FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
April 16, 2022

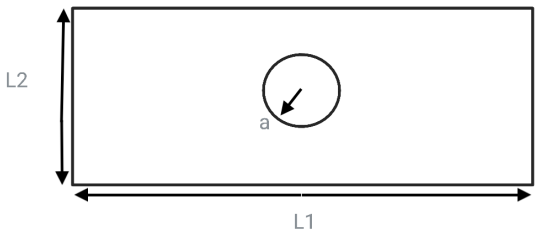
Numerical simulation of breakup of a viscous drop in simple shear flow through a volume-of-fluid method

Shreya Makkar

Chemical Engineering, IIT Bombay

Synopsis

This research migration project aims to do numerical simulations of the deformation experienced by a spherical drop placed in a viscous fluid when a shear rate is applied on parallel plates. The flow is investigated through a volume-of-fluid (VOF) method. The three dimensional case has been simulated using OpenFOAM. The scheme incorporates a semi-implicit Stokes solver (interFoam) to perform the transient analysis at low Reynolds number. The solver is PIMPLE algorithm-based. The geometry and mesh were defined using blockMesh utility. Simulation results were obtained by changing various parameters for validating the results in the paper. The analysis executed by Jie Li, Yuriko Y. Renardy, and Michael Renardy [1] using numerical analysis utilising the projection method and three-dimensional semi-implicit scheme is taken as a reference.
The dimensions of the geometry stated in figure 1 are: $L1 = 3m, L2 = 2m, Thickness = 1m$ and $a = 0.25m$. The spherical drop is in the centre of the cuboidal box and the top and bottom plates of the domain have a shear rate $\dot{\gamma}$. Fluid properties and boundary conditions are discussed in the report.



References

[1] Jie Li, Yuriko Y. Renardy, and Michael Renardy. “Numerical simulation of breakup of a viscous drop in simple shear flow through a volume-of-fluid method.” In: Physics of Fluids (1994-present) 12, 269 (2000); DOI: http://dx.doi.org/10.1063/1.870305

1 Introduction

In the reference paper [1], the case study of a spherical drop experiencing a shear rate in low Reynolds number is analysed. The effect is observed on drops and domains of various sizes for a range of Capillary and Reynolds numbers. When a shear is applied to the drop, it elongates and undergoes deformation and eventually end pinching. The simulation is done as three dimensional initial value problems, with the volume of fluid regime. This analysis is particularly important in study of dispersion science, emulsifications and mixing[2]. The liquid drop has an undeformed radius a and viscosity μ_d , and the matrix liquid has viscosity μ_m. The matrix liquid is undergoing a shear flow between two parallel plates, placed a distance $L2$ apart. The velocity field is $\mathbf{u} = \dot{\gamma} z \mathbf{i}$, where $\dot{\gamma}$ is the imposed shear rate. Additionally, interfacial tension σ is also specified. The Capillary number $Ca = a\dot{\gamma}\mu_m/\sigma$ and the Reynolds Number $Re = \rho_m \dot{\gamma} a^2/mu_m$ are the dimensionless parameters in the study. The viscosities and densities of the fluid in the matrix and the drop are taken equal. Interface plays a major role in determining the shape of the drop as it deforms, thus, an interface tracking method is used.

2 Governing Equations and Models

The VOF regime solves the Navier-Stokes equations in the single phase regions, and for the interface it solves an additional interface equation. The placement of the two fluids is represented by a concentration function α

$$\alpha(\boldsymbol{x}) = \begin{cases} 1 & \text{fluid 1} \\ 0 & \text{fluid 2} \end{cases} \tag{1}$$

The average values of density and viscosity are interpolated by the following formulae:

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2 \tag{2}$$

$$\mu = \alpha\mu_1 + (1-\alpha)\mu_2 \tag{3}$$

The concentration function is governed by the equation

$$\frac{\partial \alpha}{\partial t} + \mathbf{u} \cdot \nabla \alpha = 0 \tag{4}$$

where **u** is the velocity of the flow. The fluids are incompressible. The source term in the Navier-Stokes equation includes the interfacial tension force, $f_{\sigma i}$, which is modelled as continuum surface force.

$$f_{\sigma i} = \sigma\kappa \frac{\partial \alpha}{\partial x_i} \tag{5}$$

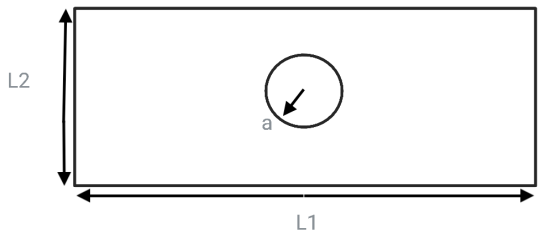
σ is the surface tension constant and κ the curvature. The curvature can be approximated as follows

