Applied Energy 174 (2016) 153–165

Contents lists available at ScienceDirect

Applied Energy

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

Numerical simulations of combustion process in a gas turbine with a single and multi-point fuel injection system

Artur Tyliszczak [a,*], Andrzej Boguslawski [a], Dariusz Nowak [b]

[a] *Czestochowa University of Technology, Faculty of Mechanical Engineering and Computer Science, Al. Armii Krajowej 21, 42-201 Czestochowa, Poland*
[b] *General Electric GmbH, Gas Power Systems, Brown Boveri Strasse 7, 5401 Baden, Switzerland*

HIGHLIGHTS

- RANS and LES simulations of a combustion process in a gas turbine are performed.
- The chemical kinetics has larger impact on the results than the turbulence model.
- The results (velocity, temperature) show good agreement with experimental data.
- Change of distribution of air and fuel causes a significant alteration of the flame.

ARTICLE INFO

Article history:
Received 12 February 2016
Received in revised form 17 April 2016
Accepted 25 April 2016

Keywords:
Can combustor optimisation
Combustion modelling
RANS and LES simulation
Flamelet model

ABSTRACT

The paper presents a numerical study of a medium size model of industrial gas turbine combustor. The research was conducted using the RANS (Reynolds Averaged Navier–Stokes) approach with k—ϵ model and LES (Large Eddy Simulation) with WALE (Wall Adapting Local Eddy viscosity) subgrid model. The simulations were performed in cold and reacting flow conditions. In the latter case, the combustion process was modelled using a steady flamelet model with chemical mechanisms of Smooke with 16 species and 25 elementary reactions, and the GRI-2.11 with 49 species and 277 elementary reactions including NO chemistry. In the first part of the paper, the numerical results were validated against experimental data including velocity field, temperature and species concentrations. The velocity components predicted for the cold flow agree very well with measurements. In the case of the simulations of the reacting flow, some discrepancies were observed in both the temperature field and species concentrations. However, the main flame characteristics were captured correctly. It turned out that the chemical kinetics had a larger impact on the results than the turbulence model. In the second part of the paper, we modified the fuel and air injection method and analysed how the changes introduced affect the flame dynamics. It was shown that: (i) depending on the distribution of air, the velocity, temperature and species composition in the upper part of the combustion chamber can be significantly altered; (ii) more substantial changes can be achieved by shifting the fuel injection points; their location outside the main recirculation zone leads to a dangerous situation resulting in overheating of the walls; (iii) it turns out that substantial differences in the flame characteristics in the upper part of the combustion chamber vanish approaching the outlet plane and the resulting mixture compositions are very similar.

1. Introduction

Since the installation of the first industrial gas turbine set at Neuchâtel in 1939, combustion chambers have evolved significantly in terms of combustion intensity and emission levels. In particular, over the past 30 years, combustors have developed gradually from large, single diffusion burners to compact premix burners in multi-burner arrangements [1]. Premix combustors offer significantly lower emissions of nitrogen oxides compared to single diffusion burners, but they are more difficult to operate under start-up and low load conditions. To overcome these limitations, most gas turbines manufacturers have developed a concept of advanced fuel staging principles [2]. Another key issue in the gas turbine burner design is the method to stabilize flame, which is most often achieved by vortex breakdown and hot gas recirculation in a swirling flow.

---

[*] Corresponding author.
E-mail address: atyl@imc.pcz.pl (A. Tyliszczak).

http://dx.doi.org/10.1016/j.apenergy.2016.04.106
0306-2619/© 2016 Elsevier Ltd. All rights reserved.

An alternative way to fuel staging to improve gas turbine stability at low load with low nitric oxides and carbon monoxide emissions is a concept of the reheat combustion used in GT24/GT26 engines in 1990 [1] and then widely applied in the industry [3,4]. A variety of new combustion concepts similar to the reheat combustion idea have been under experimental and numerical investigations for last two decades, including Flameless Oxidation (FLOX) [5–7], High Temperature Air Combustion (HiTAC) [8], Stagnation Point Reverse Flow combustion (SPRF) [9], Colourless Distributed Combustion (CDC) [10] and MILD combustion [11–16]. A common key aspect of all these new technologies is that the mixing of burnt gases, fresh air and fuel should be completed before the combustion process begins under diluted oxidizer conditions. Therefore, a variety of combustor concepts for premixed and non-premixed combustion have been investigated with mixing enhanced by swirling [17–19], reverse flows [20,15], reverse cross flow [21] and forward flow configurations [22,23]. Recent studies on CDC combustors with mixing enhanced by swirling have demonstrated that NO emissions in non-premixed combustors were comparable to these encountered in premixed combustion mode [24–27]. Moreover, this type of combustor turned out to be flexible for a range of fuels of different physical and chemical composition [28]. Another method to stabilize flame that received growing attention recently is the trapped vortex combustor, in which flame stabilization is achieved through the use of cavities [29,30]. This type of combustor is characterized by a low total pressure drop compared to swirl burners.

