Particuology 16 (2014) 29–40

Contents lists available at ScienceDirect

Particuology

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

Modeling of particle transport and combustion phenomena in a large-scale circulating fluidized bed boiler using a hybrid Euler–Lagrange approach

Wojciech P. Adamczyk[a,*], Gabriel Węcel[a], Marcin Klajny[b], Paweł Kozołub[a], Adam Klimanek[a], Ryszard A. Białecki[a]

[a] Institute of Thermal Technology, Silesian University of Technology, Konarskiego 22, 44-100 Gliwice, Poland
[b] Foster Wheeler Energia Polska Sp. z o.o., Staszica 31, 41-200 Sosnowiec, Poland

ARTICLE INFO

Article history:
Received 28 April 2013
Received in revised form 27 August 2013
Accepted 9 October 2013

Keywords:
CFB
Fluidization
Combustion
Particles
Large boiler
CFD

ABSTRACT

The constantly developing fluidized combustion technology has become competitive with a conventional pulverized coal (PC) combustion. Circulating fluidized bed (CFB) boilers can be a good alternative to PC boilers due to their robustness and lower sensitivity to the fuel quality. However, appropriate engineering tools that can be used to model and optimize the construction and operating parameters of a CFB boiler still require development. This paper presents the application of a relatively novel hybrid Euler–Lagrange approach to model the dense gas–solid flow combined with a combustion process in a large-scale industrial CFB boiler. In this work, this complex flow has been resolved by applying the ANSYS FLUENT 14.0 commercial computational fluid dynamics (CFD) code. To accurately resolve the multiphase flow, the original CFD code has been extended by additional user-defined functions. These functions were used to control the boiler mass load, particle recirculation process (simplified boiler geometry), and interphase hydrodynamic properties. This work was split into two parts. In the first part, which is referred to as pseudo combustion, the combustion process was not directly simulated. Instead, the effect of the chemical reactions was simulated by modifying the density of the continuous phase so that it corresponded to the mean temperature and composition of the flue gases. In this stage, the particle transport was simulated using the standard Euler–Euler and novel hybrid Euler–Lagrange approaches. The obtained results were compared against measured data, and both models were compared to each other. In the second part, the numerical model was enhanced by including the chemistry and physics of combustion. To the best of the authors' knowledge, the use of the hybrid Euler–Lagrange approach to model combustion is a new engineering application of this model. In this work, the combustion process was modeled for air-fuel combustion. The simulation results were compared with experimental data. The performed numerical simulations showed the applicability of the hybrid dense discrete phase model approach to model the combustion process in large-scale industrial CFB boilers.

1. Introduction

Due to their maturity, fluidized bed boilers can be extensively used for various applications, such as gasification, the coating of particles, the drying of solids, combustion, and incineration. To date, fluidization technology has taken on a continuously increasing role in the electricity generation sector. This technology is very attractive because it can be applied to the combustion of low quality coals, biomass, sewage sludge, and waste materials. Due to a relatively low combustion temperature (between approximately 800 and 950 °C), the level of thermal NOx in fluidized bed boilers is considerably reduced compared to that of more frequently used pulverized coal (PC) boilers. This reduction is accomplished

Abbreviations: AR, as received; BC, boundary condition; CFB, circulating fluidized bed; CFD, computational fluid dynamics; DDPM, dense discrete phase model; DO, discrete ordinate; ESSH, external solid super heaters; KTGF, kinetic theory of granular flow; MFIX, multiphase flow with interphase exchanges; MP-PIC, multiphase particle in cell; PSD, particle size distribution; PC, pulverized coal; QMOM, quadrature method of moments; UDFs,, user defined functions; WSGG, weighted sum of gray gases.

* Corresponding author. Tel.: 0048 32 237 2316.
E-mail address: wojciech.adamczyk@polsl.pl (W.P. Adamczyk).

1674-2001/$ – see front matter © 2014 Published by Elsevier B.V. on behalf of Chinese Society of Particuology and Institute of Process Engineering, Chinese Academy of Sciences.
http://dx.doi.org/10.1016/j.partic.2013.10.007

without the introduction of complex burners or additional flue gas treatment facilities. The in situ SO_2 removal is also an important aspect of emission control in fluidized bed boilers.

Significant effort has been devoted to improving numerical tools, such as the computational fluid dynamics (CFD) tool, which is used to model fluidized bed boilers. Numerical tools can also simulate the boiler response to a change of some operational parameters, which can be used to optimize boiler operation. In general, mathematical models used in simulations have been developed based on data collected during physical experiments. These experiments have usually been performed at small-scale facilities (Czakiert et al., 2012; Wang et al., 2010), and their results after analysis have been scaled up to large-scale boilers. The particle transport and combustion processes are computationally very expensive and demanding, mainly due to a high concentration of particle matter and the complexity of the reacting flow. To better understand the circulating fluidized bed (CFB) boiler behavior, detailed information, such as the dynamic mixing effect of gas and solid fuels, the effects of various non-uniform geometries, the impact of coal-feed inlets, air-injection nozzles, and the pressure balance over the whole loop of CFB circulation, should be known. This information can be retrieved from CFD simulations that can even be viewed as a *virtual experiment* when a well-validated numerical model is used.

