Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 834–841

Energy Procedia

www.elsevier.com/locate/procedia

72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6–8 September 2017, Lecce, Italy

Methane swirl-stabilized lean burn flames: assessment of scale-resolving simulations

Daniele Pampaloni[a,*], Davide Bertini[a], Stefano Puggelli[a], Lorenzo Mazzei[a], Antonio Andreini[a]

[a] *Department of Industrial Engineering, University of Florence, Via di S. Marta, 3, Firenze 50139, Italy*

Abstract

The reliable prediction of the turbulent combustion process in lean flames is of paramount importance in the design of gas turbine combustors. The present work presents an assessment of the capabilities of Flamelet Generated Manifold (FGM) in the framework of Reynolds-Averaged Navier-Stokes (RANS) and Large-Eddy Simulation (LES) At this purpose the TECFLAM swirl burner consisting of a strongly swirling, unconfined natural gas flame was chosen. Results highlight that RANS-FGM succeeds in predicting the main characteristics of the reacting flow field and species concentrations. However, only LES is capable of reproducing the actual turbulent mixing between swirling flow and co-flow, thus leading to appreciable enhancements with respect to RANS results.

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association

Keywords: Gas turbine; Combustion; Swirl flame; Methane; CFD; LES; FGM.

---

1. Introduction

Lean premixed combustion is by now a reference technology in heavy-duty gas turbine framework, given its ability in reducing NOx when compared to non-premixed systems. Fuel and air are directly premixed within the injector to avoid the creation of non-uniform near-stoichiometric mixture inside the combustor and allowing to local control the flame temperature. Particular attention has to be therefore devoted to the preparation of the fuel-air mixture to avoid also the appeareance of combustion instabilities that can lead to flashback or lean blow-out due to local flame quenching. In this framework, considering the complexity of the physical phenomena involved, Computational Fluid Dynamics (CFD) has become a powerful tool in the design of the combustion chamber since it is able to provide detailed information about the reacting flow-field. The numerical simulation of gas turbine involves the interaction of many complex physical processes that strongly affect the prediction accuracy, such as turbulent mixing and chemical

---

* Corresponding author. Tel.: +39-055-275-8772.
E-mail address: daniele.pampaloni@htc.de.unifi.it

1876-6102
Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.274

reactions. In an industrial framework, RANS (Reynolds Averaged Navier Stokes) approaches, where only the mean flow is solved and turbulence effects are introduced by means of ad hoc models, represents a standard tool. However, due to the relevance of turbulent mixing processes, RANS techniques are often insufficient or not completely able to properly characterize the complexity of such devices [1].

Therefore, Scale-Resolving Simulations such as Large-Eddy Simulation (LES) and hybrid RANS-LES models (e.g. Detached Eddy Simulation (DES) or Scale Adaptive Simulation (SAS)) are achieving a growing attention and they have been already widely applied for the simulation of both premixed and non-premixed gaseous flames (see [1] and references therein).

As far as the combustion modelling is concerned, to reproduce the complex topology and the stabilization mechanism shown by the flame in lean combustors as well as to properly characterize pollutant emission, approaches characterized by a detailed description of the kinetic mechanisms are required. To this end, the Flamelet Generated Manifold (FGM) model, where a pre-computed laminar flamelet solution is integrated through a pre-defined probability density function (PDF) to account for turbulence effects, has proved to be capable of accurately describing the flame evolution since it can locally consider finite rate and non-equilibrium effects [2, 3].

In the present work, a Large-Eddy Simulation of the unconfined TECFLAM swirl burner, experimentally studied at the Institute for Energy and Power-plant Technology at Darmstadt University of Technology [4, 5], has been performed using Flamelet Generated Manifold for combustion modelling. The commercial code ANSYS Fluent v18.0 has been employed in this study. The burner operates at ambient pressure and it has been chosen since detailed measurements of velocity, temperature and main species fields are available on radial planes at several axial distances from the inlet. Obtained results in LES framework have been compared with the available experimental data and with steady state solutions in order to show the improvements that can be obtained with scale-resolving approaches.

