Available online at www.sciencedirect.com

ScienceDirect

Proceedings of the Combustion Institute 35 (2015) 1657–1665

Proceedings of the Combustion Institute

www.elsevier.com/locate/proci

On the dynamics of spray flames in turbulent flows

Yuval Dagan [a,*], Eran Arad [b,1], Yoram Tambour [a,2]

[a] Faculty of Aerospace Engineering, Technion – Israel Institute of Technology, Haifa 32000, Israel
[b] Aeronautical Systems (39), RAFAEL, Haifa 31021, Israel

Abstract

Unsteady turbulent spray-flame structures were computationally investigated, employing the same flow configuration used in previous studies for analyzing the structure of gaseous turbulent flames. The presence of a recirculation zone, which is common in jet engine combustion chambers, has a significant role in spray and flame dynamics, diverting the flame in a cyclic motion. Two repetitive developmental stages of flame structures were identified and analyzed. Droplet grouping was found in the vicinity of large vortical structures, and flames surrounding groups of droplets were identified. Backflow of droplets was found to have ligament structures, similar to those found in turbulent shear flow. The local statistics of fuel droplet dispersion is discussed. A new approach is used to investigate the unsteady changes in flame structure by reducing the dimensionality of the problem, revealing low frequency flame repetitive motions, while retaining its turbulent characteristics.

Keywords: Spray combustion; Turbulent flames; Unsteady combustion

---

1. Introduction

Unsteady changes in spray-flame structures in the vicinity of a large recirculation zone is investigated in the present study. Flame instabilities play an important role in turbulent combustion. Matalon [1] has shown that flame instabilities such as cellular structures and oscillating flames appear in premixed as well as nonpremixed diffusion flames. Strong coupling with turbulent flow occurs and significantly influences the flame dynamics. Cases in which the flow is turbulent, which are inherently three-dimensional and unsteady, impose crucial unsteady characteristics on the combustion process which complicates the instability analysis even more. In previous work, we have studied three-dimensional unsteady turbulent gaseous flame structures using large eddy simulations Dagan et al. [2]. The test case investigated was a co-annular, gaseous methane fuel jet combustor, following the configuration of Owen et al. [3] in swirling and non-swirling inflow conditions. Mean statistics of the computed flow field were found to be in very good agreement with experimental data. Then, detailed time-dependent monitoring of flame structures over long periods were collected. Intermittent flame liftoff phenomena were found and quantified

---

* Corresponding author.
E-mail address: yuvalda@technion.ac.il (Y. Dagan).

[1] Head CFD Group.

[2] Professor, Stephen E. Berger Chair in Aeronautical Engineering. Currently on Sabbatical at Columbia University in New York, in the Department of Civil Engineering and Engineering Mechanics.

http://dx.doi.org/10.1016/j.proci.2014.07.044

by using newly developed criteria for flame tracking. High correlation was also found between the frequencies of oscillations in the circumferential and radial directions.

Introduction of liquid fuel into the configuration complicates the problem of turbulent combustion. Santoro et al. [4] have shown that spray flames have much longer temporal delays than the gas-phase between the onset of local extinction and reconstruction of the flame. The effects of droplet clustering on evaporation have been thoroughly discussed by Sirignano [5] and Harstad and Bellan [6]. Katoshevski et al. [7] have shown that evaporation can lead to droplet grouping, even in the case when the non-evaporating droplets are not grouped. In their comprehensive study, Reveillon and Vervisch [8] analyzed a large number of spray-flame structures in weakly turbulent flows using Direct Numerical Simulations (DNS), and various combustion regimes were identified. In addition, DNS of statistically stationary, spatially decaying turbulence was conducted by Reveillon et al. [9]. In their study, the turbulent dispersion of evaporating polydispersed sprays was characterized. Recently, in detailed direct numerical simulations (DNS) of Diesel spray combustion in the vicinity of a recirculation zone, Shinjo and Umemura [10] have shown that when droplets are larger than the Kolmogorov microscale, mixing is strongly enhanced by the presence of droplets and fuel vapor clusters are likely to form quickly when the droplet number density is high. They have suggested that external group combustion is likely to occur near the recirculation zone.

In the present study, the global unsteady structure of turbulent spray flames, in the vicinity of a large unsteady recirculation zone, is investigated. Hence, we use the same co-annular configuration of Owen et al. [3] and use a liquid fuel spray instead of the gaseous one of our previous study Dagan et al. [2]. It should be noted that our test case was conducted for highly unsteady flame gaseous dynamics. To the best of our knowledge, a similar test case that incorporates liquid fuel sprays is currently not accessible in the open literature. Tambour, Greenberg and their coworkers developed and implemented a sectional approach in order to treat fuel sprays of arbitrary drop-size distributions [11–15]. The sectional approach was found to be a very useful tool in handling the local statistics of fuel droplets dispersion, as shown below (although this approach was not directly employed in our present computation).

2. Flow field and computational configuration

The combustion chamber configuration is presented in Fig. 1. As stated before, we use the test case of Owen et al. [3]. The configuration presented is non-dimensional with a reference value of $R = 4.685$ cm. Air enters at $U_{ref} = 20.63$ m/s in the outer tube of radius $r_3 = 1.3$R and at 1 m/s in the inner tube. Fuel droplets are injected from a small ring of radius 0.05R located at $x/R = -0.9$. The liquid fuel is injected at a velocity of $2U_{ref} = 41.26$ m/s and a mass flow rate of 0.001 kg/s. Angular velocity is imposed on the droplets to simulate swirl jet injection of 45°. The spray injected is initially distributed according to Rosin–Rammler distribution with a minimum droplet size of $d_{min} = 0.1$ μm, a maximum of $d_{max} = 10$ μm and a mean of $d_m = 5$ μm. This method of injection imposes a non-symmetric spatial distribution of droplets. The walls are assumed to be adiabatic. These values are typical for fine spray injection systems [10].

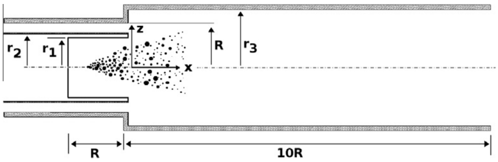

Fig. 1. Combustion chamber configuration.

The mesh used in the current computation comprises of about 7 million hexahedral cells. A time-step of $3.4 \cdot 10^{-7}$ s was used, which is two orders of magnitude smaller than the one used in our previous study of gaseous fuel flame Dagan et al. [2]. This fine mesh was required for proper resolution of the droplet dynamics and coupling with the gas-phase flow.

3. Computational method

A well validated Large Eddy Simulation (LES) code, CDP, developed at Stanford University was used. This package has been successfully applied for the solution of large scale turbulent combustion problems [16,17]. A coupled Lagrangian–Eulerian approach is adopted to treat the reactive spray, using the formulation of Apte et al. [18,19]. The challenge of capturing the turbulent structures responsible for the rapid mixing between fuel and oxidizer, while maintaining numerical stability, is met by the use of a non-dissipative, energy conserving scheme [16]. The non-dissipative nature of the scheme is especially important for long integration periods, needed for proper analysis of combustion dynamics. The spray equations are briefly described below. The method comprises a two-way coupling between gas-phase Eulerian equations and the Lagrangian particle tracking equations [18].

3.1. Gas phase equations

The flow field is modeled by the unsteady, low Mach number Navier–Stokes equations. The flow