NUMERICAL INVESTIGATION OF CONFINEMENT EFFECTS ON A SUPERCRITICAL LOX-METHANE FLAME

A. Remiddi*, G. Indelicato*, R. Lamioni*, P. E. Lapenna*,, F. Creta***
pasquale.lapenna@uniroma1; lapenna.pe@gmail.com; rachele.lamioni@uniroma1.it
*Department of Mechanical and Aerospace Engineering, Sapienza University, Rome, Italy
**ENEA Research Center Casaccia, Rome, Italy

Abstract

In this contribution we numerically investigate the effects of lateral confinement on a turbulent, non-premixed, LOx-methane jet-flame at supercritical pressure. The simulations are carried out using a flamelet-based numerical framework capable of including both non-adiabatic and real-fluid effects. It is found that the confinement strongly influences the flame flow-field and as the confinement distance increases the average heat flux insisting on the injector face plate also increases in a more than linear fashion.

Introduction

Practical applications requiring high energy densities are often based on chemical energy conversion based on combustion at elevated pressures. These conditions are commonly encountered in the combustion chambers of liquid rocket engines (LRE) which are designed to operate at high pressure in order to achieve elevated performance and reasonably geometrical dimensions [1]. However, two relevant technological drawbacks are caused by high-pressure conditions: (i) the chamber pressure can exceed the critical pressure of the propellants causing mixing and combustion to occur under supercritical conditions under which the gas mixture behave as a complex real fluid rather than an ideal gas complicating substantially the injectors design [2], (ii) increasing the operating pressure implies an almost linear increase in the wall heat transfer at the walls causing a significant work load for both the structure and the cooling system [3]. Both of these drawbacks are currently being tackled by space agencies and related industries in order to develop low-cost and possibly re-usable high-performance LRE coping with shortened development time and restricted budgets. In this context computational fluid dynamics (CFD) is rising as an important tool for the prediction of combustion chambers operations, such as the estimation of wall heat loads, in order to cope with restricted budgets and shortened development time of a LRE.

In this work we present a numerical investigation on the geometrical confinement of a turbulent non-premixed flame emanating from a single element injector in a combustion chamber at supercritical pressure $p_0>p_{cr}$. In particular, we focus on the recently proposed propellant combination consisting in liquid-like oxygen (LOx) and gaseous methane (GCH4) in which the latter has shown some advantages over the standard space propulsion fuels such as RP1 kerosene and liquid hydrogen. The

LOx is injected in a liquid-like or transcritical thermodynamic state [4] characterized by $T_{LOx}<T_{cr}$, being T_{cr} its critical temperature, while GCH4 is injected in a gas-like or supercritical thermodynamic state with $T_{GCH4}<T_{cr}$.

The numerical simulations are carried out using an unsteady Reynolds Averaged Navier Stokes (uRANS) approach coupled with non-adiabatic flamelet-based thermochemical manifolds for both real fluids and combustion modeling. Such an approach, described in detail elsewhere [5,6], allows a consistent and efficient way of modeling turbulent combustion under trans- and supercritical thermodynamic conditions. The aim of the present contribution is to understand the effect of confinement on the jet flame structure and on the resulting heat flux on the injector face plate.

Configuration and numerical formulation

The configuration of interest is a standard shear coaxial injector [1] consisting in a LOx jet surrounded by a faster GCH4 stream. A schematic description of the two-dimensional (2D) axis-symmetric configuration used for the numerical simulations is given in Fig. 1. The lip between the two streams anchor the flame [7] while the recess of the injector enhances the mixing process allowing a fully established turbulent non-premixed flame and a complete pseudo-boiling of the liquid-like LOx-core [4] in the first part of the representative combustion chamber. The coaxial jets and the ensuing inverse diffusion flame create a large recirculation region between the injector faceplate and the upper wall of the chamber. These two walls are considered isothermal and characterized by the same temperature of the fuel stream while the other walls, such as the lip and recessed part of the injector are considered adiabatic.

In the present study a fixed recess length *R* is employed while the lateral confinement *C* of the flame is varied in a range of interest in order to understand its impact on the wall heat flux insisting on the injector faceplate. Moreover two boundary conditions are used for the upper wall, namely a symmetry condition and wall condition. The former conditions assume the presence of a symmetrical injector and therefore is expected to be representative of the injectors clustered in the central part of an injector plate. The latter is expected to be representative of the injectors in the proximity of the LRE combustion chamber lateral walls. Throughout the paper the injector characteristics, such as the velocity and temperature of the LOx and GCH4 streams, have been kept constant together with a chamber pressure that is supercritical for both the fuel and oxidizer.