Nomenclature

D_{cb}	Diameter of the center-body
D_{ext}	External diameter of the annular duct
S	Swirl Number
T	Temperature
ϕ	Equivalence Ratio
CFD	Computational Fluid Dynamics
FGM	Flamelet Generated Manifold
LES	Large-Eddy Simulation
PDF	Probability Density Function
RANS	Reynolds Averaged Navier Stokes

2. Modelling

LES simulations performed in this work are based on a density-weighted localized spatial filter applied to the governing equations in order to separate the large-scale and small-scale turbulence. The unclosed sub-grid stress tensor, which appears in the filtered system, has been closed through a dynamic Smagorinsky-Lilly model [7]. In order to stabilize the model, the modification proposed in [8] is applied. Regarding combustion modelling the Flamelet Generated Manifold (FGM) model has been considered. In FGM a multi-dimensional manifold is generated solving a set of laminar adiabatic one-dimensional flamelets and describing the chemical state and reaction progress space only as function of few control variables. In the present work the mixture fraction Z and the normalized progress variable c have been considered. A premixed flamelet behavior has been considered and flamelet equations have been solved using the dedicated solver integrated in ANSYS Fluent and exploiting the Gas Research Institute reaction mechanism *GRImech 3.0* for CH_4, with 325 reactions and 53 species. The flame turbulence interaction has been accounted for through a Presumed Probability Density Function (PDF) approach, where variances of progress variable and mixture fraction have been modelled respectively through a transport equation and an algebraic closure. Finally, the source term of progress variable has been modelled in the present work using a finite rate formulation.

3. Description of the test case

The TECFLAM swirl burner has been studied experimentally at the Institute for Energy and Powerplant Technology at Darmstadt University of Technology [4] [5] and consists of a swirled, unconfined gaseous flame. A schematic of the burner is shown in Figure 1. The flame burns a natural gas/air mixture at ambient temperature with an equivalence ratio $\phi = 0.83$, leaving the nozzle through an annulus with an outer diameter of $D_{ext} = 60mm$, which surrounds a water-cooled center-body with a diameter of $D_{cb} = 30mm$. Homogeneous mixing is ensured by the injection of natural gas to the air flow far upstream of the nozzle exit and has been confirmed experimentally by Schneider et al. [4]. Upstream of the burner exit, swirl was generated by a moveable block geometry, that allows for tuning of the swirl intensity: in the investigated configuration the Swirl Number, defined following Gupta et al. [9], was set to $S = 0.7$, close to the stability limit of the flame. The operating conditions ensure that the stratified mixture burns in a predominantly fuel-lean premixed mode [4] [5], with a thermal power of $30kW$. Detailed experimental measurements of velocity, temperature and species profiles are available at several axial positions above the burner exit.

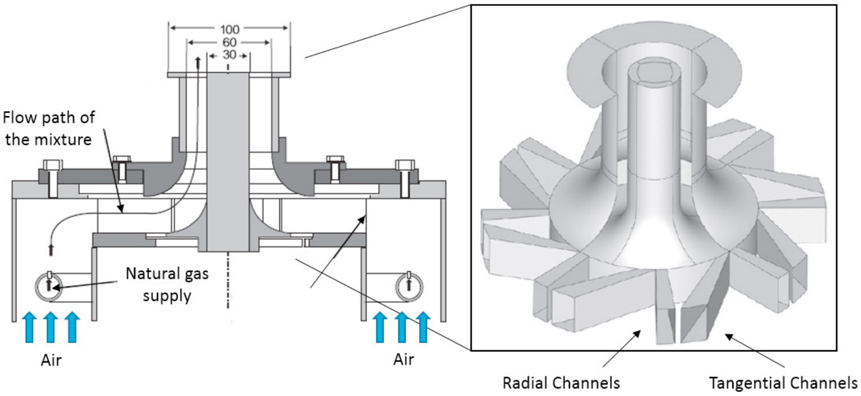

Fig. 1. Schematic of the TECFLAM premixed swirl burner with details of the swirler geometry (adapted from [10])

4. Numerical setup

The computational domain exploited in LES calculation consists of a cylindrical combustion chamber with a diameter of 660 mm and a lenght of 600 mm. To allow for a fully developed turbulent flow the upstream geometry, including the channels of the swirler device was included in the computational domain. The simulations reported in this paper were performed with the commercial code ANSYS Fluent v18.0, on the computational grid depicted in Figure 2. The grid was generated in ANSYS Meshing and consists of about 5M elements (tetrahedrons with a layer of 5 prisms close to the wall) with a size of 0.9 mm at the burner exit. Regarding the impact of mesh sizing on the results, the criterion proposed by Pope [6] was calculated and resulted satisfied for the LES simulation.

