DNS of a diffusional jet flame in turbulent cross-flow using a low Mach number solver

Z. H. Zhang[1], A. Abdelsamie[2], D. Thévenin [2], K. H. Luo [3]

[1]Center for Combustion Energy, Tsinghua University

Beijing, 100084, China

[2] Laboratory of Fluid Dynamics and Technical Flows, University of Magdeburg "Otto von Guericke"

Universitätsplatz 2, Magdeburg, 39106, Germany

[3] Department of Mechanical Engineering, University College London

Torrington Place, London WC1E 7JE, United Kingdom

Abstract

Understanding of flame anchoring in a jet in crossflow (JICF) configuration is vital to the design of fuel injectors in combustion devices. The present study numerically investigates a hydrogen rich jet injecting perpendicularly into hot vitiated crossflow using direct numerical simulation (DNS). Development of the reacting flow field and flame shape along the jet trajectory is scrutinised. The flame is found to be anchored around the jet exit, and downstream only on the windward side. Heat release rate and Chemical Explosive Mode Analysis (CEMA) are used to identify combustion modes. Distinct from flames stabilizing in non-vitiated crossflow where combustion is mainly partially premixed, diffusion flame is significant under the current condition, though some premixed or partially premixed regions are found on the leeward side of the jet due to large scale turbulent mixing.

1 Introduction

Jet in cross flow (JICF), also referred to as transverse jet, is a configuration where a jet injects perpendicularly into a crossflowing fluid and interacts with it [1, 2]. JICF is common in both industrial devices and the natural world. For industrial applications, examples are film cooling on turbine blades, fuel injection in combustors, and air injection in staged combustors. A fascinating example of JICF in nature is the jet plume from volcano transported by the crosswind. Due to its broad applicability, the JICF configuration has been studied as a topic of significant technological interest over the past few decades. According to whether there is a chemical reaction during the subsequent process, jets in crossflow can be categorized into non-reacting JICF and reacting JICF.

Many early research on reacting JICF focus on developing empirical formula of the jet trajectory. Later on, flame structure, flame stabilization and emissions in different conditions attract more research attention [1]. For instance, Weinzierl et al. [3] used the LES method together with a NO_x model to simulate a reacting JICF and predicted the NO_x emission from an axial-staged combustion system. Pinchak et al. [4] experimentally investigated the effects of jet equivalence ratio, momentum flux ratio and jet geometry on the stability characteristics of a premixed ethylene-air jet injection to a vitiated crossflow. It is reported that at their experiment conditions the flame stabilization process is flame propagation controlled rather than autoigniton assisted. Wagner et al. [5] also studied the flame stabilization behavior of a premixed ethylene-air jet injected transversely to a vitiated crossflow of lean combustion products. In their experiments, it was observed that the windward flame branch was unsteady and showed both attached and lifted flame behavior, and chemiluminescence images of lifted flame showed that both propagation and auto-ignition contributed to flame anchoring.

Among numerous studies of reacting JICF, we refer here to some of those conducted by the direct numerical simulation (DNS) method. DNS uniquely provides full scale information regarding instantaneous reacting flow fields and flame structures. Grout et al. [6] reported a three-dimensional DNS of a nitrogen-diluted hydrogen transverse jet injecting into a cross-flow of heated air. Results indicated that the flame stabilized in a low velocity region only on the jet lee side. Lyra et al. [7] reported a DNS study of reacting JICF together with an experiment, where a hydrogen rich jet injected transversely to a turbulent vitiated crossflow of lean methane combustion products. Different from the previous simulations conducted by the same code [6], the flame structure of non-premixed fuel jet into vitiated crossflow was uniformly stabilized near to the jet exit around its entire circumference.

In the DNS studies mentioned above, the Navier-Stokes equations were solved in their compressible formulation. Since most reacting JICF in previous studies take place at low Mach numbers, Ma<<1, a significant speedup can be obtained by using a low-Mach-number solver compared to a fully compressible one. In the present study, three-dimensional DNS of reacting JICF is conducted using a low-Mach-number solver [8]. The jet and crossflow conditions are set similar to [7] so that the results can be qualitatively compared between the two methods. Furthermore, DNS databases of reacting JICF are still rare, so the present study also enriches the databases of reacting JICF and can serve as validation data for LES and RANS models.

2 Numerical setup

2.1 Description of the physical problem

The jet composition is 70% H2 and 30% He by volume and the crossflow is comprised of combustion products of lean methane. The temperatures of the jet and crossflow are 407 K and 1640 K, respectively. The velocity of the jet is 291 m/s and the maximum velocity of the crossflow is 59 m/s, resulting in a momentum flux ratio of J = 9.

