Engineering Conferences International
ECI Digital Archives

10th International Conference on Circulating Fluidized Beds and Fluidization Technology - CFB-10

Refereed Proceedings

Spring 5-3-2011

3D CFD Simulation of Combustion in a 150 MWe Circulating Fluidized Bed Boiler

Nan Zhang
State Key Laboratory of Multiphase Complex Systems, Institute of Process Engineering, Chinese Academy of Sciences

Bona Lu
State Key Laboratory of Multiphase Complex Systems, Institute of Process Engineering, Chinese Academy of Sciences

Wei Wang
State Key Laboratory of Multiphase Complex Systems, Institute of Process Engineering, Chinese Academy of Sciences

Jinghai Li
State Key Laboratory of Multiphase Complex Systems, Institute of Process Engineering, Chinese Academy of Sciences

3D CFD SIMULATION OF COMBUSTION IN A 150 MW_e CIRCULATING FLUIDIZED BED BOILER

Nan Zhang, Bona Lu, Wei Wang, Jinghai Li
State Key Laboratory of Multiphase Complex Systems, Institute of Process Engineering, Chinese Academy of Sciences, Beijing 100190, China

ABSTRACT

Eulerian granular multiphase model with meso-scale modeling of drag coefficient and mass transfer coefficient, based on the energy minimization multi-scale (EMMS) model, was presented to simulate a 150 MW_e CFB boiler. The three-dimensional (3D), full-loop, time-dependent simulation results were presented in terms of the profiles of pressure, solids volume fraction and solids vertical velocity, the distributions of carbon and oxygen, as well as the temperature. The EMMS-based sub-grid modeling allows using coarse grid with proven accuracy, and hence it is suitable for simulation of such large-scale industrial reactors.

INTRODUCTION

The gas-solid flow, heat /mass transfer and chemical reactions in Circulating Fluidized Bed (CFB) reactors are inherently coupled with spatio-temporal multi-scale structures, and featured with non-linear non-equilibrium behavior (1). For such a multi-phase complex system, it is inadequate to reproduce the real physical process by refining the grid in the conventional Two-Fluid Model (TFM), thus calling for establishing meso-scale models (2).

The present study firstly presents a 3D full-loop simulation for a 150 MW_e CFB boiler, showing the powerful capability of the sub-grid EMMS model in predicting the hydrodynamics in a CFB. Then, a reduced EMMS/mass model is incorporated into the mass conservation equations for combustion species, and a 3D combustion simulation is realized for the furnace part.

MODEL DESCRIPTION AND SIMULATION SETTING

Governing equations

The Eulerian granular model in Fluent®6.3.26 was used to study the flow behavior in the full-loop of the boiler, in which the stress of the solid phase was described with the kinetic theory of granular flow; the drag coefficient correlation was corrected with consideration of particle clusters, which was detailed in (3) and would not be stressed here. By neglecting of energy due to viscous dissipation, compression or expansion and interfacial flow, the internal energy balances and the conservation equations of chemical species in gas and solid phase were used to study the combustion in the furnace. As a first step to combustion simulation, only the main reaction of carbon react with oxygen to carbon dioxide was considered here. The volatile combustion and moisture release was considered in the heat balance through a UDF which filled the gap between the low heating value of the coal and the reaction heat as carbon was combusted completely, and the value was added averagely on the solids phase in the lower section ($h <= 6m$) of the furnace. The volatile and moisture was neglected in the mass balance equation, as they are less than 2% compared with the total air input, detailed in (4).

Geometry and mesh

The 150 MW_e CFB boiler was designed by Harbin Boiler Co. Ltd. and installed in Guangdong, China. The main cross section of the furnace is a rectangle of 15.32 × 7.22 m^2; the chamber height is 36.5 m; detailed information can be found in (3).

The geometry and surface mesh of the whole loop of the solids material was shown in Fig. 1. As can be seen from the figure, most of the boiler was meshed with hexahedron, others meshed with polyhedron, all with size scale of 0.1 m. For convenience, the primary air and the loop-seal aeration were assumed in plug flow from their bottoms. To balance the solid inventory in the boiler, the solids exiting from the cyclone outlets, which was about 6%(mass fraction) of the circulating solids during the simulation, were returned via the coal-feed inlets. The origin point is set at the center of the primary inlet at the bottom of the furnace; the x-axis is along the front-to-back wall direction (width direction); the y-axis is along the side-to-side wall direction (depth direction), and the z-axis is against the gravity direction.

