Applied Thermal Engineering 76 (2015) 344–356

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research paper

Coupling of a radiative heat transfer model and a three-dimensional combustion model for a circulating fluidized bed furnace

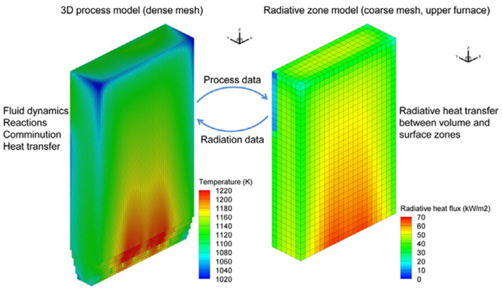
Mohammad Hadi Bordbar[*], Kari Myöhänen, Timo Hyppänen

Department of Energy Technology, Lappeenranta University of Technology, Lappeenranta, Finland

HIGHLIGHTS

- Radiative zone model is used to analyze a large scale CFB furnace.
- A semi-empirical model for CFB processes is presented.
- The radiative effect of long distance is taken into account.
- The geometric optic is used for radiative properties of particles.
- The WSGGM is used for radiative properties of combustion gases.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Received 2 July 2014
Accepted 5 November 2014
Available online 14 November 2014

Keywords:
Radiation model
Zonal method
Circulating fluidized bed
Combustion
Numerical modeling
Oxygen fired combustion

ABSTRACT

A 3D semi-empirical model for reactive two-phase flow in a circulating fluidized bed furnace (CFB3D) is modified by implementing the radiative zone method to solve the radiation heat transfer. The radiative properties of the gas and particle phase have been calculated using detailed information of gas and particle distribution obtained from the CFB3D model. A recently published WSGGM for oxygen-fired combustion has been used to calculate the absorption coefficient of gaseous combustion products. The results of implementing the radiative zonal approach have been compared with those obtained using empirical radiative correlations. The temperature field obtained by using the radiative zone method is more uniform than the one obtained by empirical correlation, and the total heat flux to the wall is slightly higher. The long distance effect of radiation has been found more important in the upper furnace where the gas is the dominant phase. Detailed discussion concerning the obtained results is presented.

1. Introduction

Circulating fluidized bed (CFB) combustion technology is known for its many advantages, including fuel flexibility, good combustion efficiency, and low emissions. The typical unit sizes are on the order of 20–200 MWe, but recently, the maximum unit capacities have reached 550 MWe utilizing supercritical steam conditions [1,2]. Currently, CFB technology is being developed for new cleaner energy production approaches, such as oxygen-fired combustion [3,4] and chemical looping combustion [5,6], which enable the capture and storage of carbon dioxide from flue gas.

CFB furnaces are typically coupled with several complex physical phenomena, such as chemical reactions, turbulent two phase flow, phase transition, attrition of particles, and heat transfer. A proper understanding of heat and mass transfer in CFB furnaces is essential in order to obtain better designs and performance for this type of system. The physical phenomena, such as gas particle flow,

---

[*] Corresponding author. Laboratory of Modeling of Energy Systems, Department of Energy, Lappeenranta University of Technology, FIN-53851 Lappeenranta, Finland. Tel.: +358 400 38 7607; fax: +358 5 6216399.
E-mail address: Bordbar@lut.fi (M.H. Bordbar).

http://dx.doi.org/10.1016/j.applthermaleng.2014.11.008
1359-4311/© 2014 Elsevier Ltd. All rights reserved.

turbulence, chemical reactions, and heat transfer, are individually difficult phenomena to predict and analyze. This analysis is more difficult and challenging when the phenomena are coupled in large-scale systems, such as CFB boilers. Fig. 1 schematically shows how different aspects of modeling are coupled in CFB furnaces.

The modeling methods for CFB furnaces can be classified as fundamentals-oriented and practice-oriented models [7,8]. In the fundamentals-oriented models, the modeling of the fluid dynamics is attempted with the currently available fundamental theories, such as the Eulerian–Eulerian multiphase models applying the kinetic theory of granular flow [9]. In the practice-oriented models (or engineering models), the theories are simplified and adjusted by empirical correlations to improve the calculation speed and fit the model results with the measured data.

Due to their high computational cost, the application of the fundamentals-oriented models for simulation of industrial-scale CFB furnaces has been small. Moreover, most of the large-scale 3D studies have been limited to modeling of the fluid dynamics only [10–12]. The following paragraphs present the few published studies about large-scale 3D simulations that include chemical reactions and heat transfer.

Weng et al. [13] simulated a 250 MWth unit applying the MP-PIC (multiphase particle in cell) method, which is a hybrid Eulerian–Lagrangian approach [14]. The model assumed instantaneous evaporation and devolatilization of fuel. The modeled reactions included gasification and combustion of char (as carbon), shift conversion, combustion of devolatilized hydrocarbons, and combustion of carbon monoxide. Temperature profiles were simulated as well, but the applied heat transfer model was not described.

