Energy 160 (2018) 693–709

Contents lists available at ScienceDirect

Energy

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

Application of LES-CFD for predicting pulverized-coal working conditions after installation of NOx control system

Wojciech P. Adamczyk [a, b, c, *], Benjamin Isaac [b], John Parra-Alvarez [b], Sean T. Smith [b], Derek Harris [b], Jeremy N. Thornock [b], Minmin Zhou [b, **], Philip J. Smith [b], Robert Żmuda [c]

[a] Institute of Thermal Technology, Silesian University of Technology, Gliwice, Poland
[b] Institute for Clean and Secure Energy, University of Utah, Utah, USA
[c] SBB ENERGY S.A. DeNox Technology Department, Sowinskiego 11, 44-121 Gliwice, Poland

ARTICLE INFO

Article history:
Received 16 January 2018
Received in revised form
25 June 2018
Accepted 8 July 2018
Available online 12 July 2018

Keywords:
CFD-LES
Combustion
Multiphase
DQMOM
NOx reduction

ABSTRACT

The upcoming environmental requirements outlined at the 2017 European Commission Integrated Pollution Prevention and Control (IPPC) are becoming more and more strict in comparison to the existing standards – IED 2010/75/EU. Questions arise about changes in boiler operating conditions after the adoption of the regulations to fulfill specified emission limits. Of initial concern is the question of which technology to pursue in order to reach the specified level of NOx emissions at 150 mg/Nm^3@6%O_2. By introducing an air staging technique the NOx limit can be partially achieved; however these changes affect the boiler operation in a detrimental way. In order to investigate impact of de-NOx installation on the temperature and heat flux distribution, numerical techniques can be used. In order to determine the impact of the changes in the boiler operation, this paper presents the application of an advanced, multiphase open-source code Arches, developed at the University of Utah for modeling pulverized-coal combustion using Large-Eddy Simulation (LES). Numerical simulations showed the effect on the boiler's working parameters e.g., heat-transfer rate, temperature distribution, species concentration, and NOx emission before and after proposed modifications.

1. Introduction

Increasingly rigid EU regulations require coal-fired boilers to adapt to a wide range of new environmental standards. Coming in 2017, the European Commission Integrated Pollution Prevention and Control (IPPC)/Industrial Emissions Directive (IED) Best Available Techniques (BAT)/Best Available Techniques Reference Documents (BREF) proposes a more strict emission regulations outlined in 2010/75/EU [1] regarding SO_2, NOx, particulate matter and heavy metals. The NOx and SO_2 emissions are limited to a level of 150 mg/Nm^3 and 200 mg/Nm^3 @6%O_2, respectively. To deal with high NOx emissions, different technologies may be implemented to achieve compliance. The most commonly used are primary reduction methods – based on controlling the fuel combustion process. This can be achieved mainly by manipulating coal-burner fuel and air dynamics ensuring stable control of the combustion zones where oxidizing and reducing atmospheres usually occur. Miller [2], and Xu et al. [3] proposed air staging throughout the height of the combustion chamber extending the reduction zone and decreasing total NOx. Nonetheless, the standard over-fired air (OFA) system is insufficient in modern coal-fired furnaces in order to reach currently defined emission limits for NOx, and alternative solutions are desired. Instead of using the standard OFA configuration, the boosted over-fire air (BOFA) system developed by Doosan Babcock or the rotating opposed-fire air (ROFA) system from Mobotec LLC. Higgins et al., 2010 [4] have shown application of a ROFA system for NOx control within a large scale utility boiler, showing the NOx reduction at levels of 20–40%.

When attempting NOx reduction through air staging it is important to understand how the boiler modifications (not only installation of additional air boxes but also burner modifications) will affect operating conditions. In order to understand these impacts, advanced computational fluid dynamic (CFD) modeling can

* Corresponding author. Institute of Thermal Technology, Silesian University of Technology, Gliwice, Poland.
** Corresponding author. Institute for Clean and Secure Energy, University of Utah, Utah, USA.
E-mail addresses: wojciech.adamczyk@polsl.pl (W.P. Adamczyk), zhouminmin7@gmail.com (M. Zhou).

https://doi.org/10.1016/j.energy.2018.07.031
0360-5442/

be used. CFD allows one to investigate a variety of flow properties across the entire boiler for different boiler operating conditions without executing expensive and time consuming *in situ* tests. CFD has been shown to be a useful and sometimes accurate modeling framework for the description of particulate systems and for the description of turbulent flows in general, for instance in Ref. [5] the changes of particles concentration due to the turbulence was studied while in Ref. [6] the two-way coupling between gaseous and particulate phases in terms on isotropic turbulence was carefully investigated. In addition, an advantage of using CFD modeling is the possibility to study different configurations of air staging for the same unit including: box positions, mass flow rates or burner design changes. Burner design changes can be important in air-stagging systems because up to 30% of the total air flow rate may be diverted for staging leaving a reduced amount of air for mixing and stable combustion near the burner tips in the wind-box. The wind-box is designed to enhance the fuel-air mixing, resulting in a controlled transition from the fuel-rich to fuel-lean regions of the boiler. Naturally, the conversion of fuel-bound nitrogen to NOx is limited by the fuel-lean conditions. In this environment, NOx experiences the so-called reburning reactions, where NOx is reduced to N2 by CHi. However, the transition to fuel-lean conditions promotes nitric oxide formation [7] due to the introduction of additional oxygen. In practice, maintaining the reducing environment in one section of the boiler and smoothly transitioning to the oxidizing section is a non-trivial task. A chief actor among these complexities is the turbulence: the large-scale eddies couple with the motion of the individual-coal particles and the smaller-scale eddies, influence heat-transfer and mixing as the particles produce fuel into the gas and burn themselves. Each of these mechanisms must be correctly coupled, which leads to an understanding of flame stability, heat fluxes (desired product), and pollutant formation (undesirable product).

