Mixture Preheating Effects in a Bluff-Body Burner

R. Potami, E. Giacomazzi, B. Favini, F.R. Picchia, N. Arcidiacono

Dept. of Mechanics and Aeronautics, University “La Sapienza”, Rome – ITALY
ENEA-IMP, ENEA, C. R. Casaccia, Rome – ITALY

ABSTRACT

We simulate by means of Large Eddy Simulation a premixed burner consisting in a straight channel (rectangular cross-section) with a prismatic bluff-body (triangular cross-section). In a previous work we simulated this burner without preheating the mixture; we showed the importance of simulating the side wall confinement effect to correctly predict the shortening of the wake and the wall heat transfer. In this paper we simulate the same burner operated with a preheated mixture to show the effect of preheating on flame stabilization and on turbulence – chemistry interaction. We simulate both simple mixing and reacting cases; results are compared with the experimental data. The time-dependent LES results show three-dimensional vortex structures periodically shortening the recirculation zone downstream of the bluff-body and entraining fresh mixture into the hot recirculating region.

INTRODUCTION

A method widely used in combustor to hold and stabilize a turbulent flame consists of creating a recirculation region where oxidation kinetics has sufficient time to burn most or all of the fuel. A backward-facing step or a bluff-body are typical flameholder geometries. Combustors with bluff-body flameholders, such that analysed in the present study, are characterized by a shear layer where vortices are shed due to Kelvin-Helmhotz instability [1]. This shear layer separates the regions of high speed fresh mixture from the wake region of lower speed hot products. Turbulence enhances mixing downstream of the bluff-body, stabilizing the flame in an unsteady manner.

We numerically simulate a premixed bluff-body burner built and instrumented by VOLVO-FLYGMOTOR [2,3]. The purpose is to investigate how a flameholder works. We perform LES simulations of the turbulent reacting flow to capture its dynamics; we use the Fractal Model (FM) [4,5] as SGS model for turbulence and chemical kinetics closure. Comparison of numerical results with experimental data validate again the FM itself.

TEST CASE

The burner is a straight channel with a rectangular cross section; the inlet section is used for flow straightening, turbulence control and for fuel and seeding injection. The air entering in the inlet section is distributed over the cross-section by a critical plate that isolates the combustor acoustically from the air supply system. The combustor section ends in a circular duct with a large diameter. The flame is stabilized by means of a typical V-Flameholder whose height is 0.04m and width is 0.24m; this flameholder is centered in the channel, 0.682 m upstream of combustor exit. A detailed description of the combustor is given in [3].

The fuel is a mixture of propane and air at 600 K and \(\phi = 0.65\) (equivalence ratio); the operative pressure is 1 atm and the initial combustor pressure is 100KPa. The inlet flow velocity before bluff-body is \(U_{bulk} = 37 m/s\), the airflow 0.6Kg/s and the Reynolds number \(Re = 28000\) based on the hydraulic diameter. This flow condition has been experimentally studied only in the reactive case in [2,3].

NUMERICS

The numerical simulation is made by a 3-D code developed for turbulent, reacting, unsteady flows at low Mach number, parallelized using the MPI procedures. The finite difference numerical scheme is explicit, third order accurate in time and second in space.
Our computational domain starts just downstream of the bluff-body and ends before the sudden expansion in the final part of the combustor; dimensions are $0.24m \times 0.12m \times 0.5m$ (spanwise, height, length). The grid (100x100x150 nodes along X, Y and Z) is structured with nodes uniformly distributed in the X and Y directions; nodes along Z are stretched, the grid becoming coarser towards the burner exit.

Walls are adiabatic and viscous. At the outlet we use the NSCBC (Navier-Stokes Characteristic Boundary Conditions) method [6]; it is particularly useful because it reduce numerical reflection of acoustics waves. At the inlet section we impose the three components of velocity, the flow temperature and composition; pressure is extrapolated by the internal domain. The velocity profile has been derived from a previous simulation that took into account the whole bluff-body triangular shape.

At the start-up of the nonreactive simulation the mixture enters into the burner filled with still air. Transition to turbulence is forced overimposing some disturbances turned off after transition. After we used the nonreactive developed flow as initial condition for the reactive simulation. We ignited the mixture by using a fast chemistry approach, then switched to the two step mechanism proposed in [7].

The SGS model adopted is the Fractal Model, described in [4,5].

**RESULTS**

**Nonreactive Case**

We simulated the nonreactive flow to show the effects of combustion on turbulent structures and turbulence dynamics.

Results show that the flow is characterized by a recirculation zone downstream of the bluff-body. The recirculation zone is long approximately $z = 0.09m$ and consists of two counter-rotating vortices asymetrically shed from the flame-holder edges. These vortices are quickly dissipated after shedding and cause oscillation in the wake. The wake oscillation has a characteristic frequency $f = 522 Hz$, whilst the shedding frequency of the two asymmetric vortices is $f = 261 Hz$. The shedding Strouhal number ($St = f \cdot h / U_{in}$, where $h$ is the bluff-body height and $U_{in}$ is the inlet velocity) estimated from our simulation is $St = 0.282$. We note that in the actual preheated flow, the Strouhal number and frequency are higher than in the non-preheated flow; in fact, results in [8] show $St = 0.25$ and $f = 105 Hz$ for the nonpreheated flow.

Figure 1 shows an instantaneous axial velocity field in the middle plane of the burner; the streamlines show the coherent structures shed by the bluff-body. The recirculation zone dynamics is not symmetric in the spanwise direction.

The time averaged flow field reported in Fig. 2 is obtained by means of 40 sampled fields corresponding to a time window of 0.067s. The average recirculation zone is characterized by two symmetric counter-rotating vortices whilst instantaneously it has only one eddy (see Fig. 1).

**Reactive Case**

Starting from the nonreactive simulation we ignited the mixture. Due to combustion the recirculation zone topology and dynamics change (see Fig. 3): its length is approximately $z = 0.14m$, greater than the nonreactive case, and it is characterized by two counter-rotating vortices that pulse asymmetrically without being shed (whilst the nonreactive flow experienced asymmetric shedding). The two eddies stretch each other, produce intense mixing of hot combustion products and fresh mixture and bring them towards the bluff-body; the residence time and temperature are high enough for chemical reactions to take place. Therefore the wake is a hot spot anchoring the flame. Downstream of the bluff-body out of
the reacting zone the flow accelerates up to 50-60 m/s as in the nonreactive flow; acceleration is larger in the wake.

Figure 3 shows an instantaneous temperature field and some streamlines in the middle plane of the burner. Temperature is maximum in the recirculation zone that anchors the flame. The stretching of the two main eddies produces other small structures not present in the time averaged flowfield shown in Fig. 4.

**Fig. 1:** Instantaneous axial velocity field and streamlines in the nonreactive case.

**Fig. 2:** Time averaged axial velocity field and streamlines in the nonreactive case.

**Fig. 3:** Instantaneous temperature field and streamlines in the reactive case.
Looking at the three-dimensional structure of the recirculation zone we note that its length is greater in the middle plane of the burner due to the side walls effect; therefore the main anchoring is achieved at the center, where the largest heat release takes place. Expansion at this location produces lateral motion of the flow from the side walls towards the center, as shown in Fig. 5. This effect is less evident in the nonreactive simulation.

We note that the actual preheated case does not experience any shedding; the wake oscillates at frequency less than $30 \text{ Hz}$. Instead, nonpreheated flow of a slightly lean mixture at slower inlet velocity in the same burner [8] has characterized by a symmetric shedding at $105 \text{ Hz}$ and $St = 0.33$.

REFERENCES