power conditions of operation. A typical solution with 4000 steps requires about 60 hours on a SGI workstation.

RESULTS AND DISCUSSIONS

To validate the KIVA code, computations were conducted over a wide range of conditions (from the idle to the maximum conditions with $P_3$ from 0.1 to 1.2 MPa, $T_3$ from 400 to 730 K and FAR from 0.015 to 0.024, correspondingly). The parametric studies were based on the medium condition ($P_3 = 8$ MPa, $T_3 = 600$ K and FAR=0.02) which is the most common operating condition. Jet A fuel was used in computation. The log-normal distribution rule was assumed for fuel spray distributions. Fuel droplets were injected with SMD of 50 $\mu$m, initial velocity of 50 m/s, injection angle of 90°, and distribution constant of 1.5.

Combustion Efficiency and OTDF

The gas turbine combustion performance, including the combustion efficiency and the overall temperature distribution factor (OTDF), was analyzed from the numerical results of combustor flows. Combustion efficiency is defined as

$$\eta_c = \frac{\text{heat released in combustion}}{\text{heat available in fuel}} \quad (8)$$

The overall temperature distribution factor is defined as

$$OTDF = \frac{T_{\text{max}} - T_3}{T_4 - T_3} \quad (9)$$

where $T_{\text{max}}$ is the maximum temperature at the exit of combustor, $T_3$ is the mean inlet air temperature, and $T_4$ is the mean exit temperature. Higher combustion efficiency and more uniform exit temperature distribution generally are the most important requirements for combustor performance.

The semi-empirical correlations (Lefebvre, 1984, 1985) were used to validate the CFD code. A wide range of operating conditions from the idle to the maximum was chosen for the validation. Figs. 2 and 3 show the comparison between the results obtained from the numerical computations and the semi-empirical formula. It is seen that the data are reasonably correlated. In general, the agreement of the predictions from two methods is within 2% for combustion efficiency and 0.02 for exit temperature distribution, which indicates a satisfactory accuracy for the code to predict the combustion performance. This confirms the capability of KIVA-3V code in the gas turbine combustion simulation.
primary and dilution penetrations, and flow accelerations in flow channels. Airflow from compressor first enters the prediffuser with a high velocity. In the prediffuser, the flow is decelerated and the reduction in velocity head is converted to a rise in static pressure. After being discharged from the prediffuser, air flow is split into three branches: two streams that feed the outer and inner combustor annuli, i.e., the dump, and a third that supplies air to the combustor dome. As a result of the expansion in the dump region, flow velocity is further reduced and static pressure continues to rise. Two recirculation zones form at the corners in this region (Fig. 4a). The lower recirculation zone is stronger than the upper one. There is strong flow acceleration around the cowl. Airflow in the outer and inner annuli then goes into the liner through the primary and dilution holes. It can be seen that the diffuser flow, the secondary flows in outer and inner annuli, and the combustion flow in the liner are closely coupled together. This is the most important advantage of the integrated simulation of the entire combustor. The numerical errors produced in the exchange of flow physical properties between liner and diffuser, outer and inner annuli are eliminated.

The primary combustion zone is located between the swirler and the primary holes (the first hole on the plane of J=8). The dilution zone begins from the dilution holes (the second hole) until the exit of combustor. In the primary zone, mixing and recirculation provide an ideal aerodynamic condition for the evaporation of fuel spray and the ignition of the mixture. The near unity equivalence ratio created in this zone is an important factor in promoting the flame stability and the complete combustion of the fuel-air mixture.

Fig. 5 illustrates that the combustion process is basically completed in the primary zone. In the dilution zone, the fresh air flows into the liner through the dilution holes (the second hole on the plane of J=8 and the hole on the plane of J=13). High temperature gas is diluted in this zone and temperature distribution becomes more uniform, as shown in Fig. 5 (J=13 and K=42, the exit plane). In order to increase the combustion efficiency and improve the uniformity of the exit temperature distribution, flow fields in the primary and dilution zones need to be carefully controlled.

