Chemical Engineering Science 120 (2014) 10–21

Contents lists available at ScienceDirect

Chemical Engineering Science

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

# Comparison of full-loop and riser-only simulations for a pilot-scale circulating fluidized bed riser

Tingwen Li [a,b,*], Jean-François Dietiker [a,c], Lawrence Shadle [a]

[a] National Energy Technology Laboratory, Morgantown, WV 26505, USA
[b] URS Corporation, Morgantown, WV 26505, USA
[c] West Virginia University Research Corporation, Morgantown, WV 26506, USA

## HIGHLIGHTS

- Both full-loop and riser-only simulations of a CFB are carried out.
- Flow hydrodynamics are compared for full-loop and riser-only simulations.
- The mean solids circulation rate is the most critical parameter for both simulations.
- The riser-only simulation provides reasonable prediction of steady CFB riser flow.
- The full-loop simulation is promising but more challenging.

## ARTICLE INFO

*Article history:*
Received 13 April 2014
Received in revised form
30 July 2014
Accepted 19 August 2014
Available online 27 August 2014

*Keywords:*
Gas–solids flow
Circulating fluidized bed
Computational fluid dynamics
Full-loop simulation
Flow hydrodynamics

## ABSTRACT

In this paper, both full-loop and riser-only simulations of a pilot-scale circulating fluidized bed (CFB) system carried out by the open-source code MFIX (Multiphase Flow with Interphase eXchanges) are presented. Detailed comparison between full-loop and riser-only numerical simulations has been conducted with respect to the flow hydrodynamics inside the riser. The mean solids circulation rate is found to be the most critical parameter for both riser-only and full-loop simulations. On one hand, the mean solids circulation rate is needed for specifying the solids inlet boundary condition for the riser-only simulation. On the other hand, a reasonable prediction of the solids circulation rate is a prerequisite for the full-loop simulation to properly predict the flow hydrodynamics inside the riser. To better account for the full-loop dynamics in the riser-only simulation, the transient solids circulation rate measured from an experimental facility is imposed in the simulation in addition to the mean solids circulation rate. Consistent numerical predictions of the flow hydrodynamics inside the riser are obtained by different types of simulations. In-depth discussion on the advantage and disadvantages of each approach is presented and the riser-only simulation with appropriate boundary conditions is shown to be sufficient for investigating the steady CFB riser flow. The full-loop simulation is promising but more challenging as all major components and associated complicated physics have to be included and correctly modeled which require both advanced model capability and high computer power.


## 1. Introduction

Circulating fluidized beds (CFB) have been widely utilized in a wide variety of industrial applications including coal combustion, gasification, fluid catalytic cracking, owning to several attractive advantages—such as fuel flexibility, increased through-put, in-bed sulfur capture, and relatively low NOx emissions with high efficiencies in combustion and gasification (Grace et al., 1997). To accomplish successful and reliable design and operation of CFBs in various industrial processes, numerous investigations pertaining to different hydrodynamic aspects of CFBs have been undertaken over the past few decades (e.g. Werther et al., 2008; Knowlton, 2011). Among various research tools, computational fluid dynamics (CFD) is playing an increasingly important role in studying the complex flow hydrodynamics in CFBs (Grace and Li., 2010).

Extensive CFD work on various CFB systems can be found in the open literature. Most of it focused on the riser part, which is usually of the most practical interest, by isolating it from the whole CFB system. Usually, the riser is simulated by applying proper boundary conditions for the gas distributor, the solids inlet and the top exit. Clearly, to properly reflect the operation of the rest of the CFB system,

* Corresponding author at: National Energy Technology Laboratory, Morgantown, WV 26505, USA. Tel.: +1 304 285 4538.
*E-mail addresses:* tingwen.li@contr.netl.doe.gov, litingwen@gmail.com (T. Li).

http://dx.doi.org/10.1016/j.ces.2014.08.041
0009-2509/

accurate boundary conditions are needed in the riser flow simulations. One of the key boundary conditions needed by the typical riser flow simulation is the solids circulation rate at the solids inlet of the riser, which is also a critical design and control parameter in the CFB reactor systems. Usually, the average solids circulation rate measured from the experimental facility is used to specify a steady inflow condition for the solids inlet. Generally speaking, the riser simulation allows the modeler to concentrate on the flow hydrodynamics inside the riser with the available computational resource and effort. On the other hand, the operation of CFB system is a complex and integrated process coupling all inter-connected components including the riser, air compressor and feeding system, cyclones, standpipe, L-valve and associated aeration system (Grace et al., 1997). Accordingly, it is preferable to model the CFB as a whole dynamic system to fully account for interactions between different components, rather than isolated components.