Adamczyk et al. [15] applied a DDPM model (dense discrete phase model) to simulate a 460 MWe CFB. The DDPM model was generated with a modified MP-PIC method, which has been implemented using the commercial CFD software of Ansys Fluent. The system of reactions was simplified to heterogeneous combustion of char (as carbon) and homogeneous combustion reactions of devolatilized hydrocarbons and carbon monoxide. The solution of the energy balance included the heat due to reactions and the heat transfer between particles, gas, and the boiler walls. The calculation times with 2 × 4 core processors were on the order of 3 months.

Zhang et al. [16] recently demonstrated an application of a EMMS model (energy minimization multi-scale) for calculating combustion in a 150 MWe unit. The reaction system was simplified to combustion of carbon, and the thermal effect of evaporation and devolatilization was considered to be an average heat sink, which affected the lower part of the furnace.

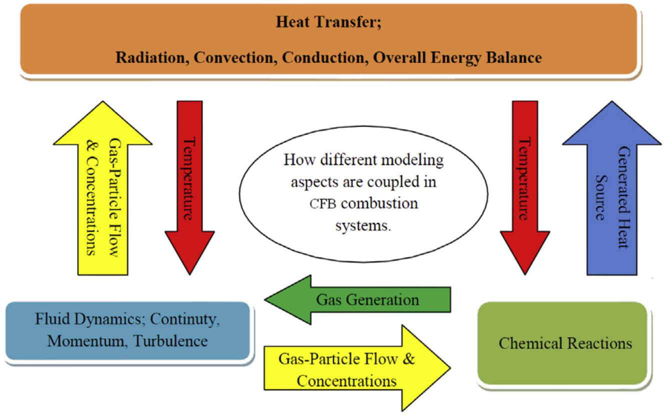


Fig. 1. Coupling of different modeling aspects in CFB combustion systems.

The calculation times for large-scale furnaces with fundamentals-oriented models are unfeasibly long. Thus, the empirical and semi-empirical models are used for comprehensive calculations, which include the simulation of reactions and heat transfer in addition to multiphase flow. A common method is a core-annulus approach in which the gas–solid suspension is divided into dilute (core) and dense (annulus) sections. These models are often referred as 1.5D models [17,18].

Only a few semi-empirical models, which solve the CFB furnace in three dimensions, have been published. One model, which was applied in this study, was originally developed by Hyppänen et al. [19] and later updated by Myöhänen [20] (CFB3D model hereafter). Other similar comprehensive 3D models have been presented by Wischnewski et al. [21], Ratschow et al. [22] and Pallarès et al. [23]. In these models, the modeling of fluid dynamics is simplified, but the description of the combustion chemistry can be quite detailed, including the different heterogeneous and homogeneous gasification and combustion reactions. The energy equation solution generally consists of terms related to reaction heats, convection of gas and solids, heat transfer to walls, and mixing of energy by diffusive flux [20,22].

In all of the above reviewed three-dimensional models, the radiative heat transfer inside the furnace is not solved separately, but it has been included in the overall heat flux between the suspension and the walls. In a CFB furnace, the radiative heat transfer occurs between particles, radiative gases, and heat transfer surfaces [24]. Under full load conditions and especially at the lower regions, the solid concentration inside the furnace is high, and the suspension can be considered optically thick, which simplifies the treatment of the radiative heat transfer. However, under low load conditions, the solid concentration at the upper part of the furnace can be small: the conditions can be comparable with a freeboard of a bubbling fluidized bed boiler. Moreover, in oxygen-fired combustion, the share of the radiative gases (CO_2 and H_2O) is considerably higher (>90%) than in air-fired conditions (<30%), which will affect the radiative heat transfer. The higher absorbing gas concentration in oxygen-fired systems limits the use of previously developed gas radiative properties models, which have been basically developed to support air-fired combustion systems [25].

Consequently, a more detailed treatment of the radiative heat transfer may be required for accurate prediction of the heat flux to furnace walls. Especially in large supercritical once-through units, the heat flux distribution should be known in all conditions to avoid problems with steam circuits.

The detailed radiative models are based on solving an integro-differential radiative transfer equation (RTE). Except for a few very simple problems, such an equation cannot be solved analytically and instead must be solved using approximate solution methods. In the multiflux and discrete ordinate methods, the continuous angular variation of the radiation intensity is discretized into a finite number of directions, and the RTE is transformed into a set of differential equations. In the Monte Carlo method, the radiative heat flux is simulated by randomly sampling the passage of photons through a medium until they are absorbed or escaped. In the zone method, the calculation domain is divided into a finite number of volume and surface area zones. An energy balance is then performed for the radiative heat exchange between different zones using pre-calculated exchange areas. The zone method was first developed by Hottel and Sarofim [26]. It has been further developed, used and modified by many researchers; see for instance [27,28]. To the best of the authors' knowledge, this method has not been applied to circulating fluidized bed furnaces until this study.

Several models have been published for simulating the heat transfer mechanism and the radiative heat transfer in a CFB. What