Energy 92 (2015) 77–86

Contents lists available at ScienceDirect

Energy

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

A validation of computational fluid dynamics temperature distribution prediction in a pulverized coal boiler with acoustic temperature measurement

Norbert Modliński [a, *], Pawel Madejski [b], Tomasz Janda [b], Krzysztof Szczepanek [b], Wlodzimierz Kordylewski [a]

[a] Division of Boilers, Combustion and Energy Processes, Faculty of Mechanical and Power Engineering, Wroclaw University of Technology, Poland
[b] Research and Development, EDF Polska S.A., Poland

ARTICLE INFO

Article history:
Received 15 December 2014
Received in revised form
25 April 2015
Accepted 21 May 2015
Available online 29 June 2015

Keywords:
Computational Fluid Dynamics
Pulverized coal
Front-fired boiler
Acoustic Pyrometry

ABSTRACT

The main objective of this work was to examine the capability of CFD (Computational Fluid Dynamics) on properly predicting temperature distribution in the combustion chamber. Numerous approaches were employed to verify CFD models of large-scale utility boilers. Furnace Exit Gas Temperature is one of the key values used for verification studies. Harsh environment and large dimensions inside the furnace make temperature measurement a complex task. Traditionally used suction pyrometry provides only local information. With this technique, while extremely accurate, it is practically impossible to obtain a representative temperature distribution at the furnace exit as measurements in different locations are not taken at the same time. Acoustic Pyrometry technique is the most appropriate for comprehensive CFD flame shape prediction verification. Not only average temperature value in a certain boiler cross-section can be continuously measured but also its complete two-dimensional distribution. CFD code was used to simulate the OP-650 front-fired boiler operation. The boiler is equipped with Acoustic Gas Temperature Measuring system located in a horizontal plane approximately 4 m under the furnace exit. Comparison of simulation results with measurements proves good accuracy of CFD results.

1. Introduction

According to the International Energy Agency, coal will become the world's top source of energy, before oil, in the coming years [1]. Global coal consumption will grow by 1.1% per annum by 2035, driven mainly by non-OECD countries [2]. Although deployment of renewables, need to improve coal plant efficiency and increase in natural gas utilization tends to decrease coal consumption in OECD, coal will long remain a key energy fuel for electricity generation in a number of developed countries.

Albeit pulverized-fuel firing technology was first established almost a century ago, researchers and boiler operators still look for a reliable tool able to describe complex phenomena inside the furnace, including gas–solid flow, combustion and heat transfer. Performance and environmental concerns as well as utility maintenance issues have increased the use of CFD (Computational Fluid Dynamics) codes to investigate and understand processes inside large scale boilers.

CFD application to pulverized coal combustion has been extensively applied. Boyd [3] presented a fully three-dimensional model of a tangentially-fired furnace almost 30 years ago. However, detailed validation studies of pulverized coal combustion simulations have been mostly concerned with pilot scale combustors. Andre et al. [4] carried out a mathematical modeling of a 2.4 MW swirling pulverized coal flame. Experimental measurements provided comprehensive data on velocity components in the near-burner zone, temperature, radiative heat flux and species distribution along the furnace. Hashimoto et al. [5] proposed a novel approach to devolatilization modeling. Suggested tabulated-devolatilization-process model was validated by performing simulation of a pulverized coal combustion field behind a low-NO_x burner in a 100 kg-coal/h test furnace. The results show that drastic differences in the gas flow patterns and coal particle behavior

* Corresponding author. Division of Boilers, Combustion and Energy Processes, Faculty of Mechanical and Power Engineering, Wroclaw University of Technology, 27 Wybrzeze Wyspianskiego St, 50-370 Wroclaw, Poland. Tel.: +48 606 219 270; fax: +48 71 328 38 18.
E-mail address: norbert.modlinski@pwr.edu.pl (N. Modliński).

http://dx.doi.org/10.1016/j.energy.2015.05.124
0360-5442/

appear between novel and standard approach. Numerous measurement ports provided detailed distribution of temperature and O_2 at the furnace axial cross section.

A detailed validation of large-scale utility boiler modeling studies is difficult. Majority of the CFD validation studies utilize measurements provided by the Distributed Control System. However, the useful information is limited to average species concentrations at the outlet, local temperatures, unburned carbon in ash. Very often these information are supplemented by design information on FEGT (Furnace Exit Gas Temperature) and heat absorbed by the furnace. FEGT can also be evaluated from zero-dimensional boiler energy balance model [6]. Additional local data can be provided using suction pyrometers or aspiration probes. Xu et al. [7] predicted flow with combustion in a front fired boiler. Measured (estimated) data of furnace outlet temperature, unburned carbon, O_2, CO_2, CO and NO_x concentrations at the furnace outlet as well as heat absorbed by walls and platen super heaters have been used for model validation. Yin et al. [8] investigated a furnace and part of the rear pass in the tangentially fired boiler. The simulation have been validated with global design parameters including O_2 at the furnace outlet, heat transfer in the furnace and furnace exit temperature. Site operation data was used to verify NO_x predictions. Pallares et al. [9] simulated a front fired boiler. The work concentrated on char burnout predictions. A limited furnace modeling validation included only O_2 plant measurements. Choi et al. [10] have used various measured and design values to validate tangentially-fired furnace. Local temperatures at different furnace locations, total heat flux to the furnace walls, O_2, CO_2, and NO_x concentrations at the boiler exit have been compared with computed values. Karampinis et al. [11] have evaluated the effect of co-combustion of cardoon with lignite in a 300 MWe boiler. Validation of the simulations was performed using plant data for the reference case of pure lignite combustion (furnace outlet temperature, O_2 and NO_x concentrations). Asotani et al. [12] predicted pulverized coal ignition behavior in a 40 MW tangentially fired boiler. Ignition image was obtained from high temperature resistant camera and compared to simulation results. Accuracy of general simulation approach was confirmed by available operating and design data. Gubba et al. [13] have applied CFD simulations to the tangentially-fired boiler co-firing coal with biomass. Predicted temperatures have been compared to local measurements at three boiler heights.

