INTISARI

Di dalam industri migas hingga reaktor nuklir, studi terkait aliran dua fase gas-cairan sangatlah penting karena banyak ditemui pada berbagai peralatan yang digunakan dalam industri tersebut. Pola aliran dua fase *bubbly flow* adalah pola yang banyak ditemukan, di mana kondisi aliran dan interaksi antar fase yang terjadi cenderung kompleks. Salah satu fenomena yang dapat terjadi pada aliran dua fase adalah *bubble breakup and coalescence*. Hal ini membuat studi terkait pemodelannya penting untuk dilakukan. Belum banyak studi numerik yang dilakukan pada pipa vertikal dengan penghalang. Aliran yang kompleks akibat penghalang penting untuk ditelili karena di dalam industri, aliran yang terjadi juga cenderung kompleks. Untuk mengetahui pengaruh penggunaan model-model *breakup* dan *coalescence* pada aliran dilakukan studi numerik dengan menggunakan CFD dan *Population Balance Model*.

Simulasi dilakukan menggunakan ANSYS CFX dengan *Eulerian-Eulerian two-fluid model*. Fase gas sebagai *polydispersed fluid* dimodelkan menggunakan *homogenous Multiple Size Group Model* (MUSIG). Dilakukan simulasi menggunakan perbedaan model *breakup* dan *coalescence*. Digunakan model *breakup* Luo & Svendsen (1995) dan Liao, dkk. (2015) serta model *coalescence* Prince & Blanch (1990) dan Liao, dkk. (2015). Simulasi dilakukan dengan kondisi kecepatan superfisial cairan (j_l) yang berbeda, kasus 01 dengan j_l =0,4050 m/s, sementara itu kasus 02 dengan j_l =1,0170 m/s. Kecepatan superfisial gas untuk kedua kasus bernilai sama, yaitu j_g =0,0368 m/s. Hasil simulasi berupa nilai distribusi *void fraction*, distribusi ukuran *bubble*, dan *gas velocity* dibandingkan dengan data eksperimen.

Hasil simulasi menunjukkan bahwa penggunaan model *breakup* dan *coalescence* perlu dipertimbangkan dalam pemodelan aliran dua fase gas-cairan di dalam pipa vertikal dengan penghalang. Penggunaan model *breakup* dan *coalescence* Liao, dkk. (2015) mampu memberikan prediksi distribusi *void fraction* yang baik untuk kasus dengan kecepatan superfisial cairan yang lebih rendah. Untuk memperbaiki prediksi distribusi *void fraction* pada kasus dengan kecepatan superfisial cairan yang lebih tinggi, dilakukan simulasi menggunakan variasi model turbulensi, yaitu *shear-induced turbulence* (SIT) dan *bubble-induced turbulence* (BIT) serta variasi model *drag*, yaitu Grace, dkk. (1976), Schiller & Naumann (1935), dan Ishii & Zuber (1979). Hasil simulasi menunjukkan bahwa penggunaan variasi model-model tersebut belum dapat memperbaiki prediksi distribusi *void fraction*.

Kata kunci: CFD, MUSIG, model *breakup* dan *coalescence*, *void fraction*

ABSTRACT

In industries such as oil and gas as well as nuclear reactors, the study of gas-liquid two-phase flow is critically important due to its frequent occurrence in various equipment used in these sectors. One of the most commonly observed flow patterns is bubbly flow, where the flow conditions and interphase interactions are often complex. A key phenomenon in such flows is bubble breakup and coalescence, making its modeling an essential area of study. However, there have been limited numerical studies conducted on vertical pipes with obstacles. The complex flow caused by obstacles is important to investigate, as industrial flows are often similarly complex. To understand the effects of different breakup and coalescence models on the flow, a numerical study was carried out using CFD and the Population Balance Model.

The simulation was carried out using ANSYS CFX with the Eulerian-Eulerian two-fluid model. The gas phase, treated as a polydispersed fluid, was modeled using the homogeneous Multiple Size Group Model (MUSIG). Simulations were performed using different breakup and coalescence models. The breakup models used were Luo & Svendsen (1995) and Liao, et al. (2015), while the coalescence models used were Prince & Blanch (1990) and Liao, et al. (2015). Simulations were conducted under different liquid superficial velocities (jl): Case 01 with jl = 0.4050 m/s, and Case 02 with jl = 1.0170 m/s. The gas superficial velocity (jg) was kept the same for both cases, at jg = 0.0368 m/s. The simulation results, including void fraction distribution, bubble size distribution, and gas velocity, were compared with experimental data.

The simulation results indicate that the use of breakup and coalescence models should be considered in modeling gas-liquid two-phase flow in a vertical pipe with an obstruction. The Liao, et al. (2015) breakup and coalescence model provides a good prediction of void fraction distribution for cases with lower liquid superficial velocity. To improve the prediction of void fraction distribution in cases with higher liquid superficial velocity, simulations were conducted using variations of turbulence models, namely shear-induced turbulence (SIT) and bubble-induced turbulence (BIT), as well as different drag models, including Grace et al. (1976), Schiller & Naumann (1935), and Ishii & Zuber (1979). However, the results show that using these model variations has not yet improved the prediction of void fraction distribution.

Keywords: CFD, MUSIG, breakup and coalescence model, void fraction