Coupled smoothed particle hydrodynamics and discrete element method for simulating coarse food particles in a non-Newtonian conveying fluid
Lian, Xue; Savari, Chiya; Li, Kun; Barigou, Mostafa

DOI: 10.1063/5.0144992

License: Creative Commons: Attribution (CC BY)

Document Version
Publisher's PDF, also known as Version of record

Citation for published version (Harvard):

Link to publication on Research at Birmingham portal

General rights
Unless a licence is specified above, all rights (including copyright and moral rights) in this document are retained by the authors and/or the copyright holders. The express permission of the copyright holder must be obtained for any use of this material other than for purposes permitted by law.

• Users may freely distribute the URL that is used to identify this publication.
• Users may download and/or print one copy of the publication from the University of Birmingham research portal for the purpose of private study or non-commercial research.
• Users may use extracts from the document in line with the concept of ‘fair dealing’ under the Copyright, Designs and Patents Act 1988 (?)
• Users may not further distribute the material nor use it for the purposes of commercial gain.

Where a licence is displayed above, please note the terms and conditions of the licence govern your use of this document.

When citing, please reference the published version.

Take down policy
While the University of Birmingham exercises care and attention in making items available there are rare occasions when an item has been uploaded in error or has been deemed to be commercially or otherwise sensitive.

If you believe that this is the case for this document, please contact UBIRA@lists.bham.ac.uk providing details and we will remove access to the work immediately and investigate.

Download date: 15. Aug. 2023
Coupled smoothed particle hydrodynamics and discrete element method for simulating coarse food particles in a non-Newtonian conveying fluid

Cite as: Phys. Fluids 35, 043325 (2023); doi: 10.1063/5.0144992
Submitted: 2 February 2023 · Accepted: 30 March 2023 · Published Online: 19 April 2023

Xue Lian (廉雪), Chiya Savari, Kun Li (库坤), and Mostafa Barigou

AFFILIATIONS
School of Chemical Engineering, University of Birmingham, Edgbaston, Birmingham B15 2TT, United Kingdom

Note: This paper is part of the special topic, Special Issue on Food Physics.

Author to whom correspondence should be addressed: m.barigou@bham.ac.uk

ABSTRACT

A Lagrangian particle-based numerical framework based on smoothed particle hydrodynamics (SPH) coupled with a discrete element method (DEM) was used to simulate the flow behavior of coarse food particles in a non-Newtonian conveying fluid in a horizontal pipe. Nearly neutrally buoyant nearly spherical calcium-alginate particles were used as model food particles. The capability of the SPH–DEM methodology was successfully validated in non-Newtonian single-phase as well as in two-phase particle–liquid flows by comparing the local phase velocity flow field, radial particle distribution, and particle passage times with experimental Lagrangian measurements obtained by a technique of positron emission particle tracking. The simulations also yielded accurate predictions of flow pressure drop. In addition, detailed information was afforded on local particle spin, fluid pressure, and carrier fluid vorticity. The results demonstrate the high capability of the proposed numerical framework to predict the complex features of complex particle–liquid flows in pipes.

© 2023 Author(s). All article content, except where otherwise noted, is licensed under a Creative Commons Attribution (CC BY) license (http://creativecommons.org/licenses/by/4.0/). https://doi.org/10.1063/5.0144992

I. INTRODUCTION

Pipe transport of food materials is a common operation in food manufacture. For example, continuous in-line processing of particulate foods relies on the conveying of particle–liquid mixtures through pipes. Typically, the challenge in a heat-hold-cool process is to ensure a safe and uniform level of sterility while preventing over-processing. Often, in such processes, the food particles are coarse solids (up to 20 mm), their loading is high, and the carrier fluid is viscous of non-Newtonian rheology. Compared to single-phase flow, the dynamical behavior of these particle–liquid flows is extremely varied and complex and depends on many factors including particle properties (size, shape, and density), liquid properties (density and rheology), and other process parameters (pipe size, flow rate, and particle concentration). Particle flow behavior critically affects product heating, cooling, and microbial sterilization. Moreover, information on the two-phase flow pressure drop and particle residence time distribution is also necessary for process optimization. Thus, the knowledge of the detailed two-phase flow behavior is necessary for optimum food processing as the final product quality is highly dependent on the usually complex flow patterns involved. Understanding this complexity and being able to model it would undoubtedly represent a big step forward toward improving current food engineering practices. To achieve this aim, advanced and reliable experimental and numerical methodologies are required.

Imaging of food flows is very challenging because of their opacity which prevents the use of leading laser-based optical measurement techniques, such as particle image velocimetry (PIV) and laser Doppler velocimetry (LDV), except for unrealistically dilute situations. Positron emission particle tracking (PEPT) is able to visualize concentrated opaque particle–liquid flows by providing the long-term three-dimensional (3D) trajectories of all components, while being noninvasive. In PEPT, a tiny radiolabeled tracer is utilized to track the flow components in space and time with comparable accuracy to LDV and PIV. PEPT has extensively been used to study single-phase as well as complex multiphase flows and the data provided have successfully been exploited to inform, drive, and validate various modeling methodologies. The technique, however, is very
expensive to run and not widely available, which makes any available data scarce and precious. Hence, there is a need for the development of advanced theoretical/numerical modeling strategies to reduce reliance on experimental work in such a challenging field.

Various modeling approaches have been attempted over the years, each having its own advantages and disadvantages, focusing on certain aspects of the flow and oversimplifying others. There are two main approaches, mesh-based and particle-based, to simulate the carrier fluid in a particle–liquid flow. In mesh-based computational fluid dynamics (CFD) methods, the fluid is treated as a continuum and a mesh is used to describe the flow domain. In many cases, the two-fluid or Eulerian–Eulerian (E–E) model has been used to study particle–liquid food flow in pipes under different flow regimes and solid loadings.25 Data in the form of radial velocity profiles and radial solid phase distributions determined by different experimental techniques were used to validate the simulations. However, as this numerical approach relies on a mesh to evaluate the development of the flow field and structural deformation, the large deformation of the mesh can lead to mesh skewness which affects the accuracy of the predictions. This limitation is even more challenging when the mesh needs to be modified in complex flow geometries due to topological changes and the meshing process can be extremely complicated.22 Moreover, in this approach, the distribution of local flow properties in a domain is represented without reference to the history of the fluid or particle trajectories. Thus, where localized particle effects are dominant such as in heterogeneous dense flows of coarse particles, such effects cannot be accurately modeled by conventional mesh-based CFD approaches.

Smoothed particle hydrodynamics (SPH) is a more recent mesh-free alternative to traditional CFD techniques and is a fully Lagrangian particle-based method. The continuum is divided into a set of computational particles, and an appropriate set of governing equations, such as Navier–Stokes, are solved for each computational particle. This technique has shown a lot of promise and has recently been used in a number of studies to simulate single-phase and particle–liquid flows in different geometries.23–25 In contrast to traditional mesh-based CFD approaches, SPH promises to be particularly efficient in solid–liquid flows where particle and flow characteristic dimensions are comparable. It has the added advantage of being able to model stresses within fluid and solid, but not particle–particle or particle–wall contact forces. The discrete element method (DEM), however, is a robust numerical approach for modeling particulate systems, where particles are treated as individual entities and their interactions are usually modeled using established contact laws.26 Thus, complete quantitative information about the time evolution and spatial distribution of local solid fraction, particle kinematics, and both collisional stresses and streaming stresses can be obtained. Such detailed, physically sound information cannot be obtained through continuum modeling, such as CFD.

A number of other modeling techniques exist, such as lattice Boltzmann method (LBM),27 finite element method (FEM),28 boundary element method (BEM),29 and coarse-grained molecular dynamics (CGMD).30 However, there is currently no single modeling methodology which can effectively tackle these complex particle–liquid flows. The continuum approach used in CFD is more efficient in handling the fluid phase, low solid concentrations, and small particles, while DEM is much better at modeling high solid concentrations, but it cannot handle the continuous phase well. LBM, on the other hand, is able to handle complex boundary conditions, such as those found on the endothelium, and can also be applied to microscale flow phenomena, but it is not efficient for particles with complex shapes; while SPH is a relatively general method which can be applied to both fluid and solids, can handle surface flows well, but it has some limitations in dealing with solid boundaries. The best solution, therefore, lies in a methodology which can exploit the strengths of each technique and assemble these methods in an optimized hybrid fashion. Selecting methods which are particle-based facilitates their linking in a hybrid fashion. In particular, the SPH and DEM algorithms are almost identical, essentially the only difference being the routines needed to calculate the forces used for updating particle trajectories, which simplifies code development and makes it more efficient.

A coupled SPH–DEM framework, therefore, should be able to appropriately and efficiently handle the interactions between the carrier fluid and solid particles to yield flow details, such as the rotational angular velocity (or spin) of the particles and Lagrangian trajectories of both particles and fluid. The coupled SPH–DEM model is highly robust and flexible with regard to the shape and size of the solid particles considered compared to conventional mesh-based CFD approaches31 and has recently been successfully applied to simulate different flow systems, including packed beds,32 particle sedimentation,33 and debris flow.34 However, few studies have been validated due to the lack of appropriate experimental data, and food flows have not been investigated in this way. Here, the PEPT experimental method, which is a fully Lagrangian measurement technique, provides a unique opportunity to examine the capability of the SPH–DEM methodology to simulate the complex pipe flow of particle–liquid food suspensions.

In this paper, a coupled SPH–DEM scheme is used to investigate the behavior of coarse model food particles in a viscous non-Newtonian conveying fluid inside a horizontal pipe. The model is fully validated by PEPT-measured radial profiles of particle velocity and solid phase as well as particle passage time distribution and pressure drop, under varying conditions of solid loading. A detailed analysis of localized particle–liquid dynamics is obtained from the simulation results for different conditions of the two-phase flow. Such information is invaluable to inform the protocols of food particle transport and processing in pipelines.

II. EXPERIMENTAL

A. Pipe flow rig

Particle–liquid flow was driven by a vortex pump (T21–32 HF4 LB1, Turo vortex pump, EGGER, Switzerland) through a 10 m long flow loop of 40 mm internal diameter, schematically represented in Fig. 1. Food particles were represented by model nearly neutrally buoyant calcium alginate beads, fabricated in-house using the protocol reported by Fairhurst et al.,2 being nearly spherical with 4.0 mm diameter. The carrier fluid was a 0.80 wt. % aqueous carboxymethylcellulose (CMC) solution of non-Newtonian rheology used to mimic the usual behavior of industrial food fluids. The rheological properties of the carrier fluid were determined before each experiment and checked at the end using a rheometer (Discovery Hybrid Rheometer, HR-2, TA Instruments). The mixture was circulated via a large cone-bottom tank where particles were kept in suspension by a mechanical mixer. Flow was studied in a 4 m long horizontal section of the flow loop. Flow visualization by the PEPT technique was conducted over a pipe length of 0.40 m which was located 3 m downstream of the inlet and
0.6 m upstream of the outlet to ensure that the flow was fully developed and was not affected by pipe bends. The particle–liquid flow rate was measured by an ultrasonic Doppler flowmeter (UDF D5500, Doppler flowmeter, Micronics), and the mean particle delivery concentration in the pipe was measured with a bucket and stopwatch method. The mean delivery concentration of solids was in the range of 10–40 vol. %, with mean mixture velocities of 0.056–0.092 m s$^{-1}$. The experimental conditions are summarized in Table I.

### TABLE I. Experimental conditions of particle–liquid flows.

| Flow case | CMC (wt. %) | C$_l$ (vol. %) | $d_p$ (mm) | $u_{mean}$ (m s$^{-1}$) | $D$ (m) | Re = $\mu$Du$_{mean}$ | $\rho_f$/|$\rho_p$ |
|-----------|-----------|--------------|--------|-----------------|-------|----------------|---------|
| 1         | 0.80      | 10           | 4.0    | 0.056           | 0.040 | 11.2           | 1.01    |
| 2         | 0.80      | 20           | 4.0    | 0.072           | 0.040 | 14.4           | 1.01    |
| 3         | 0.80      | 30           | 4.0    | 0.083           | 0.040 | 16.6           | 1.01    |
| 4         | 0.80      | 40           | 4.0    | 0.092           | 0.040 | 18.4           | 1.01    |

#### B. Positron emission particle tracking

PEPT is a noninvasive flow measurement technique. In PEPT, a tiny tracer radiolabelled by a positron-emitting radionuclide (here $^{18}$F) is introduced in the flow. The tracer position is then determined by detecting a set of coincident gamma rays associated with the annihilation of emitted positrons with nearby electrons. PEPT can be used to visualize flow in opaque flow/apparatus, a unique advantage, and its accuracy is on a par with leading laser-based optical methods. Detailed information about PEPT technology and data processing algorithms has been reported in our earlier papers. In pipe flow, imaging by PEPT consists in letting the particle tracer flow in a closed loop until it maps the entire area of interest, therefore requiring a statistically representative number of Lagrangian trajectories, typically $>50$. In this study, since flow was viscous, a long time would have needed to collect enough trajectories in each experiment with a single particle tracer. Hence, several identical tracers (usually 7–8 tracers) were introduced in the flow loop and simultaneously tracked. In PEPT, both carrier fluid and solid particles can be separately tracked by using suitable radiolabelled tracers, but in this study only the solid phase was tracked using a tiny (100 $\mu$m) radiolabelled resin-particle tracer encapsulated within a representative alginate bead.

### III. PEPT DATA ANALYSIS

PEPT provides 3D Lagrangian trajectories which can be used to calculate particle velocity profiles, particle concentration distribution, and particle passage times. In the following sections, the procedures for analyzing PEPT data to infer these parameters are presented.

#### A. Radial particle velocity profile

PEPT provides data arrays of tracer positions as a function of time in the form of $f(x, y, z, t)$. The local velocity in each direction is estimated using the time derivatives of positions by a differencing approach. Since flow in the pipe is laminar, the principal velocity component is the axial component ($u_x$), and any radial fluctuations can be ignored. The local axial velocity at each point is estimated from the slope of a straight line fitted to a number of $x$-positions vs time using regression analysis. In this case, 20 points were used covering a small distance of 25 mm. The radial velocity profile was obtained by dividing the pipe cross section into 20 semi-annular regions of equal area, as shown in Fig. 2. As the solid particles used were slightly heavier than the carrier fluid ($\rho_s = 1.01$), 10 regions above and 10 regions below the centerline of the pipe were used to account for axial asymmetry of the flow. The radial velocity profile was constructed by normalizing with the mean mixture velocity, $u_{mean}$, and the mean and standard deviations were calculated in each region.

#### B. Radial particle concentration profile

We previously demonstrated a method by which the Lagrangian PEPT trajectories can be used to determine the spatial distribution of solids, utilizing the occupancy of the particle tracer. Traditionally, by dividing the flow domain into a grid, occupancy is defined as the fraction of the total experimental time ($t_{exp}$) a tracer spends in each grid cell during the experiment, which makes it dependent on the density of the grid used. To mitigate this bias, an ergodic time ($t_{erg}$) is defined, the time which a tracer would spend in a cell, given the system

\[
\sigma_i = \sqrt{\frac{\sum (u_{x,i} - u_{x})^2}{N}}
\]

\[
\bar{u}_{x,i}: \text{mean velocity in region } i \\
\sigma_i: \text{axial velocity of particle } j \\
N: \text{total number of particles in region } i \\
\sigma_i: \text{standard deviation of region } i
\]
is single-phase and ergodic. Since the tracer has equal probability of presence anywhere within the pipe flow, the ergodic time can be expressed as the total experimental time divided by the number of cells, i.e., $t_e = t_\text{exp}/N_e$. Thus, the local occupancy ($O_e$) can, therefore, be defined as

$$O_e = \frac{\Delta t}{t_e},$$

where $\Delta t$ is the time the tracer spends inside a given cell. Guida et al. demonstrated that the local occupancy is equal to the ratio of the local particle concentration, $c$, to the mean particle concentration in the vessel, $C$, as follows:

$$O_e = \frac{c}{C}.$$

Using these definitions and dividing the pipe into a 3D grid of equal volume cells, the local particle concentration was estimated and the local radial particle concentration profile constructed. An average radial concentration profile was calculated from the local concentration profiles over 40 equidistant cross sections along the length of the pipe.

IV. SPH-DEM SIMULATION

The non-Newtonian carrier fluid was modeled by the SPH technique while the solid particles were modeled by the distributed contact DEM (DCDEM) approach, a variant of DEM, which is able to simulate irregularly shaped solid particles, if necessary. DCDEM models each solid particle using a cluster of numerical particles, keeping the relative positions of these inner constituent particles constant.

A. SPH formulation of carrier fluid

1. Kernel function

SPH is a Lagrangian numerical framework that is based on numerical particle interpolation to compute smooth field variables. The numerical particles act as control masses and carry all physical properties of the system to be simulated in the computational domain. The governing equations at each SPH particle are solved in a weighted averaged formulation in an influenced domain $\Omega$, as shown in Fig. 3.

The influence domain and weighting of neighboring particles are defined by a kernel function $W$, whose derivative is used to calculate the gradient and divergence operators. The kernel is expressed as a function of the normalized distance between SPH particles ($q = |r_{ab}|/h$), where $r_{ab}$ is the distance between any two given numerical particles $a$ and $b$ and the parameter $h$ is the smoothing length (Fig. 3). The smoothing length controls the size of the area around the target numerical particle in which neighboring particles are considered. The choice of kernel function is not unique, but a quintic class 2 Wendland function is widely used as kernel function in SPH simulations,

$$W(r_{ab}, h) = \phi_d \begin{cases} \left(1 - \frac{q^2}{2}\right)^4 (2q + 1), & 0 \leq q \leq 2, \\ 0, & q > 2, \end{cases}$$

where $\phi_d = 7/4\pi h^2$ in 2D simulations and $\phi_d = 21/16\pi h^2$ in 3D simulations.

2. Continuity and momentum equations

The motion of the liquid phase is governed by continuity and momentum conservation equations. The Lagrangian form of the momentum conservation equation in a continuum is

$$\frac{d\rho_a}{dt} = -\frac{1}{\rho_a} \nabla P + \mathbf{g} + \Gamma + S_C,$$

where $\Gamma$ represents the dissipative terms; $P_a$ and $\rho_a$ are the pressure and density that correspond to a target particle $a$, and $P_b$ and $\rho_b$ are the pressure and density corresponding to a given neighboring particle $b$ (Fig. 3). In particle-liquid flow $S_C = f_p/m_p$, where $f_p$ represents the whole coupling force on the liquid particle, because of the DEM particles. The laminar viscous stresses $(\nu_0 \nabla^2 \mathbf{v})_a$ in the momentum conversation can be represented by

$$(\nu_0 \nabla^2 \mathbf{v})_a = \sum_b m_b \frac{4\nu_0 r_{ab} \cdot \nabla \mathbf{v}_b W_{ab}}{(\rho_a + \rho_b)(r_{ab}^2 + \eta_0^2)} \mathbf{r}_{ab},$$

where $\nu_0$ is the kinematic viscosity, $r_{ab} = r_a - r_b$ is the separation distance between numerical particles, and $\eta_0 = 0.01 h^2$. During the whole simulation process, the mass of each particle remains constant.

Mass continuity in the SPH form is given by

$$\frac{d\rho_a}{dt} = \rho_a \sum_b \frac{m_b}{\rho_b} \mathbf{v}_{ab} \cdot \nabla \mathbf{v}_b W_{ab}. $$

3. Equation of state

There are two common approaches for solving the momentum conservation equation in SPH, namely, incompressible SPH (ISPH) and weakly compressible SPH (WCSPH). In ISPH, the pressure term in the equation is computed by solving the pressure Poisson’s equation, while the pressure is computed explicitly from the density variations using an artificial equation of state in WCSPH.

$$P = B \left( \frac{\rho}{\rho_0} \right)^7 - 1, $
where $P$ is the pressure, $\gamma$ is the fluid polytropic index (here, $\gamma = 7$), $B = \frac{c_0^2 \rho_0}{\gamma}$, $\rho_0$ is the reference density, and $c_0$ refers to the speed of sound at the reference density. It should be noted that it is common practice in WCSPH not to actually use the physical speed of sound but sound at the reference density. It should be noted that it is common for the system. In WCSPH simulations, the density variation factor ($\|\rho/\rho_0 - 1\|$) must be less than 1% to keep the relative incompressibility of the fluid.21

### 4. Carrier fluid rheology

The non-Newtonian rheology of the carrier fluid was modeled using the Herschel–Bulkley–Papanastasiou (HBP) model,25 which includes an apparent yield stress. This is a general viscoplastic model which has the advantage of being able to represent various time-independent rheological behaviors including Newtonian, pseudoplastic, and dilatant with and without a yield stress, making it popular for use in numerical simulations. In this model, the effective viscosity ($\mu_{\text{eff}}$) is a function of shear rate ($\dot{\gamma}$) and yield stress ($\tau_y$), thus,

$$\mu_{\text{eff}} = \mu(\dot{\gamma})^{\eta - 1} + \frac{\tau_y}{\gamma} (1 - e^{-\eta \dot{\gamma}}),$$

(8)

where $\mu$ is the apparent dynamic viscosity. Details of shear rate calculations are widely discussed in the literature.49–51 Compared to the well-known Bingham model, two additional coefficients, the Papanastasiou parameter ($m$) and power-law index ($n$), are included in the HBP model. As $m$ tends to infinity and $n \neq 1$, the HBP model reduces to the original Herschel–Bulkley model. When $\tau_y \neq 0, m \neq 0$, and $n = 1$, the model reduces to the simple Bingham model. When $\tau_y = 0$, and $n \neq 1$, the model reduces to the power-law model and, consequently, when $\tau_y = 0$, and $n = 1$ it represents Newtonian behavior.

### B. DEM formulation of dispersed solid phase

In DEM, the suspended solid particles are treated as a discrete phase whose motion is described by Newton’s second law:

$$m_i \frac{d^2 r_i}{dt^2} = \sum_j f_{ij} + m_i g,$$

(9)

where $m_i$ represents the mass of particle $i$ at position $r_i$, $c_{ij}$ represents the contact force between solid particles, and $f_{ij}$ is the coupling force of the particle–liquid flow acting on solid particle $i$. In the coupled SPH-DEM approach, the force exerted on solid particle $i$ by the numerical SPH particles is estimated as22

$$f_{ij} = V_i (-\nabla P + \nabla \cdot \tau) + f_d(\epsilon_i, u_i),$$

(10)

where $V_i$ is the volume of the solid particle. The first two terms account for fluid forces, such as shear stress, pressure, and buoyancy. The force $f_d$ is the particle drag force which is affected by the local porosity $\epsilon_i$ and the superficial fluid velocity $u_i$ given by

$$u_i = \epsilon_i (u_{li} - u_{ip}),$$

(11)

where $u_{li}$ and $u_{ip}$ are, respectively, the velocity of the SPH fluid and DEM solid particles. The drag force is defined as22

$$f_d = \frac{1}{8} C_d \epsilon_i \rho_l \frac{\rho_j}{\rho_i} u_i \left| u_s \right|,$$

(12)

where $\rho_j$ is the fluid density. The drag coefficient $C_d$ and the constant $\varphi$ are related to the particle Reynolds number, $Re_p = \rho u_d \mu,$

$$C_d = \left( \frac{2.43}{Re_p} \right)^{1/2},$$

(13)

$$\varphi = 3.7 - 0.65 \times \exp \left[ \frac{- (1.5 - \log_{10} Re_p)^2}{2} \right].$$

The total contact force ($F_{ij}$) acting on particle $i$ resulting from collision with particle $j$ is resolved into normal ($F_{nij}$) and tangential ($F_{tij}$) components, as shown in Fig. 4. Both forces are further decomposed into a repulsion force ($F_r$) and a damping force ($F_d$) arising, respectively, from deformation of the material and dissipation of energy during the deformation.

The normal force is expressed as

$$F_{nij} = F_n + F_d = k_{nij} \delta_{ij}^{1/2} \epsilon_{ij}^{1/2} - \gamma_{nij} \delta_{ij}^{1/2} \delta_{ij}^{1/2} e_{ij},$$

(15)

where the stiffness is given by

$$k_{nij} = -\frac{4}{3} E^* \sqrt{R^*},$$

(16)

and the damping coefficient is

$$\gamma_{nij} = -\frac{\log c_d}{\pi^2 + \log^2 c_y},$$

(17)

where $\delta_{ij} = \max(0, (d_i + d_j)/2 - |r_{ij}|)$ is the particle overlap (approximating deformation), $\epsilon_{ij}^{1/2}$ is the unit vector between the centers of particles $i$ and $j$, and $\delta_{ij} = \nu_{ij} \cdot \nu_{ij}$ is the rate of normal deformation. The parameter $c_{dy}$ is the restitution coefficient defined as the ratio of the velocities after and before particle collision, and its value controls viscous dissipation during collision. The reduced radius $R^*$ and reduced elasticity $E^*$ are defined as

$$R^* = \left( \frac{1}{r_i + 1/r_j} \right)^{-1}; \quad E^* = \left( \frac{1 - V_p}{E_p} + \frac{1 - V_p}{E_p} \right)^{-1},$$

(18)
where $r$ is the particle radius, $E$ is the Young's modulus, and $v_p$ is the Poisson's ratio.

Similarly, the tangential force is also decomposed into a repulsion $F_r$ and a damping force $F_d$, thus,

$$F_{t,ij} = F_r^{ij} + F_d^{ij} = k_{t,ij} \delta \hat{e}_y^{ij} - \gamma_{t,ij} \delta \hat{\theta}_{y}^{ij},$$

where the stiffness and damping constants for tangential force are

$$k_{t,ij} = \frac{2}{7} k_{n,ij}, \quad \gamma_{t,ij} = \frac{2}{7} \gamma_{n,ij}.$$ (20)

### C. SPH–DEM coupling

The SPH–DEM coupling process is illustrated in the flow chart of Fig. 5. The governing equations for the liquid and solid phases are solved for each numerical particle. At the pre-processing stage, the physical parameters, including the rheological parameters of the carrier fluid, mean mixture velocity, and solid particle positions, are fed into the framework. The number of solid particles is estimated based on the solid loading and their positions are randomly assigned. The pipe containing the particle–liquid flow is constrained by periodic boundaries at the inlet and outlet. Three types of simulation particles are generated within the simulation domain: fluid, solid, and boundary particles. During the course of the simulation, each fluid and solid particle is independently tracked.

The main numerical calculations are implemented in the processing stage, where the data for each simulation particle are read and a list of neighboring particles is generated by searching the particle's surrounding. Then, the model variables at each numerical particle are estimated based on the weightings assigned to the neighboring particles.
particles by the kernel function [Eq. (3); Fig. 3]. Once the quantities for each simulation particle are computed in the SPH module within the time step \( t_{SPH} \), the linear and angular accelerations of each solid particle are computed based on the fluid–solid interactions and are fed into the DEM module. A collision condition is checked by the framework, and if a collision happens with other solid particles, DEM calculates the after-collision linear and angular velocities and updates the positions of the particles. In the case of no collision, the density, position, and velocity of the numerical solid particles are updated by the SPH module. At each time step, all physical information pertaining to the simulation particles is saved and carried forward to the next time step. The modeling parameters used in the SPH–DEM simulations are summarized in Table II; the parameters of the solid phase, i.e., alginate beads, were obtained from Chan et al.\textsuperscript{55}

D. Simulation set-up

1. Geometry and total simulation time

The simulations were conducted in 2D using DualSPHysics software. Flow in a pipe is essentially unidimensional and, therefore, 2D simulations are adequate for its description. There are a number of works in the literature which have used similar 2D simulations to study various types of flows.\textsuperscript{56–58} In 2D, solid particles are circular rather than spherical and the pipe wall is reduced to two lines on the cylindrical surface. The pipe geometry was identical to that of the experimental pipe, i.e., circular with a 40 cm diameter, as shown in Fig. 6. Simulations were conducted over a pipe length of 0.07 m. The effects of particle spacing, \( s_p \), for each solid concentration were studied via a sensitivity analysis and the optimum values obtained were 4, 3, and 2 \( \times 10^{-5} \) m for 10, 20, 30, and 40 vol. % mean solid loadings, respectively. The total simulation time was 15 s to ensure that the particle-liquid flow was fully developed.

2. Boundary conditions

Using periodic boundary conditions at the inlet and outlet of the pipe can substantially reduce the computational effort.\textsuperscript{59} To enhance computational accuracy near open boundaries, the support domain of the kernel function of a particle is clipped by the nearest open boundary and the remainder of its clipped support applied at the complementary open boundary.\textsuperscript{59}

The wall boundary is also described by numerical boundary particles (Fig. 6). Twenty layers of such particles were used. The dynamic boundary condition (DBC) method assumes that the boundary particles satisfy the same equations as the fluid particles, but they do not move in response to the forces exerted on them.\textsuperscript{60} The stability of the DBC method depends on the time step used in the numerical calculations, which should be suitability short to deal with the highest velocity expected for any fluid particle interacting with the boundary particles.

3. Simulation time step

To ensure the robustness of the SPH–DEM simulations, the Courant–Friedrichs–Lewy (CFL) condition needs to be satisfied.\textsuperscript{3} There are viscous diffusion and forcing terms in the CFL condition. The simulation time step, \( \Delta t_{SPH} \), is estimated as

\[
\Delta t = \min\left(\frac{h}{C_1|\nabla f|}ight)
\]

\[
\Delta t_v = \min\left(\frac{h}{c_n + \max_n \left|\frac{\mathbf{r}_a \cdot \mathbf{r}_b}{r_{ab}^2 + \eta}\right|}ight)
\]

\[
\Delta t_{SPH} = C_{CFL} \cdot \min\left(\Delta t_i, \Delta t_v\right)
\]

where \( h^2 = 0.01 h^2 \), and \( \Delta t_i \) is influenced by the force per unit mass \((|f|)/M)\); \( \Delta t_v \) combines the Courant and the viscous time step controls. As the SPH calculations are coupled with the DEM module, another restriction should be considered in optimizing the time step

\[
\Delta t_{ij} = \frac{3.21}{50} \left(\frac{M^*}{\kappa_{n,ij}}\right)^{2/5} v_{n,ij}^{1/5}
\]

where \( v_n \) represents the normal relative velocity and \( M^* \) denotes the mass reduction of the whole system.

**TABLE II.** Modeling parameters used in SPH-DEM simulations.

<table>
<thead>
<tr>
<th>Solid phase</th>
<th>1014</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg m(^{-3}))</td>
<td>400</td>
</tr>
<tr>
<td>Young’s modulus (kPa)</td>
<td>0.5</td>
</tr>
<tr>
<td>Passion ratio (-)</td>
<td>0.75</td>
</tr>
<tr>
<td>Friction coefficient (-)</td>
<td>0.4</td>
</tr>
<tr>
<td>Numerical particle spacing, ( s_p ) (m)</td>
<td>0.000 02–0.000 04</td>
</tr>
<tr>
<td>Overlap distance (m)</td>
<td>0.5 ( s_p )</td>
</tr>
<tr>
<td>Liquid phase</td>
<td></td>
</tr>
<tr>
<td>Density (kg m(^{-3}))</td>
<td>1000</td>
</tr>
<tr>
<td>Papanastasiou parameter, ( m ) (-)</td>
<td>10</td>
</tr>
<tr>
<td>Power-law index, ( n ) (-)</td>
<td>0.89</td>
</tr>
<tr>
<td>Numerical particle spacing, ( s_p ) (m)</td>
<td>0.000 02–0.000 04</td>
</tr>
<tr>
<td>Kernel function</td>
<td>Quintic class 2 Wendland</td>
</tr>
<tr>
<td>Kinematic viscosity (Pa s)</td>
<td>0.1</td>
</tr>
<tr>
<td>Kernel smooth length, ( h ) (m)</td>
<td>( h = 0.1 \sqrt{3 s_p} )</td>
</tr>
<tr>
<td>Time step, ( \Delta t_{SPH} ) (s)</td>
<td>0.000 04</td>
</tr>
</tbody>
</table>

**FIG. 6.** Schematic diagram of SPH-DEM setup for simulation of particle–liquid flow. Note how particle (A) moves through the periodic boundaries of the pipe.

**Phys. Fluids** 35, 043325 (2023); doi: 10.1063/5.0144992

© Author(s) 2023
V. RESULTS AND DISCUSSION
A. Validation of simulations
1. Single-phase flow of fluids of different rheology

The first step in the validation process evaluated the capability of the SPH module in the framework to simulate the single-phase flow of fluids of different rheology. The simulations were performed at a mean fluid velocity of 0.075 m s\(^{-1}\), typical of the experimental mean particle–liquid mixture velocities used. The SPH-predicted velocity profiles are compared with those obtained from analytical solutions for the various rheologies considered in Figs. 7 and 8. By appropriately selecting the values of the parameters \(\tau_y, m,\) and \(n\) in the HBP rheological model [Eq. (8)], different rheological fluids were implemented. Thus, for a Newtonian fluid (\(\tau_y = 0\) and \(n = 1\)), the numerical and theoretical profiles are identical [Fig. 7(a)]. For power-law rheology, the simulation results are presented in Fig. 7(b) for a wide range of cases (e.g., \(\tau_y = 0\) and \(n = 0.5, 0.8, 1.3, \) and \(1.7\)), showing an excellent agreement with the theoretical solutions. Simulations of typical Bingham fluids (e.g., \(n = 1\) and \(m = 100\)) with different yield stress values (\(\tau_y = 1, 3,\) and \(5\) Pa) are depicted in Fig. 7(c), again exhibiting a high degree of accuracy.

The rheological behavior of the CMC solution used in the particle–liquid flow experiments was also represented by the HBP model, where, \(\tau_y = 6.4\) Pa, \(n = 0.89,\) and \(m = 10\). The SPH-predicted velocity profile shown in Fig. 8 coincides very well with the analytical solution provided by the well-known Herschel–Bulkley equation. In conclusion, SPH is able to predict the pipe flow of non-Newtonian fluids with a high degree of accuracy.

2. Particle–liquid flow

The SPH–DEM simulations provide the Lagrangian trajectories for both the fluid and solid particles in the flow. Typical trajectories are depicted in Fig. 9 illustrating the different behavior of the fluid and particles in different regions of the flow. The SPH–DEM simulations were validated by comparing in each case the predicted local radial particle velocity profile, radial solid phase distribution and particle passage time distributions with those estimated from the PEPT experimental data, as shown, respectively, in Figs. 10–12. The simulation results were treated using the same procedure described above for analyzing the PEPT data to construct radial profiles of local particle velocity and solid volume fraction, and particle passage times.
1. Radial particle velocity profile

The SPH–DEM predicted radial particle velocity profiles are compared with PEPT determined profiles in Fig. 10 at different solid loadings from 10 to 40 vol.%. The local velocity error bars for both simulations and experimental data are too small to be shown. There is a generally good agreement between the SPH–DEM predictions and PEPT measurements at all solid concentrations. The results show the capability of the SPH–DEM coupling to predict particle velocities in a...
complex two-phase flow consisting of a non-Newtonian carrier fluid with high solid loadings. The plots exhibit a minor asymmetry in the velocity profiles as the maximum appears slightly above the centerline. As the density of the particles is slightly higher than that of the carrier fluid, particle settling effects would be expected to distort the particle velocity profile causing particles to travel faster in the top part of the pipe.

2. Radial solid phase distribution

The SPH–DEM predicted and PEPT determined radial solid phase distributions corresponding to the same particle size and concentrations are compared in Fig. 11. The local particle concentration in the flow was inferred from the Lagrangian particle trajectories by the method described above in Sec. IIB, at 20 radial positions over 40 equidistant cross sections along the length of the pipe and an average obtained along each radial position. The radial particle concentration profiles, thus, constructed are shown in Fig. 11. The error bars indicate the axial standard deviation of the local particle concentration at each radial position along the pipe, reflecting the complex dynamic fluctuating nature of the solid phase distribution. Such variations tend to reduce as the mean particle concentration increases. Comparison of the SPH–DEM predicted and PEPT determined solid phase profiles shows a good agreement, confirming again the capability of the coupled SPH–DEM methodology to deal with such complex flows.

The effects of particle sedimentation, even though small, are apparent in the distorted profiles corresponding to $C_s = 10$ and 20 vol. %, showing a maximum in the bottom region of the pipe cross section. Particles become more uniformly distributed across the pipe at higher solid loadings, exhibiting a nearly flat profile at $C_s = 40$ vol. % as the pipe cross-section fills up with particles (approaching the packing fraction) and inter-particle interactions increase.

3. Particle passage time distribution

As both SPH–DEM and PEPT provide Lagrangian trajectories of solid particles in the flow, they allow extraction of particle passage times, a very important parameter in pipeline transport of particle-liquid flows, e.g., in continuous inline processing of particulate food mixtures. Each trajectory within the experimental data set was used to extract the particle passage time over a given length of pipe, which was then normalized by the mean mixture passage time estimated from the mean mixture velocity (Table 1). The distributions obtained by both methods are presented in Fig. 12 at the different particle concentrations investigated. Results show a good agreement between the numerical predictions and experimental measurements.
4. Pressure-drop in particle–liquid flow

Prediction of pressure drop is important in the design and operation of pumped flow systems. Some empirical and semi-empirical correlations have been proposed for estimating pressure drop in particle–liquid pipe flows. Two empirical correlations whose experimental conditions are remarkably close to the flow cases studied here are used to verify the pressure drop estimated from the SPH–DEM simulations. When the difference in density between particles and liquid is small ($\leq 1\%$) suspensions can usually be treated as a pseudo-homogeneous fluid whose density is equal to the mean density of the mixture, i.e., the two-phase flow is approximated by a single-phase flow. Gradeck et al. developed a correlation to estimate the pressure drop per unit length $D_P/L$ in particle–liquid flow consisting of coarse alginate particles (4.4 mm) in a CMC solution, thus,

$$C_f = \frac{1}{2} \frac{d_p}{\rho_{\text{sus}} u_{\text{mean}}} \frac{\Delta P}{L}, \quad (23)$$

where $C_f$ is the fanning friction factor, $\rho_{\text{sus}}$ is the mean suspension density. In laminar flow, the friction factor is given by

$$C_f = \frac{16}{Re}, \quad (24)$$

where $Re$ represents the Reynolds number of the (pseudo-homogeneous) suspension.

Earlier Rasteiro et al. had developed a semi-theoretical solution to predict the pressure drop in particle–liquid flow based on the kinetic energy loss, $J_k$, the viscous energy loss, $J_v$, and the energy loss from particle–particle interactions, $J_p$, thus,

$$\frac{\Delta P}{L}_{\text{sus}} = \theta_1 J_k + \theta_2 J_v + \theta_3 J_p, \quad (25)$$

$$J_k = \frac{\rho_{\text{pp}} u^2}{d(1 - C_s)}; \quad (26)$$

$$J_v = \frac{\mu_{\text{pp}} u^2}{d^2(1 - C_s)^2}; \quad (27)$$
these complex flows even under conditions of high concentration. While there is a fair agreement with Rasteiro's empirical correlation, there is a very good agreement with Gradeck et al.'s empirical correlation over the particle concentration range 10–30 vol. %. From 30 to 40 vol. %, the two results seem to gradually diverge reaching a maximum difference of about 15% at 40 vol. %; such a discrepancy may be attributed to the fact that the correlation is based mostly on data in the concentration range of 10–20 vol. %. Overall, the SPH–DEM predictions fall within the range of estimations provided by these two correlations, which is further evidence of the accuracy and reliability of the coupled method to simulate these complex flows even under conditions of high concentration which are typical of industrial food processing.3

B. Localized dynamics of particle–liquid flow

The Lagrangian particle-based nature of the SPH–DEM simulations provides important information on the local hydrodynamic properties of both particles and carrier fluid. One of these properties which can be estimated is the local particle–liquid slip velocity, \( u_{slip} = u_p - u_l \). While such a slip velocity can have significant values in dense flows, because the particles here are nearly neutrally buoyant, the values estimated from both experimental and simulated trajectories were, as expected, vanishingly small.

1. Particle angular velocity or spin

One of these properties which can be estimated is particle angular (or spin) velocity, \( \omega \), which is defined as

\[
\omega = \frac{d\theta}{dt},
\]

where \( \theta \) is the angular displacement. In a number of applications including food processing, flow is often accompanied by heating or cooling. Similarly, in chemical reactors mass transfer occurs frequently. The heat/mass transfer coefficient between particle and liquid is critical and directly affected by the local relative (slip) velocity between particle and fluid. The rotational (spin) motion of particles enhances slip and is, thus, also significant in defining heat/mass transfer. It also produces a lift force which affects particle trajectories and, hence, the distribution of particles and flow field.6,5 Thus, to understand particle–liquid flow, both rotational and translational behaviors of particles need to be known. Rigorous studies of particle spin motion are virtually non-existent.

The local particle angular velocities simulated at different mean solid concentrations are presented in Fig. 14(a), while the corresponding averaged radial profiles are plotted in Fig. 14(b). The spin velocities are positive (anticlockwise) in the top half of the pipe cross section and negative (clockwise) in the bottom half. Particle spin is fastest closest to the wall where the liquid velocity gradients are highest, reducing to zero at the center of the pipe before switching direction. At the solid loading of 40 vol. %, spin becomes negligible everywhere except close to the wall. As the solid concentration approaches the particle packing fraction, velocity gradients in the core section dramatically weaken and inter-particle interactions increase such that particle spin is prevented.

2. Flow pressure field

Another important parameter which can be extracted from the Lagrangian simulations is the flow pressure field. A typical contour plot is shown in Fig. 15(a). Enlarged views of the pressure field and isobars surrounding particles located in the top (A) and bottom (B) parts of the pipe are presented in Figs. 15(a) and 15(b). A close inspection of the isobars around particle A shows that the pressure \( P_1 \) to the right of the particle is slightly higher than the pressure \( P_2 \) to the left. This means that the liquid around particle A exerts a retarding form drag force on particle A which causes the particle to lag behind the liquid. On the other hand, the pressure \( P_1 \) to the right of particle B is slightly lower than the pressure \( P_2 \) to left. Thus, particle B is subject to an accelerating form drag force causing it to lead the liquid. Such detailed information is valuable for analyzing the local behavior of individual particles as well as particle clusters.

3. Fluid vorticity

Vorticity is another parameter of fundamental importance which can be extracted from the simulations and is a measure of the rotation or degree of deformation of a fluid and is defined as the curl of the velocity. A typical simulated liquid vorticity contour map is depicted in Fig. 16. Vorticity values are highest near the pipe walls showing a high degree of fluid deformation, falling to zero in the central region. Vorticity is positive in the top part of the pipe cross section and negative in the bottom part, confirming the reason for the anticlockwise and clockwise spin of particles in the top and bottom regions, respectively, as discussed above. A close inspection of the vorticity field around a single particle and around two adjacent particles is shown in Figs. 16(a) and 16(b). The fluid around a single particle exhibits a symmetrical four-petal shape of vorticity which is deformed when two particles approach each other squeezing the liquid film in between.
VI. CONCLUSION

A Lagrangian–Lagrangian particle-based SPH–DEM numerical approach was applied to simulate the horizontal laminar pipe flow of coarse particle–liquid food suspensions. SPH–DEM was able to predict the single-phase flow of various non-Newtonian fluid rheologies including the carrier fluid used to convey food particles with a high degree of accuracy, as validated by analytical solutions of radial velocity profiles. The particle–liquid flow simulations were also successfully
FIG. 15. Typical SPH–DEM predicted pressure field: (a) pipe flow on 1:1 scale; (b) enlarged pressure field around particle A; and (c) enlarged pressure field around particle B; $C_s = 10$ vol.%. $P_1$, $P_2$, $P_3$, and $P_4$ indicate local fluid pressure.

FIG. 16. Typical SPH–DEM predicted liquid vorticity field: (a) pipe flow on 1:1 scale; (b) enlarged liquid vorticity field around particle A; and (c) enlarged liquid vorticity field around particles C and D; $C_s = 10$ vol.%.
validated using experimental measurements obtained by a technique of positron emission particle tracking. The radial particle velocity profiles, radial solid phase distributions, as well as particle passage time distributions were accurately predicted at solid loadings varying from 10 to 40 vol.%. Radial particle velocity profiles are asymmetric, with the maximum being located above the centerline. Such asymmetry disappears at high solid concentrations, and the maximum moves to the center.

Simulations yielded detailed information on the local dynamics of the two-phase flows investigated, including translational as well as rotational particle slip velocities, flow pressure field, and fluid vorticity. While local translational particle slip velocities, as expected for nearly neutrally buoyant particles, were vanishingly small, particle angular (slip) velocities were significant. Spin is positive (anticlockwise) in the top half of the pipe cross section and negative (clockwise) in the bottom half. Particle spin is fastest closest to the wall where liquid velocity gradients are highest, reducing to zero at the center of the pipe before switching direction. As solid loading increases, approaching the particle packing fraction, spin becomes negligible everywhere except close to the wall.

The pressure drop predicted by SPH–DEM at different particle concentrations showed a very good agreement with empirical correlations from the literature, which is further evidence of the accuracy and reliability of the coupled method to simulate these complex flows even under conditions of high concentration which are typical of industrial food processing. With further development, the framework has potential for simulating flows involving particles with more complex physical properties and fluids with more complex rheologies.

ACKNOWLEDGMENTS

This work was supported by EPSRC Programme (Grant No. EP/R045046/1); Probing Multiscale Complex Multiphase Flows with Positrons for Engineering and Biomedical Applications (PI: Professor M. Barigou, University of Birmingham). Xue Lian’s Ph.D. was funded by the University of Birmingham and China Scholarship Council (CSC).

AUTHOR DECLARATIONS

Conflict of Interest

The authors have no conflicts to disclose.

Author Contributions

Xue Lian: Conceptualization (equal); Data curation (equal); Formal analysis (lead); Investigation (lead); Methodology (equal); Validation (equal); Visualization (lead); Writing – original draft (equal); Writing – review & editing (equal). Chiya Savari: Conceptualization (supporting); Data curation (supporting); Formal analysis (supporting); Methodology (supporting); Visualization (supporting); Writing – original draft (supporting); Writing – review & editing (equal). Kun Li: Conceptualization (supporting); Data curation (supporting); Formal analysis (supporting); Methodology (supporting); Visualization (supporting); Writing – original draft (supporting); Writing – review & editing (equal). Mostafa Barigou: Conceptualization (lead); Funding acquisition (lead); Investigation (equal); Methodology (lead); Project administration (lead); Resources (lead); Supervision (lead); Validation (lead); Writing – review & editing (equal).

DATA AVAILABILITY

The data that support the findings of this study are available within the article.

NOMENCLATURE

Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>Local solid volume concentration (–)</td>
</tr>
<tr>
<td>C_v</td>
<td>Mean solid volume concentration (–)</td>
</tr>
<tr>
<td>c_n</td>
<td>Normalized solid phase concentration (–)</td>
</tr>
<tr>
<td>D</td>
<td>Pipe diameter (m)</td>
</tr>
<tr>
<td>d_p</td>
<td>Particle diameter (m)</td>
</tr>
<tr>
<td>E</td>
<td>Young modulus (Pa)</td>
</tr>
<tr>
<td>h</td>
<td>Kernel smooth length (–)</td>
</tr>
<tr>
<td>L</td>
<td>Pipe length (m)</td>
</tr>
<tr>
<td>m</td>
<td>Papanastasiou parameter (–)</td>
</tr>
<tr>
<td>n</td>
<td>Power law index (–)</td>
</tr>
<tr>
<td>O_E</td>
<td>Local occupancy (–)</td>
</tr>
<tr>
<td>P</td>
<td>Pressure (Pa)</td>
</tr>
<tr>
<td>P_t_n</td>
<td>Normalized particle passage time (–)</td>
</tr>
<tr>
<td>r</td>
<td>Radial position (m)</td>
</tr>
<tr>
<td>R</td>
<td>Pipe radius (m)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number (–)</td>
</tr>
<tr>
<td>s_p</td>
<td>Numerical particle spacing (m)</td>
</tr>
<tr>
<td>u_L</td>
<td>Liquid velocity (m s^{-1})</td>
</tr>
<tr>
<td>u_{mean}</td>
<td>Mean mixture velocity (m s^{-1})</td>
</tr>
<tr>
<td>u_p</td>
<td>Solid particle velocity (m s^{-1})</td>
</tr>
<tr>
<td>u_s</td>
<td>Superficial velocity (m s^{-1})</td>
</tr>
<tr>
<td>u_{slip}</td>
<td>Slip velocity (m s^{-1})</td>
</tr>
<tr>
<td>\Delta t_{SPET}</td>
<td>Time step of simulation (ms)</td>
</tr>
<tr>
<td>\Delta t</td>
<td>Tracer spent time in each cell (s)</td>
</tr>
<tr>
<td>\Delta P</td>
<td>Pressure difference (Pa)</td>
</tr>
<tr>
<td>t_{\infty}</td>
<td>PEPT experiment runtime (s)</td>
</tr>
<tr>
<td>t_E</td>
<td>Ergodic time (s)</td>
</tr>
<tr>
<td>W</td>
<td>Kernel function</td>
</tr>
</tbody>
</table>

Greek Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>\omega</td>
<td>Angular velocity (rad s^{-1})</td>
</tr>
<tr>
<td>\theta</td>
<td>Angular displacement (rad)</td>
</tr>
<tr>
<td>\mu</td>
<td>Apparent viscosity (Pa s)</td>
</tr>
<tr>
<td>\mu_{eff}</td>
<td>Effective viscosity (Pa s)</td>
</tr>
<tr>
<td>\sigma</td>
<td>Standard deviation (–)</td>
</tr>
<tr>
<td>\rho_f</td>
<td>Fluid density (kg m^{-3})</td>
</tr>
<tr>
<td>\rho_p</td>
<td>Particle density (kg m^{-3})</td>
</tr>
<tr>
<td>\rho_p</td>
<td>Particle to fluid density ratio (–)</td>
</tr>
<tr>
<td>\nu_p</td>
<td>Poisson coefficient of particle (–)</td>
</tr>
<tr>
<td>\tau_e</td>
<td>Yield stress (Pa)</td>
</tr>
</tbody>
</table>

Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant–Friedrichs–Lewy</td>
</tr>
<tr>
<td>CGMD</td>
<td>Coarse-grained molecular dynamics</td>
</tr>
<tr>
<td>CMC</td>
<td>Carboxymethylcellulose</td>
</tr>
<tr>
<td>DCDEM</td>
<td>Distributed contact discrete element model</td>
</tr>
</tbody>
</table>
WCSPH Weakly compressible smoothed particle hydrodynamics

REFERENCES


M. J. Robinson, Turbulence and Viscous Mixing Using Smoothed Particle Hydrodynamics (Monash University, 2009).


