Influence of inlet and outlet configurations for two-dimensional CFD simulation of circulating fluidized bed riser

MUKESH UPADHYAY[1,2], 서명원[3], 박종호[1,2,*]
[1]Advanced Energy Technology, University of Science & Technology; [2]Petroleum and Gas Laboratory, Korea Institute of Energy Research; [3]Clean Fuel Department, Korea Institute of Energy Research
(jongho@kier.re.kr*)

Understanding of hydrodynamics inside the circulating fluidized bed (CFB) riser have been widely performed using 2D domain. In the present work effect of inlet and outlet configuration has been investigated using the kinetic theory approach for the particulate phase. In addition, grid convergence of numerical simulation depends on the flow field chosen for verification and also checked for various operating conditions. Model parameters were tuned to fit numerical result with experimental data in terms of axial solid holdup distribution. Effect of gas velocity and solid circulation rate, on solid hold distribution has been investigated. The presented experimental and CFD prediction will be useful to map out model prediction capability for wide range of operating conditions and for further work on understanding riser flow.