Currently, CFD simulations are usually limited to a 2D geometry (Wang & Liu, 2010), for which the granular flow is modeled using a standard multi-fluid Euler–Euler model. To better understand the CFB boiler behavior, the 2D geometry should be extended to 3D. Thus, detailed information can be acquired regarding the dynamic mixing effect of gas and solid fuels, the effects of various non-uniform geometries, the impact of coal-feed inlets, air-injection nozzles, and the pressure balance over the CFB boiler. Only a few studies that focused on the 3D full-loop simulation of industrial CFB boilers have been reported in the literature (Hansen & Madsen, 2001; Myohanen & Hyppanen, 2011; Wischnewski, Ratschow, Hartge, & Werther, 2010; Zhang, Lu, Wang, & Li, 2010).

In this work, the particle transport in a large industrial CFB boiler installed in Poland was simulated by applying the ANSYS FLUENT 14.0 commercial CFD package with a relatively new implementation of the hybrid Euler–Lagrange approach, namely, the dense discrete phase model (DDPM) (Cloete et al., 2010). The original code was enhanced with a set of user-defined functions (UDFs). These functions were used for several purposes: the solid phase recirculation procedure, calculation of the suspension density distribution within the combustion chamber, and implementation of the interphase exchange model to ensure a smooth transition between the dense and dilute solid ranges in the CFB boiler. Moreover, the UDFs were applied to control mass discrepancies during the solution procedure, which will be presented later in more detail. The numerical simulations obtained within this work have been divided into two parts. In the first part, the combustion process was not directly simulated. The constant gas density, which was calculated for the average combustion temperature and flue gas composition, was used to maintain the hydrodynamic properties similar to those that occur during combustion in the fluidized bed boiler. To test the sensitivity of the numerical model to changes in the operating conditions of the boiler, two boiler thermal loads were investigated, namely, 80% and 100% of the nominal thermal load. The hydrodynamic conditions within the boiler for both the investigated thermal loads slightly differed from each other due to differences in the injected amounts of the oxidizer and solid mass tracked. The hybrid Euler–Lagrange DDPM approach was used for both of these cases (80% and 100% of the nominal boiler thermal load). Additionally, the 100% load case was investigated using the standard Euler–Euler approach. Mesh independence studies were performed for both the Euler–Lagrange (DDPM) and Euler–Euler approaches. The evaluated results were compared against the measured data collected during normal boiler operating conditions (Blaszczuk, Komorowski, & Nowak, 2012).

The numerical simulations carried out for the *pseudo combustion* conditions were focused on gathering experience on the proper use of the numerical techniques, the selection of the appropriate mesh size and distribution, as well as the time step size used in computations of the unsteady flow. The experience gained was used in later simulations of the multiphase reacting flow. The introduction of all complexities to the reacting flow model at one stage is not likely to result in a stable solution process. In the second part of the research, only the hybrid Euler–Lagrange approach was used to simulate the combustion process for conditions at 100% of the boiler thermal load. The evaluated results were compared with measured data that were acquired during the normal operation of the boiler.

2. Modeling the particle transport phenomena

The gas and solid particle flow in the CFBs has been frequently modeled by techniques known as the multi-fluid approach. A detailed description of this method, as well as its advantages, disadvantages, and limitations can be found in the literature (Gidaspow, 1994; Myohanen & Hyppanen, 2011; Syamlal, Rogers, & O'Brien, 1993; Zhang et al., 2010). The main disadvantage of the multi-fluid approach is a long calculation time required to evaluate the time-averaged solution. Considering the particle size distribution (PSD) can significantly increase this time. An additional dispersed phase must be used for each characteristic diameter. Alternatively, the methods of moments, such as the quadrature method of moments (QMOM) or the direct quadrature method of moments (DQMOM), can be used. However, they introduce additional complexities (Fan, Marchisio, & Fox, 2004; Marchisio, Vigil, & Fox, 2003). Instead of using the Euler–Euler approach, the hybrid Euler–Lagrange DDPM technique can be applied to model the granular flow in CFB units. This approach considers the PSD in a natural way by tracking groups of particles of various sizes. As mentioned earlier, the hybrid Euler–Lagrange DDPM was applied to model particle and combustion processes within a large-scale industrial CFB boiler in this work. The hybrid Euler–Lagrange approach has common roots with the multiphase particle in cell (MP-PIC) technique (Andrews & O'Rourke, 1996; Snider, O'Rourke, & Andrews, 1998). The four-way coupling (Crowe, Schwarzkopf, Sommerfeld, & Tsuji, 2012) was taken into account to represent the relationship between the continuous and dispersed phases in the mass, momentum, and energy transfer. The interactions between particles were taken into account by using models based on the kinetic theory of granular flow (KTGF) (Gidaspow, 1994). The influence of the particle movement and energy transfer between the particles and the fluid carrier were considered in the conservation equations of the continuous phase via additional source terms. The data were transferred between the Eulerian grid and particle position by the use of interpolation operators (Andrews & O'Rourke, 1996; Snider, 2001). The interpolation method was utilized to resolve the solid stresses, which were difficult to calculate for each particle in the dense flow. The gradient of solid stress could be assumed to be calculated on the Eulerian grid. Thus, its value could be interpolated to the discrete particle position (Andrews & O'Rourke, 1996; Snider et al., 1998).

In the hybrid Euler–Lagrange approach, the motion of each individual particle is not solved. Instead, groups of particles, or parcels, are tracked. Each parcel contains several particles of identical mass, velocity, position, temperature, composition, etc. The number of individual particles contained in the injected parcel can be easily calculated from the following: