MODELING OF GAS TURBINE SWIRL CUP DYNAMICS, PART 5: LARGE EDDY SIMULATION OF COLD FLOW

Shanwu Wang and Vigor Yang

Department of Mechanical Engineering
The Pennsylvania State University
University Park, PA 16802

Hukam Mongia, Shih-Yang Hsieh and George Hsiao

GE Aircraft Engines
Cincinnati, OH 45215

41st Aerospace Sciences Meeting & Exhibit
6-9 January 2003/ Reno, NV
MODELING OF GAS TURBINE SWIRL CUP DYNAMICS, PART 5: LARGE EDDY SIMULATION OF COLD FLOW

Shanwu Wang and Vigor Yang
Pennsylvania State University, University Park, PA 16802

Hukam Mongia, Shih-Yang Hsieh and George Hsiao
GE Aircraft Engines, Cincinnati, OH 45215

Abstract

The flow dynamics in a gas-turbine swirl cup is investigated by means of a large-eddy-simulation (LES) technique. The formulation includes the complete conservation equations of mass, momentum, and energy in three dimensions. The analysis employs a preconditioned, density-based, finite-volume approach with dual-time stepping integration. The code is further equipped with a multi-block domain decomposition feature to facilitate parallel processing in a distributed computing environment using the Message Passing Interface (MPI) library. Vortex breakdown in a helical form is observed in the injector and it considerately influences the injector dynamics.

Introduction

Low pollutant emissions are a basic requirement for modern gas turbine engines. A unique prefilming airblast injector, referred to as swirl cup herein, is implemented in several combustion systems as an aid to stabilize high-intensity combustion process for efficient clean combustion because of its unique characteristics for low-power emissions, lean blow out, and smoke formation at high power. While an excellent set of empirical design rules and tools has been employed in the low-emissions engine design for many years, physics-based design rules and tools are currently being developed. In particular, the CFD-based swirl cup model should be utilized with confidence during both the preliminary and detailed design phases by combustor designers.

The theoretical formulation and numerical method implemented in the current work are given in Section 2. A preconditioning technique is described. In Section 3, the computational domain and grid system is addressed. Some primary results are presented and discussed. Finally, a brief summary is given in Section 4.

Theoretical Formulation and Numerical Method

Theoretical Formulation

The present analysis is based on a large-eddy-simulation (LES) technique, in which large-scale turbulent structures are directly computed and small dissipative structures are modeled. Mathematically, the LES methodology begins with filtering of small-scale effects from large-scale motions in the full conservation equations.

The Favre-filtered conservation equations of mass, momentum, and energy can be expressed in the following conservative form:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_j} = 0 \]

\[ \frac{\partial (\bar{p} \bar{u}_i)}{\partial t} + \frac{\partial (\bar{p} \bar{u}_i \bar{u}_j + \bar{p} \bar{\delta}_{ij})}{\partial x_j} = \frac{\partial (\tau_{ij} - \tau_{ij}^{sgs})}{\partial x_j} \]

\[ \frac{\partial (\bar{e}^t)}{\partial t} + \frac{\partial (\bar{e}^t \bar{u}_i + \bar{p} \bar{\delta}_{ij})}{\partial x_j} = \frac{\partial (\bar{u}_i T^t + \bar{q}_j - H_j^{sgs})}{\partial x_j} \]

where \( \rho, u_i, p, e_t, q_j \) and \( \tau_{ij} \) represent the density, velocity components, pressure, specific total energy, heat flux, and viscous stress tensor, respectively. The subgrid-scale terms in Eqs. (2)-(3), i.e., the subgrid stress \( \tau_{ij}^{sgs} \) and subgrid energy fluxes \( H_j^{sgs} \), are closed by implementing an improved Smagorinsky model.
proposed by Erlebacher et al (1992). Details of the filtered equations and the subgrid closure employed have been reported in the cited papers and therefore not presented here for brevity.

The method of characteristics (MOC) is used to treat the boundary conditions. The flow velocity \( \mathbf{u} \) and total temperature \( T_0 \) are specified at the inlet and the pressure is determined using a simplified one-dimensional momentum equation in the axial direction. At the outlet, the back pressure is fixed at 1 atm and the other flow variables are calculated by mean of zero-gradient extrapolation.

### Numerical Method

The presence of low Mach number regimes where large disparities exist between the acoustic and flow speeds could substantially degrade the numerical convergence and efficiency of computational algorithms. A time derivative preconditioning technique is employed to circumvent this difficulty.

To facilitate numerical treatment, Eqs. (1)-(3), are written in a general vector notation

\[
\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial (\mathbf{E} - \mathbf{E}_0)}{\partial x} + \frac{\partial (\mathbf{F} - \mathbf{F}_0)}{\partial y} + \frac{\partial (\mathbf{G} - \mathbf{G}_0)}{\partial z} = 0 \tag{4}
\]

where \( \mathbf{Q} = [\rho, \rho \mathbf{u}, \rho \mathbf{v}, \rho \mathbf{w}, \rho E]^T \) and the superscript \( T \) stands for the transpose of a vector. \( \mathbf{E}, \mathbf{F}, \) and \( \mathbf{G} \) represent the convective flux vectors in the \( x-, y-, \) and \( z- \) directions, respectively, and \( \mathbf{E}_0, \mathbf{F}_0, \) and \( \mathbf{G}_0 \) the diffusion-flux vectors in the \( x-, y-, \) and \( z- \) directions, respectively.