At the inlets of the domain, being unknown the exact mass flow partitioning through the radial and tangential channels during the experiments, but only the overall mixture mass flow, several preliminary RANS simulations have been performed exploiting different mass flow partitioning. A mass flow partitioning of 48% through the tangential channels and the residual 52% through the radial channels was found to reproduce the experimental measurements most accurately. No turbulence generator has been employed at the inlet boundaries because the main turbulent structures are generated when crossing the swirler geometry. A coflow of pure air surrounding the swirler device has been also considered while a uniform static pressure value has been imposed at the outlet and on the lateral walls of the combustion chamber. All the other boundaries have been considered as smooth, no slip and adiabatic walls.

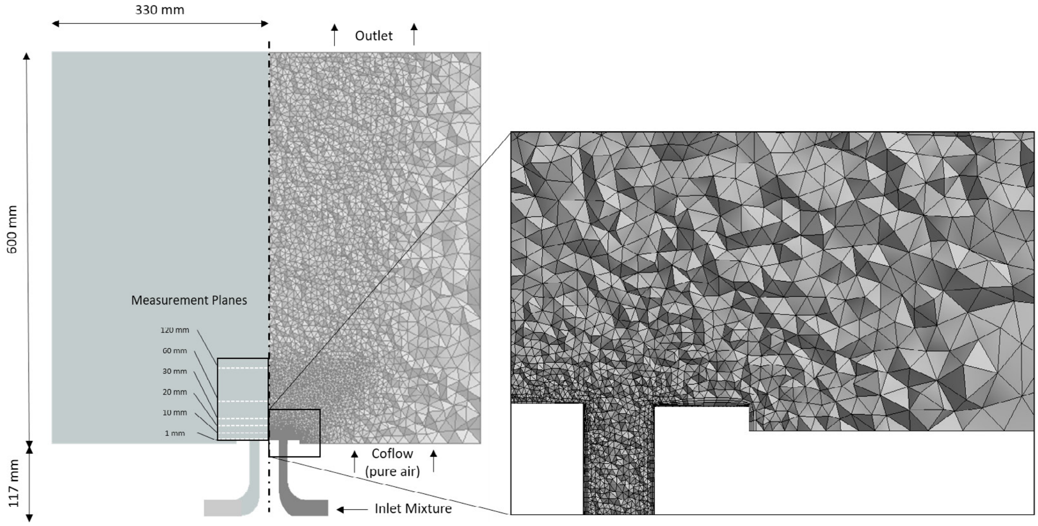

Fig. 2. Computational grid for LES simulation and measurement planes

A time step of $\Delta t = 1.5e-5s$ has been chosen so as to ensure a control on Courant number in the region of the swirler and appropriately resolve the turbulent flow structures.

In terms of numerical schemes and solution algorithms, a PISO algorithm together with second order upwind schemes for spatial discretization and a bounded second order implicit formulation for time have been used. Finally, convergence at each time step has been evaluated considering a reduction of two orders of magnitude for residuals of the continuity equation requiring at least 10 iterations per time step.

5. Results

Preliminary RANS simulations have been performed in order to investigate the main flow features, choose the optimal numerical setup for LES simulation, both in terms of model settings and time/space scales, and assess the capabilities of LES in the predicition of methane swirl-stabilized turbulent flames. Figure 3 shows the swirled flow field typical of these configurations, in terms of axial velocity (left) and temperature (right) fields. A premixed air/CH_4 jet exits from the radial swirler with a high axial velocity core. The swirling component spreads the jet outward, generating an inner recirculation zone that acts to stabilize the flame. Downstream of the jet exit plane the mixture is ignitied by recirculating exhaust hot gases. The rise of gas temperature shown in Figure 3 leads to a prompt acceleration of the flow.

