Prediction of Gas Turbine Combustor Performance Based on Optimal Reduced Kinetics and Dynamic Mode Decomposition

Abdallah Abou-Taouk
Department of Applied Mechanics
Division of Fluid Mechanics
Chalmers University of Technology
abdallah@chalmers.se

ABSTRACT

In the gas turbine community the development of combustor technology is currently following the general trend towards fuel-flexibility. Fully detailed mechanisms for these fuels are expensive in terms of computer time when coupled with CFD, and the complexity of combustion chemistry must hence be downsized. The research field hence may be described as CFD modeling of unsteady combustor flows in the context of finite-rate kinetics and the use of such a tool for eigenmode extraction and analysis. The research includes refinement of optimized global multi-step mechanisms for several different fuels as well as improvement and evaluation of the Dynamic Mode Decomposition in practical combustor settings.

INTRODUCTION

Combustion is a multi-disciplinary field involving areas such as chemical kinetics, fluid dynamics and thermodynamics. In real life, most industrial gas turbine combustors are difficult to study due to complex geometries and multiple inlets, fast chemistry, dilution with burnt gases and high pressures. This implies that robust and accurate models for combustion chemistry and its interaction with turbulence are needed for the design and development of future gas turbine combustors. In order to obtain accurate and validated models one also needs experimental data which is difficult to obtain since these devices often operate at high pressure. The rapid increase in computer power in recent years has made reacting flow simulations feasible using more sophisticated models such as hybrid Unsteady Reynolds Averaged Navier Stokes /Large Eddy Simulation model (URANS/LES) and LES, which resolves the large scales and flow-flame interactions, and the uncertainty of the combustion modelling is then narrowed to the unresolved sub-grid scale motions. LES of reacting flow is under intense development and investigated in numerous studies [1–2]. Reacting flows are dominated by different mechanisms (fluid dynamics, kinetics, interfaces, and structures) which exhibit instabilities. A significant challenge in the development and operation of gas turbines is to prevent different combustion instabilities by predicting their dominating frequencies. In a premixed confined flame the acoustic waves produced by combustion might reflect on the boundaries and propagates back into the flame zone. The flame itself is sensitive to waves and may feed energy into the oscillations and thus cause an amplification of one or several resonance modes. The lean premixed swirled combustion burners are therefore prone to instabilities, which can cause structural damages due to large amplitude vibrations and/or high thermal loads at walls. There exist numerous methods of predicting such instabilities, where the DMD algorithm [3] is a recently developed method which enables to extract aero-acoustic modes from a data set recorded using CFD or experiments. To evaluate the DMD approach together with the SAS-SST turbulence model and a global reaction mechanism, a reference multi-point injection and premixed burner (SGT-100 DLE [4]) is simulated and the results are compared with experiments.

SGT-100 burner

In this work the numerical simulations are conducted using the SAS-SST. In the SAS model, the von Karman length scale is added within the turbulence length scale equation, so that the model changes dynamically from a URANS to LES resolution mode. The majority of the published combustion articles on gas turbine burners based on the hybrid URANS/LES and LES are tested at atmospheric pressure and are for relatively simple geometrical configurations. The work described here aims to validate the SAS-SST model together with the M4 [5] global reaction mechanism for the experimental version of SGT-100 industrial gas turbine combustor which is designed and manufactured at Siemens Industrial Turbomachinery Ltd, UK. In this work, an operating pressure of 6 bar is used. One-dimensional Raman for scalar and PIV velocity measurements profiles are available at four streamwise locations [4].

Numerical simulation

The flame is anchored on the pilot face, while the main flame is located in the combustion chamber and takes the shape of a hollow cone. The flame topology consists of regions of corrugated flame fronts due to the high turbulence levels in this shear layer. The obtained Damköhler and Karlovitz numbers for this operating point corresponds to a regime where thickened flame fronts and local extinctions are to be expected according to the regime diagrams by Borghi. Figure 1 presents a comparison of PIV measured data [4] and simulated streamwise mean velocities at four different locations. The CFD results agree well with the measured main flow field. The outer recirculation zone is well captured at all positions, however the inner recirculation zone is not fully reproduced at third and fourth location. The inner recirculation variation in velocity is explained by a periodic vortex shedding by Stopper et al. [4].

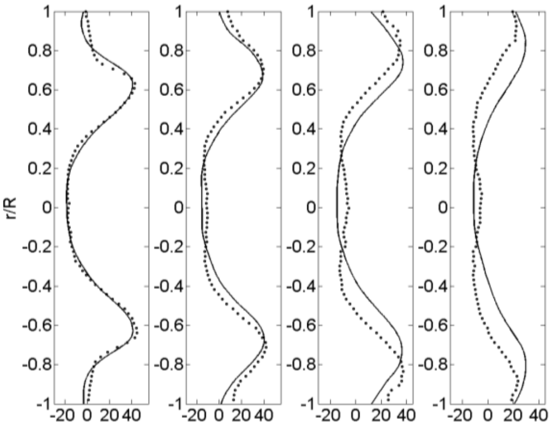

Figure 1: Streamwise averaged velocity at four different locations. Symbol: Experiments [4]. Solid-line: CFD.