Fig. 4 Velocity vector in combustor

Fig. 5 Temperature contours in combustor

(a) Longitudinal sections
(b) Transversal section

Fig. 4 Velocity vector in combustor (cont.)
The static pressure recovery coefficient is defined as

\[ C_p = \frac{P - P_0}{0.5 \rho U_0^2} \]  

(10)

where \( P \) is the pressure at the inlet of the diffuser, \( P_0 \) is the pressure at the inner and outer walls of the prediffuser and combustor casing, \( U_0 \) is the average velocity at the inlet of the diffuser. From numerical simulations, the pressure recovery coefficient was obtained. The predicted static pressure recovery coefficients along the outer and inner walls of the prediffuser combustor casings are presented in Fig. 7 respectively with the measurements from Karki (1992). It is seen that the predictions closely follow the experimentally obtained trends. Considering the limitation of \( k-\varepsilon \) model for flows with separation and adverse pressure gradient, the computation and measurement agreement is considered satisfactory.

Fig. 7 Static pressure contours (in 0.1Pa)

Static Pressure Recovery

Static pressure contours in longitudinal and transversal sections are demonstrated in Fig. 6. The exit pressure of combustor is 8 MPa and the predicted inlet pressure is about 8.37 MPa. Static pressure of flow increases in the diffuser. It has a maximum value of 8.48 MPa inside the dome and 8.42 MPa in outer and inner annuli. In liner, gas pressure is almost constant. It is about 8.12 MPa in combustion zone, then slightly decreases towards the exit of combustor. The most notable variations of pressure occur around swirler, primary and dilution holes where the fresh air penetrates into the liner. There is no significant circumferential variation of pressure (Fig. 6b).

The diffuser is functioned to reduce velocity and to convert kinetic energy or dynamic pressure into a rise in static pressure with a less pressure loss. Pressure recovery in diffuser and connected outer and inner annuli are of interest from the viewpoint of designers.
Effects of Non-Uniform Velocity Profiles on Combustion Performance

Inlet conditions of diffuser significantly influence the subsequent flow development. Because the flow pattern in diffuser, outer and inner annuli determines the mass flow distributions along liner walls, the inlet flow conditions have a significant effect on the combustion performance. The sensitivity of combustor performance to inlet conditions is important for investigation of the flow interactions between compressor and combustor.

Two linear profiles of non-uniform velocities at the inlet of diffuser were accepted in the study: radial and circumferential linear profiles. The non-uniform profile factor of flow velocity was defined as

$$\sigma = \frac{U_{\text{max}} - U_{\text{min}}}{U_{\text{ave}}}$$  (11)

where $U_{\text{max}}$ and $U_{\text{min}}$ are the maximum and minimum velocities in radial or circumferential direction, $U_{\text{ave}}$ is the averaged velocity. The linear profiles of non-uniform velocities are demonstrated in Fig. 8. Non-uniform factors of 0, 0.29, 0.86, 1.43 and 2 were accepted in simulations. Effects of non-uniform velocities on OTDF in radial and circumferential directions are shown in Figs. 9 and 10 respectively. It is found that OTDF increases with the non-uniform factor for both radial and circumferential profiles. The effect of non-uniform profile in the circumferential direction is more intensive. As the flow from compressor goes downstream, it becomes more and more uniform due to the transportation of fluid properties. The non-uniform pattern of flow with a larger $\sigma$ could be carried further down, which then leads to a more deterioration of exit temperature distribution. The deteriorated flow field at the section of the primary penetration holes is illustrated in Fig. 11.

Interactions between diffuser and combustion flows are important characteristics of gas turbine combustor. One function of diffuser and the secondary channels is to inhibit the influence of non-uniform profiles and fluctuations of the incoming compressor flow on combustion performance. The transient behavior of flows in compressor-combustor system will be investigated based on this study.
CONCLUSIONS

Numerical study of combustor-diffuser flow interactions was conducted using KIVA-3V code. The entire combustor from the inlet of diffuser to the exit of liner was considered. The swirler was simplified to reduce the complication of the mesh. The effects of non-uniform profiles of inlet velocity on the overall temperature distribution factor were investigated. Following conclusions from this study were obtained:

1. The results from the numerical analysis are in reasonable agreement with those from semi-empirical correlation and measured data.
2. OTDF increases with the non-uniform factor of velocity profiles at the inlet of diffuser. The circumferential non-uniform profile has stronger influence on the exit temperature distribution.

It is concluded that the non-uniform velocity profile at the diffuser inlet has an influence on temperature distribution. The transient behavior of flows in compressor-combustor system will be investigated based on this study.

ACKNOWLEDGEMENTS

The authors wish to thank the Department of Energy for the support of this work under contract #540-6288-1121, and A.A. Amsden of the Los Alamos National Laboratory for all the help with the KIVA-3V code.

REFERENCES