Simulations on coarse and fine computational grids respectively of 0.5M and 1.0M elements stated as the solution is mesh insensitive. This is confirmed by Figure 4-5, where respectively axial velocity and temperature fields are compared with experimental data. As shown in Figure 4 the velocity field at the first measurement plane (x=1 mm) follows experimental profile, highlighting how the mass flow is properly distributed in the radial direction along the feeding duct radius. In the downstream measurement planes jet spreading is overestimated causing a weaker recirculating zone and a wrong prediction of the temperature field, as reported in Figure 5. The disagreement at higher radii can be ascribed to the well-known inability of steady simulations in modeling the mixing in shear stress zones, where recirculating vortices are generated entraining fresh air within the reacting zone. In Figure 6 profiles of the main species molar fractions are reported at two different axial planes. At plane x=10 mm FGM model predicts accurately the species concentrations. However moving downstream (x=30 mm) the wrong jet spreading negatively impacts also on product species profiles. In addition, the underprediction of mixing in RANS framework leads to reduced burning rate, resulting in higher peak values of reactant species, i.e. O_2 and CH_4.

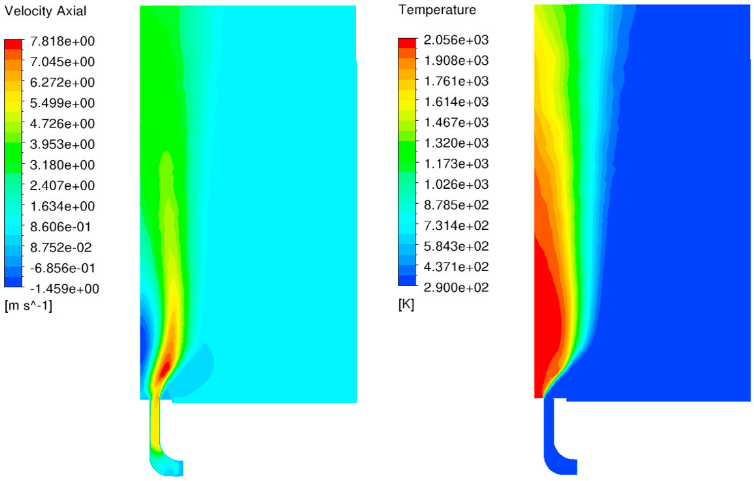

Fig. 3. Axial velocity (left) and temperature (right) distributions with RANS.

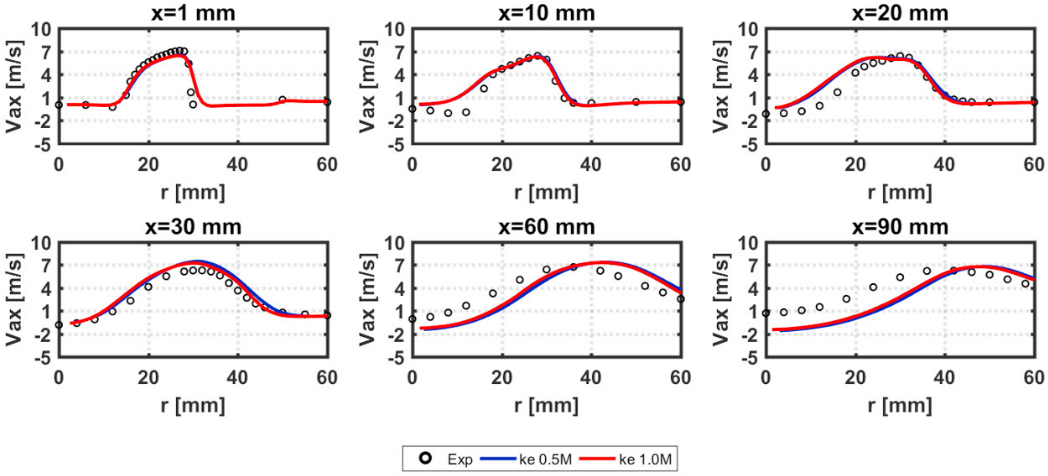

Fig. 4. Radial profiles of axial velocity at several axial plane with RANS.

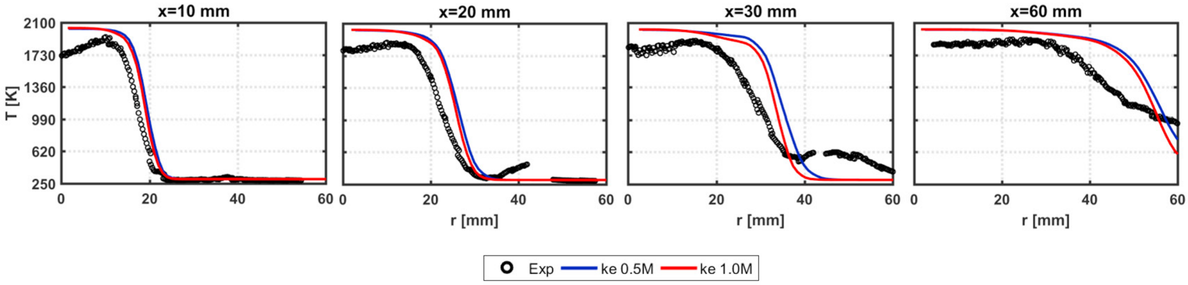

Fig. 5. Radial profiles of temperature at several axial plane with RANS.

