Large Eddy Simulation and experimental study of a Trapped Vortex Combustor

C. Merlin, P. Domingo, L. Vervisch, J. Burguburu, G. Cabot, B. Renou
CORIA-CNRS & INSA de Rouen
Rouen, France

1 Introduction

Combustion stability may be achieved with recirculation zones providing a continuous source of ignition, through the mixing of hots products with the incoming air and fuel mixture. To generate these recirculation zones, swirl vanes, bluff bodies and rearward facing steps are commonly used. As opposed to conventional combustion systems which rely on swirl stabilization, the Trapped Vortex Combustor (TVC) is a challenging configuration where flame stabilization is achieved through the presence of a cavity. The flow is then trapped within a cavity where the reactants are mixed. This challenging configuration provides high combustion efficiency, low emission and low pressure drop. Some experimental and numerical studies have also been devoted to the effective use of cavities to suppress combustion instability [1–4]. Generally, a bluff body is located upstream to initiate shear layer instabilities which are then trapped in the cavity region and provides a wake to help the flame stabilization.

The numerical study of a non rectangular configuration with a bluff body is possible in a non-structured mesh solver. Another approach retained in this paper is to keep a structured mesh with the addition of the immersed boundary concept. Immersed Boundary Methods employ cartesian meshes that do not conform to the shape of the body in the flow and modify the governing equations to incorporate the boundary conditions. First introduced by Peskin [5], IBMs involve either continuous or discrete forcing approaches. Only discrete forcing approach preserves good performance at high Reynolds number. A ghost-cell method (a discrete forcing approach) is used here with a sharp boundary and with a modification of the computational stencil and an extrapolation scheme to deduce the state variables in cells bordering the computational domain.

The present work describes an unsteady non-reacting and reacting annular TVC flow studied with Large Eddy Simulation and experimental approach. In the following sections, the numerical method as well as the combustor configuration are briefly described. The cold flow properties are first compared with their experimental counterparts to assess the ability of the immersed boundary to deal with complex compressible flows. The flame dynamics is then investigated with a combustion model based on detailed tabulated chemistry associated with a presumed PDF (PCM-FPI).

Correspondence to: cindy.merlin@coria.fr

2 Combustor configuration and computational details

As shown in Fig.1, the cylindrical combustion chamber consists of a 22 mm long cavity. Cavity flows are often categorized depending on the length to depth ratio L/D. The experiment includes a deep cavity of lenght-to-depth ratio of L/D=0.8. Flame holders (rods immerged in the main flow) are added to enhance the mixing of the burnt gases with the main flow in the wake region and to improve the main flame stability. The methane and air mixture is perfectly premixed upstream of the burner entrance. Fuel and air are also injected on the forward and afterward faces of the cavity to reinforce the stable vortex generated by the main flow over the cavity. The main air mass flow rate is 20g/s. The air mass flowrate is 0.7 g/s for the mixture entering the cavity and 1g/s for the after body air injector. Experiments were conducted on this new TVC combustor. The Particles Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV) methods were used to measure the velocity fields for the cold flow. Reactive conditions measurements are in progress.

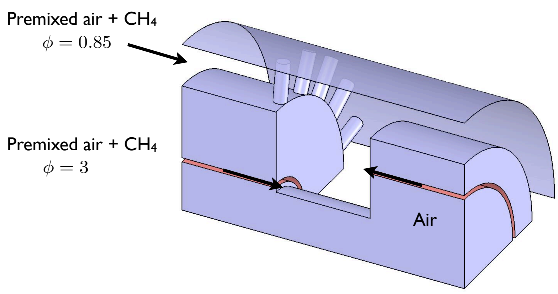

Figure 1: 1/4 sector TVC geometry

The governing equations considered are the unsteady Navier-Stokes equations for a viscous compressible flow. Computations are performed with a parallel solver based on a fourth-order finite volume scheme for cartesian grids. To avoid the appearance of spurious reflections at the open boundaries, the three-dimensional Navier-Stokes characteristic boundary conditions are used to describe non-reflective boundary conditions [6]. The annular TVC is investigated using Immersed Boundaries Methods. The approach employs a ghost-cell method [7–9] for imposing the boundary conditions on the immersed boundaries. Image points are expressed resorting to bilinear or trillinear interpolations. The immersed boundary conditions are described by a Neumann condition for pressure ($\partial P/\partial n = 0$, where n is the direction normal to the immersed surface) and Dirichlet for the no-slip conditions. The wall temperature is calculated by using an adiabatic hypothesis.