Simulation settings

For full-loop hydrodynamic simulation, the boiler was considered operated at the design temperature of 917℃ and atmospheric pressure, thus the physical properties of the gas was considered as a const, while for combustion simulation in the furnace, the mixing laws of the gas mixture were summarized in Table 1. The boundary and initial conditions of the hydrodynamic simulation and the combustion simulation are listed in Table 2 and Table 3 separately.

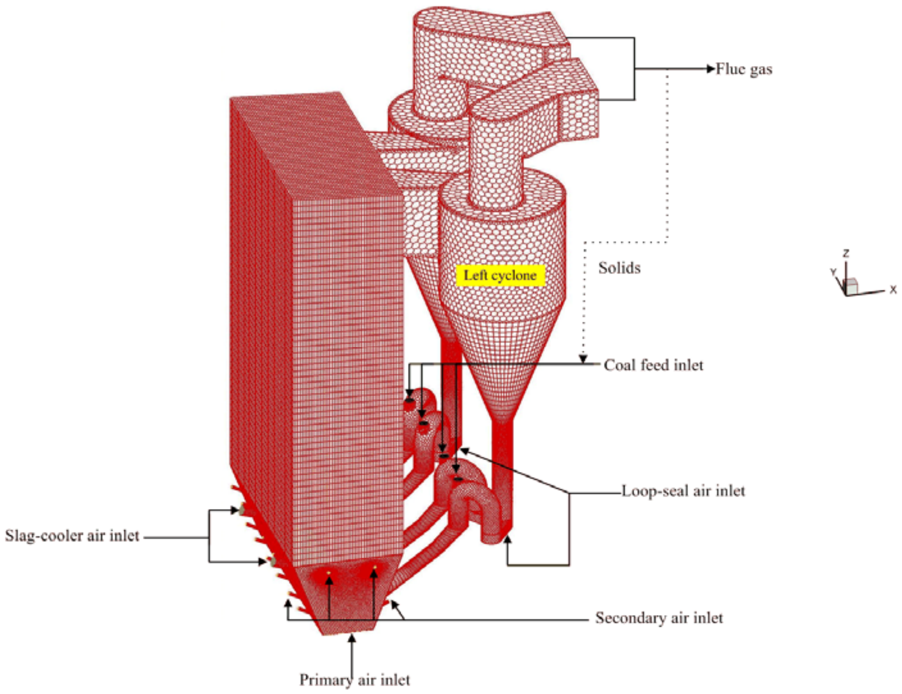

Fig. 1. Geometry and surface mesh of the whole loop of the 150 MW$_e$ CFB boiler.

Table 1. Mixing laws of the gas mixture

Property	Unit	Mixing law	Equation
Density	(kg/m^3)	Incompressible ideal gas law	$\rho = p_{op} \big/ RT \sum_i (Y_i / MW_i)$
Specific Heat Capacity	(J/kg·k)	Mixing law	$c_p = \sum_i Y_i c_{p,i}$
Thermal Conductivity	(w/m·k)	mass weighted mixing law	$k = \sum_i Y_i k_i$
Viscosity	(kg/m·s)	mass weighted mixing law	$\mu = \sum_i Y_i \mu_i$
Mass Diffusion	(m^2/s)	Dilute Approximation	$D_{i,m}$

Table 2. Boundary and initial conditions of hydrodynamic simulation in full-loop of the boiler

Boundary and Initial Conditions	Gas Phase			Solid Phase
	Flow rate (kg/s)	Area (m^2)	Velocity (m/s)	
Primary air inlet	94.16	50.88	6.32	0
Secondary air Inlet	53.21	0.92	198.01	0
Slag-cooler inlet	8.00	0.75	36.32	0
Loop-seal inlet	2.32	8.02	0.99	0
Coal-feed inlet	12.48	1.16	36.70	UDF
Initial solid packing height				2.5 m
Cyclone Outlet	101325 Pa			
Wall	No-slip			Partial slip

