



LARGE EDDY SIMULATION OF TURBULENT PREMIXED COMBUSTION

Ashoke De
Ph.D. Candidate

Faculty Advisor: Dr. Sumanta Acharya

ABSTRACT

Turbulent premixed combustion is of great interest because of its widespread practical applications including gas turbine combustion, furnaces, boilers, and automobile engines. To improve design, higher fidelity, low-cost computational procedures are needed. In the present research, the application of our interest is gas turbine combustion. It is evitable that in a gas turbine combustor, the flow and pressure fields, the mixing of fuel and air and the associated combustion and heat release, are all essentially unsteady non-linear processes and are intrinsically interlinked with each other. Understanding this complex interaction and its implication on key performance metrics such as emissions, thrust or volumetric heat release, and pressure oscillations is a challenging task. Therefore, the objective of the present work is to perform series of numerical simulations to have a clear understanding of the above-mentioned phenomena, and throw light on the fundamental issues associated with, and ultimately use of improved turbulence models. In recent times, LES has become more promising tool over RANS and DNS to provide accurate and cost-effective predictions for studying combustion dynamics. Our numerical approach here is based on LES with a thickened flame (TF) model for simulating such flows. In the LES-TF approach, the flame front is resolved on the computational grid through artificial thickening and the individual species transport equations are directly solved with the reaction rates specified using Arrhenius chemistry. With this approach, reaction rate modeling does not require any ad-hoc closure assumptions. However, suitable modifications have to be made to compensate for flame thickening.

We are currently considering premixed combustion of natural gas/hydrogen and air, since such systems are being proposed by the department of energy for use in future land based gas turbines for power generation. A key goal is to enable computations with high accuracy and short turnaround times. Both these metrics are important from an industrial perspective. In the present study, a multi-block compressible flow code, Chem3D, developed in house, is used. The code runs in a scalable parallel manner on parallel

Linux clusters and solves for an arbitrary number of reacting species in generalized curvilinear coordinate systems.

Due to the complexities brought about by the nonlinear aspects of the dynamic response of the flame, it is very difficult to build a simple yet effective model from first principles to describe the flame dynamics in full scale gas turbines. The developed models cannot be used as accurate models for full scale gas turbine combustors unless the effect of physical parameters on the behavior of the empirical model is well understood, so as to enable the mapping over to full scale gas turbines. With this philosophy in mind, it was decided to first study the LES of a simple premixed flame, then LES of a premixed cold flow for their dynamic response at atmospheric conditions, thus minimizing the number of physical variables involved. The techniques developed during these studies will later applied to the investigation of the swirl stabilized turbulent flames under atmospheric conditions. Finally, the complexity of unmixedness generated due to fuel injection in swirling flow just upstream of the combustor, will be added to the problem. Thus, this research effort first concentrated on evaluating flame structures of simple premixed flame and the dynamics of a cold flow and then added complexities such as turbulence, swirl and fuel injection in an effort to move closer towards simulating full scale gas turbine conditions.

RESULTS

To validate the LES-TF approach, the considered configuration here is the Bunsen burner geometry investigated by Chen et al. [1]. The flame is a stoichiometric premixed methane-air flame, stabilized by an outer pilot. More details on computational domain and grids will be found in the literature [2].

Figure 1 shows the comparison between cold and reacting flow predictions. As observed, the radial profiles of the mean axial velocity are broadened in the reacting case, due the effect of the flame front, pushing the shear layer outward in the radial direction. Furthermore, it is observed that the peak center line velocity remains almost constant in

the axial direction, and exhibits a longer potential core compared to the cold flow case. These effects are reasonably well reproduced by the present simulations.

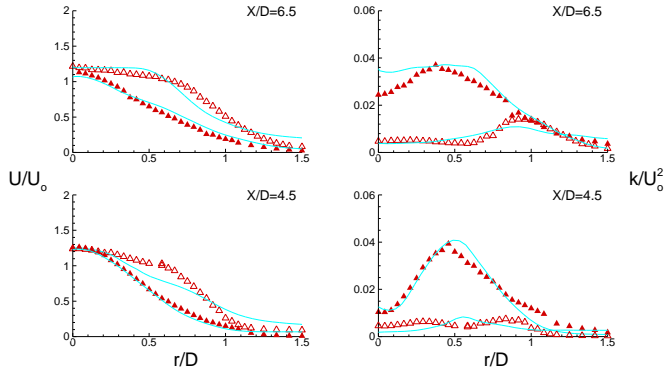


Figure 1. Mean axial velocity U/U_0 and turbulent kinetic energy k/U_0^2 , at $Re=24000$: Experimental data: reacting flow (Δ), cold flow (\blacktriangle); Lines are LES predictions