Many studies are dedicated to the study of cavities of parallepipedic shape [10]. There is nevertheless a lack on the experiment and simulation of more complex geometry (axisymmetric cavity) where optical diagnostics are challenging. To evaluate the accuracy of the present solver and especially the boundary layers reconstruction with immersed boundaries, a transonic cavity of parallelepiped shape was investigated for which measurements are available [11]. The simulations performed [12] accurately reproduce the flow features (velocity fields and pressure spectra). Various LES investigations were conducted with different subgrid scale strategies. The implicit LES (MiLES approach) leads to the best agreement with experimental data. According to this subgrid scale modeling comparison, turbulence is addressed in the present study with the MiLES technique.

The combustion model is based on detailed tabulated chemistry associated with a presumed PDF (PCM-FPI model [13]). This method relies on the tabulation of chemistry based on laminar premixed flames at different equivalence ratios. The species mass fraction and the reaction rates are then expressed with first and second order moment of two variables: the progress of reaction Y_c and the mixture fraction Z, which stands for the equivalence ratio of a flamelet. The SGS flame properties are modeled with a probability density function (PDF) presumed with a beta-shape. To describe the progress variable and the mixture fraction on the near wall region, the aforementioned IBM formalism is used with a Dirichlet like condition for the progress variable on the wall (zero progress variable is assigned) and Neumann like conditions for the Z variable. For the methane-air combustion, the chemical table has been generated from a collection of freely propagating premixed flamelets computed with the PREMIX software and the detailed GRI-3.0 mechanism.

To reduce computational expense, simulations are performed over a sector spanning $1/4^{th}$ of the domain with the use of axisymmetric boundary conditions. A mesh of 13 millions of cells is then used among them 63% are fluids and others immersed boundaries.

3 Results and discussion

3.1 Cold flow results

The comparison between the experimental measurements and the LES are done in a plane located behind a rod and in a plane between two rods. The mean and fluctuating axial velocity profiles behind rods at different longitudinal locations are shown in Fig. 2 and in Fig. 3, respectively. A fair agreement is observed implying that the main features of the flow are reproduced by the LES. The flow seems uniform across the entire span. Compared to the parallelepiped cavity from the previous section investigated by Forestier et al. [11], the measurements are much more difficult in the actual set-up and we may advance that the uncertainties are thus more important. This may partially explain the slight disagreement found at certain locations between LES and experiment especially in the vicinity of the rods or in the bottom of the cavity. The structure of the flow with the Q-criterion is is shown in Fig. 4. The presence of a double vortex structure is highlighted.

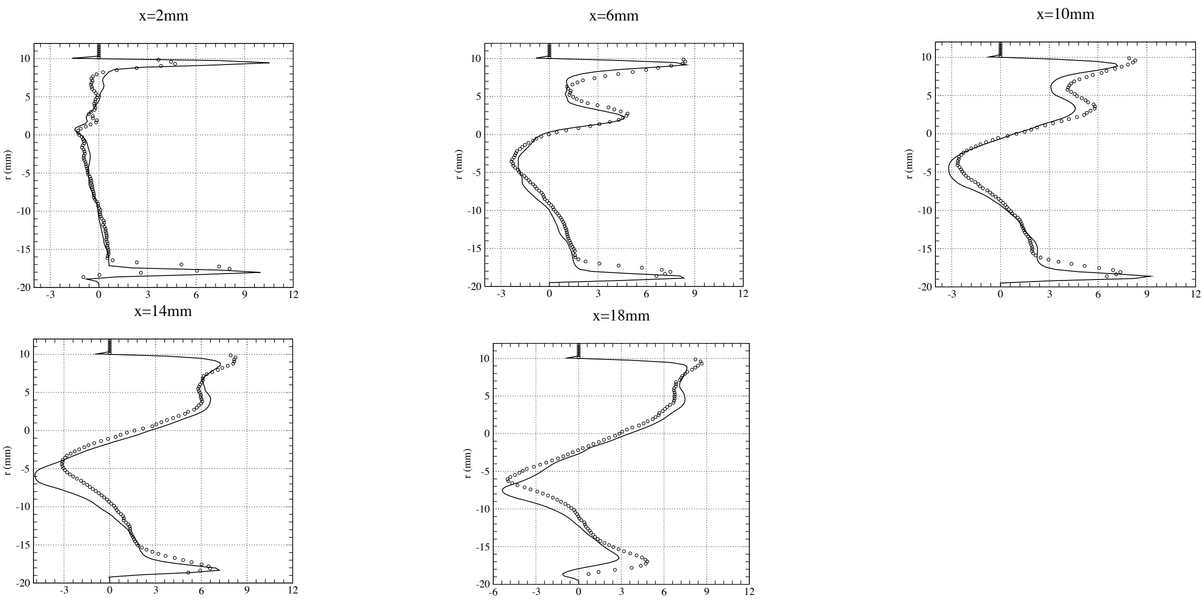

Figure 2: Radial profiles of mean axial velocity behind a rod at different streamwise locations. Symbol: Measurements. Solid line: LES.

