Advanced Powder Technology xxx (2018) xxx–xxx

Contents lists available at ScienceDirect

Advanced Powder Technology

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

Original Research Paper

Verification of optimal models for 2D-full loop simulation of circulating fluidized bed

Yujian Lu [a], Yefeng Zhou [a,*], Lei Yang [a], Xiayi Hu [a], Xiao Luo [b], Hongbo Chen [a,*]

[a] College of Chemical Engineering, Xiangtan University, Xiangtan 411105, PR China
[b] College of Chemistry and Chemical Engineering, Hunan University, Changsha 410082, PR China

A R T I C L E I N F O

Article history:
Received 31 January 2018
Received in revised form 27 June 2018
Accepted 28 July 2018
Available online xxxx

Keywords:
Full loop simulation
Circulating fluidized bed
Turbulent model
Drag model

A B S T R A C T

To verify the optimal models for a two-dimensional (2D) full loop simulation of a circulating fluidized bed (CFB), different turbulent models and drag models are studied according to relevant pressure profile, voidage distribution and particle collision energy. With regard to a laminar model and turbulent models including Standard k-ε, RNG k-ε and Realizable k-ε, the experimental data reveals that the RNG k-ε model is the best at predicting pressure, voidage, axial solid velocity and granular temperature. Besides, through the comparison of four drag models, it is found that the Gidaspow model can achieve a higher accuracy of prediction. Therefore, it can be concluded that the combination of the RNG and Gidaspaw models is suitable for the 2D full loop simulation of a CFB, and therefore potential models for the prediction of flow characteristics.

1. Introduction

Due to the high efficiency and the non-bubble contact with highly complex hydrodynamics, gas-solid circulating fluidized beds (CFBs) are widely applied in different fields, such as chemical, petroleum, metallurgy, material, energy, biochemical, environmental protection and pharmaceutical industries [1–4]. Therefore, it is essential to study the characteristics of the gas-solid two-phase flow in the CFB, which is also beneficial for optimizing the CFB structure and improving industrial production efficiency.

Nowadays, computational fluid dynamics (CFD) is believed to be a promising tool for studying the CFB because of its convenience and efficiency [5]. Therefore, a large number of researchers have been devoted to exploring the most suitable CFD model. In the past decades, different turbulent models and drag models have been developed based on experiments, which were widely used to investigate the structure and flow characteristics of the CFB [6–11]. According to the expression of stress tensor of viscous fluid and deformation rate tensor, the early study of the turbulent models tended to directly associate fluctuation velocity with the average velocity. Initially, Jones [7] put forward two equations for the turbulent kinetic energy, k, and the turbulent dissipation rate, ε, when investigating the local turbulent viscosity, producing this way the k-ε model. As a typical turbulent model, k-ε has been utilized in many different aspects, including boundary layer flow, shear flow, and rotation flow. On the basis of the original k-ε model, a new k-ε model was developed by Launder [8], which is known as the Standard k-ε model and is applicable to flows with a small Reynolds number. However, in the case of rotation flow and curved flow, the Standard k-ε model was no longer applicable. Then, the RNG k-ε model and the Realizable k-ε model were developed in succession by researchers. The development of the three models greatly promoted the study of gas flow characteristics. For instance, Guan et al. [12] adopted the Standard k-ε model to study the effects of superficial gas velocity, particle size and inventory on the solids circulating rate in a three-dimensional full loop CFB. Zi et al. [13] investigated the hydrodynamics characterization of solids oscillation behavior based on the RNG k-ε model with the aid of a CFB. By adopting the Eulerian-Eulerian (two-fluid) model with the k-ε model and Gidaspow drag model, Seo et al. [14] conducted simulations to study the characteristics of circulation by changing the velocity of the riser and the loop seal. Overall, the k-ε model has been regarded as a suitable turbulent model for the simulation of a CFB. As indicated by relevant literatures, different k-ε models have been directly adopted in most researches, without a systematic study of the models themselves. Up until now, it remains unclear that which specific type of k-ε model will be suitable for certain operating system. In addition, in most of the reported CFD simulation of gas-solid fluidized beds, a laminar model for gas phase have been adopted. Therefore, the simulation results with a laminar model are also included in this study.

* Corresponding authors.
E-mail addresses: zhouyf@xtu.edu.cn (Y. Zhou), chenhb@xtu.edu.cn (H. Chen).

https://doi.org/10.1016/j.apt.2018.07.024
0921-8831/

Please cite this article in press as: Y. Lu et al., Verification of optimal models for 2D-full loop simulation of circulating fluidized bed, Advanced Powder Technology (2018), https://doi.org/10.1016/j.apt.2018.07.024

