Particuology xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Particuology

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

Numerical approach for modeling particle transport phenomena in a closed loop of a circulating fluidized bed

Wojciech P. Adamczyk[a,*], Paweł Kozołub[b], Grzegorz Kruczek[a], Monika Pilorz[a], Adam Klimanek[a], Tomasz Czakiert[c], Gabriel Węcel[a]

[a] Institute of Thermal Technology, Silesian University of Technology, Gliwice, Poland
[b] AMEC Foster Wheeler, Staszica 31, 41-200 Sosnowiec, Poland
[c] Institute of Advanced Energy Technologies, Częstochowa University of Technology, Częstochowa, Poland

ARTICLE INFO

Article history:
Received 3 August 2015
Received in revised form 9 December 2015
Accepted 11 December 2015
Available online xxx

Keywords:
Fluidization
CFB
Numerical modeling
Multiphase flow
Particle transport
Cyclone

ABSTRACT

Numerical modeling of a large scale circulating fluidized bed (CFB) imposes many complexities and difficulties. Presence of a dense solid phase, a variety of spatial and time scales as well as complex model geometries requires advanced numerical techniques. Moreover, the appropriate selection of a numerical model capable of solving granular flow, and geometrical model simplification can have a huge impact on the predicted flow field within the CFB boiler. In order to reduce the cost of the numerical simulations, the complex CFB boiler geometry is reduced to that of the combustion chamber. However, a question arises as to how much one can simplify the geometrical model without losing accuracy of numerical simulations. To accurately predict the gas–solid and solid–solid mixing processes within subsequent sections of the CFB boiler (combustion chamber, solid separator, drain section), a complete 3D geometrical model should be used. Nevertheless, because of the presence of various spatial and temporal scales within subsequent boiler sections, the complete model of the 3D CFB boiler is practically unrealizable in numerical simulations. To resolve the aforementioned problems, this paper describes a new approach that can be applied for complete boiler modeling. The proposed approach enables complex particle transport and gas flow problems within each of the boiler sections to be accurately resolved. It has been achieved by dividing the CFB boiler geometry into several submodels, where different numerical approaches can be used to resolve gas–solid transport. The interactions between computational domains were taken into account by connecting the *inlets/outlets* of each section using a set of user-defined functions implemented into the solution procedure. The proposed approach ensures stable and accurate solution within the separated boiler zones.

Introduction

To understand how a fluidized bed installation works, experiments and numerical simulations are usually performed applying 2D models for testing and development of the mathematical model. Afterwards, those models are used for modeling the particle transport phenomena (Cloete, Johansen, & Amini, 2012; Cloete, Johansen, & Amini, 2015) in pilot and industrial CFB boilers. A high concentration of the solid phase suspended in the gas phase significantly influences the particle behavior and combustion processes. To tackle this complexity, special numerical techniques and solution procedures have to be used. In order to better understand the CFB boiler behavior, a 2D geometry should be extended to a 3D model, which allows the prediction of a strong non-uniform dynamic mixing process between phases. The literature reports only a few cases devoted to 3D full-loop simulation of the industrial CFB boilers. In Zhang, Lu, Wang, and Li (2010), the application of the standard Euler–Euler approach for modeling isothermal flow within large scale CFB boiler was presented. Shah, Myöhänen, Kallio, and Hyppänen (2015a) and Shah, Myöhänen, Kallio, Ritvanen, and Hyppänen (2015b) also applied Euler–Euler model for modeling two large-scale industrial CFB boilers where the mesh sensitivity study was performed together with implementation of the subgrid-scale and energy-minimization multi-scale (EMMS) drag models (Wang et al., 2010). The applicability of an EMMS model to large boilers was also studied in Lu et al.

* Corresponding author. Tel.: +48 32 237 2316.
E-mail address: wojciech.adamczyk@polsl.pl (W.P. Adamczyk).

http://dx.doi.org/10.1016/j.partic.2015.12.006

(2013). Wischnewski, Ratschow, Hartge, and Werther (2010) have shown applicability of a standard two fluid model for modeling the combustion process. Some application of a semi empirical model for predicting the combustion process within industrial CFB boilers has been presented (Bordbar, Myöhänen, & Hyppänen, 2015; Myöhänen & Hyppänen, 2011).

The main problem encountered when developing the numerical model of the CFB boiler are the large differences of the spatial and temporal scales, which are directly related to the boiler zones (e.g. combustion chamber, cyclone, and external solid heaters). Using a closed loop of the CFB boiler geometry it is assumed that within each of the boiler sections, the same time steps, mathematical models responsible for predicting particle–particle interactions and particle–wall interactions, as well as turbulent model have to be used. These introduce errors and very often destabilize numerical simulations. In order to circumvent these difficulties, each boiler section has to be simulated separately. Nevertheless, in that approach the influence and interactions between modeled boiler sections are not taken into account. In order to deal with these difficulties a new computational technique was proposed. The idea is as follows: the closed loop of the boiler can be divided into the several subsections; each section can be modeled separately. The results from one section can be used as the inlet boundary condition for the next section. After several iterations of the solution procedure, the results should be stabilized and a pseudo-steady state solution within each of the boiler section should be achieved.

