Applied Energy 160 (2015) 489–501

Contents lists available at ScienceDirect

Applied Energy

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

Three-dimensional full-loop simulation of a dual fluidized-bed biomass gasifier

Hui Liu [a], Robert J. Cattolica [a,*], Reinhard Seiser [a], Chang-hsien Liao [b]

[a] Department of Mechanical and Aerospace Engineering, University of California, San Diego, 9500 Gilman Drive, La Jolla, CA 92093, USA
[b] West Biofuels, LLC, Woodland Biomass Research Center, 14958 County Road 100B, Woodland, CA 95776, USA

HIGHLIGHTS

- CFD simulation of biomass gasification in a dual fluidized-bed.
- The CFD model predicts the gas composition and the reactor temperature distribution.
- The CFD model has been validated by experimental data.
- The effects of the particle size distribution and drag models have been investigated.

ARTICLE INFO

Article history:
Received 12 May 2015
Received in revised form 10 September 2015
Accepted 15 September 2015

Keywords:
Biomass gasification
Fluidization
CFD modeling

ABSTRACT

A three-dimensional CFD model was developed to simulate the full-loop of a dual fluidized-bed biomass gasification system consisting of a gasifier, a combustor, a cyclone separator, and a loop-seal. This full-loop simulation includes the chemical kinetic modeling of biomass drying and pyrolysis, heterogeneous char reactions, and homogeneous gas-phase reactions. In the model, the gas phase is described using Large Eddy Simulation (LES) and the particle phase is described with the Multiphase Particle-In-Cell (MP-PIC) method. The simulation was performed using the GPU-accelerated computing and the simulation results were compared with the gas composition and temperature measurements from a pilot-scale biomass gasification power plant (1 MW_{th}, 6 tons biomass/day). The independence of the accuracy of the model on mesh resolution and computational particle number was determined. The impacts of the particle size distributions (PSD) and drag models on the reactive flows were also investigated.

1. Introduction

Fossil fuels are the primary energy source in industry. These natural resources, however, are limited and will be depleted in the future. Biomass as a renewable energy source can be an alternative to fossil fuels [1–5]. Biomass resources are abundant and can be derived from many sectors such as agricultural residues, food waste, and industrial by-products [6].

Bioenergy can be released from biomass through thermal conversion technologies such as pyrolysis, gasification, and combustion [7,8]. Among these technologies, biomass gasification is an attractive option, because it can generate heat and can also be applied to produce syngas for electricity generation and chemical synthesis. A variety of gasification technologies such as fixed-bed, fluidized-bed, and entrained-flow gasifiers have been developed and applied in various industries [9–11].

Compared to other types of gasification processes, fluidized-bed gasification is attractive due to its efficient mass and energy transfer [12–15]. However, because of the complexity of gas-particle interactions and gasification reaction kinetics, designing fluidized-bed gasifiers is arduous. In recent years, owing to the developments of computer technologies, computational fluid dynamics (CFD) is now capable of simulating biomass gasification to assist with process design, scale-up, and optimization. Currently, there are mainly three CFD methods for the simulations of fluidized-bed biomass gasifiers: the Eulerian–Eulerian (EE) approach, the Eulerian–Lagrangian (EL) approach, and the hybrid Eulerian–Lagrangian approach.

In the Eulerian–Eulerian approach, the particle phase is treated as a continuum. The Eulerian–Eulerian approach requires less computing power because it treats particles as a continuous phase and does not track each of them. Due to its computational effectiveness, this method can be used to simulate large-scale fluidized-bed

* Corresponding author. Tel.: +1 858 5342984.
E-mail address: rjcat@ucsd.edu (R.J. Cattolica).

http://dx.doi.org/10.1016/j.apenergy.2015.09.065
0306-2619/

Nomenclature

A_p	particle surface area (m^2)
$C_{p,i}$	concentration of particle species i ($kmol/m^3$)
C_V	specific heat (kJ/(kg K))
D_t	turbulent mass diffusivity (m^2/s)
D_p	aerodynamic drag function
E	Enthalpy (kJ/kg)
f	particle size distribution function
F	interphase force between the gas and particle phases
g	gravity (m/s^2)
k_d	the thermal conductivity of the particle phase (W/(m K))
δm_p	mass source term ($kg/(m^3\ s)$)
Mw	molecular weight (kg/mole)
Nu	Nusselt number
Re	Reynolds number
u	velocity (m/s)
V	computational cell volume (m^3)
Y_i	mass fraction of gas species i