One of the most difficult challenges in combustion modeling is to accurately predict flow turbulence with its large range of active length & time scales. There are essentially three categories for numerical representation of turbulence: Reynolds-Averaged Navier-Stokes (RANS), Large-Eddy Simulation (LES) and Direct Numerical Simulation (DNS). DNS requires very fine grid spacing & time steps. Even though it potentially has the highest numerical accuracy, its application to practical combustion systems has been unrealistic due to the computational cost required. Models based on the RANS approach are most commonly used to simulate coal or gas combustion. Adamczyk et al., 2013 have shown the application of Ansys Fluent for modeling the oxy-fuel combustion process within an industrial scale boiler equipped with complex swirl burners. Other examples can be found in Ref. [8] where the pulverized coal combustion process was modeled within a tangentially fired utility boiler under various conditions. There is however, growing evidence that RANS has systematic inaccuracy predicting particle-laden flows which was studied by Sommerfeled et al., 1992 [9] where particle-laden flows through a pipe expansion was modeled. Similar study was conducted by Apte et al., 2003 [10] where LES approach was used for modeling swirling particle-laden flows in a coaxial-jet combustor. Furthermore, it is common for RANS to require fitting of geometric and reaction-specific parameters – preventing it from being an accurate *predictive* tool. The root cause of this systematic error is the requirement to model the entirety of the turbulent-kinetic-energy spectrum. The main advantage of using LES over RANS is that a large Bportion of the energy spectrum (preferably more than 80%) is resolved, leaving only the subgrid (unresolved) scales to modeling. To clarify, LES directly solves the transport equations for the larger eddies (in the same way DNS does) and models only the smallest eddies. Length- & time-scale analysis shows that the vast majority (by orders of magnitude) of the computational resources in DNS is expended on these small scales. As a result, interest for LES is growing. Franchetti et al. [11] applied LES to study a pulverized-coal jet flame at the Japanese Central Research Institute of Electric Power (CRIEPI) with overall good agreement with experimental data. Rabacal et al., 2015 [12] used LES to study devolatilization parameters in a large-scale combustion facility, overall good agreement with experimental data was achieved. Shen et al. [13] used LES coupled with a simplified version of the direct quadrature method of moments (DQMOM) to study a coal-flame ignition distance in a Hencken-type entrained flow reactor, the effects of the particle-size distribution were quantified and comparison against experimental measurements was performed. In general, LES has been convincingly shown to be superior to RANS in accurately predicting turbulent mixing and combustion dynamics, for instance in Ref. [14] the application of the CFD approch for modeling pulverized coal combustion using RANS technique is studied. Other example can be found in Ref. [15] where RANS technique was used for modeling coal combustion, co-firing of the waste gas, and emission in industrial scale boiler.

In this paper, the air-staging process (ROFA) of an industrial scale, pulverized-coal boiler (OP-430, 100% load: 342 MWt), was simulated using an LES-CFD tool developed at the University of Utah (Uintah-Arches) for modeling particle-laden combustion problems [16]. The dispersed phase is represented in a Eulerian framework by the direct quadrature method of moments (DQMOM) in which the coal-particle physical processes include: devolatilization, char oxidation, swelling, shrinkage and body forces. The application of the LES-CFD tool allowed to investigate the proposed design modifications and characterize the resulting boiler behavior. The original configuration (baseline construction) consisted of the standard configuration of OFA nozzles one level of OFA ports, while the modified boiler configuration involves air staging by distributing air over the upper part of the combustion chamber (four levels of the ROFA ports). For the baseline construction, simulation results compared to measurement data collected at the boiler using a high velocity thermocouple (HVT) probe. As it will be discussed later the difference between simulated and measured data changes in range from 2% to 20% depending on investigated quantity of interested (O_2, NOx, T). After this stage the simulation was run for the retrofired boiler to investigate the impact of the provided modification on the temperature, heat flux distribution, as well as achieved NOx reduction level. Based on carried out simulations main differences between investigated constructions were observed in temperature profiles which will be discussed in results section of the paper.

2. Investigated utility unit

The boiler under consideration is a mid-sized industrial boiler OP – 430 producing steam used for electricity generation ~342 MWt. The unit fires a hard coal with low moisture content 12.4%. It generates steam at a maximum continuous rate of ~430 t/h (532 °C, 12.7 MPa). The boiler system is fired using pulverized coal where pulverization is achieved with five on-site mills. In order to achieve a boiler efficiency up of 90%, and given a Lower Heating Value (LHV) of 21 828 kJ/kg, pulverized coal from only four coal mills is used. The general isometric view of the boiler construction with highlighted heat-transfer surfaces, burner configuration, marked over fired ports (OFA) can be seen in Fig. 1. The boiler height is ~39 m, the horizontal cross section of the combustion chamber takes an almost square shape with an edge length of ~9.7 m. The center of the wind box is located at 16 m from the ground. The burner box consist of five sections, where each has two secondary air nozzles and each primary tip is surrounded by cooling air. The