NUMERICAL INVESTIGATION OF THE NEAR HEAD-ON COLLISION DYNAMICS OF NON-NEWTONIAN DROPLETS: A COMPARATIVE STUDY USING THE VOLUME OF FLUID AND THE LOCAL FRONT RECONSTRUCTION METHOD

Cristina García Llamas, Penbe Merve Durubal, Arie Huijgen, Kay Buist, Hans Kuipers, & Maike Baltussen

Department of Chemical Engineering and Chemistry, Multiphase Reactors Group, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

*Address all correspondence to: Maike Baltussen, Department of Chemical Engineering and Chemistry, Multiphase Reactors Group, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands, E-mail: m.w.baltussen@tue.nl

Original Manuscript Submitted: 11/22/2023; Final Draft Received: 4/10/2024

Droplet-droplet interactions of highly viscous liquid suspensions have a major impact on industrial processes such as spray drying, fuel combustion, or waste treatment. The efficiency of these processes depends heavily on the morphology of the droplets after the collision (i.e., surface area and volume). Although often encountered, the physical mechanisms governing merging and break-up of non-Newtonian droplets are largely unknown. It is therefore of paramount importance to gain a better understanding of the complex physics dominating the collision of non-Newtonian droplets. In this research, we investigate numerically the collision of droplets using the local front reconstruction method (LFRM) and the volume of fluid (VOF) method. The coalescence and stretching separation regime are studied using a xanthan solution, whose shear-thinning rheology is described with the Carreau–Yasuda model. The capabilities of the two methods to capture the complex topological changes are assessed by a one-to-one comparison of the numerical results with experiments for near head-on collisions at various We numbers.

KEY WORDS: non-Newtonian, droplet collisions, volume of fluid, local front reconstruction method, direct numerical simulation

1. INTRODUCTION

The droplet size distribution of droplet-droplet collisions of highly viscous liquids and suspensions is of foremost importance in various industrial applications such as spray drying, fuel combustion, spray coating, and irrigation (Brazier-Smith et al., 1972; Herskowitz and Smith, 1983; Jiang et al., 2010; Post and Abraham, 2002; Yao and Takei, 2017). The collision outcome is
influenced by various impact parameters such as the inertia, the relative spatial distribution upon interaction, or the non-Newtonian behavior of the interacting droplets (Motzigemba et al., 2002). Generally, droplet-droplet collisions can be classified into four regimes: coalescence, bouncing, reflexive, and stretching separation.

Coalescence occurs when the droplets merge into a single larger droplet, while in bouncing the droplets rebound without merging. In the reflexive and stretching separation regime, the droplets combine into one single droplet, which eventually breaks apart into two or more (satellite) droplets. To classify the collisions in these regimes, a collision map is used, which describes the interaction of droplets based on two factors: the impact Weber number \( \text{We} \) and the impact parameter \( B \) [see Eqs. (1) and (3), respectively] at a given or apparent Oh number [see Eq. (2)]. \( \text{We} \) is determined by the relative velocity \( u_{\text{rel}} \) of the droplets, liquid density \( \rho \), surface tension coefficient \( \sigma \), and the droplet diameter \( d \), while \( B \) is the sine of the angle between the droplets, which can be calculated by the perpendicular distance \( b \) between the droplets and the position vector of their center of mass in the direction of motion (see Fig. 1). Alternatively, the Oh represents the ratio of viscous to surface tension forces, where \( \mu_l \) represents the effective viscosity of the droplet.

\[
\text{We} = \frac{\rho d u_{\text{rel}}^2}{\sigma},
\]

FIG. 1: Schematic representation of two colliding droplets
In a collision map, the boundaries between the collision regimes are defined (Adam et al., 1968; Ashgriz and Poo, 1990; Estrade et al., 1999; Jiang et al., 1992; Qian and Law, 1997). However, the lines demarcating the transition between regimes are not sharp as they depend on the type of liquid. In particular, the non-Newtonian behavior of the liquid plays an important role in the collision as pointed out by Motzigemba et al. (2002) and Finotello et al. (2018), which observed that the transition from coalescence to stretching separation occurs at much higher Weber numbers.