Pseudo time derivatives are added to Eq. (4) to implement the dual-time stepping procedure. This procedure yields the following equation.

\[
\Gamma \frac{\partial \hat{\mathbf{Q}}}{\partial \tau} + \frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial (\mathbf{E} - \mathbf{E}_0)}{\partial x} + \frac{\partial (\mathbf{F} - \mathbf{F}_0)}{\partial y} + \frac{\partial (\mathbf{G} - \mathbf{G}_0)}{\partial z} = 0 \tag{5}
\]

The quantities \( \hat{\mathbf{Q}} \) and \( \Gamma \) in the above equation represent the primitive variable vector and preconditioning matrix, respectively. These terms are defined as,

\[
\hat{\mathbf{Q}} = \begin{bmatrix} \gamma / \beta & 0 & 0 & 0 & -\bar{\rho} / \bar{T} \\ \bar{u}_x / \beta & \rho & 0 & 0 & -\bar{u}_x / \bar{T} \\ \bar{u}_y / \beta & 0 & \rho & 0 & -\bar{u}_y / \bar{T} \\ \bar{u}_z / \beta & 0 & 0 & \rho & -\bar{u}_z / \bar{T} \\ \bar{h} / \beta - 1 & \bar{p} \bar{\rho}_x & \bar{p} \bar{\rho}_y & \bar{p} \bar{\rho}_z & \bar{p} c_p - \bar{h} / \bar{T} \end{bmatrix}
\]

Here, \( \rho_p \) is the gauge pressure, i.e.,

\[
p_p(x,t) = p(x,t) - p_0,
\]

which is introduced to avoid the pressure singularity in the momentum equations, and

\[
\beta = \gamma \varepsilon c^2 / [1 + (\gamma - 1) \varepsilon] \tag{8}
\]

where \( c \) is the speed of sound, \( \gamma \) the specific heat ratio, and \( \varepsilon \) the preconditioning factor, which is proportional to the local Mach number \( \bar{M} \) and provides a control of both the inviscid and viscous time scales. In the present study, the methodology developed by Choi and Merkle (1993), Buelow (1995), and Venkateswaran and Merkle (1995) is implemented to select the preconditioning factor.

Equation (5) is discretized using an explicit four-step Runge-Kutta method to evaluate the pseudo time derivatives. A generalized backward differencing is used to evaluate the physical time derivatives.

\[
\Gamma \left( \hat{\mathbf{Q}}^{n+1} - \hat{\mathbf{Q}}^n \right) + \frac{\Delta \tau}{\Delta t} \left[ \alpha \mathbf{Q}^n - \phi(\mathbf{Q}^n, \mathbf{Q}^{n+1}, \cdots) \right] + \Delta \tau \left[ \frac{\partial (\mathbf{E} - \mathbf{E}_0)}{\partial x} + \frac{\partial (\mathbf{F} - \mathbf{F}_0)}{\partial y} + \frac{\partial (\mathbf{G} - \mathbf{G}_0)}{\partial z} \right]^p = 0
\]

and \( \mathbf{Q}^{n+1} = \mathbf{Q}^n \) at convergence in pseudo time. Here, the efficient \( \alpha \) and the fundamental relationship \( \phi \) are determined by the accuracy of the time integration. Implementation of the dual-time stepping technique allows for the flexibility in selecting these two time steps.

The spatial discretization is obtained using the fourth-order accurate flux-differencing methodology developed by Rai and Chakravarthy (1993). A sixth-order artificial dissipation is employed in the discretized formulation in order to prevent numerical oscillation and subsequently to improve numerical convergence. The numerical simulation is implemented on an in-house parallel computing facility consisting of 500 Pentium II/III/4 processors at Penn State. A multi-block technique based on a domain decomposition method (DDM) is employed to parallelize the code. The message passing interface (MPI) library is used to exchange information among processors, i.e., blocks.

### Results and Discussions

The swirl cup considered in the present work is the GE CFM56 injector, as shown in Fig. 1. A pressurized fuel stream is delivered to a simplex atomizer. As the liquid fuel flows from the atomizer, it encounters a swirling airflow and impinges onto a liquid filming surface. Subsequently, a second, counter-rotating airflow produces secondary atomization, and the fuel spray issued forms a conical sleeve where the flame is
The purpose of this paper is to conduct a comprehensive numerical analysis of the detailed injector dynamics under cold flow conditions. Figure 1 shows the configuration of concern, consisting of two counter-rotating swirlers: the primary jets and the secondary swirler.