The numerical approach employed for the present study is based on uRANS approach coupled with a flamelet-based turbulent combustion model. Such an approach is based on the assumption that a three-dimensional turbulent non-premixed flame can be globally treated as an ensemble of precomputed, one-dimensional, laminar, non-premixed flames, commonly denoted as *flamelets*. The flamelet assumption holds as long as the turbulence small scale vortices remain larger than the chemical reaction regions [8]. Such assumption is commonly employed dealing with high pressure combustion and in particular in supercritical pressure environments [5,9].

Operatively the laminar flamelets are obtained as steady state solutions of a

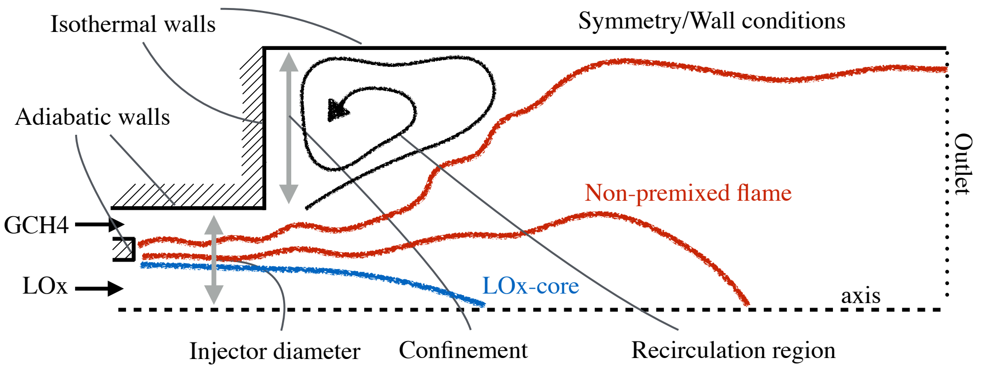

Figure 1. Schematic description of the confined injector configuration.

prototypical 1D laminar problem using a detailed chemical kinetic mechanism and a proper real gas equation of state at a given constant thermodynamic pressure p_0 [5]. Such flamelets or, more generally, *thermodynamic manifolds*, can be expressed, for a non-adiabatic non-premixed flame at supercritical pressure, as $\psi=\psi(Z;\chi_{st},\phi_H,p_0)$ where ψ represents a generic thermodynamic variable, Z is the so-called mixture fraction, χ_{st} is the stoichiometric scalar dissipation rate of the mixture fraction and ϕ_{H} is the enthalpy defect simply defined as the difference between the transported enthalpy and the enthalpy, at the same Z, of an adiabatic mixture [10]. Then the averaged values needed by the uRANS solver are obtained by means of a multi-variate probability density function (p.d.f.) representing, at a sub grid scale (SGS) level, turbulence-chemistry interactions, see for instance [8] for the standard procedure. The resulting averaged values are stored in look-up tables that can be efficiently accessed at run-time by the CFD solver.

The described approach is implemented in the pressure-based uRANS solver *RflameletSmoke* [5,6,11] recently developed in the context of the OpenFOAM and OpenSMOKE [12] open source frameworks. The flameletSmoke solver [12] has been upgraded with the introduction of a modified Pressure Implicit with Splitting of Operators (PISO) [13] algorithm as described in [14] in order to increase the solver stability in the presence of strong thermodynamic non-linearities. Turbulent fluctuations are modeled using the standard k−ε model while wall functions are used at the walls avoiding the boundary layers resolution [15].

Results

The effect of confinement is investigated parametrically, in particular 6 simulations are carried out using the symmetric boundary condition for the upper wall while 3 runs have been performed using the wall condition.

Figure 2 shows the temperature fields for 3 representative confinements of the symmetry case with the superimposition of streamlines and mixture fraction isolines to evidence respectively the recirculation region and flame region. It is evident that as the confinement increases the recirculation region gets wider and