Table 3. Boundary and initial conditions of combustion simulation in the furnace

Boundary and Initial Conditions	Gas Phase		Solid Phase	
	Temperature (K)	Velocity (m/s)	Temperature (K)	Velocity (m/s)
Primary air inlet	500.15	2.662	1190	0
Secondary air Inlet	500.15	83.397	1190	0
Slag-cooler inlet	293.15	9.018	1190	0
Loop-seal inlet	293.15	0.245	1190	0
Coal feed inlet	293.15	15.458	1190	0.013+returned
Initial solid packing height			1.5 m	
Cyclone Outlet	101325 Pa			
Wall	No-slip		Partial slip	

HYDRODYNAMIC FLOW IN THE FULL-LOOP

Fig. 2(a) shows the simulated pressure balance in the boiler: pressure gradient is large at the bottom and comparatively small at the top in the furnace, while the largest gradient occurs at the return legs. Fig. 2(b) shows that the general trends of the simulated data were comparable with experimental data. Fig. 3 and Fig. 4 show profiles of solids volume fraction and solids vertical velocity at different heights, respectively. Fig. 3(a) shows the so-called core-annulus structure, i.e. a denser solids concentration near the wall than in the center. Fig. 3(b) shows that solids concentration profiles along the depth direction are flatter than those along the width direction. Fig. 4(a) shows falling clusters near the walls while rising particles in the center. Fig. 4(b) shows two maximum rising velocities not far from the walls, which may be affected by the secondary air inlets at the side walls.

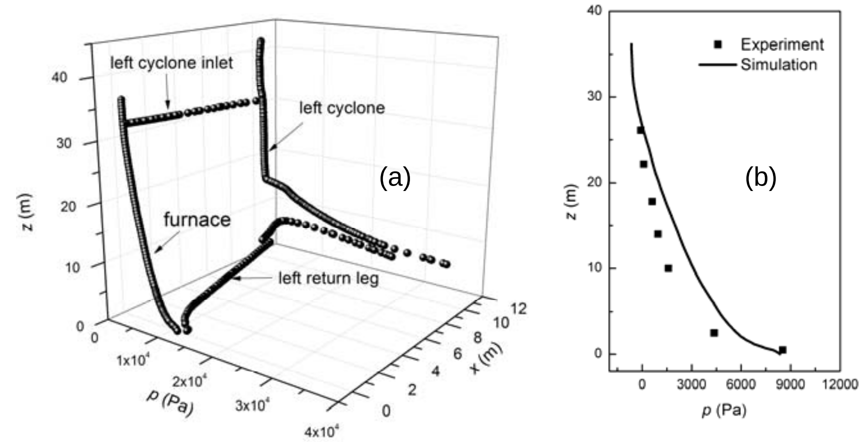

Fig. 2. Pressure distribution: (a): in the whole loop (b): compared with experimental data in the furnace (simulation data were taken from the center line of the furnace).

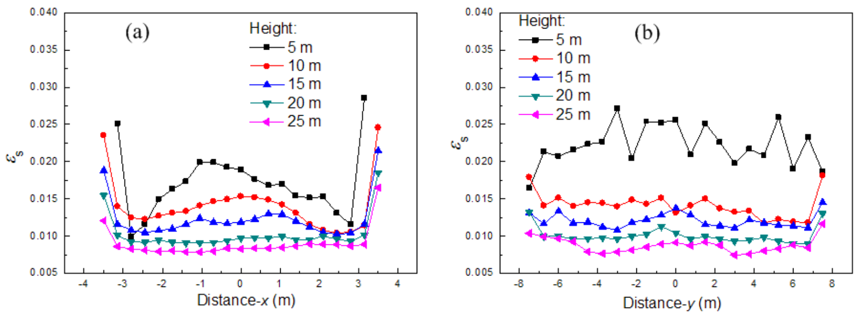

Fig. 3. Solids volume fraction at different heights in the boiler.