APT 2036
4 August 2018
ARTICLE IN PRESS
No. of Pages 10, Model 5G

Nomenclature

Symbol	Description
U_{mf}	minimum fluidization velocity (m/s)
$U_{g,r}$	superficial gas velocity at the riser (m/s)
$U_{g,b}$	superficial gas velocity at the bubbling fluidized bed (m/s)
$U_{g,pt}$	superficial gas velocity at the pot-seal (m/s)
$U_{g,pt\text{-}s}$	superficial gas velocity at the supply chamber (m/s)
$U_{g,pt\text{-}r}$	superficial gas velocity at the recycle chamber (m/s)
d_p	particle mean diameter (μm)
P	Pressure (kPa)
ΔP	pressure drop (kPa)
ΔP_r	pressure drops across the riser (kPa)
ΔP_c	pressure drops across the cyclone (kPa)
ΔP_b	pressure drops across the bubbling fluidized bed (kPa)
ΔP_{pt}	pressure drops across the pot-seal (kPa)
ΔP_{ud1}	pressure drops across under duct connecting the pot-seal to the bubbling fluidized bed (kPa)
ΔP_{ud2}	pressure drops across under duct connecting the pot-seal to the riser (kPa)
H	height from the riser bottom (m)
ΔH	height difference (m)
H_d	correction factor
g	gravitational acceleration (m/s^2)
e_{ss}	particle-particle restitution coefficient
$g_{0,ss}$	radial distribution function
$\vec{u}_g$	gas velocity vector (m/s)
$\vec{u}_p$	particle velocity vector (m/s)
C_D	particle drag force coefficient
C_p	specific heat (j/kg k)
a_{pf}	acceleration of particle in the dilute (m/s^2)
k	turbulent kinetic energy (m^2/s^2)
Re	Reynolds number
t	time (s)

Greek letters

ε_g	volume fraction of gas
ε_p	volume fraction of solid
β	inter-phase momentum transfer coefficient
ρ_p	density of particle (kg/m^3)
μ	viscosity (Pa·s)
Θ	granular temperature (m^2/s^2)
λ	bulk viscosity (Pa·s)
T	stress-strain tensor (Pa)
τ_p	particle shear stress

Subscripts

g	gas phase
p	solid phase
r	riser
b	bubbling bed
c	cyclone
pt	pot seal
pf	particle phase in the dilute

Additionally, the drag model has also achieved great development in the recent years [6,9–11,15]. Since phase interaction has a significant impact on the flow behaviors in a CFB based on different approaches for inter-phase interaction, researchers have developed some typical drag models, including Wen-Yu model [6], Syamlal-O'Brien model [10] and Gidaspow model [11]. They are called the traditional or homogeneous drag models, applicable to many gas-solid two-phase simulations. For example, Upadhyay et al. [16] adopted those drag models to study the effects of specularity and restitution coefficient on the hydrodynamics of the riser. They found that the Syamlal-O'Brien model was suitable for the dilute zone while the Gidaspaw model helped with the prediction of the solid concentration accurately. Fariborz [17] pointed out that Syamlal-O'Brien model would be inaccurate under a low gas flow rate while the simulation results agreed with the prediction of Gidaspow model at a high gas flow rate. As research further developed, some restrictions were found in these traditional drag models due to simplification used in these models. In order to improve the drag model, Li et al. [9] put forward the Energy Minimization Multi-Scale (EMMS) model by taking clusters into consideration. The EMMS model has performed well in a variety of simulations including 3D full loop simulation of the CFB [18,19]. Recently, researchers have found that EMMS model has advantages in simulating gas-solid fluidized bed reactors [20–22]. Nevertheless, the EMMS models were applied to the riser in most cases and few of them only have been applied in the 2D full loop simulation of a CFB as the EMMS model has more restrictions in 2D simulation. Hence, investigation of the applicability of EMMS model to a 2D full loop CFB is very important for accurate predictions.

However, among the published literature most researchers have used the turbulent model and drag model directly without explanation and verification, which is not rigorous [12–14,23,24]. Due to the limitations of numerical simulation, some differences would be obtained with the use of an inappropriate model. Therefore, it is indispensable to testify the applicability of the models before investigation. In this study, the experiment was carried out in a CFB to verify the applicability of certain models in the simulation. Generally, a CFB is composed of riser, cyclone, standpipe and loop seal. Unlike most CFBs in previous studies, the pot-seal in this study is divided into two parts by a vertical baffle and is fed with a certain amount of fluidizing gas, making the solid circulation smoother. Moreover, with changes in the riser, bubbling fluidized bed and pot-seal gas velocity, solids circulation rate and bed structure, different flow regimes will occur in the riser, such as bubbling fluidization, turbulent fluidization, fast fluidization and dense phase pneumatic conveying. Understandably, there exist interactions among different parts of the CFB due to various operating conditions, which have a significant impact on the flow regime. Therefore, a thorough understanding of the interaction is essential for CFB design and operation.

This study is to explore the optimal models for 2D full loop simulation of CFB by investigating flow characteristics in CFB. For this, a 2D CFB simulation is conducted in which different turbulent models and drag models are compared. To verify the models used in this study, data measured by pressure transducers and acoustic emissions (AE) under relevant conditions are used.

2. CFD modeling

2.1. Mathematical models

2.1.1. Turbulent model

Over the years, researchers have developed different turbulent models, including the k-ε model, the k-ω model and the RES model. As mentioned above, application of the k-ε model, with its special advantages, is broad in the gas-solids flow [12–14,23,24]. In this study, three k-ε models, namely Standard k-ε model, RNG k-ε model and Realizable k-ε model, are used respectively. Standard k-ε model, as a typical turbulent model, is adopted

Please cite this article in press as: Y. Lu et al., Verification of optimal models for 2D-full loop simulation of circulating fluidized bed, Advanced Powder Technology (2018), https://doi.org/10.1016/j.apt.2018.07.024