Energy 143 (2018) 219–240

Contents lists available at ScienceDirect

Energy

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

Generation of data sets for semi-empirical models of circulated fluidized bed boilers using hybrid Euler-Lagrange technique

Wojciech P. Adamczyk [a, *], Kari Myöhänen [b], Ernst-Ulrich Hartge [c], Jouni Ritvanen [b], Adam Klimanek [a], Timo Hyppänen [b], Ryszard A. Białecki [a]

[a] Institute of Thermal Technology, Silesian University of Technology, 44-100 Gliwice, Konarskiego 22, Poland
[b] LUT School of Energy Systems, Lappeenranta University of Technology, Post Office Box 20, FI-53851 Lappeenranta, Finland
[c] Institute of Solids Process Engineering and Particle Technology, Hamburg University of Technology, Denickestrasse 15, 21073 Hamburg, Germany

ARTICLE INFO

Article history:
Received 2 May 2017
Received in revised form
23 September 2017
Accepted 7 October 2017

Keywords:
Hybrid Euler-Lagrange
Fluidization
Combustion
Particles
CFB boiler
Semi-empirical model

ABSTRACT

The primary goal when designing a CFB boiler is to ensure its high energy efficiency. Additional constraints come from environmental regulations, which are constantly becoming more difficult to assess, and require improvements in the boiler design process. Local on-site measurements are often restricted to short distances inward from the furnace wall, leaving most of the core unmeasured. Computer simulations are practically the only feasible tools to investigate the combustion processes and to support the design of CFB units. Three-dimensional steady-state semi-empirical CFB furnace models predict adequate results within reasonable times but these models require tuning to existing objects, which obviously excludes their application for new object design. The present study represents a step toward using the hybrid Euler–Lagrange (HEL) technique to model flow variables in order to replace the measured data. The information collected using the HEL model for an arbitrary furnace can be used to tune a semi-empirical model. To validate the HEL model, measurements of a 235 MWe CFB were used. The simulations show good agreement between the numerical and experimental results, indicating that the presented idea is worth investigating in the future.

1. Introduction

Fluidization technology has found various applications in different industries, e.g., power generation, petroleum, chemical, and food processing. The application of fluidized beds in the energy sector gives significant fuel flexibility of the units, with good efficiency and performance factors. The main variants of fluidization are circulating fluidized beds (CFBs) and bubbling fluidized beds. The present work is concentrated on the CFB boiler. In the CFB boiler, the combustion process proceeds at very uniform and relatively low temperature (the average flue gas temperature is maintained at the level of 800–900 °C) preventing the formation of thermal NOx [1,2] and ash melting. Moreover, the direct injection of limestone in the combustion chamber bounds the sulfur dioxide already during the fuel combustion [3,4], which is also seen as a positive feature of this type of boiler.

Increasingly stringent environmental regulations are forcing the power generation sector to improve the efficiency of fuel-conversion processes and limit the emission of pollutants to the atmosphere. Various primary and secondary technologies are applied to deal with these requirements. The price to pay is an efficiency penalty and the emission of other harmful substances such as ammonia in the case of non-selective catalytic reduction technologies. As an alternative to the post combustion CO_2 absorption, the retrofitting of existing air-fired boilers to oxy-combustion mode is also a topic of research [5].

CFB boilers, owing to their ability to combust various types of fuels, are becoming popular devices used in waste utilization and biomass combustion. To design new units or efficiently upgrade existing ones, reliable simulation tools are necessary, with computational fluid dynamics (CFD) models being the most natural choice. The role of the experiments is to generate data to tune the submodels and validate the results of the simulations of pilot-scale and finally large-scale installations. Owing to the presence of various spatial and temporal scales, strong coupling between the gaseous and solid phases, and interactions between the particles themselves, building a reliable mathematical model of CFB boilers

* Corresponding author.
E-mail address: wojciech.adamczyk@polsl.pl (W.P. Adamczyk).

https://doi.org/10.1016/j.energy.2017.10.029
0360-5442/

is not a trivial task. The presence of combustion, gasification, and other reactions makes the task even more demanding. Another important aspect that needs to be accounted for is the heat transfer mode, mainly from the solid phase to the boiler membrane walls. To predict this feature, a technique based on fuzzy logic has been applied [6,7]. Another option is to resort to CFD, where this energy exchange mode arises in a natural way.

The available computational techniques capable of dealing with such a complex process are divided into three main groups:

- macroscale empirical and semi-empirical models,
- meso- and macroscale multifluid models like Euler–Euler models,
- macroscale hybrid Euler–Lagrange models.

In spite of the availability of various computational techniques, validated examples of modeling the complex fluidization process in large-scale industrial CFB boilers are very scarce.