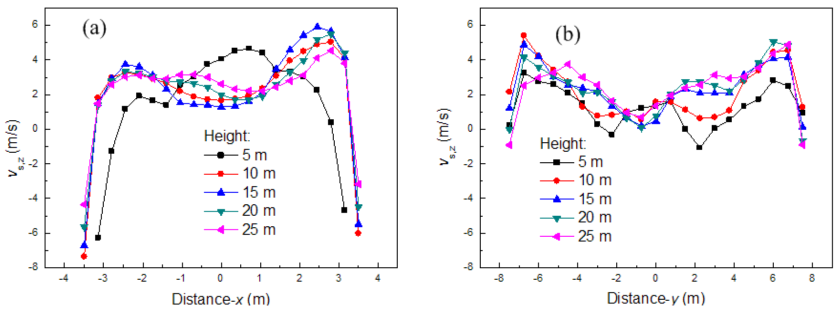

Fig. 4. Solids vertical velocity at different heights in the boiler.

COMBUSTION IN THE FURNACE PART

Fig. 5 shows the simulated temperature in the furnace, which is low at the bottom because of the injected cold air and is high at the top because of combustion energy released. Fig. 6 and Fig. 7 show snapshots of simulated carbon concentration and oxygen concentration with several slices in different directions, respectively. Fig. 6(a) and Fig. 6 (b) Show that carbon concentration is large at the bottom and small at the top, Fig. 6(b) and Fig. 6 (c) show that carbon concentration reaches local maximum near the solids return ports and causes non-uniform in the bottom of the furnace, and this local non-uniform caused by local ports will decrease along the height because of combustion and dispersion. Fig. 7(a) shows that oxygen concentration is large at the bottom and is small at the top because of combustion. Fig. 7(b) shows that the injected secondary air causes local maximum of oxygen concentration, while the maximum oxygen is not at the center line of the furnace but there are two maximum near the center line, which is caused by the fact that the injection does not reach the center of the furnace, this phenomenon was also reported by Myöhänen et al. (5). Fig. 7(c) shows that the injected secondary air leads to non-uniform oxygen concentration at the bottom and this non-uniformity decreases along the height but still exists at the top of the furnace.

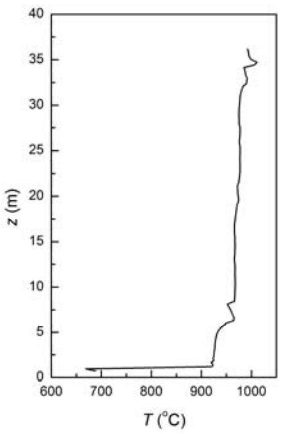

Fig. 5. Temperature profile in the furnace.

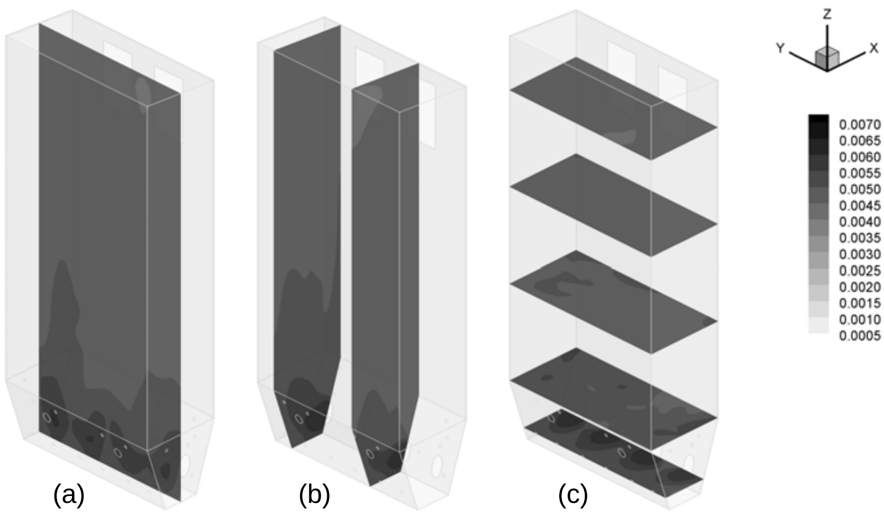

Fig. 6. Simulated carbon concentration distributions with vertical slices (a): side-to-side slice ，(b): front-to-back slices and (c): horizontal slices.

