Numerical study of hydrodynamics and heat transfer in a spouted bed

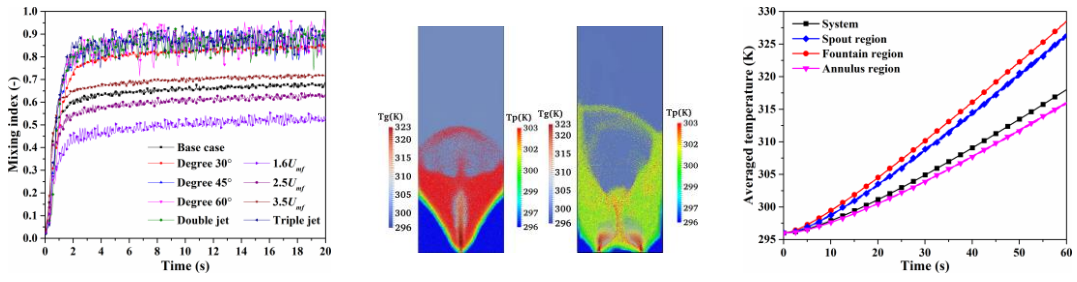
Junjie Lin, Kun Luo*, Shuai Wang, Chenshu Hu, Jianren Fan

State Key Laboratory of Clean Energy Utilization, **Zhejiang University,**
Hangzhou 310027, P. R. China
Email: 21627020@zju.edu.cn

Abstract

The hydrodynamics and heat transfer behavior in a spouted bed is modeled by the coupling of computational fluid dynamic and discrete element method (CFD-DEM). In this method, each particle is tracked under the Lagrangian framework whilst the gas motion is solved under the Eulerian framework. The gas-solid flow patterns, mixing properties, pressure drop, velocity distribution and temperature evolution in the spouted bed are deeply studied. Moreover, effects of the number of orifices, superficial velocities and inclined angles of distributor on heat transfer performance are comprehensively investigated. Results show that the solid temperature increases with the increase of the number of orifices and the temperature distribution in the multiple jets system become more homogeneous than the single jet system. In addition, the similar conclusion can be drawn in the spouted bed with the increase of the superficial velocity and incline angle of the distributor. These phenomena are mostly due to the fact that the disturbance in the spouted bed becomes more intense with the increase of those simulation parameters, and the interactions between gas and solid phases are more frequent. The results shed light on the design, operation and optimization of spout beds.

Keywords: heat transfer; CFD-DEM; multiphase flows; spouted bed



(a) (b) (c)

Fig. 1. (a) Mixing indexes under different operating conditions; (b) Snapshots of gas-solid temperature distributions; (c) evolutions of averaged temperature in four regions.