Progress in Energy and Combustion Science 38 (2012) 782–817

Contents lists available at SciVerse ScienceDirect

Progress in Energy and Combustion Science

journal homepage: www.elsevier.com/locate/pecs

Review

Large Eddy Simulations of gaseous flames in gas turbine combustion chambers

L.Y.M. Gicquel*, G. Staffelbach, T. Poinsot

CERFACS, 42 Avenue G. Coriolis, 31057 Toulouse Cedex 1, France

ARTICLE INFO

Article history:
Received 5 October 2011
Accepted 30 April 2012
Available online 9 August 2012

Keywords:
Large Eddy Simulations
Complex geometry
Swirled flows
Gaseous combustion
Turbulent combustion
Gas turbine

ABSTRACT

Recent developments in numerical schemes, turbulent combustion models and the regular increase of computing power allow Large Eddy Simulation (LES) to be applied to real industrial burners. In this paper, two types of LES in complex geometry combustors and of specific interest for aeronautical gas turbine burners are reviewed: (1) laboratory-scale combustors, without compressor or turbine, in which advanced measurements are possible and (2) combustion chambers of existing engines operated in realistic operating conditions. Laboratory-scale burners are designed to assess modeling and fundamental flow aspects in controlled configurations. They are necessary to gauge LES strategies and identify potential limitations. In specific circumstances, they even offer near model-free or DNS-like LES computations. LES in real engines illustrate the potential of the approach in the context of industrial burners but are more difficult to validate due to the limited set of available measurements. Usual approaches for turbulence and combustion sub-grid models including chemistry modeling are first recalled. Limiting cases and range of validity of the models are specifically recalled before a discussion on the numerical breakthrough which have allowed LES to be applied to these complex cases. Specific issues linked to real gas turbine chambers are discussed: multi-perforation, complex acoustic impedances at inlet and outlet, annular chambers.... Examples are provided for mean flow predictions (velocity, temperature and species) as well as unsteady mechanisms (quenching, ignition, combustion instabilities). Finally, potential perspectives are proposed to further improve the use of LES for real gas turbine combustor designs.

Contents

1. Introduction . . . 783
2. Fundamentals of LES for complex burner simulations . . . 784
   2.1. The filtering approach: implicit, explicit and no-model approaches . . . 784
   2.2. LES transport equations and sub-models . . . 785
      2.2.1. Turbulence models for velocity and scalars . . . 786
      2.2.2. Combustion models . . . 786
      2.2.3. LES and the DNS limit modeling constraint . . . 788
   2.3. Numerical methods . . . 789
      2.3.1. High-order schemes, mesh type and complex geometries . . . 789
      2.3.2. Implicit versus explicit time integration, incompressible, low Mach and fully compressible approaches . . . 790
      2.3.3. Boundary condition treatment and stability . . . 790
   2.4. The massively parallel context, mesh generation and data management . . . 790
3. Laboratory-scale burner simulations . . . 791
   3.1. Key geometrical features . . . 792
   3.2. Flow validations . . . 792
      3.2.1. Predictions of the mean statistical flow features . . . 794
      3.2.2. Predictions of the unsteadiness of swirl flow features . . . 794

* Corresponding author. Tel.: +33 (0)5 61 19 30 46; fax: +33 (0)5 61 19 30 00.
E-mail addresses: Laurent.Gicquel@cerfacs.fr, lgicquel@cefacs.fr (L.Y.M. Gicquel).
URL: http://www.cerfacs.fr

0360-1285/$ – see front matter
http://dx.doi.org/10.1016/j.pecs.2012.04.004

3.3. Reacting flow validations 794
3.3.1. Statistically stationary flow conditions 795
3.3.2. Leadership-class LES modeling and predictions 799
3.3.3. Thermo-acoustic instabilities 802
3.3.4. Transient operating conditions 803
4. Real engine combustor simulations 805
4.1. Specific features and missing links 805
4.2. Current state-of-the-art LES for real engine combustors 806
4.2.1. Swirler simulations 806
4.2.2. Single sector simulations 807
4.2.3. Full annular burner simulations 809
5. Conclusions and perspectives 810
Acknowledgments 811
References 811