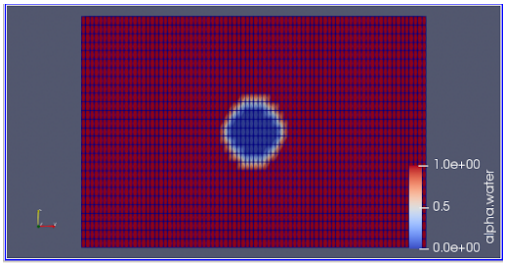
$$\kappa = -\frac{\partial n_i}{\partial x_i} = -\frac{\partial}{\partial x_i}\left(\frac{\partial \alpha/\partial x_i}{|\partial \alpha/\partial x_i|}\right) \tag{6}$$

3 Simulation Procedure

3.1 Geometry and Mesh

The geometry presented has two sections: the matrix fluid and the drop fluid. A three-dimensional geometry has been considered. The cuboid defined is $3m \times 2m \times 1m$ and the drop has a radius $a = 0.25m$. Using `blockMesh` utility, geometry, and mesh are defined. A grid of $80 \times 27 \times 14$ cells has been used with `simpleGrading (1 1 1)`. Cyclic boundary condition has been imposed on the left and right walls.





3.2 Initial and Boundary Conditions

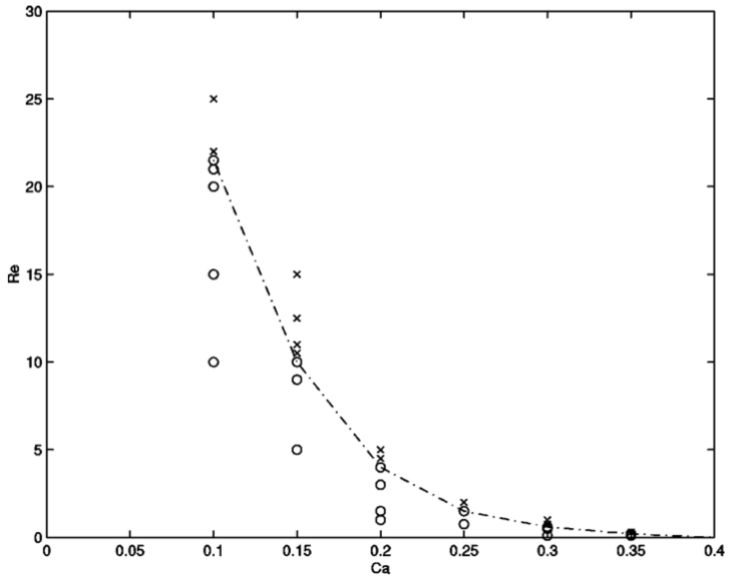
The cyclic walls have been assigned cyclic boundary conditions for both velocity and pressure. For the top and bottom wall, velocity is specified according to the Capillary and Reynolds number. The parallel plates have velocities in the opposite directions. For the front and back walls, no slip boundary condition has been imposed. Since it is not a pressure driven flow, the pressure is specified as `fixedFluxPressure` of a uniform value 0. For both the matrix and drop fluid, the kinematic viscosity and density are taken as 1.3793×10^{-3} and 1.45×10^{3} respectively. The interfacial tension σ is calculated from the Capillary and Reynolds number.

3.3 Solver

`interFoam` is a solver which can be used for two incompressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach. The solver uses PIMPLE algorithm to evaluate NS equations. The PIMPLE Algorithm is a combination of PISO (Pressure Implicit with Splitting of Operator) and SIMPLE (Semi-Implicit Method for Pressure-Linked Equations). The end time for the simulations was specified as 200 with a timestep of 0.001.