Secondly, an unconfined strongly swirled stabilized flow is investigated for different Reynolds using LES-TF methodology. The detailed description of the geometry and computational domain can be found in the paper [3]. Fig. 2 shows the streamline patterns for non-reacting flow at different Reynolds numbers. Three distinct recirculation regions are observed in the high Reynolds number case, $Re=13339$, that include a separation wake recirculation zone (WRZ) behind the center body, a corner recirculation zone (CRZ) due to sudden expansion of combustor configuration, and a central toroidal recirculation zone (CTRZ) formed due to vortex breakdown. The symmetric structure of CTRZ, however, becomes more clearly visible at higher Reynolds number ($Re=13339$). The asymmetry at the lower Reynolds number indicates a low-frequency unsteadiness that is not averaged out despite the long integration times (15-25 flow through times) used for statistical averaging. Thus the origins of the CTRZ at the lower Re appear to be in the form of a flapping vortical structure that becomes more steady and well defined at higher Reynolds numbers.

Fig. 3 shows the distributions of the normalized axial velocity profiles, and axial fluctuations at different axial locations for two Reynolds numbers. The overall agreement of the predictions with the data is found to be quite reasonable, considering the complexity of the physical processes and the configuration. With increasing axial distance the magnitude of the peak velocity decreases and the location of the peak is moved further outwards radially. Moreover, both axial and tangential turbulence levels are enhanced with reaction. Predicted RMS fluctuations clearly exhibit two peaks. The location of the peaks corresponds to the burnt (low peak) and un-burnt (high peak) regions in the inner part of the shear layer and associated with the high velocity gradients and temperature. More detailed distributions of tangential velocity, major species mass fractions can be found in the paper [3].

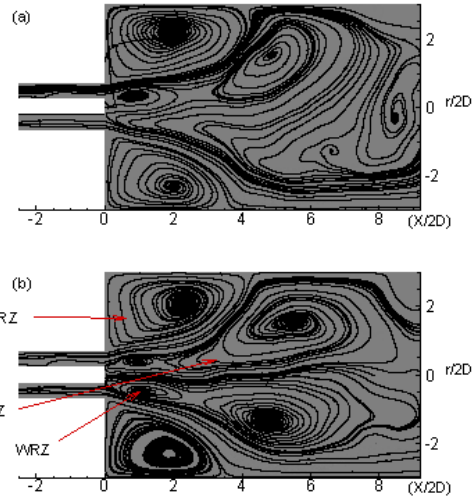


Figure 2. Streamline patterns for non-reacting flow condition [D : center-body diameter]: (a) $Re=10144$, (b) $Re=13339$

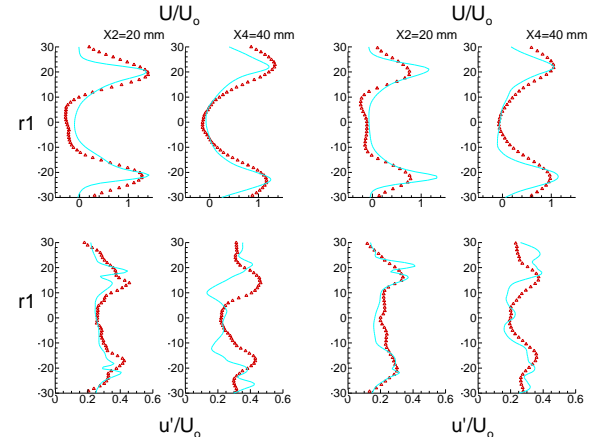


Figure 3. Reacting flow results for $Re=10144$ (left) and $Re=13339$ (right) at different axial locations [$r1=(r/2D) \times 25.4$; $X2, X4=(X/2D) \times 25.4$]: Experimental data (Δ), Lines are LES predictions. Axial velocity U/U_0 , Axial velocity fluctuation u'/U_0 .

ACKNOWLEDGMENTS

This work was supported by the Clean Power and Energy Research Consortium (CPERC) of Louisiana through a grant from the Louisiana Board of Regents. Simulations are carried out on the computers provided by LONI network at Louisiana, USA (www.loni.org) and HPC resources at LSU, USA (www.hpc.lsu.edu). This support is gratefully acknowledged.

REFERENCES

1. Chen, Y. C., Peters, N., Schneemann, G. A., Wruck, N., Renz, U. and Mansour, M. S. 1996. *Combustion and Flame*, 107, 233.
2. De, A. and Acharya, S. July 2009. *ASME J. of Engg. for Gas Turbines and Power*, 131.
3. De, A., Zhu, S. and Acharya, S. 2009. *ASME J. of Engg. for Gas Turbines and Power*, submitted, March 2009; *ASME Turbo Expo 2009: Power for Land, Sea and Air*, ASME Paper GT2009-60230.