Volume 2018, Issue 59, 2018, Pages 118-125

Computational study of droplet breakup in a trapped channel configurationusing volume of fluid method

T. CHEKIFI

Abstract: Computational Fluid Dynamics is performed to numerically investigate the droplet breakup of water in oil intrapped channel configuration. The volume-of-fluid (VOF) method based the commercial code FLUENT isadopted to track the interface. Two designs are suggested to study the effect of flow conditions parameters andouter channel size on the droplet breakup mode, droplet generation frequency and size. As a function of thevelocity ratio, droplets are formed in two modes, dripping mode: droplets were generated closed to the nozzle, itwas identified at low capillary number (Ca<0.005) and jetting mode: droplets were produced after a long jet, where the capillary number Ca varies from 0.01 to 0.025. The numerical results indicated the collection channeldiameter plays potential role in the determination of droplet size and droplet generation frequency, the shearforces excreted by the continuous phase on the dispersed thread are reduced in the wider model leading to havedroplets much bigger than the narrow model, the latter produced small droplets due the high shear stressgenerated in the confinement region. Furthermore, the droplet frequency and size are found to be stronglydependent on the capillary velocity ratio. However, increasing the flow velocity ratio in both models leadsdroplets to be generated in high frequency, while the droplet length was decreasing. This work also demonstratesthat the VOF method is an effective way to simulate the droplets breakup in trapped channel geometry.

Keywords : T. Chekifi