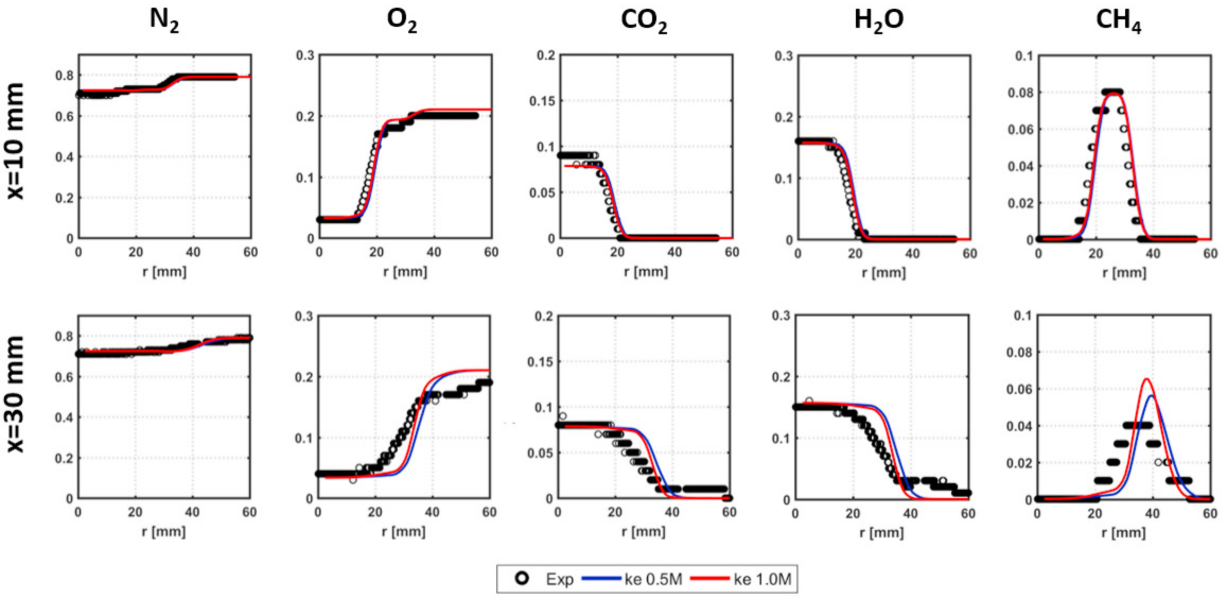

Fig. 6. Radial profiles of main species molar fraction at two axial plane with RANS.

RANS results highlight the need to move toward scale-resolving simulations in order to resolve the large energy-carrying vortices and predict a more physical mixing and jet spreading. This last statement is confirmed by Figure 7 that shows as the velocity profiles are physically reproduced in LES framework and the jet spreading is properly catched. Consequently, the temperature field obtained in LES has a trend more similar to experimental data at higher radii, as reported in Figure 8. Now, mixing acts smoothing the high temperature gradients along the shear zone, especially at the downstream measurement planes. Instead, closer to the axis, the temperature drop observed in experimental data is not captured by the simulation, due to the lack of the cooling effect of the bluff body surfaces. In fact, being unknown the experimental temperature distribution of the bluff body surfaces, an adiabatic wall tretment has been set. In the downstream region, where the influence of bluff body is negligible, the results show a better

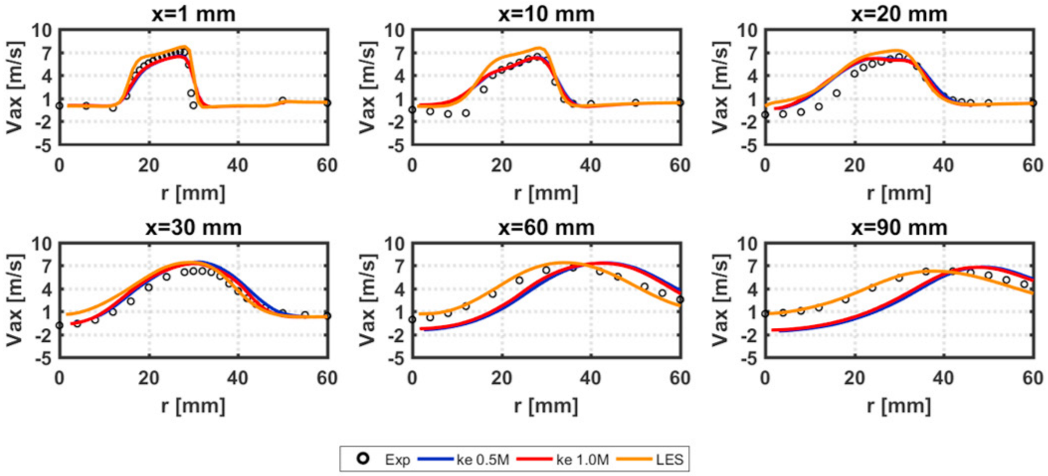

Fig. 7. Comparison of radial profiles of axial velocity at several axial plane between LES and RANS.

agreement. The discrepancy between RANS and LES simulations above the bluff body (i.e. x=10 mm) is caused by the flame anchoring: as shown in Figure 9 in LES simulation the flame anchors at a more upstream location within the premixer duct with respect to RANS (Figure 3). In Figure 10 species molar fractions obtained by LES simulation are

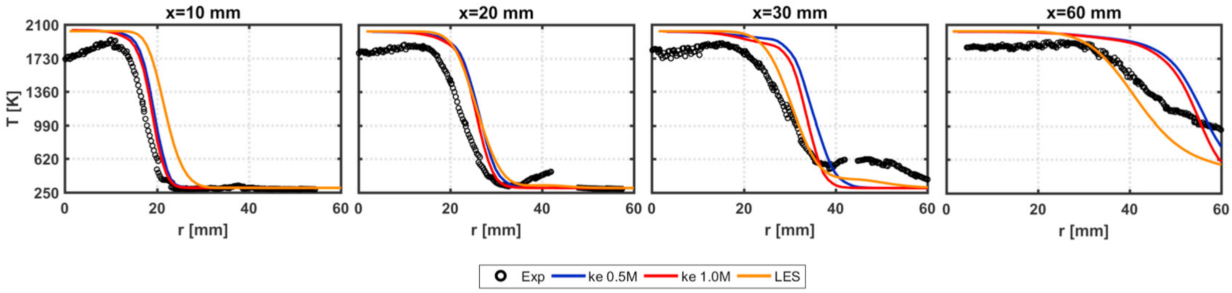

Fig. 8. Comparison of radial profiles of temperature at several axial plane between LES and RANS.

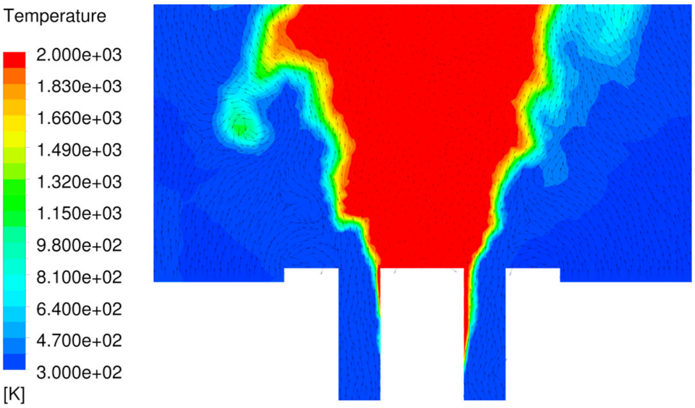

Fig. 9. Instantaneous velocity vectors superimposed on temperature contours in LES.

reported. Results show a general improvement in the estimation of species concentration, especially at downstream planes where the experimental burning rate is recovered, as shown by CH_4 profile.

6. Conclusions

In this work, LES calculations of a swirler stabilized turbulent gas flame have been performed. The comparison with experimental data and time-averaged solutions shows the improvements that can be obtained with scale-resolving approaches in terms of flow field and species concentration. In fact LES has proved to be able to reproduce the large energy-carrying vorticies and predict a more physical mixing and jet spreading with respect to RANS results, especially at dowstream locations. Effect of bluff body cooling should be further investigated.

