TURBULENT REACTING FLOW ANALYSIS FROM GAS TURBINE CAN-TYPE COMBUSTOR CHAMBER USING COMPUTATIONAL FLUID DYNAMICS CODE

GUESSAB A.1, ARIS A. 2, TAWFI B.1, MADANI Y. H. 3, BAKI T.2

1I.P. S. I. L, Oran, Algérie.
2 L.C.G. E, USTOMB, Oran, Algérie.
3 Laboratoire de Mécanique des fluides, ENP d’Oran, Algérie.

Introduction

The turbines are widely used in modern industry to deliver shaft power or thrust power. The combustor is part of the gas turbine. The main goal of the combustor design is lower emissions with less volume. Flow field simulation in the combustor is a challenging subject to both academics and industries. It is of commercial importance to understand and to predict various phenomena in the combustor. Gas turbine combustion systems need to be designed and developed to meet many mutually conflicting design requirements, including high combustion efficiency over a wide operating envelope and low NOx emission, low smoke, low lean flame stability limits and good starting characteristics; low combustion system pressure loss, low pattern factor, and sufficient cooling air to maintain low wall temperature levels and gradients commensurate with structural durability. Numerical simulations of the flows in gas turbine combustors had become an unavoidable way to accelerate the design of this type of modern engines and optimize their performances. The calculations also facilitate the understanding and the visualization of physical phenomena often inaccessible by the experimental measurements. The use of Computational fluid dynamics (CFD) codes to predict the reactive flow within a gas turbine combustor has been the center of many studies: Lysenko and Solomatnikov (2003), Lysenko and Solomatnikov (2006), Palm et al (2006), Fureby et al (2007), Widendorf et al (2009), (2010).

In this works the three-dimensional reacting turbulent flow fields of a swirl-stabilized gas turbine model combustor was analysed with compressible CFD. Computations were performed using the commercial finite-volume code ANSYS FLUENT 15, employing the Detached Eddy Simulation model (DES) with the shear stress transport (SST) k-ω turbulence model for closure and the eddy dissipation model for combustion.

Geometry

The basic geometry of the gas turbine can-type combustor chamber is shown in Fig. 1. The size of the combustor is 590 mm in the Z direction, 250 mm in the Y direction, and 230 mm in the X direction. The primary inlet air is guided by vanes to give the air a swirling velocity component. The total surface area of primary main air inlet is 57 cm². The fuel is injected through six fuel inlets in the swirling primary air flow. There are six small fuel inlets, each with a surface area of 0.14 cm². The secondary air is injected in the combustion chamber through six side air inlets each with an area of 2 cm². The secondary air or dilution air is injected at 0.1 m from the fuel injector to control the flame temperature and NOx emissions. The can-type combustor outlet has a rectangular shape with an area of 0.0150 m².

Figure 1: 3-D geometry of the gas turbine combustor.

Computational method

The three-dimensional geometry was created using GAMBIT—a FLUENT pre-processor. A view of the grid system is shown in Fig.2. The meshed geometry contained 255652 mixed hexahedral and tetrahedral cells. ANSYS Fluent 15 offers a variety of different models for turbulence and combustion. The presented results were obtained using the Detached Eddy Simulation model for turbulence and Eddy Dissipation model for combustion. The DES approach was employed using a modification of the shear stress transport (SST) k-ω turbulence model for closure. The turbulence-chemistry interaction model called the eddy-dissipation model is based on the work of Magnussen and Hjertager (1976). In this model combustion proceeds whenever turbulence is present and an ignition source is not required to initiate combustion. This is usually acceptable for non-premixed flames. In this work, a two-step mechanism to model the combustion of methane in air was employed (Westbrook and Drayer (1981)) was used to compute the rate of fuel oxidation.

\[
\begin{align*}
2\text{CH}_4+0.22\text{N}_2+3(\text{O}_2+3.76\text{N}_2) & \rightarrow 2\text{CO}+ 4\text{H}_2\text{O} +11.5\text{N}_2 \quad (1) \\
2\text{CO} + (\text{O}_2+3.67\text{N}_2) & \rightarrow 2\text{CO}_2 +3.76\text{N}_2 \quad (2)
\end{align*}
\]

The constants of Arrhenius equations (1) and (2) are reported in Table 1.

<table>
<thead>
<tr>
<th>Eqs.</th>
<th>(A_k \quad [s^{-1}])</th>
<th>(E_k)</th>
<th>(\beta_k)</th>
<th>(\gamma_{\text{CH}_4})</th>
<th>(\gamma_{\text{O}_2})</th>
<th>(\gamma_{\text{CO}})</th>
</tr>
</thead>
<tbody>
<tr>
<td>28</td>
<td>(1.50\times10^7)</td>
<td>30</td>
<td>0</td>
<td>-0.3</td>
<td>1.3</td>
<td>___</td>
</tr>
<tr>
<td>29</td>
<td>(1.0\times10^{20.75})</td>
<td>40</td>
<td>0</td>
<td>0.25</td>
<td>1</td>
<td>___</td>
</tr>
</tbody>
</table>

Table1: The constants of the Arrhenius equations.
The unsteady simulations were performed using the commercial CFD code ANSYS FLUENT 15, which is based on finite volume discretization of the conservative governing equations. For the time discretization an implicit second order time differencing scheme was used. For the spatial discretization a bounded central difference scheme was used for the momentum equation, a second order upwind scheme for turbulence equations and a first order upwind scheme for the energy and species equations. The algorithm SIMPLE was used for the coupling of pressure and velocity. The boundary conditions are summarized in Table 2.