With the continued advances in computational hardware as well as numerical models, three-dimensional simulations of the full CFB loop are more common for investigating the complex system (Sundaresan, 2011). Quite a few numerical studies of whole CFB system can be found in the literature. Chu et al. (2007) presented a numerical study of the gas–solid flow in a whole-loop three-dimensional circulating fluidized bed by incorporating discrete particle method (DPM) codes into the commercial CFD software package, Fluent. It was shown that the discrete particle simulation can qualitatively capture the key flow features in the CFB such as core-annulus structure, axial solid segregation and S-shaped axial solid concentration. Zhang et al. (2008) conducted three-dimensional, time-dependent full-loop simulations of a semi-industry scale CFB using the Eulerian granular multiphase model with the drag correlation based on the energy-minimization multi-scale (EMMS) model. The predicted axial profiles of pressure gradient and the radial profiles of solid velocity and solid volume fraction in the riser were in reasonable agreement with the experimental data. Fluid regime diagrams and pressure balance around the CFB loop were derived accordingly. Zhang et al. (2010) further extended their full-loop simulation to a 150 MW CFB boiler to better understand the overall flow behavior inside the system. Liu et al. (2010) simulated the fluid dynamics of a 50 MWe CFB combustor and investigated the effect of secondary air on the flow behavior in the combustor. Nguyen et al. (2012) investigated the gas and particles hydrodynamic behaviors in a pilot-scale cold-model gasifier consisting of a CFB riser and a bubbling fluidized bed by means of experiment and two-dimensional full-loop simulation. Nikolopoulos et al. (2013) reported a high-resolution three-dimensional full-loop simulation of a CFB carbonator cold flow model. Their full-loop simulation results agreed quite well with the experimental data, regarding the recirculation flux and the pressure profile along the full-loop. Lu et al. (2013) reported a three-dimensional full-loop simulation of an industrial-scale circulating fluidized bed boiler with and without fluidized-bed heat exchanger. Wang et al. (2014a, 2014b) simulated a CFB system with a loop seal to investigate the influences of several operating parameters, including loop seal aeration rate, fluidized air velocity in the riser and total bed inventory on the solid circulation characteristics.

In 2010, U.S. Department of Energy's National Energy Technology Laboratory (NETL) in collaboration with Particulate Solid Research Inc. (PSRI), generated a third challenge problem from data collected in NETL's circulating fluidized bed and PSRI's bubbling fluid bed (Breault, 2010). Model predictions by different research groups have been compared against the experimental measurements and summarized by Panday et al. (2014). Li et al. (2012) reported comprehensive numerical simulations of NETL/PSRI challenge problem for CFB riser using the open-source code Multiphase Flow with Interphase eXchange (MFIX). A full-loop simulation of that pilot-scale CFB was also conducted which compared favorably to the experimental measurement as well as their riser-only simulations. A brief discussion on advantages and disadvantages of the full-loop simulation of CFB system was presented in Li et al. (2012). On one hand, it is desirable to include the entire system so that the global features of the flow are captured in CFD simulations. On the other hand, modeling the entire CFB systems is very challenging as it requires both advanced model capability and high computer power. It is necessary to appropriately simplify the target problem to facilitate the numerical simulation. Hence, it is a compromise between model complexity and computational cost which needs a proper

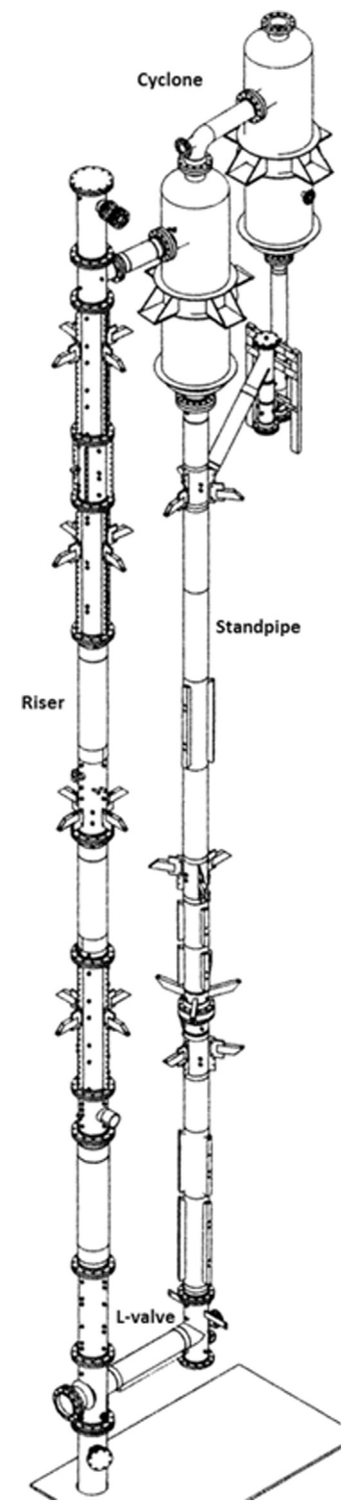


**Fig. 1.** Schematic of the NETL pilot-scale CFB system.