4 Results and Discussions

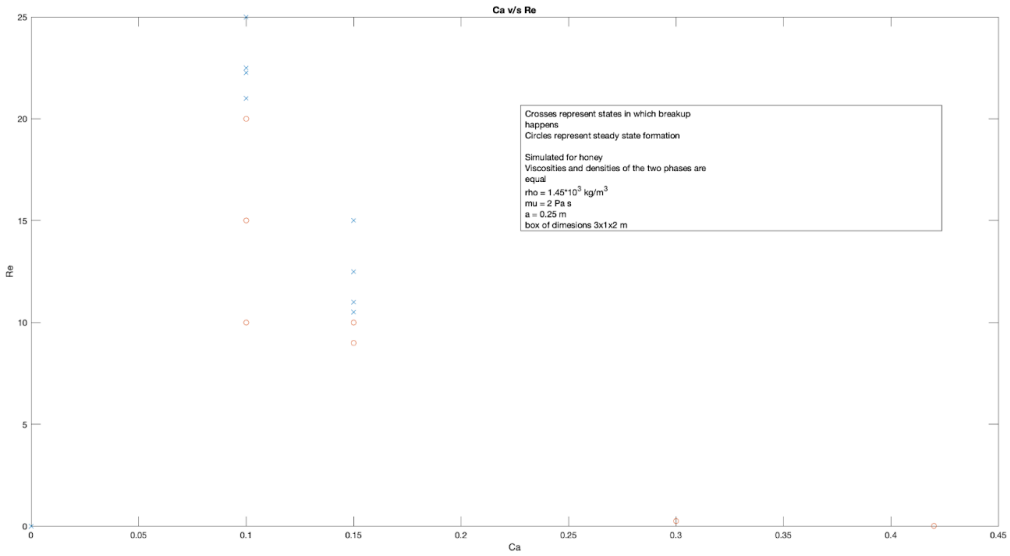
The graph presented in the paper that illustrates the state of the drop at different values of Capillary and Reynolds numbers:



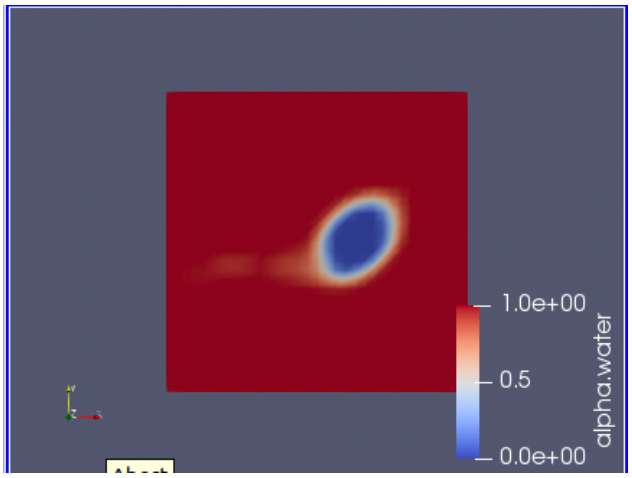
Circles represent evolution to steady state and crosses represent breakup. Dashed curve interpolates critical values.
From these values, velocity and interfacial tension values were obtained. Code for the same:

```
Ca_circles = [0.1 0.1 0.1 0.1 0.1 0.15 0.15 0.15 0.2 0.2 0.2 0.2 0.25 0.25
0.3 0.3 0.35];
Re_circles = [22.25 21 20 15 10 10 9 5 4 2.5 1.5 1 1.5 0.5 0.5 0.25 0.25];
Ca_crosses = [0.1 0.1 0.15 0.15 0.15 0.15 0.2 0.2 0.25 0.3 0.42];
Re_crosses = [25 22.5 15 12.5 11 10.5 5 4.5 2.5 1 0.00625];
rho = 1.45*10^3;
mu = 2;
a = 0.25;

gamma_circles = (mu.*Re_circles)/(rho*a^2)
sigma_circles = a*gamma_circles*mu./Ca_circles
velocity_circles = 2.*gamma_circles

gamma_crosses = (mu.*Re_crosses)/(rho*a^2)
sigma_crosses = a*gamma_crosses*mu./Ca_crosses
velocity_crosses = 2.*gamma_crosses
```

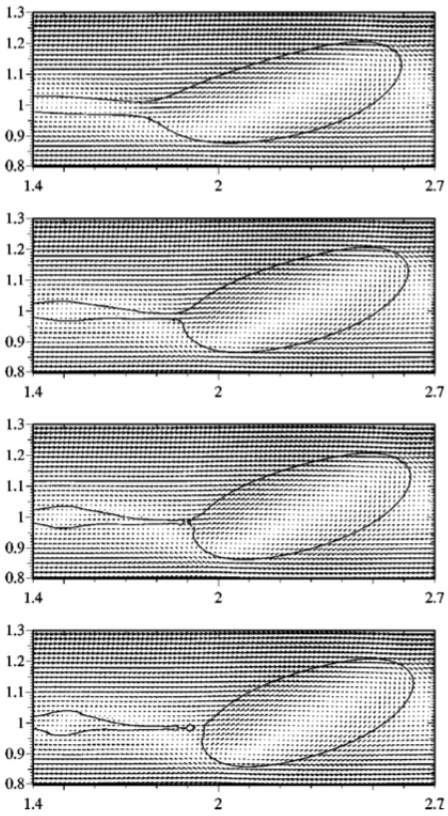
Simulations for the values of u and σ were performed. The corresponding graph obtained is shown.