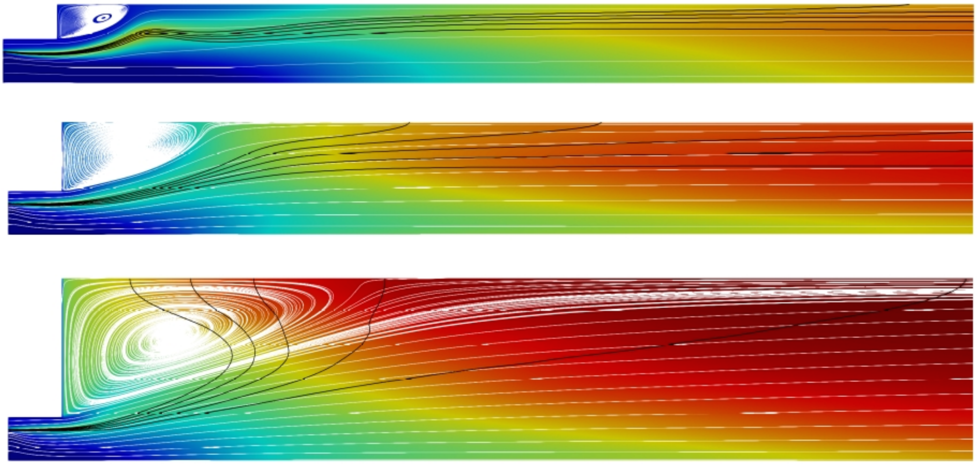

Figure 1. Temperature flow fields for three different confinements in the symmetric boundary condition case, the color code goes from T_{LOx} (blue) to the maximum temperature (red). The superimposed black isoline represent the flame region ($0.2<Z<0.4$) while the white lines represent the flow streamline.

longer. In the presence of a small confinement *C* the flame remain parallel to the upper wall and the recirculation is not able to collect hot gas pockets from the flame. Conversely for large confinement the flame impinges directly on the upper wall and therefore the recirculation region is capable of including some high

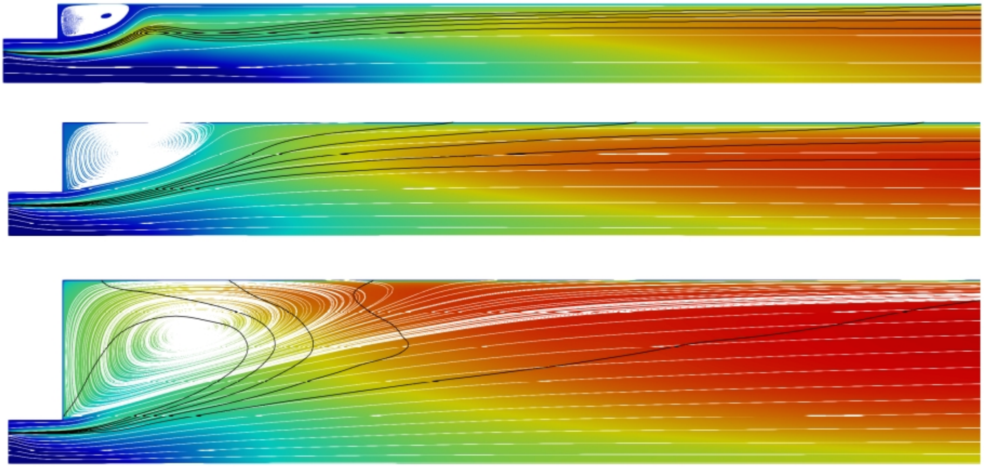

Figure 2. Temperature flow fields for three different confinements in the wall boundary condition case, the color code goes from T_{LOx} (blue) to the maximum temperature (red). The superimposed black isoline represent the flame region ($0.2<Z<0.4$) while the white lines represent the flow streamline.

temperature fluid particle. As a result the injector face-plate is invested by a substantially cold flow for small values of *C* while higher temperature in the recirculation region are achieved in the case of high *C*.

The same phenomenology can be observed, as shown in Fig. 3, if the upper-wall boundary condition is a non-slip, isothermal wall. Moreover, a slightly longer recirculation region is shown for the higher *C* value considered compared to the symmetric case. This leads to a larger portion of flame that is captured by the recirculation bubble. Finally the impact of confinement on the wall heat flux insisting on the injector flace-plate is shown in Fig. 4 where a more than linear trend is observed.