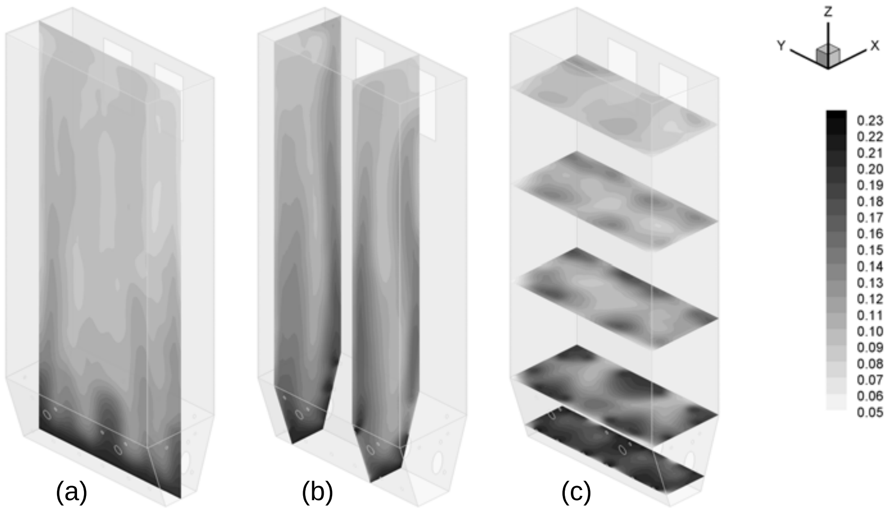

Fig. 7. Simulated oxygen concentration distributions with vertical slices (a): side-to-side slice ，(b): front-to-back slices and (c): horizontal slices.

CONCLUSIONS

Simulation results show the capability of the current model, with emphasis on the EMMS-corrected drag coefficient, in predicting the two-phase flow behavior. The reduced multi-scale mass transfer model and the drag coefficient correction based on the EMMS model were coupled to realize combustion simulation in the furnace. It is an extension to our experience on virtual experimentation to investigate the combustion within an industrial reactor.

ACKNOWLEDGMENT

The authors are grateful to Professor Junfu Lu of Tsinghua University for providing the blueprint of the boiler. The financial supports from NSFC under Grant No. 20821092, MOST under Nos. 2007AA050302-03 and 2008BAF33B01, and CAS under No. KGCX2-YW-222 are also gratefully acknowledged.

NOTATION

c_p	specific heat capacity, J/kg·k
D	mass diffusion coefficient, m^2/s
k	thermal conductivity, w/m·k
p	pressure, Pa

R	gas constant, 8.314 J / mol·K
MW	molecular weight, kg/mol
T	temperature, K or ℃
V	velocity, m/s
x	coordinate, m
y	coordinate, m
Y	mass fraction, dimensionless
z	coordinate, m
Greek letters	
ε	volume fraction, dimensionless
μ	viscosity, kg/(m·s)
ρ	density, kg/m^3
Subscripts	
s	solid phase
i	species

REFERENCES

1. J. Li, M. Kwauk, Particle-fluid two-phase flow: The energy-minimization multi-scale method. Beijing: Metallurgical Industry Press, 1994.
2. B. Lu, W. Wang, J. Li, Searching for a mesh-independent sub-grid model for CFD simulation of gas-solid riser flows. Chemical Engineering Science 2009, 64 (15), 3437-3447.
3. N. Zhang, B. Lu, W. Wang, J. Li, 3D CFD simulation of hydrodynamics of a 150 MW_e circulating fluidized bed boiler. Chemical Engineering Journal 2010, 162 (2), 821-828.
4. N. Zhang, B. Lu, W. Wang, J. Li, 3D CFD simulation of combustion in a 150 MW_e circulating fluidized bed boiler. To be submitted.
5. K. Myöhänen, T. Hyppänen, M. Loschkin, Converting Measurement Data to Process Knowledge by Using Three-Dimensional CFB Furnace Model, in: K. Cen (Eds.), Circulating Fluidized Bed Technology VIII - Proceedings of the Eighth International Conference on Circulating Fluidized Beds, International Academic Publishers, World Publishing Corp., Hangzhou, 2005, pp. 306-312.