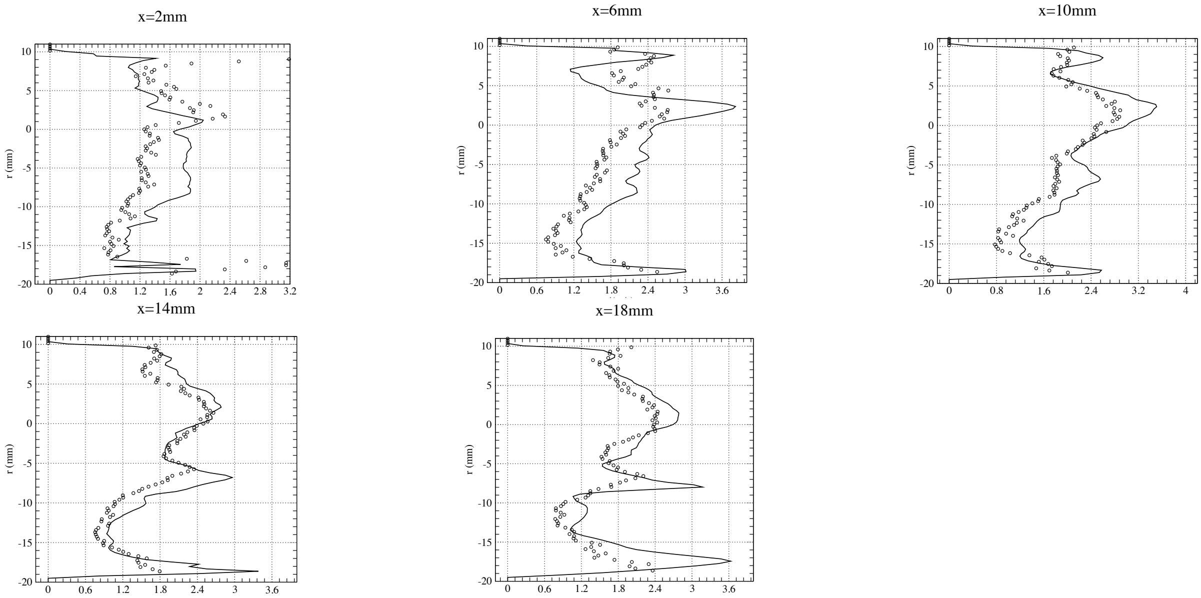

Figure 3: Radial profiles of fluctuating axial velocity behind a rod at different streamwise locations. Symbol: Measurements. Solid line: LES.

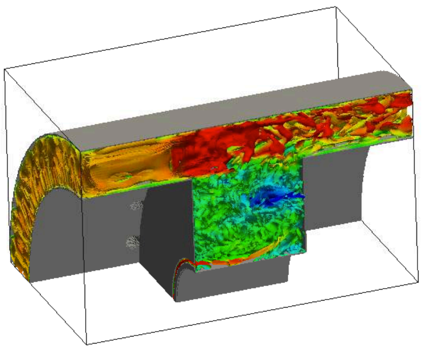

Figure 4: Iso-surfaces of Q-criterion

3.2 Burning flow simulation

The flame structure is shown in Fig. 5 with the iso-surfaces of progress variable. The flame is trapped into the cavity. The injection of air and fuel directly into the cavity provides a stable recirculation zone. The temperature and mixture fraction maps in Fig. 6 reveal that most of the combustion process occurs within the fuel-rich cavity region. The cavity air-injection provides an efficient turbulent mixing of the reactants and compensates the lack of oxidizer which may result from a low fluid exchange between the main lean flow and the cavity region. Hence, the cavity seems to be an interesting option for fuel-lean burning.

4 Summary

Unsteady compressible simulations have been conducted on a TVC combustor for the non-reacting and reacting flows. The essential features of the cold flow are recovered. This demonstrates the ability

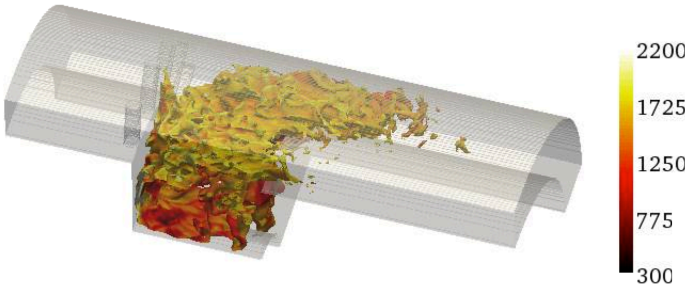

Figure 5: Iso-surfaces of progress variable $Y_c = 0.2$ colored by temperature

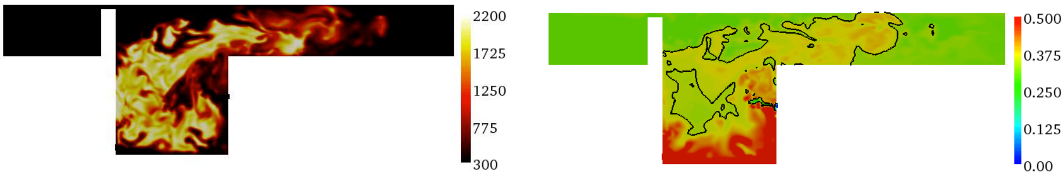

Figure 6: Maps of the instantaneous resolved temperature (left) and of the resolved mixture fraction with the iso-line of stoechiometric mixture fraction (right).