In view of the rapid progress in development new combustion technologies for gas turbine applications, reflected in recent experimental and numerical investigations, much less attention has been paid to the problem of advanced numerical modelling of combustion in traditional silos or can-type combustors with the single diffusion burners. As it was mentioned above, this technology is currently not in progress due to excessive NOx emissions caused by a high temperature flame front. Hence, although it might seem that advanced numerical models are not needed for new combustors design, there is a huge number of gas turbines of old type still in use all over the world and they could possibly be improved. A question arises whether a low cost retrofit of such a single diffusion burner could lead to a significant decrease of nitric oxides emission. Such a redesign of the combustion chamber requires CFD simulations of non-premixed combustion stabilized by a swirling flow with validated flow and combustion models. There are relatively few such attempts in recent literature. Nemitallah and Habib [31] studied the oxy-combustion non-premixed flame using RANS (Reynolds Averaged Navier–Stokes) for turbulence modelling, a simple two-step reaction mechanism and quite old eddy-dissipation model for combustion modelling. They showed quite limited experimental verification taking into account only averaged exhaust gas emissions. Another example of a recent numerical analysis of the single can combustor was presented by Gobbato et al. [32]. However, their model turned out to be not very accurate as the results obtained exhibited significant discrepancies with the experimental data.

Compared to RANS, the use of LES (Large Eddy Simulation) method offers much deeper and more accurate insight into physics of combustion. It enables study of very complex problems, eg. auto- and forced ignition [33–36], local extinctions [37] or active flame control [38]. In these cases, the LES with advanced combustion models and complex reaction mechanisms provide an excellent agreement with the experimental data. Concerning modelling of gas turbines, the use of LES is limited mainly to laboratory small-scale problems with characteristic dimensions of at most several centimetres [39–46]. Though feasible, the studies devoted to medium-size configurations are very rare. One example is the study by Boileau et al. [47] on numerical analysis of complete ignition sequence in a full annular combustion chamber of a helicopter gas turbine. LES of a full scale stationary SGT-100 Siemens gas turbine with the Eulerian PDF method and complex chemistry, described by the mechanism derived from GRI 3.0 chemistry including 15 reaction steps and 19 species, was recently presented by Bulat et al. [48]. The main barrier to use LES applications in industrial practice is huge computational costs. It turns out that even for small scale combustor models, the LES requires computational meshes with at least a couple or tens of millions of nodes (eg. 19×10^6 in [47] or 329×10^6 in [40]), involvement of thousands of CPUs and simulation times taking several weeks. Recent status and perspective of LES modelling of gas turbines was presented in [49]. As discussed by the authors, a typical LES is at least 100 times more expensive than RANS, so it seems that even with a very fast and continuous increase of the computing power, the use of LES for design or modernization of large scale industrial installations is not feasible in the nearest future.

The present paper is aimed at the applications of both RANS and LES (under-resolved) approaches for analysis of the combustion process in a medium scale model (1:4) of silos type industrial gas turbine combustor. To the best knowledge of the authors, the present work constitutes one of the first attempts aiming to combine sophisticated experimental measurements and advanced numerical tools to improve characteristics of the large-size industrial devices. The research was conducted in both cold flow and reacting flow conditions. The results were compared to the experimental data in terms of local values of velocity and temperature as well as concentrations of selected species. Having the validated numerical model for a basic configuration, an attempt was made to modify the fuel and air distribution while keeping their overall mass flow rates constant. The idea behind the modified configurations was to improve a fuel/air mixing characteristics in such a way that the regions of hot spots were limited leading to a certain decrease of the nitrogen oxide emissions without an excess of the carbon monoxide level.

2. Combustor chamber geometry and inlet parameters

Fig. 1 shows a CAD view of a half of the combustor geometry. The arrows point to the fuel injection system, air supply system, effusion cooling slots and observation windows through which measuring probes were introduced. The internal sizes of the combustion chamber are the following: (i) vertical length measured from the end of the swirler section to the outlet equals to 1.6 m; (ii) internal maximum and minimum diameters are 0.38 m and 0.46 m, respectively. The preheated air is injected through the cylindrical channels mounted above the main large swirler consisting of 12 vanes and through a small central swirler with 12 vanes. The air entering the combustion chamber rotates clockwise when looking from the inlet side. The air swirlers are shown in Fig. 2. The fuel is supplied to a cylindrical nozzle placed between the air channels. The fuel (natural gas: 96.9% CH_4, 1.9% N_2, 1.2% higher hydrocarbons) is injected to the combustion chamber without swirl and in the direction inclined 60° with respect to the vertical axis.

The research was performed for the atmospheric pressure conditions, first for cold flow regimes (i.e. without the fuel injection) and then for reacting regimes for a fully developed combustion process, such that ignition and flame propagation processes were not taken into account. The experimental data for the cold flow conditions were needed to validate numerical predictions of aerodynamics inside the burner and also to assess whether the applied numerical meshes provide accurate representation of the main flow characteristics. Table 1 gives details of the inlet parameters of the fuel and air (temperature and mass flow rates) and also