Figure 2 presents a comparison of PIV measured data [4] and simulated mean temperature at four different locations. The temperature has been normalized by the maximum temperature. The CFD results agree well with the measured mean temperature. Some small deviations can be observed around r/R=0.6. This area corresponds to the inflow region of the fuel air mixing stream.

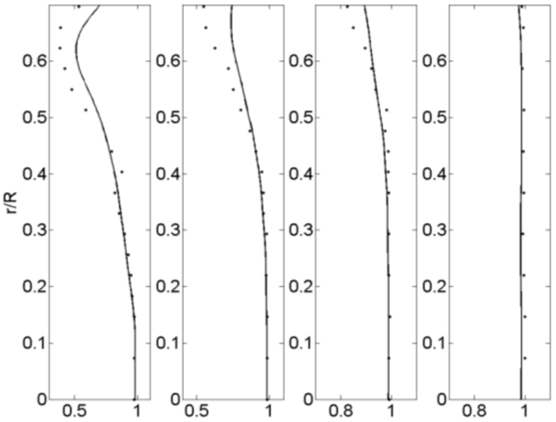

Figure 2: Mean normalized temperature at different locations. Symbol: Experiments [4]. Solid-line: CFD.

The DMD technique is applied here to identify the resonant modes in the burner, to extract the mode structures and to quantify the respective damping modes. Previously published results using CFD and experiment [4, 6] show that the combustion chamber features a small (20 mbar) periodic pressure amplitude at an operating pressure of 3 bar. However, at an operating pressure of 6 bar used in this work, the pressure amplitude is increased (113 mbar in the experiment). This oscillation was observed in the experiment [4] at the frequency 220 Hz with the variations ± 19 Hz. The CFD predicted a pressure amplitude of 100 mbar without any excitation of the flow field. In the present DMD simulation, the sampling time interval between two snapshots was selected such that a pressure fluctuation/oscillation at high frequency, e.g. 1000 Hz could be predicted. The time step in the CFD simulations was chosen to 25 µs and a snapshot is recorded every five time steps. The DMD is performed with 400 snapshots, corresponding to the lowest detectable frequency of 20. Figure 3 shows normalized axial velocity at different planes. The plots are extracted from the DMD algorithm for the mode with the frequency corresponding to 226Hz. A circumferential rotating mode can be observed. The transient data suggest that vortices are created close to the prechamber wall and form a periodic vortex shedding which downstream in the burner displays merging and dissipation.

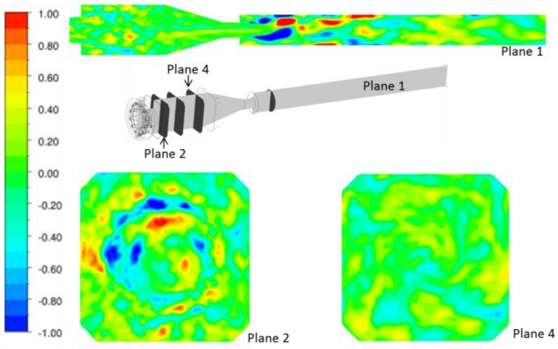

Figure 3: DMD mode at f=226Hz showing the normalized axial velocity at three different planes.

REFERENCES

[1] Moureau. V., Domingo. P., Vervisch. L. *Combust. Flame*, 158(7):1340–1357, 2011.

[2] Abou-Taouk. A., Farcy. B., Domingo. P., Vervisch. L., Sadasivuni. S., Eriksson. L.E., "*A new approach for LES of gas turbine partially premixed combustion using optimized chemistry*, Submitted to Fuel, 2014.

[3] Schmidt. P. J, *Journal of Fluid Mechanics*, 656:5–28, 2010.

[4] Stopper. U., Meier. W., Sadanandan. R., Stöhr M., Aigner. M., Bulat. G. *Combust. Flame,* 160(10):2103–2118, 2013.

[5] Abou-Taouk. A., Sadasivuni. S., Lörstad. D., Eriksson. L. E, "*Evaluation of global mechanisms for LES analysis of SGT-100 DLE combustion system",* In ASME Turbo Expo 2013 USA, 2013.

[6] Bulat. G., Jones. W. P., Marquis. A. J. *Combust. Flame*, 161(7):1804–1825, 2014.