Furnace Exit Gas Temperature is one of the key values used for verification studies. A tool being able to measure not only the average value but also the temperature distribution is needed for a comprehensive CFD model validation. The aggressive environment of high temperatures and ash particles in addition to large dimensions of the furnace make temperature measurement a complex task [9]. Traditionally used suction pyrometry is extremely accurate. However, single probe provides only local information [14]. As the combustion chamber is a dynamic and turbulent environment, representative temperature distribution in the furnace cross-section can be obtained only by performing simultaneous measurements. Number of available test ports pose a technical limitation. Using six pyrometers simultaneously has been reported [14], which would probably not be enough for the cross-section of a large scale furnace.

A turning point in CFD furnace models general accuracy assessment came with Acoustic Pyrometry [15]. This technology is more appropriate for two-dimensional temperature mapping than suction pyrometry. It can provide average value in the selected cross-section and the information is available on-line.

Flame shape and its location is a critical parameter influencing combustion process performance. Combustion process optimization can improve thermal efficiency of existing boilers up to 0.84% [16]. Homogenous temperature field promotes lower emissions of NO_x, CO and minimizes loss on ignition. The plant operators often rely on 3-D simulation tools to acquire information necessary for combustion process improvement. A comprehensive large scale furnace CFD model should be capable of properly predicting the temperature field and peaks associated with high combustion rates.

In the current work a commercial CFD code Ansys Fluent was used to simulate the OP-650 front-fired boiler operation. The boiler is equipped with AGAM (Acoustic Gas Temperature Measuring) system located in a horizontal plane approximately 4 m under the furnace exit. The simulation results were compared with measurements in terms of average temperature as well as temperature profile.

2. Case study boiler and operating conditions

The evaluations were performed for a drum type radiant unit installed at the EDF power plant in Rybnik (Poland) utilizing bituminous coal. The case-study boiler (OP-650) is a front-wall pulverized coal boiler with maximum capacity of 220 MWe. The boiler produces 650 tons of steam per hour. Main/reheat steam temperature is 540 °C. The boiler is a dual-pass type with unique air/fuel supply system.

General scheme of OP-650 boiler low-emission installation with the firing system is depicted in Fig. 1. 12 swirl burners are located on the front wall in two rows (6 in Burner Level I and 6 in Level II). Combustion air is separated into core, primary, secondary and tertiary air as demonstrated in Fig. 2. Swirl vanes fixed at 45° angle are mounted in primary air-coal mixture tube to ensure burner stability. Additional diluted coal-air mixture is provided through 12 drop tubes located in a single row above vortex burners with additional 6 nozzles for optional biomass injection (Burner Level III) as shown in Fig. 1. Approximately 40% of fuel and primary air is transported to the top level. Burners design details are shown in Fig. 2.

Coal is supplied through five pulverizers (notation A to E in Fig. 1). During normal operation one pulverizer is always out of service to reassure undisturbed fuel supply in case of unexpected break-down. OFA (Over-fire air) ports are located on three levels (Fig. 1) to complete the oxidation of any unburned combustibles and assure minimum NO_x formation. In current boiler configuration the OFA I is shut down. 6 OFA II ports and 10 OFA III ports are installed on front and rear wall respectively. Additionally, air-slots installed on front, rear and side walls are used to provide protective air to the membrane walls against over temperature operations and the influence of the combustion products. To minimize unburned carbon in bottom ash and rear wall corrosion 20 air nozzles were installed on the front wall of the ash pit (Fig. 3).

Simulations were conducted for two boiler loads of 135 and 200 MWe. The excess air levels were 15 and 20%, respectively. For lower load two pulverizers were out of service. Operating conditions have been retrieved from plant on-line monitoring system by averaging two-hours measurements during steady-state boiler operation. Air/fuel distribution, temperatures and pulverizers activity are given in Table 1.

In certain conditions, due to dynamic boiler environment, deposits are unevenly formed on the platen superheaters surface. As a result, uneven cooling water injection level between left and right live steam line was noticed. The boiler operator tends to adjust flame shape to compensate water injection imbalance. To cool the specific side of the furnace more over-fire air is injected through nozzles on that side. That was the situation for 135 MWe case. Operator directed more air to the left side of combustion chamber to lower the temperature in that region. The OFA II and OFA III distribution through specific ports is given in Table 2. In 200 MWe case the over-fire air was distributed evenly on each level.