**Table 2: Inlet boundary conditions.**

| Primary air | • The injection velocity is 10 m/s;  
|            | • The injector diameter is 85 mm.  
|            | • Heat transfer: Static temperature:  
|            | 300K  
|            | • The turbulence intensity is 10%;  
|            | • Mass fraction of oxygen: YO₂=0.232  
| Fuel       | • Flow direction: Normal to boundary  
|            | condition;  
|            | • The injection velocity is 40 m/s;  
|            | • The injector diameter is 4.2 mm.  
|            | • The temperature is 300K;  
|            | • The turbulence intensity is 10%;  
|            | • Thermal radiation: Local temperature;  
| Secondary | • The injection velocity is 6 m/s;  
| air (dilution) | • The injector diameter is 16 mm.  
|            | • The temperature is 300K;  
|            | • The turbulence intensity is 10%;  
| Outlet     | • Flow regime: Subsonic;  
|            | • The relative pressure is 0 Pa;  
|            | • Mass fraction of oxygen: YO₂=0.232;  
|            | • Thermal radiation: Local temperature;  
| Wall       | • Wall boundary condition was no slip;  
|            | • Wall heat transfer was adiabatic;  
|            | • Wall emissivity was 0.95  

**Results and discussion**

**Aerodynamics characteristics**

Figure 3, shows the streamline contours produced by the numerical model. The color of the streamline patterns represents the axial values of velocity. These streamlines indicate the path which a fluid particle would follow. Flame stability, combustion intensity, and performance are directly associated with the size and shape of this recirculation vortex.

In primary zone, the negative axial velocities in the center of the combustion chamber indicate the existence of an inner recirculation zone. Swirl vanes around the fuel nozzle generate a strong vortex flow in the combustion air within the combustor. The flow field is typical of confined swirl flames and consists of a cone-shaped stream of burning gas in the inlet of the chamber. For sufficiently high swirl number (in this study, \( S=1.5 \)), recirculation zone appears as a result of the vortex breakdown. The length of the recirculation zone is about (150 mm). The recirculation zone operates normally at a rich mixture. This recirculation region is deeply involved in the flame stabilization process as it constantly puts hot burnt gases in contact with fresh gases allowing permanent ignition.

The central recirculation zone results from the vortex breakdown generated by the swirl. This large toroidal central recirculation zone plays a main role in the flame stabilization process by acting as a store for heat and chemically active species. Negative axial velocities in corner regions indicate the existence of two corner recirculation zones resulting from sudden expansion. Corner recirculation zone above the shear layer of the recirculation zone and takes it shape from the neighboring boundary walls. The length of the corner recirculation zone is about (76 mm). Strong velocity gradients occur in the throat of the nozzle and, in the outer shear layer between inflow and recirculation zone, because of the strong turbulent mixing in those locations. The secondary zone adds air to weaken the mixture. More air is added in the dilution zone to decrease the hot gas temperature for meeting the requirements of turbine guide vanes. This hot gas helps stabilize the flame by providing a continual source of oxygen to the incoming fuel. It also serves as a zone of intense mixing within the combustor by promoting turbulence through high levels of shear between the forward and reverse flows. Recirculation zone is formed just at the downstream of the secondary air inlet (secairin). Fuel velocity coming out from the injector at the plan 1 (Fig. 4) has a magnitude of 40m/s. Air coming through the swirled jet has a maximum velocity of 10 m/s and this flow spreads in the radial direction occupying whole space without formation of recirculation zone. This is due to the blockage effect created by the secondary air inlet jets forcing the primary air inlet jets and fuel jet to spread in radial direction. Velocity of fuel-air mixture reduces as it enters the liner axis of combustor. From Figure 4, it can be seen that the modelling of dilution holes (secairin) results in the corresponding air jets being generated. The secairin jets help in recirculating a part of the primary air to aid in combustion in the primary zone. This zone is a prerequisite for the stable combustion. The function of the dilution air jet is to mix with the products of combustion to cool these to a value acceptable by the turbine and also to reduce the pattern factor at the exit of the combustor.
Conclusion

In this work, numerical simulations of complex reactive turbulent swirled flow correspondent to a gas turbine can-type combustor were established by means of the ANSYS-FLUENT 15. The main aim was to study the vortex structures within the gas turbine combustion chamber. Mean flow field showed a large inner recirculation zone (IRZ) and an outer recirculation zone (ORZ) which represents a typical result of confined swirl flames. Mean temperature contours plots showed that the IRZ is formed by two counter rotating vortex pair, while instantaneous temperature contours plots showed clearly the existence of many small vortex that change in size and position in that zone.

References