2.2 Algorithm

Corresponding author. Fax: +44 (0)20 7388 0180
E-mail address: K.Luo@ucl.ac.uk

The low-Mach-number Navier-Stokes system coupled with detailed physicochemical models is solved with an in-house code named DINO, which is a high-order finite-difference code for low-Mach-number reactive flows, and has been used and validated in a variety of applications [8]. A six-order centered explicit scheme is used to compute the spatial derivatives. An explicit fourth-order Runge-Kutta time integrator is employed for temporal integration. Chemical reactions are solved by Cantera [9]. The mechanism by Li et al. [10], which consists of 13 species and 23 reactions, is used to represent the hydrogen-air chemical kinetics.

2.3 Computational domain and boundary conditions

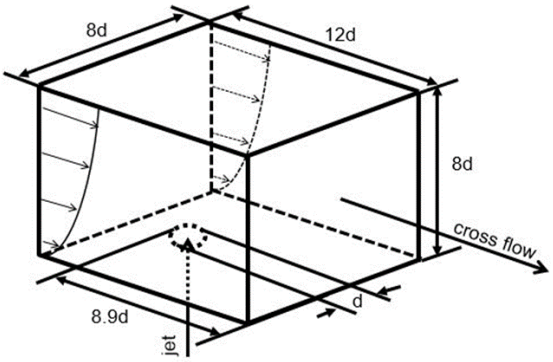

Fig.1: Schematic diagram of the 3-dimensional (3D) simulation domain

DNS is conducted to study a round jet injecting into a turbulent crossflow. The computational domain is shown in Fig. 1, where the domain lengths in streamwise, spanwise and wall-normal direction are 12d, 8d, and 8d, respectively, where d is the jet diameter. A uniform grid comprised of $900 \times 600 \times 600$ points is used. A spatial resolution of 20 μm is reached to ensure that there are sufficient near-wall grid points to resolve the boundary layer. A mean velocity profile from an experimental result is used to represent the velocity profile for a fully developed turbulent flow in a channel. Turbulent fluctuations are then superimposed on the mean velocity field to generalize a turbulent channel flow. During the simulation this pre-calculated turbulent velocity field is used as the inflow boundary condition. In the streamwise direction, the turbulent flow enters the domain from an inflow boundary and exits the domain from an outlet boundary where pressure is kept as a constant. In the spanwise direction, the periodic boundary conditions are applied. On the top, a symmetry boundary condition is used. At the lower border, a mixed boundary condition is used. For the jet exit area, the velocity is assumed in the wall normal direction. For the area other than the jet exit, the boundary is assumed as no-slip isothermal wall, velocity on the wall is therefore zero, and a hyperbolic tangent function is used to smooth the profile between the isothermal wall boundary and jet exit boundary. Simulation time lasts longer than 2τ to ensure the simulation has reached a statistically stabilized condition, where flow through time τ is defined as streamwise length divided by crossflow velocity.

3 Results and discussion

3.1 Mean and instantaneous reacting flow fields

JICF flow fields are characterized by a jet injecting transverse to the crossflow direction. After ejecting from the exit, the jet is deflected and curves in the direction of free stream, as shown by the jet trajectory (black line) in Fig. 2. The jet trajectory is defined as the stream line starting from the center of the jet exit.

In order to demonstrate how the flow field and flame structure develop along the jet trajectory, properties on five different cross-sections perpendicular to the jet trajectory are plotted in Fig 3. Distance between the center of jet exit and the intersections of jet trajectory and the slices ranges from 0.2d to 6d (0.2d, 1d, 2d, 4d, 6d). Locations of the cross-sections are shown by dashed white lines in Fig. 2. Near the jet exit (0.2d slices), the momentum of jet does not have enough time to be changed by the impact of crossflow. The flow field near this area is very much like a flow around a circular cylinder. At a little distance from the entry point, the originally round jet cross-section becomes oval, as shown by the velocity contour on the 1d slice. This can be easily understood if we recall the flow around a circular cylinder, where there are stagnation points and high pressure zones upstream and downstream. Lower pressures on both sides attract the jet to expand at its lateral edges. The shearing between the jet's lateral edges and crossflow results a kidney shaped jet cross-section. The development of this kidney shaped jet cross section can be seen from the 1d slice to the 4d slice. As the jet trajectory further bends, the jet is flanked by shading vortice, those vortice on both sides then develop into a counter rotating vortex pair (CRVP), as shown by the development of vorticity magnitude in Fig. 3. CRVP is the most prominent vortex structures of JICF and is the primary mechanism for fast mixing between the fuel jet and crossflow [2].