Despite these findings, the research on non-Newtonian droplet collisions is relatively limited. This is primarily due to the technical difficulties associated with studying experimentally non-Newtonian fluids, such as accurately capturing local changes and handling the materials. In contrast, interface-resolved numerical simulations provide an excellent tool to study non-Newtonian droplet collisions as it allows for a complete characterization of the fields (see a summary of the available research in Table 1). Despite this, only a few numerical works have been devoted to the study of non-Newtonian droplet collisions. França et al. (2022) investigated non-Newtonian droplet-droplet collisions using a 2D lattice Boltzmann method (LBM) to represent the interface and the modified smoothed particle hydrodynamics (SPH) for the flow field. Focke and Bothe (2011) and Motzigemba et al. (2002) used the volume of fluid (VOF) method to track the interface and the power-law model to model the non-Newtonian behavior.

As evident from the presented review, front-capturing methods [i.e., VOF (van Sint Annaland et al., 2004), level set (LS) (Gibou et al., 2018) or combined level set VOF (CLSVOF)]

<table>
<thead>
<tr>
<th>Source</th>
<th>Model</th>
<th>Viscosity model</th>
<th>Validation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Motzigemba et al. (2002)</td>
<td>VOF</td>
<td>Power-law</td>
<td>water/carboxy-methylcellulose (CMC)</td>
</tr>
<tr>
<td>Focke and Bothe (2011)</td>
<td>VOF</td>
<td>Power-law</td>
<td>CMC Motzigemba et al. (2002)</td>
</tr>
<tr>
<td>Focke and Bothe (2012)</td>
<td>VOF</td>
<td>Effective viscosity</td>
<td>Motzigemba et al. (2002); Qian and Law (1997); Willis and Orme (2003)</td>
</tr>
<tr>
<td>Shabankareh et al. (2017)</td>
<td>LS</td>
<td>Power-law</td>
<td>Theory Taylor (1966)</td>
</tr>
<tr>
<td>Sun et al. (2015a)</td>
<td>LBM</td>
<td>Carreau–Yasuda</td>
<td>—</td>
</tr>
<tr>
<td>Sun et al. (2015b)</td>
<td>LBM</td>
<td>Power-law</td>
<td>CMC Motzigemba et al. (2002)</td>
</tr>
<tr>
<td>Xu et al. (2020)</td>
<td>SPH</td>
<td>Cross model</td>
<td>—</td>
</tr>
<tr>
<td>Qian et al. (2020)</td>
<td>CLSVOF</td>
<td>Power-law</td>
<td>CMC Motzigemba et al. (2002)</td>
</tr>
<tr>
<td>França et al. (2022)</td>
<td>FT+MAC</td>
<td>Carreau–Yasuda</td>
<td>Xanthan gum Finotello et al. (2018)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Oldroyd-B; PTT</td>
<td>—</td>
</tr>
<tr>
<td>Mandal et al. (2023)</td>
<td>FEM</td>
<td>Power-law</td>
<td>Theory Reddy and Esmaeili (2009); Taylor (1966)</td>
</tr>
</tbody>
</table>
(van der Pijl et al., 2005)] are typically used to simulate droplet-droplet collisions as they can handle strong topological changes (i.e., merging and break-up) at relatively low computational cost. However, a highly refined grid is required to capture strong deformations of the interface such as those occurring in a high inertia collision where a thin disk shape (lamella) is formed after the merging of the droplets. The lamella quickly contracts or breaks due to the combined effects of inertia and surface tension forces, resulting in complex deformations of the gas-liquid interface. Front-tracking methods track the interface explicitly, resulting in a sharper definition of the interface, which enables the use of coarser grids compared to front-capturing methods. However, traditional front-tracking methods cannot handle merging and break-up. Hence, only a limited number of research studies have used the front-tracking method to simulate droplet-droplet collisions. For instance, Rajkotwala et al. (2018, 2020) proofed the strong capabilities of the local front reconstruction method (LFRM) for simulating Newtonian droplet-droplet collisions. Despite this, at the time this article is written, no additional studies have used LFRM to simulate droplet collisions.

In this work, we study numerically droplet-droplet collisions of a shear-thinning liquid using two interface methods to track the interface: VOF and LFRM. A comparison between VOF and LFRM is established, where the accuracy of both methods at capturing the liquid-gas interface is evaluated. The non-Newtonian behavior of the liquid is captured with the Carreau–Yasuda model and the numerical results are validated with the experiments of Finotello et al. (2018), which are performed at the same conditions as the simulations.

2. MODEL FORMULATION

The Navier–Stokes equations for the liquid and the gas phase are combined into a single equation via the one-fluid formulation, whereby both fluids are assumed to be incompressible and immiscible [see momentum and continuity equations in Eqs. (5) and (4), respectively].

\[ \nabla \cdot \mathbf{u} = 0, \tag{4} \]

\[ \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \nabla \cdot (\mathbf{u} \mathbf{u}) = -\nabla p + \rho \mathbf{g} - \nabla \cdot \mathbf{\tau} + \mathbf{F}_\sigma, \tag{5} \]

where \( \rho \) is the density, \( \mathbf{u} \) is the fluid velocity, \( t \) is the time, \( p \) is the pressure, \( \mathbf{g} \) is the gravity, \( \mathbf{F}_\sigma \) is the body force representing surface tension, and \( \mathbf{\tau} \) is the stress tensor, which is computed using Eq. (6).

\[ \mathbf{\tau} = \mu_{\text{app}}(\dot{\gamma})\dot{\gamma}, \tag{6} \]

where \( \mu_{\text{app}} \) is the apparent viscosity [see Eq. (12)] and \( \dot{\gamma} = \sqrt{(1/2)\dot{\gamma} : \dot{\gamma}} \) and \( \dot{\gamma} \) is the shear-rate tensor computed using Eq. (7).

\[ \dot{\gamma} = \frac{1}{2} \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right). \tag{7} \]

2.1 Numerical Methodology

The calculation of \( \mu_{\text{app}} \) in this equation is detailed in Section 2.2. The density and viscosity are calculated each time step by simple linear and harmonic averaging of the intrinsic properties using the phase fraction field, respectively.

The continuity and momentum equations are discretized using the finite-volume method and are solved on a staggered Cartesian grid. An adapted version of the projection method introduced
by Chorin (1968) is used to solve the equations, whereby the velocity and pressure fields are calculated in two steps. First, a tentative velocity ($u^*$) field is calculated from the momentum equation [see Eq. (5)]. The convective term is discretized using a total variation diminishing (TVD) scheme (Versteeg and Malalasekera, 1995) whereas the shear stress is discretized semi-implicitly using a second-order differencing scheme. The tentative velocity field is solved using a bi-conjugate-gradient stabilized solver 2 (Masterov, 2019). The obtained velocity field does not obey the continuity equation given by Eq. (4). Thus, the tentative velocity is corrected to ensure mass conservation by solving the Poisson equation. The resulting linear set of equations is solved using an AMG-preconditioner (Trilinos Project Team, 2020) and the bi-conjugate-gradient stabilized solver 2 (Masterov, 2019).

After calculating the fields, the interface is advected and the surface tension is recalculated. The details of the advection and surface tension calculation are discussed in the following section as the two interface methods used in this research (i.e., VOF and LFRM) treat the interface in a different manner.

2.1.1 Volume of Fluid

In VOF, the interface is captured implicitly on the basis of a color function. In the current work, an adaptation from the original method proposed by van Sint Annaland et al. (2005) and Baltussen et al. (2014) is used.

The liquid-gas interface is reconstructed from the color field by means of piecewise linear interface calculation (PLIC) (Youngs, 1982) and it is advected using Eq. (8).

$$\frac{DF}{Dt} = \frac{\partial F}{\partial t} + \mathbf{u} \cdot \nabla F = 0,$$

(8)

where $F$ is the color field, which in the current method is equal to the phase fraction field.

The surface tension is included in the governing equations, using the continuous surface force (CSF) method of Brackbill et al. (1992) [see Eq. (9)]. To reduce the spurious currents originating from the imbalance of the pressure gradient and the surface tension forces, the balanced force method of Francois et al. (2006) is used. The curvature of the interface is computed using the hybrid convolution – generalized height function (CV-GHF) method proposed by Patel et al. (2018). This method is an improved version of the height function method to calculate the curvature, which switches to the convolution method when the computed height functions are not consistent. The consistency requires that there is a monotonic variation of the color function in the columns used in the height function calculation.

$$F_\sigma = \sigma \kappa \mathbf{n},$$

(9)

where $\sigma$ is the surface tension coefficient, $\mathbf{n}$ the normal at the interface, and $\kappa$ the curvature. The normal $\mathbf{n}$ is calculated by

$$\mathbf{n} = \frac{\nabla F}{|\nabla F|}.$$  

(10)

2.1.2 Local Front Reconstruction Method

The liquid-gas interface is modeled using a mesh of marker points, which are advected using the local velocity. To determine the velocities at the marker point positions, cubic spline interpolation
is performed. The new positions of the marker points are obtained by numerically integrating the interpolated velocities with a 4th-order Runge-Kutta scheme. Following this, the phase fraction is computed using a geometrical approach in which the phase fraction is added or subtracted from the Euler cells based on the marker normal direction (Dijkstraun et al., 2010).

After the advection, marker points might become unevenly distributed across the interface surface resulting in inaccuracies in the computation of local surface tension forces. To mitigate this issue, a periodic remeshing operation is carried out to redistribute the points evenly across the interface surface. In this work, the remeshing technique employed is the LFRM of Shin et al. (2011). The advection and remeshing may introduce numerical discretization errors resulting in variations in the global mass and volume of the interface. To mitigate these errors, the volume of the interface is restored at each time step through the volume conservative smoothing proposed by Kuprat et al. (2001).

The surface tension between the two phases is handled by the surface tension term \( F_{\sigma} \) and it is calculated using the hybrid surface tension method of Shin et al. (2005).

\[
F_{\sigma} = \sigma \kappa_H \nabla F,
\]  

where \( \sigma \) is the surface tension coefficient, \( \kappa_H \) the local interface curvature, and \( F \) the phase fraction field.

### 2.2 Viscosity Models

In this work, we investigate a shear-thinning liquid, which is characterized by lower shear stress upon increasing shear rate. To model the rheology of the shear-thinning fluids, the Carreau–Yasuda model is used, mathematical description of which is given by Eq. (12). In the Carreau–Yasuda model, the viscosity of the fluid is related to the shear rate by a power-law function, where the exponent defines the degree of non-linearity with respect to the shear rate. This model considers a smooth transition between the low-shear-rate behavior of the fluid, which is dominated by its viscosity, and the high-shear-rate behavior, which is dominated by its elasticity (Bird, 1987; Finotello et al., 2018).

\[
\mu_{\text{app}} = \mu_\infty + (\mu_0 - \mu_\infty) \left[1 + \left(\frac{\dot{\gamma}}{K}\right)^{\frac{n}{\alpha}}\right]^{-\frac{1}{(n-1)}},
\]

where \( \mu_0 \) and \( \mu_\infty \) are the boundary values of shear viscosity, while \( K, n, \) and \( \alpha \) are fitting parameters.

### 3. VERIFICATION OF NON-NEWTONIAN MODEL

The Carreau–Yasuda model was verified by comparing duct flow simulation results using the settings in Table 2 with the analytical solution. Here, the rheological properties of the shear-thinning fluid are those from the experimental work of Finotello et al. (2018). The analytical solution is obtained from the Navier–Stokes equations assuming a steady, incompressible, and fully developed flow between two infinite plates which is described by Eq. (13). This equation is solved using the shooting method to satisfy the two-point boundary conditions.

\[
-\frac{\Delta p}{L} = \frac{du}{dy} \left[\mu_\infty + (\mu_0 - \mu_\infty) \left[1 + \left(\frac{K \left|\frac{du}{dy}\right|^{\alpha}}{\dot{\gamma}}\right)^{\frac{n}{\alpha}}\right]^{-\frac{1}{n-1}}\right],
\]

where \( \alpha, K, \) and \( n \) are the Carreau–Yasuda coefficients summarized in Table 2.
TABLE 2: Simulation properties duct of flow with Carreau–Yasuda model by Finotello et al. (2018)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimensions</td>
<td>10 × 2</td>
<td>cm</td>
</tr>
<tr>
<td>Grid points</td>
<td>500 × 100</td>
<td>[-]</td>
</tr>
<tr>
<td>Initial velocity</td>
<td>(0.0925–7.11) × 10⁻²</td>
<td>m/s</td>
</tr>
<tr>
<td>Courant</td>
<td>0.1</td>
<td>s</td>
</tr>
<tr>
<td>Carreau–Yasuda index, n</td>
<td>0.4</td>
<td>[-]</td>
</tr>
<tr>
<td>Carreau–Yasuda index, K</td>
<td>3.29</td>
<td>[-]</td>
</tr>
<tr>
<td>Carreau–Yasuda index, α</td>
<td>9.4</td>
<td>[-]</td>
</tr>
<tr>
<td>Infinite shear viscosity, µ∞</td>
<td>2.5 × 10⁻³</td>
<td>Pa·s</td>
</tr>
<tr>
<td>Zero shear viscosity, µ₀</td>
<td>0.228</td>
<td>Pa·s</td>
</tr>
</tbody>
</table>

Figure 2 shows both the analytical and the simulated velocity profile and effective viscosity (left and right figure, respectively) across a half cross section of the duct (i.e., from the wall to the center line of the duct). This graph shows an excellent match between simulations and analytical results for all the studied conditions. Here, the shear-thinning behavior of the fluid is evident in the intermediate region between the wall and the center line, where the shear forces are dominating. In contrast, the fluid exhibits Newtonian behavior in proximity to the wall and center line due to the presence of high and nearly zero shear rates, respectively.

4. PROBLEM DESCRIPTION

In this work, we investigate near head-on collision of a 500 ppm xanthan-solution droplets with a diameter of $D_0 = 1000 \, \mu m$, for $We \approx 40$ to 210. To capture the shear-thinning rheology of the xanthan solution, the Carreau–Yasuda model is fitted with the parameters in Table 2. The physical properties of the fluids employed in this research are summarized in Table 3.

FIG. 2: Simulation (○) and analytical solutions (-) using the Carreau–Yasuda (CY) model, where the velocity (left) and effective viscosity (right) are shown from the wall to the center line of a duct
TABLE 3: Physical properties used in the current work

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid density, $\rho_l$</td>
<td>1000</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Gas density, $\rho_g$</td>
<td>1.185</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Gas viscosity, $\mu_g$</td>
<td>$1.85 \times 10^{-5}$</td>
<td>m²/s</td>
</tr>
<tr>
<td>Surface tension, $\sigma$</td>
<td>72.2</td>
<td>mN/m</td>
</tr>
</tbody>
</table>

Additionally, the simulations are performed on a fixed Cartesian grid using the in-house code foxberry. The size of the computational domain and the grid resolution for the VOF and LFRM simulations is shown in Table 4. In the LFRM simulations, the droplets are placed at the center of the domain, while in the VOF simulations they are located along the diagonal direction of the domain to allow for a smaller computational domain. A free slip boundary condition is applied to all the domain boundaries. Table 5 shows the impact Weber (We) and impact parameters ($B$) of the colliding droplets studied in the current work.

5. RESULTS

In this section, the simulation results using VOF and LFRM are compared with the experiments from Finotello et al. (2018). Figures 3, 4, and 5 show the experimental and numerical shape of the droplets, with We = 40, We = 98.6, and We = 211, respectively. In the three figures, the simulated encounter process of the droplets show a good correspondence with the experiments until $t \sim 4$ ms, where they start deviating. Additionally, in the case with We = 98.6, shown in Fig. 4, the simulated droplets using VOF deviate from LFRM results and experiments after $t = 3$ ms as the grid size is not small enough to capture the complex deformations of the droplets. Thus, they are discarded from this study. The deviation with the experiments during the final stages of the collision could be associated with neglecting the extensional viscosity in the model. This extensional viscosity results in more pronounced energy dissipation which promotes the coalescence, as observed in the experiments of Finotello et al. (2018).

In the higher impact velocity cases with We = 98.6 and We = 211, shown in Figs. 4 and 5, respectively, the merged droplet acquires non-symmetrical shapes, which are not observed in the

TABLE 4: Simulation settings used in the current work

<table>
<thead>
<tr>
<th>Property</th>
<th>LFRM</th>
<th>VOF</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid resolution</td>
<td>30</td>
<td>40</td>
<td>Cells/$D_0$</td>
</tr>
<tr>
<td>Computational domain</td>
<td>$(5D_0, 10D_0, 10D_0)$</td>
<td>$(5D_0, 5D_0, 5D_0)$</td>
<td>m</td>
</tr>
</tbody>
</table>

TABLE 5: Impact properties of the droplets used in the simulations

<table>
<thead>
<tr>
<th>Case</th>
<th>We</th>
<th>$B$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>40.17</td>
<td>0.009</td>
</tr>
<tr>
<td>Case 2</td>
<td>98.6</td>
<td>0.045</td>
</tr>
<tr>
<td>Case 3</td>
<td>210.9</td>
<td>0.002</td>
</tr>
</tbody>
</table>
Numerical Investigation of the Near Head-on Collision Dynamics of Non-Newtonian Droplets

FIG. 3: Binary droplet collision at $We = 40$, $B = 0.01$ with LFRM and VOF compared with experiments Finotello et al. (2018). Each image corresponds to the following times $t = \{0, 0.5, 1, 1.5, 2, 2.5, 3, 3.5, 4, 4.5, 5, 6, 7\}$ ms, from left to right and from top to bottom.

<table>
<thead>
<tr>
<th>$We = 40.2$, $B = 0.01$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exp.</td>
</tr>
<tr>
<td><img src="image1.png" alt="Image 1" /></td>
</tr>
</tbody>
</table>

FIG. 4: Binary droplet collision at $We = 98.6$, $B = 0.0045$ with LFRM and VOF compared with experiments Finotello et al. (2018). Each image corresponds to the following times $t = \{0, 0.5, 1, 1.5, 2, 2.5, 3, 3.5, 4, 5, 6, 7, 8, 9, 10\}$ ms, from left to right and from top to bottom. The striped boxes indicate results excluded from this research.

<table>
<thead>
<tr>
<th>$We = 98.6$, $B = 0.0045$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exp.</td>
</tr>
<tr>
<td><img src="image10.png" alt="Image 10" /></td>
</tr>
</tbody>
</table>

lower inertia case (see Fig. 3). Upon collision, the droplets combine and subsequently a lamella shape expands radially forming a disk shape with an outer rim. The merged droplet undergoes expansion along the axis of impact, resulting in stretching separation in simulations and coalescence in experiments. Upon impact, a ring with a diameter of 3.5 times the initial droplet diameter ($D_0$) is formed for the case with $We = 98.6$ (see Fig. 4, which notably surpasses the
FIG. 5: Binary droplet collision at $\text{We} = 211$, $B = 0.002$ with LFRM and VOF compared with experiments Finotello et al. (2018). Each image corresponds to the following times $t = [0, 0.5, 1, 1.5, 2, 2.5, 3, 4, 5, 6, 7, 8]$ ms, from left to right and from top to bottom.

disk diameter of $2.2D_0$ observed for a Newtonian fluid by Finotello et al. (2018). The augmented expansion of the ring in the non-Newtonian scenario can be attributed to reduced viscosity stemming from elevated shear forces, which is consistent with the experimental observations reported by Motzigemba et al. (2002). In the simulations, the lamella shows break-up when it becomes too thin to be resolved by the interface method. However, we cannot conclude if the lamella rupture observed in the simulations is physical, as the resolution of the experimental images is not sufficiently high to capture it. After the lamella breaks, the droplets tracked with VOF and LFRM show different deformations (see $t = 3$ ms in Figs. 4 and 5), whilst LFRM shows more similarity with the experiments. The irregular deformations of the coalesced droplet following the lamella rupture may stem from the combined effects of inertia and local viscosity variations within the droplet, while in the final phases of the collision process (see droplet deformations after $t = 5$ ms in Figs. 4 and 5), the influence of surface tension forces becomes pronounced, leading to the stabilization of the droplet and consequent formation of more uniform shapes. In the case with $\text{We} = 211$, the collision outcome produced by VOF deviates from experimental results. This discrepancy is likely caused by the numerical limitations in representing the highly curved interfaces formed during the time interval of 2–4 ms. In contrast, the outcome predicted by LFRM is in line with the experiments. This discrepancy between VOF and LFRM could be associated with the difference in the resolution of the grids which directly influence break-up. Additionally, LFRM benefits from the explicit tracking of the interface, which generally results in more accurate results compared to VOF.

6. CONCLUSIONS AND OUTLOOK

In this work, near head-on collision of xanthan-solution droplets were studied numerically using LFRM and VOF to track the evolution of the complex topological changes arising from these collisions. The shear-thinning behavior of the xanthan gum solution was captured using the Carreau–Yasuda model.
The position and shape of the merged droplet obtained with VOF and LFRM were evaluated and compared with experimental data. This comparison shows an excellent match between experiments and simulations during the initial stages of the collision. The deviations between the simulations and the experiments could be associated with the effect of extensional viscosity, which is not included in the model. At high impact velocity, the merged droplet undergoes complex topological changes where LFRM produces results which are closer to reality compared to VOF. Additionally, the LFRM simulations are computationally less expensive as the explicit tracking of the interface allows for a coarser grid. Therefore, LFRM offers the advantage of reduced computational cost while maintaining an accuracy comparable to VOF simulations.

The ability of VOF and LFRM at capturing the collision of xanthan droplets should be further investigated over a broader range of impact parameters $B$. Additionally, the authors recommend applying other non-Newtonian models in droplet-droplet collisions to gain a further understanding on the capabilities and limitations of each modeling strategy.

**ACKNOWLEDGMENTS**

This work is part of the “EEMS – Energy Efficient Milky Sprays” project which took place within the framework of the Institute of Sustainable Process Technology (ISPT) and was co-financed by the TKI-Energy & Industry, Danone, DSM-firmeni ch and FrieslandCampina. More information can be found at the https://ispt.eu/projects/eems/EEMS website. In addition, this work is supported by the Netherlands Center for Multiscale Catalytic Energy Conversion (MCEC), an NWO Gravitation Program funded by the Ministry of Education, Culture, and Science of the government of the Netherlands. This project has received funding from the European Union’s Horizon 2020 Research and Innovation Program under the Marie Skłodowska-Curie Grant Agreement No. 801359.

**REFERENCES**


Estrade, J., Lavergne, G., and Biscos, Y., Experimental Investigation of Dynamic Binary Collision of