Acknowledgements

The authors would like to thank Prof. Andreas Dreizler for sharing geometry and experimental data of the TECFLAM test case.

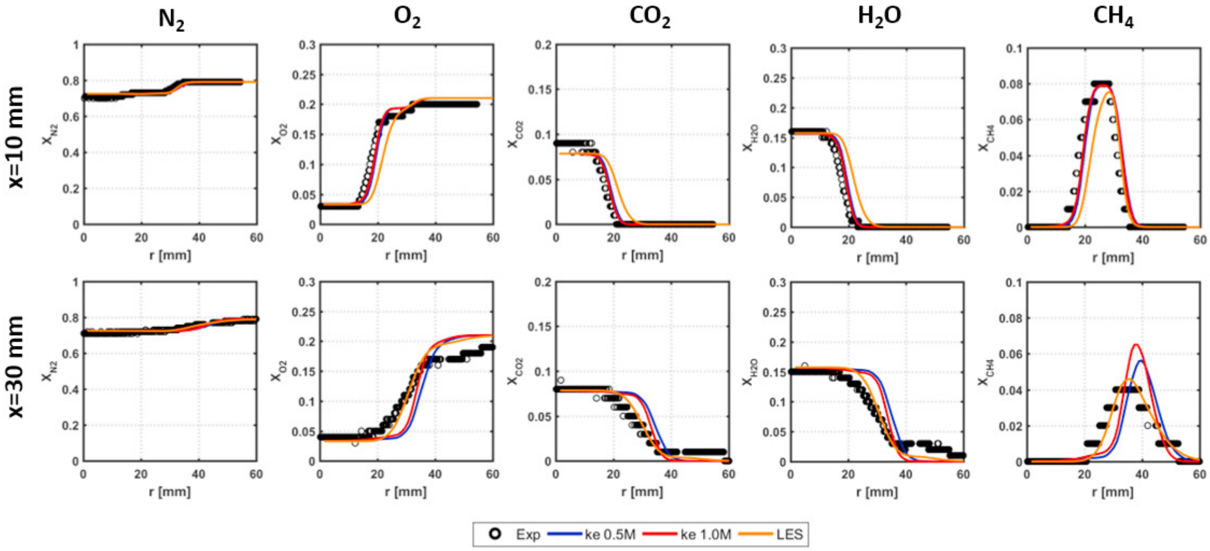

Fig. 10. Comparison of radial profiles of main species molar fraction at two axial plane between LES and RANS.

References

[1] Gicquel, L. Y. M., Staffelbach, G. and Poinsot, T. Large Eddy Simulations of gaseous flames in gas turbine combustion chambers, Progress in Energy and Combustion Science, 38 (6), (2012).

[2] Van Oijen, J. A., F. A. Lammers, and L. P. H. De Goey. ”Modeling of complex premixed burner systems by using flamelet-generated manifolds.” Combustion and Flame 127.3 (2001): 2124-2134.

[3] Van Oijen, J. A., et al. ”State-of-the-art in premixed combustion modeling using flamelet generated manifolds.” Progress in Energy and Combustion Science 57 (2016): 30-74.

[4] Schneider, C., A. Dreizler, and J. Janicka. Fluid dynamical analysis of atmospheric reacting and isothermal swirling flows. Flow, turbulence and combustion 74.1 (2005): 103-127.

[5] Gregor, M. A., et al. Multi-scalar measurements in a premixed swirl burner using 1D Raman/Rayleigh scattering. Proceedings of the Combustion Institute 32.2 (2009): 1739-1746.

[6] Pope, S. B. Ten questions concerning the large-eddy simulation of turbulent flows. New Journal of Physics, 35 (2004).

[7] Lilly, D. K. (1994) “A Proposed Modification of the Germano Subgrid-Scale Closure Model.” *Physics of Fluids*, 4: 633-635

[8] Ansys Fluent. 18.0 Theory Guide. 2017

[9] Gupta A. K., Lilley D. G., and Syred N. (1984) “Swirl Flows.” *Abacus Press*

[10] Kuenne G,, Ketelheun A. and Janicka J. (2011) “LES modeling of premixed combustion using a thickened flame approach coupled with FGM tabulated chemistry.” *Combustion and Flame* 158: 1750-1767.