1. Introduction

Aeronautical turbulent reacting flows involve a wide range of scales and complexities caused by the specific shapes of engines and the combustion regimes encountered in these devices. Because of the space and weight constraints, designers need to develop burners which ensure maximum efficiency and compactness. Over the years, manufacturers have gained significant experience and existing designs largely rely on flow recirculations to increase mixing and flow-though times despite a reduced size combustion chamber. In parallel, pollutant emissions and regulations have induced changes of technology with the emergence of partially premixed and premixed burners. Multi-point fuel injection systems and staging are also being implemented as potential solutions to the new regulations. All these concepts increase the complexity of the flow and lead to specific flow dynamics and combustion responses. Although these designs are being routinely evaluated by Computational Fluid Dynamics (CFD), most present modeling strategies rely on Reynolds Average Navier-Stokes (RANS) approaches developed for mean stationary flows [1–10]. Such models benefit from extensive research and developments from the scientific community and have been successfully calibrated on simple fundamental configurations. However, the complexity of flows in modern gas turbines adds multiple constraints on RANS and limits their precision, Fig. 1. Alternative numerical solutions are thus needed to further increase the share of CFD and decrease the number of real engine tests and design iterations.

CFD alternatives to RANS for aeronautical gas turbine applications must justify the increase in development, maintenance and computer costs. These new tools need also to be compatible with existing industrial knowledge and conception rules. The use of new CFD approaches and their future in the design chain is still unclear. It will probably depend on the computing power available to engineers as well as their ability to master and analyze ever more

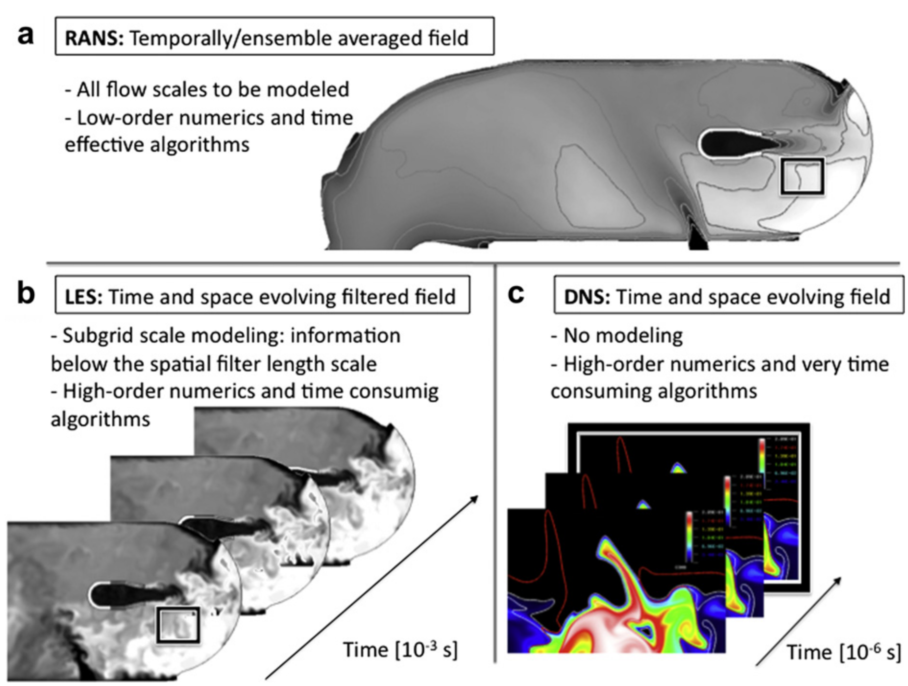

Fig. 1. Schematic representation of the three numerical methods used to simulate turbulent reacting flows: (a) RANS provide access to a temporally/ensemble averaged field representing the flow field in complex systems (extracted from [319]); (b) LES give access to a temporally and spatially evolving set of fields representative of the spatially filtered governing system of equations (extracted from [320,360]) and (c) DNS provide the exact spatially and temporally evolving field obtained by directly solving the governing equations (extracted from [361]).