Greek symbols

α	volume fraction
δ_{ij}	unit tensor
λ_{mol}	the molecular conductivity of the gas phase (W/(m K))
λ_{eddy}	the turbulent conductivity of the gas phase (W/(m K))
ρ	density (kg/m^3)
τ	shear stress tensor ($kg/(m\ s^2)$)
τ_D	particle collision damping time (s)
μ_{lam}	laminar viscosity (m^2/s)
μ_t	turbulent viscosity (m^2/s)

Subscripts

c	char
cp	close packing
g	gas phase
$i,\ j$	coordinate index
p	particle phase

reactors. The EE method, however, has limitations. Because of the assumption of the continuous solid phase, the particle diameters in one solid phase must remain the same and cannot change during the simulation [16]. This can be a serious problem for the simulations of biomass gasifiers in which particle diameters change significantly due to particle surface reactions.

The EL approach can be a better option, because each particle is tracked and has its own properties such as diameter, density, and temperature. The simulations using the EL method, however, are time-consuming. The calculations for particle collisions in dense phase require an enormous amount of computational resources. Therefore, the EL approach may not be suitable for the simulations of industrial fluidized-bed reactors which generally contain millions or billions of particles [17,18].

To simulate dense particle flows more efficiently, a hybrid Eulerian–Lagrangian approach, the Multiphase Particle-In-Cell method (MP-PIC), was developed by Andrews and O'Rourke [19]. In this method real particles are grouped into computational particles and then each computational particle is tracked. In the MP-PIC method one computational particle can represent hundreds or thousands of real particles. The particles defined in one computational particle share the same size, density, velocity, and temperature. Compared to the general EL approach, the MP-PIC method is more computational-efficient.

Furthermore, unlike the EL approach in which particle collisions are calculated by the particle collision models, the effect of particle collision in the MP-PIC method is described by an isotropic solid stress, a function of solid volume fraction [19,20]. This technique avoids intense computation for particle collisions and saves a significant amount of computing time. There are also limitations in the MP-PIC method. This method is not suitable for the simulation of particle bridging, de-fluidized beds, and non-aerated hopper flows in which the direct collisions and inter-particle contacts are critical, because in the MP-PIC method the interactions of particles are calculated with a solid stress model, rather than the collision models. For such cases, the general EL method may be a better option.

Numerous CFD models using the EE, EL, and hybrid EL approaches were previously developed to simulate fluidized-bed gasifiers, but most of them were only focused on one key-component of the fluidized bed system such as a gasifier [20–28]. Other components of the fluidized-bed system such as the cyclone separator and the loop-seal were neglected. The interactions between the key components were simplified as inlets or outlets with the fixed conditions. This simplification can cause serious errors, especially for the systems that consist of multiple reactors and cyclone separators [29]. The best solution to the problem is to simulate the full-loop of fluidized-bed system, instead of a part of the system.

Recognizing the limitations of the single-component approach, researchers have recently focused on simulating the full-loop of fluidized-bed system to improve the model accuracy. Nguyen et al. [30] developed a 2D Eulerian–Eulerian model to study the solid circulation in the full-loop of a dual fluidized-bed system. Wang et al. [31] built a 3D model to simulate the hydrodynamics in a circulating fluidized-bed using the EE approach. Other researchers have conducted similar studies by simulating the full-loop of the fluidized-bed system [32–34].

It should be noted that all of the previous full-loop models are "cold models" in which no chemical reactions were considered. Consequently, these models can only be applied to study the hydrodynamics and cannot be utilized to predict the gas production in the gasifier. Currently, "hot" or "reactive" models that simulate the full-loop of a fluidized-bed biomass gasifier have not been demonstrated.

The purpose of this work is to build a model that can simulate both the hydrodynamics and chemical reactions for a dual fluidized-bed system. To provide more comprehensive insight to the design of fluidized-bed gasifiers, a three-dimensional CFD model for a pilot-scale (6 tons/day, 1 MW_{th}) power plant is developed. In this model, the full-loop of a dual fluidized-bed biomass gasification system including a gasifier, a combustor, a cyclone separator, and a loop-seal is simulated using the MP-PIC method. The kinetics of biomass drying and pyrolysis, heterogeneous char combustion and gasification, and homogeneous gas-phase reactions are all included in this model. The momentum, mass, and energy transport equations are coupled with the reaction kinetics to predict the gas production, particle circulation, and reactor temperature within the dual fluidized-bed gasification system.

The predicted gas composition and reactor temperature profiles are compared with experimental data from the pilot power plant for model validation. Case studies of mesh resolution and particle