The empirical and semi-empirical models were used for comprehensive simulation of industrial-scale CFB units, i.e., modeling all the affecting phenomena: fluid dynamics, reactions, heat transfer, and comminution. These models have often been applied in different lower dimensions (0D … 2D) for fast modeling of the CFB process [8–14]. The three-dimensional semi-empirical model approaches are more rare [15–17]. The 3D models are computationally more demanding, but they can provide more detailed data of the local phenomena inside the furnace and the 3D approach is a prerequisite to simulate the incomplete lateral mixing in large-scale furnaces. The prediction capability of the semi-empirical 3D models is limited by the available measurement data for model validation and they cannot be reliably applied to new, deviant designs.

Euler–Euler simulations are based on submodels and as such do not require measurements of the existing large-scale objects [18–20]. The basic assumption of the Euler–Euler model is that the gas and solid phases are both treated as interpenetrating continua. In order to resolve the particle size distribution, several granular phases have to be tracked, which is computationally expensive. In the course of combustion, the diameters of the particles change, introducing additional complexity associated with the varying mass of the solid phases. Some difficulties are connected with time steps of the order of 1 ms to capture the fast movements of the solid phase and ensure stable numerical procedures. Yet another challenge is that some of the phenomena, e.g., sorbent reactions and comminution, may have very long time constants, which would require long calculations to simulate them correctly.

In various applications, such as modeling pulverized coal combustion, the detailed interactions between particles can be simply omitted. In these conditions, the low volume fraction of the solid phase can be treated as discrete particles, which flow in the continuum formed by the gas phase. The standard Euler–Lagrange (EL) approach fits these features. The application of this model allows to model even large-scale systems with moderate calculation effort [21–25]. Nevertheless, for dense-phase systems, such as a CFB furnaces, this approach is not applicable.

The hybrid EL (HEL) method, which in the Ansys FLUENT implementation has been named dense discrete phase model (DDPM), can be seen as an optimal technique to deal with dense granular transport with reactions. With this method, the simulation of multiple solid phases, e.g., fuel, ash, and sorbent, with appropriate particle size distributions, can be handled without a large increase in computational effort. This is a clear advantage over the standard EE approach, in which a large number of solid phases leads to prohibitively long calculation times. An additional benefit of this approach is the possibility to model the combustion process at the single-particle level, for which experimental data can be readily retrieved. The applicability of the HEL approach for predicting granular flow structures and combustion phenomena has been widely discussed [26–28]. The application of the HEL technique for the optimization of the coal feeder position within an industrial-scale CFB unit has been presented [29]. As in the case of the Euler–Euler model, transient simulations are necessary, but the admissible time step length is of approximately two orders of magnitude larger.

This study investigates the possibility of increasing the applicability of a semi-empirical 3D model with the support of the HEL method. The target model, which is called CFB3D, was presented by Myöhänen and Hyppänen [30]. The CFB3D model requires some empirical input parameters, e.g., data of the total solid distribution in a CFB furnace. Normally, this data is retrieved from measurements carried out at an existing CFB boiler [31]. Then, the model can be applied to study the effect of certain limited operating parameters [32]. The great advantage of this approach is the very short execution time, on the order of a few hours, making it applicable to practical optimization, sensitivity analysis, and parametric studies. However, the inaccuracy and uncertainty of the semi-empirical model increases if the modeled unit, or at least a similar one, has not yet been operated. In this case, the empirical data for the semi-empirical model tuning is not available.

The solution is to feed the CFB3D model with results of simulations obtained from more fundamental CFD simulations based on submodels. This, however, requires access to a validated simulation tool. The aim of the present work is to produce and validate modeling of a CFB furnace with the HEL method using simplified modeling of reactions. Such a model provides spatial and temporal distribution of hydrodynamics within the furnace that can be used as input data for the time-efficient CFB3D with more comprehensive modeling of chemistry for combustion and emissions. The main input data for the CFB3D model are the volume fraction fields of solids, as they have a large effect on other solved variables. In the first stage, these data can be implemented directly and use the existing submodels in CFB3D to solve the gas and solid flow fields and other process data. In the next stage, the flow fields solved by the HEL method could be utilized as well. However, this requires some additional work to ensure that the local and global mass balances are satisfied in the CFB3D model with a different computational mesh compared with the HEL model. The integration of the HEL and semi-empirical CFB3D models will be carried out in the next step of the research project.

2. Numerical model of the CFB boiler

The numerical model concerns the lignite-fired 235 MWe CFB boiler of the Turow Power Plant Unit 3. The furnace has a cross-section of 21×10 m^2 and a height of 43 m. In June 2001, the unit was investigated in an extensive measurement project in collaboration with several universities and research organizations [33]. The experimental data that were applied for validation of the present modeling study were reported in Refs. [34–38]. The measurements included particle size distribution, solid flux, solid momentum, volume fraction of solids, pressures, temperatures, and gas concentrations. The unit has been previously modeled by semi-empirical approaches [39] and by an Eulerian–Eulerian calculation of fluid dynamics [19].

Owing to the variation of spatial and temporal scales as well as the nonuniform distribution of solid material in subsequent sections of a CFB boiler, it is not recommended to simulate a closed loop of a CFB unit. Various difficulties can arise during solution procedure. To simplify and to stabilize the numerical simulations, the computational domain was limited to the combustion chamber.