The grid system and domain decomposition employed in the current study is shown in Fig. 2. Due to the geometry complexity of the computational domain, a total of 1620 blocks are used for the domain decomposition. The overall grid system includes 2 million effective cells and 4 million ghost cells for the calculation with a fourth-order spatial difference in the convection terms. All of the 1620 blocks are loaded on 48 CPUs for the parallel processing. Since each processor includes more than one block, an appropriate load balance is required to achieve computational efficiency. This is a critical feature for a distributed computing environment with a structured grid system.

Figure 3 shows a snapshot of the vorticity field at two different cross sections. The first is the vortex breakdown due to the swirling flows. A central toroidal recirculation zone (CTRZ) is found downstream of the venturi, serving as a source of low-frequency flow oscillations. Because of the strong shear between the inlet passage and the CTRZ, a vorticity layer is produced, which subsequently rolls, tilts, stretches, and breaks up into small vorticity bulbs. The second phenomenon is the high-frequency vortex shedding arising from the Kelvin-Helmholtz instabilities in both the axial and azimuthal directions. The ensuing influence on the fuel/air mixing may be significant because the shear layer will interact with the thin fuel film on the surface of the guide vane. The flow pattern in the $r-\theta$ (A-A) cross-section clearly illustrates the structures associated with the eight swirling jets, i.e., the primary swirler. This non-axisymmetric flow pattern diminishes along with the downstream distance and the flow becomes uniform in the azimuthal direction after it passes the venturi. This demonstrates the good mixing characteristics of the injector.

The structures of the mean flowfields are presented in Figs. 4-6. Figure 4 shows the contours of the axial velocity. Due to strong swirling flows through the primary swirl jets, a CTRZ is generated starting from the venturi and extending to the chamber. The presence of the CTRZ notably decreases the effective flow passage area. High axial velocity regions exist outside of the CTRZ, which further induces strong shear layers in these regions. There are two local minima of the azimuthal velocity. One is located in the flare and the other in the chamber. The detachment of the two local minima is partially due to the counter-rotating swirling flows. The primary jets induce a local minimum in the flare. The counter-rotating secondary swirler decreases the azimuthal velocity of the flow through the primary jets. In the downstream of the flare, the flows through the secondary swirler dominate the flow profile, which, hence, induces another local minimum in the chamber.
Figure 5 shows the pressure contours of the mean flowfield. The lowest pressure region is located at the central line near the primary injector. Since part of the primary swirling jets flow to this region directly, a low-pressure region is induced due to the conservation of angular momentum, which results in high azimuthal velocity near the central line. Because of the constraint of the wall, a strong negative pressure gradient exists near the wall. Similar structures were also observed by Wang et al. (2002). Figure 6 shows the contours of the angular momentum and streamlines. It clearly demonstrates the similarity between these two physical quantities. The angular momentum fields are thus employed for flow visualization in order to understand the unsteady evolution of the flowfield.

Figure 7 shows the snapshots of the angular momentum at sequential time steps. It clearly reveals the vortical interactions, which include the dynamic evolution of vortex breakdown and the interaction between the primary and secondary injectors, especially in the flare. The jets from the primary swirlers breakup due to the strong shears at the boundary the CTRZ, i.e., dynamic evolution of the vortex breakdown. When these large flow structures convect to the downstream, they interact with the counter-rotating flows through the secondary swirler, which are disturbed by the high intensity unsteady flow motions from the primary injector and even breakup intermittently. It may consequently influence the primary breakup of the fuel film in this region.

The snapshots of the pressure field on the $Z=0$ plane are shown in Fig. 8. The two-dimensional plots show that a low-pressure core exits along the central line near the wall. This core structure oscillates in the injector and small low-pressure bubbles shed from the large core structure periodically. To further examine the mechanism associated with this interesting phenomenon, a three-dimensional structure characterized by the isobaric surface is shown in Fig. 9, which clearly illustrates a helical structure in the core region. Because this structure rotates steadily, a pressure oscillation and “shedding” pattern is observed in Fig. 8. The flow motion can be recognized as a kind of helical vortex breakdown. The rotation of the helical structure modifies the effective flow passage area, and hence affects the pressure distribution in the chamber.
Fig. 7  Snapshots of angular momentum

Fig. 8  Snapshots of pressure fields
Conclusions and Perspectives

A comprehensive numerical analysis has been conducted to investigate the flow dynamics of a swirl-stabilized injector. The formulation treats the unsteady, three-dimensional conservation equations, with turbulence closure achieved using a large eddy simulation (LES) technique. Detailed flow structures and injector dynamics are studied systematically. Various unsteady flow characteristics, such as vortex shedding and vortex breakdown as well as their interactions, are investigated in depth.

The response of the injector flow to externally imposed forcing at the inlet will be investigated in the future. Such information will be used to characterize the dynamic behavior of the injector. Results of this kind are critical to injector design optimization in terms of the mixing, flame stabilization, and combustion stability characteristics.

Acknowledgment

The work was sponsored by the Pennsylvania State University.

Reference