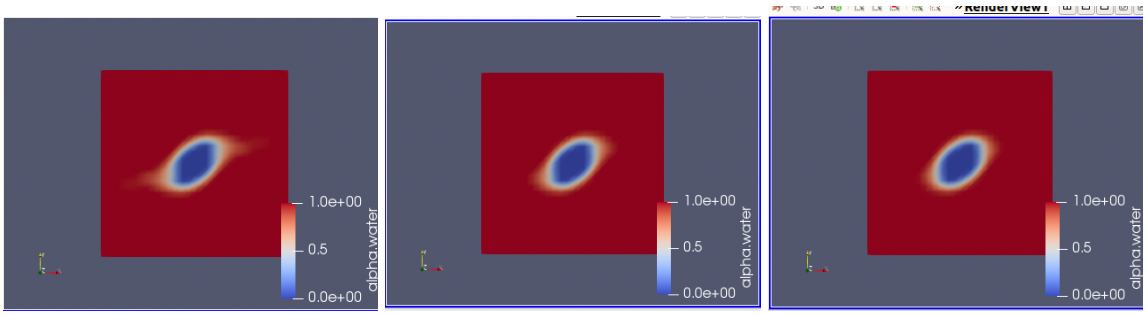
For low Capillary numbers the drop deforms into an ellipsoidal shape and does not deform further. Thus a steady state is achieved. As Ca increases, the drop starts breaking up from the edges to form small droplets[1][3]. After sufficient breakage of drop from the edges, it pinches off from the middle.
At $u = 0.9931, \sigma = 2.4828$:



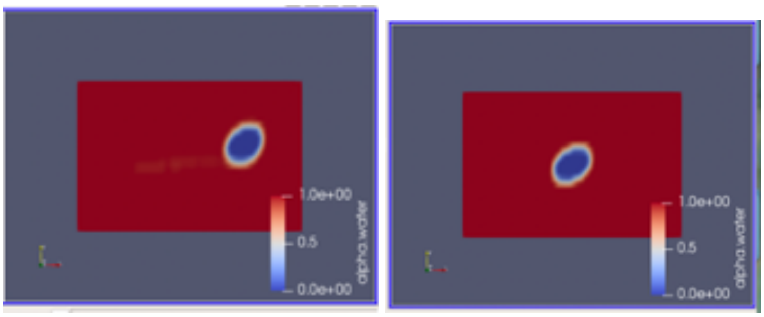
This matches with the figure shown in the paper:



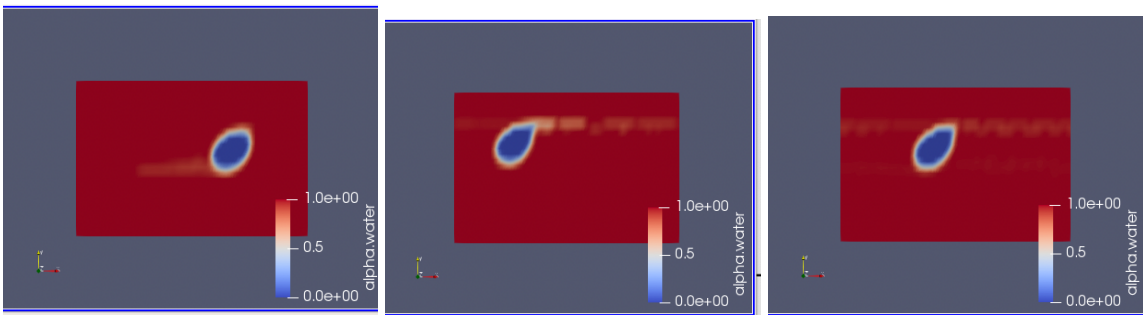
At $u = 0.6621, \sigma = 1.1034; u = 0.5517, \sigma = 0.9195; u = 0.4855, \sigma = 0.8092$ (breakage into droplets):



