

# Computational study of droplet breakup in a trapped channel configuration using 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 is adopted to track the interface. Two designs are suggested to study the effect of flow conditions parameters and outer channel size on the droplet breakup mode, droplet generation frequency and size. As a function of the velocity ratio, droplets are formed in two modes, dripping mode: droplets were generated closed to the nozzle, it was 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 channel diameter plays potential role in the determination of droplet size and droplet generation frequency, the shear forces exerted by the continuous phase on the dispersed thread are reduced in the wider model leading to have droplets much bigger than the narrow model, the latter produced small droplets due the high shear stress generated in the confinement region. Furthermore, the droplet frequency and size are found to be strongly dependent on the capillary velocity ratio. However, increasing the flow velocity ratio in both models leads droplets to be generated in high frequency, while the droplet length was decreasing. This work also demonstrates that the VOF method is an effective way to simulate the droplets breakup in trapped channel geometry.

**Keywords :** T. Chekifi