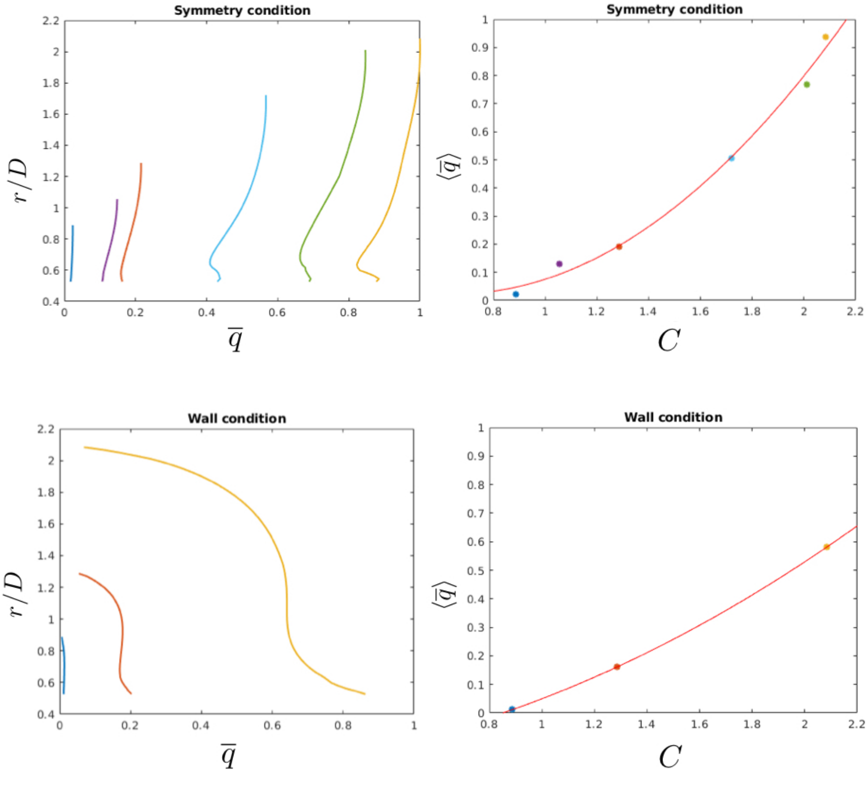

Figure 4. Left panels: radial profile of wall heat flux on the injector face plate for various confinements *C* (same color legend of points in the right panels). Right panels: spatially averaged wall heat flux as a function of *C*.

Conclusion

A numerical investigation on the effects of lateral confinement on a turbulent, non-premixed, LOx-methane jet-flame at supercritical pressure has been carried out using a flamelet-based numerical framework which includes both non-adiabatic and real-fluid effects. It has been found that the confinement geometry strongly influences the ensuing jet-flame flow-field causing warmer recirculation regions in the presence of large *C* values. As a result the heat flux insisting on the injector face plate increases more than linearly with *C*.

Acknowledgements

This work has been partially supported by MIUR and AVIO s.p.a.

References

[1] G. P. Sutton, “History of Liquid Propellant Rocket Engine”, AIAA, Reston, VA, 2005.

[2] V. Yang, “Modeling of supercritical vaporization, mixing, and combustion processes in liquid-fueled propulsion systems”, Proc. Combust. Inst. 28 (2000) 925–942.

[3] M. Pizzarelli, “The status of the research on the heat transfer deterioration in supercritical fluids: A review,” Int. Commun. Heat Mass Transfer 95, 132–138 (2018).

[4] P. E. Lapenna “Characterization of pseudo-boiling in a transcritical nitrogen jet”, Physics of Fluids 30, 077106 (2018)

[5] P. E. Lapenna, G.Indelicato, R. Lamioni, F. Creta, “Modeling the equation of state using a flamelet approach in LRE-like conditions”, Acta Astron. (2019).

[6] P. E. Lapenna, R. Amaduzzi, D. Durigon, G. Indelicato, F. Nasuti, and F. Creta. “Simulation of a single-element GCH4/GOx rocket combustor using a non-adiabatic flamelet method” Joint Propulsion Conference, Cincinnati (2018).

[7] G. Singla, P. Scouflaire, C. Rolon, S. Candel, “Transcritical oxygen/ transcritical or supercritical methane combustion”, Proc. Combust. Inst. 30 (2005) 2921–2928.

[8] T. Poinsot, D. Veynante, “Theoretical and numerical combustion”, RT Edwards Inc., Philadelphia, PA, 2005.

[9] P. E. Lapenna, F. Creta, “Mixing under transcritical conditions: an a-priori study using direct numerical simulation”, Jour. Supercr. Fluids 128 (2017).

[10] B. Marracino, D. Lentini, “Radiation modelling in non luminous non premixed turbulent flames”. Combust. Sci. Tech. 128 (1997), 23-48.

[11] G. Indelicato, P. E. Lapenna, D. Durigon, F. Creta, “Simulations of turbulent combustion and wall heat transfer in single and multi injectors GCH4/GOx rocket combustors”, 8th EUCASS Conf., Madrid (2019)

[12] A.Cuoci, A.Frassoldati, T.Faravelli, E.Ranzi, “Opensmoke++: An object-oriented framework for the numerical modeling of reactive systems with detailed kinetic mechanisms”, Comp. Phys. Commun. 192 (2015) 237–264.

[13] Ferziger, J. H. and Peric, M., Computational methods for fluid dynamics, Springer Science & Business Media, 2012.

[14] Park, T. S. and Kim, S.-K., “A pressure-based algorithm for gaseous hydrogen/liquid oxygen jet flame at supercritical pressure,” Numerical Heat Transfer, Part A: Applications, Vol. 67, (2015)

[15] D.C. Wilcox, “Turbulence modeling for CFD”. DCW industries, 1993.