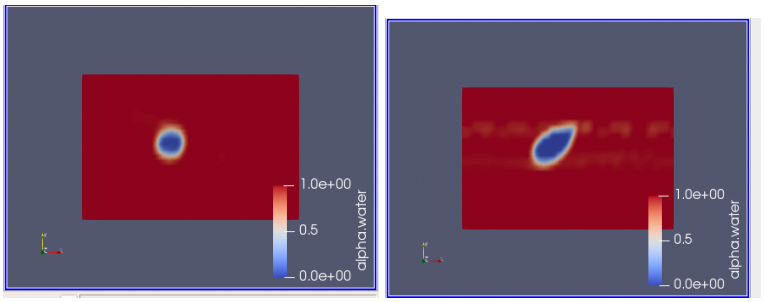
At $u = 0.4414\sigma = 1.1034; u = 0.0110\sigma = 0.0092;$ (steady state):



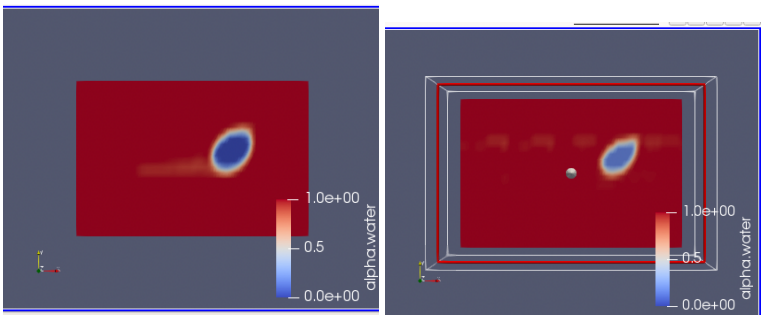
In OpenFOAM, the breakup from the middle was observed for very high Ca and Re than for the values for which it was observed in the paper.
By varying the viscosity we can see that the end pinching happens first from the bottom edge for higher μ, but from the top edge for lower μ.
At $\mu = 2, 1, 1.5$:



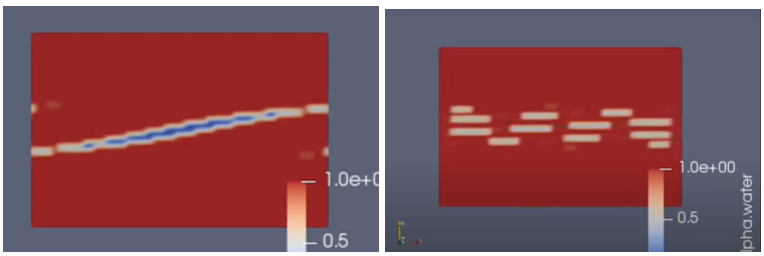
If the fluid in the matrix and the drop was instead water, we observe that no deformation happens. Thus a higher kinematic viscosity favours deformation and pinching. At $\nu = 1.0016 * 10^{-6} Pas$(water); $\nu = 1.3793 * 10^{-3}$(honey):



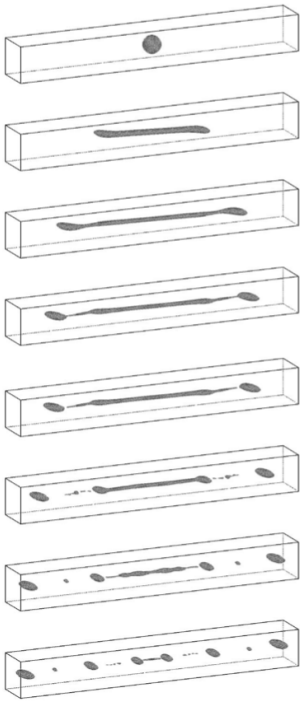
For a smaller drop, the deformation is lesser than what it is for a bigger drop.
At $a = 0.25, 0.2$:



Complete breakage into droplets (pinching from the middle also observed) at $u = 1.1034, \sigma = 0.00416667$



Complete breakup as shown in paper:



From these comparisons and test cases, we see that the results presented in the paper match with simulations obtained in OpenFOAM for similar values of the parameters.

References

[1] Jie Li, Yuriko Y. Renardy, and Michael Renardy. "Numerical simulation of breakup of a viscous drop in simple shear flow through a volume-of-fluid method." In: Physics of Fluids (1994-present) 12, 269 (2000); DOI: http://dx.doi.org/10.1063/1.870305

[2] H. A. Stone, ''Dynamics of drop deformation and breakup in viscous fluids,'' Annu. Rev. Fluid Mech. **26**, 65 (1994).

[3] V. Cristini, J. Blawzdziewicz, and M. Loewenberg, ''Drop breakup in three-dimensional viscous flows,'' Phys. Fluids 10, 1781 (1998).