In this paper, a detailed description of the solution procedure capable of solving a closed loop of the CFB boiler without losing accuracy is provided. The numerical simulations were carried out using a geometrical model of the pilot-scale CFB installation located at Częstochowa University of Technology (Czakiert, Sztekler, Karski, Markiewicz, & Nowak, 2010). The particle transport phenomena was modeled using the hybrid Euler–Lagrange model which was successfully applied for modeling particle transport in a pilot scale CFB installation (Adamczyk, & Klimanek et al., 2014). This approach was also applied for modeling the air- and oxy-fuel combustion process within a large scale CFB boiler (Adamczyk, & Węcel et al., 2014; Adamczyk et al., 2015). In addition, the impact of the geometrical model simplification on the predicted temperature field was investigated within the aforementioned papers. The calculation procedure was implemented using the ANSYS FLUENT commercial computational fluid dynamics (CFD) package, by applying a set of user-defined functions (UDFs).

Experimental facility

A schematic diagram of the experimental facility is shown in Fig. 1. The combustion chamber (riser pipe) is 4.98 m high with the inner diameter equal to 9.8 cm. The ports of the secondary gas inlet are located 55 cm above the distributor. The experimental rig works as typical CFB unit. The part of the material that is separated from the cyclone is re-circulated back to the combustion chamber through downcomer and drain section, where solid material is aerated.

In the present work, an isothermal (cold) experiment was carried out in order to measure the pressure drop over the pilot-scale installation. During the experiment, the atmospheric air was used as fluidizing gas with a constant temperature of 23 °C. The total amount of fluidizing gas delivered to the unit was equal to 120 kg/h. The majority of the gas was delivered to the primary gas inlets below the distributor in the bottom part of the riser, and the remaining part of the gas was directed to the drain section for particle aeration. The total amount of circulated solid material in the CFB pilot was approximately 5 kg.

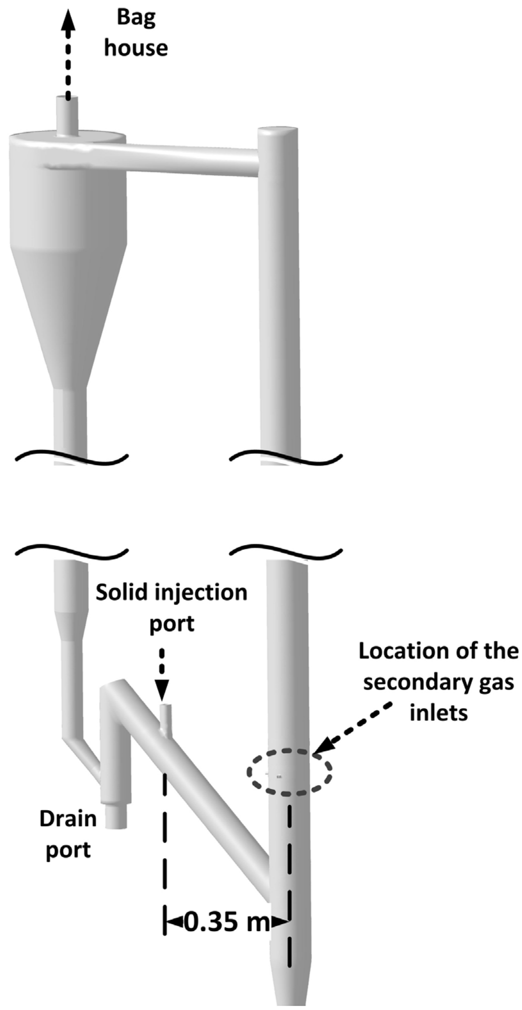

Fig. 1. Scheme of the CFB experimental rig.

Geometrical models of the simulated CFB sections

The geometrical model of the CFB pilot installation was divided into three main computational domains: riser, solid separator, and drain section.

Riser geometry

The riser geometry was limited to the combustion chamber, and part of the recirculation and outlet sections. The riser geometry is shown in Fig. 2. The 3D numerical mesh used for the riser simulations is built using 95,000 numerical cells. To reduce the cost of the numerical simulations, the geometry of the distributor located at the bottom part of the riser was not taken into account, and the primary gas enters the computational domain through whole bottom area. The gas flow rate injected to the riser geometry was set to 106.4 kg/h and it was split into the gas delivered through the distributor (95%) and solid injection port (5%). For the riser geometry, the amount of fluidizing gas delivered through the recirculation inlet was prescribed by a uniform velocity profile using a UDF.

Solid separator geometry

The cyclone geometry and generated mesh are illustrated in Fig. 3. The total number of numerical cells in this section was equal to 92,000 elements. The majority of grid elements were hexahedral; however, some wedge elements in the scroll inlet zone were