It is also interesting to note that the reaction zones and the high temperature areas do not coincide. The highest temperature exists in the recirculation zones on the jet lee side. If we compare the development of the flame shape and the fuel (H2), it seems that the flame favors the place of proper equivalence ratio. This suggests that the combustion is in a diffusion flame mode, as expected, this point will be further explained in the next section.

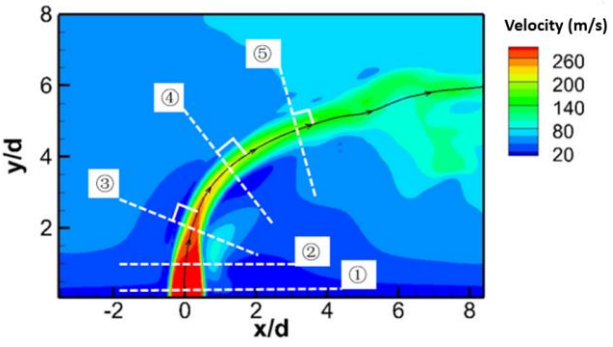

Fig.2: Time-averaged velocity magnitude on the spanwise midplane. Jet trajectory is shown by the black line. Positions of cross-sections are shown by dashed white lines. Intersections of cross-sections and jet trajectory locate at a distance of 0.2d, 1d, 2d, 4d and 6d from the jet exit.

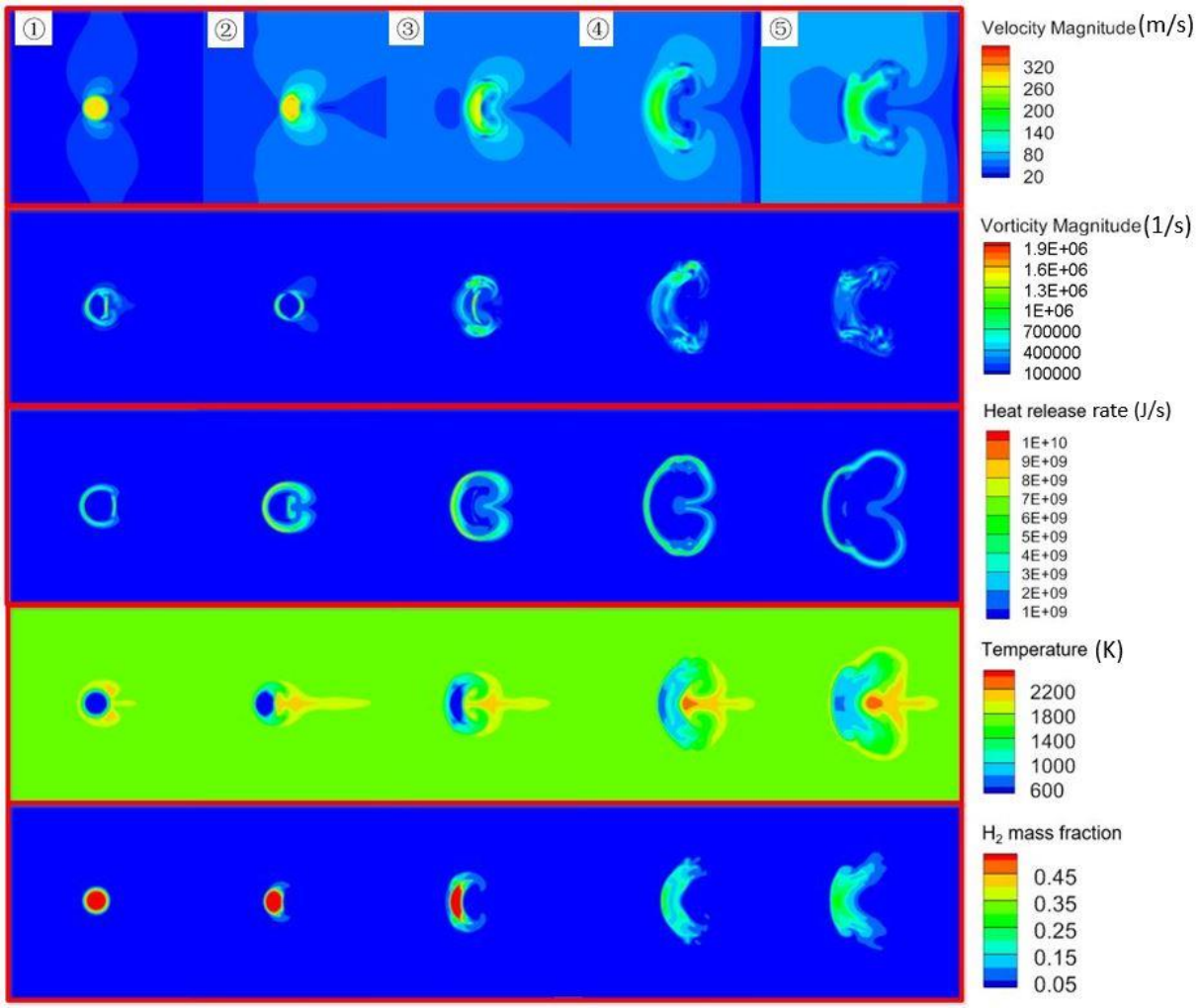

Fig.3: Velocity magnitude, vorticity magnitude, heat release rate, temperature and H_2 mass fraction distributions in different cross-sections.

3.2 Flame stabilization and combustion mode

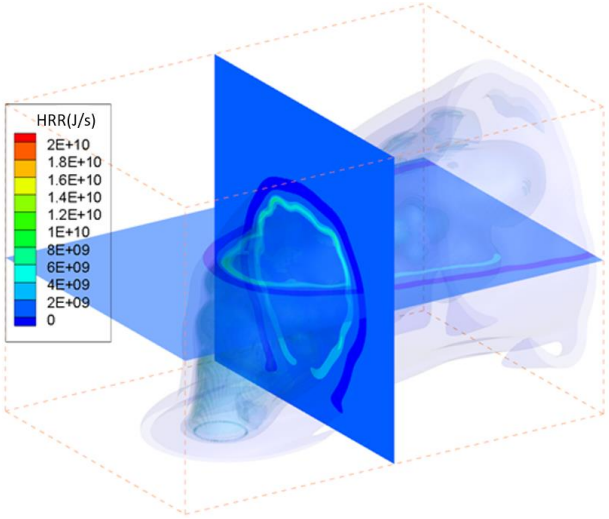

Fig.4: Volume rendering of the instantaneous heat release rate.

In order to ensure flashback safety of the fuel injection system, the flame should must not be anchored near the injector. Therefore, understanding the mechanism behind flame stabilization of reacting JICF is important to design of fuel injectors. An instantaneous snap-shot of the heat release rate is shown in Fig. 4. Contours of heat release rate are used to present reaction zones. Near the exit, the position of the flame is around the jet, and downstream, combustion takes place on the windward side of the jet. This result is consistent with that of [7].

In order to further understand the stabilization mechanism and to identify the burning mode of the jet flame, heat release rate and Chemical Explosive Mode Analysis (CEMA) [11] are used to analyze the flame structure. The reaction zone is delineated by a contour line of which the value is 20% of the maximum heat release rate, as shown by the black line in Fig. 5. The CEMA method is based on an evaluation of the largest non-conservative eigenvalues of the chemical Jacobian, and the visualization the eigenvalues is defined in the following equation,

$$\gamma_e = \text{sign}(\text{Re}(\lambda_e)) \cdot \log_{10}(1 + |\text{Re}(\lambda_e)|) \quad (1)$$

where 'sign' is the sign function and 'Re' is the real part of a complex number. The first thing we need to keep in mind is that the CEMA analysis does not take into account transport. A positive value of γ_e indicates the mixture is sufficient to support combustion under adiabatic conditions. A negative value means the mixture is chemically non-explosive in isolation. Diffusion flames can be identified by peaks of reaction rates with the non-explosive mode [12]. Figure 5 shows that for most of the

reaction zones, the flame is in diffusion mode. Near the jet exit the flame immediately forms around the nozzle, suggesting that the reaction happens very close to the exit as fuel mixes with the crossflow. It is interesting to note that on the windward side of the jet, explosive mode mixture is also found within reaction zones. Zero-crossing of γ_e identifies premixed flame where chemically explosive reactants burn into non-explosive combustion products. The presence of premixed flames can be explained by the fast mixing of fuel and oxidant due to turbulence. The vorticity magnitude in this region is found to be the highest of the whole domain, resulting from the strong shearing between jet and crossflow.

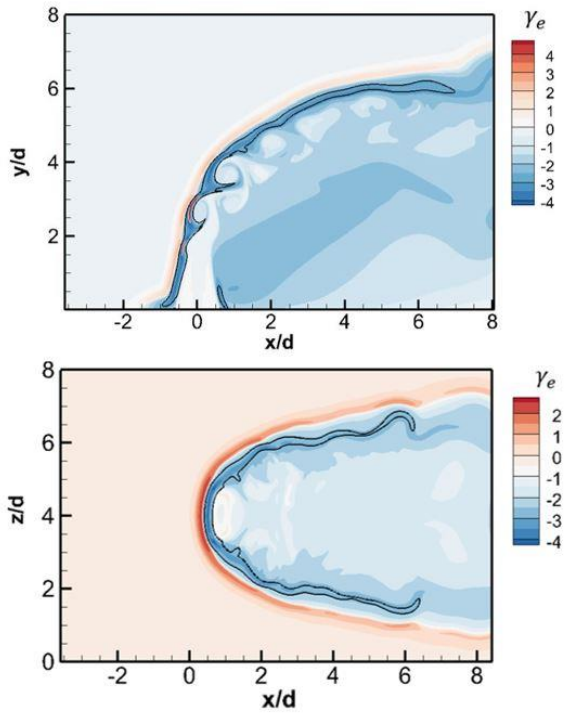

Fig.5: Chemical explosive mode at the spanwise and wall-normal midplanes. The reaction zone is delineated by black iso-contour of heat release rate at a value of 20% of the maximum.

4 Conclusions

A reacting hydrogen jet in transverse vitiated crossflow was numerically investigated using DNS. The reactive Navier-Stokes equations were solved in the low-Mach-number regime. The flow field and flame structure were analysed along the jet trajectory. Heat release rate and chemical explosive mode analysis were employed to identify the combustion mode. The main results can be summarized as follows:

• Along the jet trajectory, the cross-section of flame/fuel/velocity transitions from a circle to an ellipse and then to a kidney-like shape. Further downstream, structures are dominated by a counter-rotating vortex pair.

• The flame is anchored around the jet exit, downstream combustion happens only on the windward side. The highest temperature locates in the recirculation zone on the jet lee side.

• Diffusion flame is the predominant combustion mode, though premixed flame mode is found in the windward shear layer region.

5 Acknowledgment

Support from the UK Engineering and Physical Sciences Research Council under the project “UK Consortium on Mesoscale Engineering Sciences (UKCOMES)” (Grant No. EP/R029598/1) is gratefully acknowledged.

Financial support from the program of China Scholarships Council NO.201706210263 is acknowledged

References

[1] K. Mahesh, in: S.H. Davis, P. Moin (Eds.) Annual Review of Fluid Mechanics, Vol 45, Annual Reviews, Palo Alto, 2013, pp. 379-407.

[2] A.R. Karagozian, Prog. Energ. Combust. 36 (2010) 531-553.

[3] J. Weinzierl, M. Kolb, D. Ahrens, C. Hirsch, T. Sattelmayer, J. Eng. Gas. Turbines Power-Trans. ASME, 139 (2017) 7.

[4] M.D. Pinchak, V.G. Shaw, E.J. Gutmark, Combust. Flame 191 (2018) 353-367.

[5] J.A. Wagner, S.W. Grib, M.W. Renfro, B.M. Cetegen, Combust. Flame 162 (2015) 3711-3727.

[6] R.W. Grout, A. Gruber, C.S. Yoo, J.H. Chen, Proc. Combust. Inst. 33 (2011) 1629-1637.

[7] S. Lyra, B. Wilde, H. Kolla, J.M. Seitzman, T.C. Lieuwen, J.H. Chen, Combust. Flame 162 (2015) 1234-1248.

[8] A. Abdelsamie, G. Fru, T. Oster, F. Dietzsch, G. Janiga, D. Thévenin, Comput. Fluids 131 (2016) 123-141.

[9] D.G. Goodwin, Moffat, H.K., Speth, R.L., available at< https://cantera.org/ >

[10] J. Li, Z. Zhao, A. Kazakov, F.L. Dryer, Int. J. Chem. Kinet. 36 (2010) 566–575.

[11] L.U. T. F., C.S. Yoo, J.H. Chen, C.K. Law, J. Fluid Mech. 652 (2010) 45-64.

[12] D. Messig, M. Vascellari, C. Hasse, Combust. Theory Model. 21 (2017) 1-22.