of LES-IBM to describe complex variable density flows. Concerning the combustion modeling, the detailed chemistry approach based on the PCM-FPI model was investigated. The FPI method is based on the assumption that the flamelets are laminar and unstrained. To take into account the aerodynamic strain in the PCM-FPI method, a model to account for the effects of local strain rate will be tested [14]. As flame stabilisation is achieved in a TVC combustor by a cavity which controls the recirculation of the burnt gases, a third model including heat loss and dilution by burnt products will also be implemented [15].

Future work will focus on the description of the trapped vortex complex mechanisms depending on various parameters (structural and aerodynamic) and the comparison with further experimental data which are in progress. On a domain reduced of one fourth, only structures with fourfold symmetry or multiples can be supported. To verify that the assumption that one fourth sector is sufficient, a complete simulation of the combustor will be performed with an unstructured code.

In the final paper the flame dynamics in the cavity and implications for fuel-lean combustion will be discussed.

References

[1] Oberg C. L. (1971). Combustion stabilization with acoustic cavities (acoustic cavity resonators use for suppression of combustion instability modes, determining acoustic impedance and damping). J. Spacecraft and Rockets, Vol. 8, 1220-1225

[2] Hsu K. and Roquemore W. M. (1998). Characteristics of a trapped vortex combustor. J. Propulsion and Power, Vol.14 (1)

[3] Katta V. and Roquemore W. M. (1998). Numerical studies on trapped-vortex concepts for stable combustion. Transactions of the ASME, 120

[4] Bruno C. and Losurdo M. (2007). The Trapped Vortex Combustor : An advanced combustion technology for Aerospace and Gas Turbine Applications, in Advanced Combustion and Aerothermal Technologies, 365-384, NATO Science for Peace and Security, Series C: Environmental Security, Number 6.

[5] Peskin C. S. (1972). Flow patterns around heart valves : a numerical method. J. Comput. Phys., Vol.10, 252-271.

[6] Lodato G., Domingo P. and Vervisch L. (2008). Three-Dimensional boundary conditions for direct and large eddy simulation of compressible viscous flows. J. Comput. Phys., Vol. 227 (10), pp. 5105-5143.

[7] Fadlun E. A. , Verzicco R. , Orlandi P. and Mohd-Yusof J. (2000). Combined Immersed- Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations. J. Comput. Phys., Vol. 161, pp. 35-60.

[8] Iaccarino G. and Verzicco R. (2003). Immersed boundary technique for turbulent flow simulations. Appl. Mech. Rev., Vol. 56 (3), pp. 331-347.

[9] Balaras E. (2004). Modeling complex boundaries using an external force field on fixed Cartesian grids in large eddy simulations. Computers and Fluids, Vol. 33 (3), pp. 375-404.

[10] Gloerfelt X. (2001) Bruit rayonnée par un coulement affleurant une cavit: Simulation aroacoustique directe et application de mthodes intgrales. PhD thesis, Ecole Centrale de Lyon.

[11] Forestier N., Geffroy P. and Jacquin L. (2003). Etude exprimentale des proprits instationnaires dune couche de mlange compressible sur une cavit: cas dune cavit ouverte peu profonde. Rt 22/00153 dafe, ONERA (in French).

[12] Merlin C., Domingo P., Vervisch L. and Lodato G. (2010). An immersed boundary method for Large Eddy Simulation of fully compressible flows: Application to transonic flow. ECCOMAS CFD 2010, Lisbon, Portugal, 14-17 June.

[13] Domingo P., Vervisch L., Veynante D. (2008) Large-Eddy Simulation of a lifted methane jet flame in a vitiated coflow. Combust. Flame, Vol. 152(3): 415-432.

[14] Subramanian V., Domingo P., Vervisch L. (2010). Large-Eddy Simulation of forced ignition of an annular bluff-body burner. Combust. Flame, Vol.157(3): 579-601.

[15] Wang K., Ribert G., Domingo P., Vervisch L. (2010). Self-similar behavior and chemistry tabulation of burnt-gases diluted premixed flamelets including heat-loss. Combust. Theory and Modelling, Vol.14(4): 541-570.