Recent Progress on Mesh-free Particle Methods for Simulations of Multi-phase Flows: A Review†

Mikio Sakai1*, Yuki Mori2, Xiaosong Sun1 and Kazuya Takabatake2

1 Resilience Engineering Research Center, School of Engineering, The University of Tokyo, Japan
2 Department of Nuclear Engineering and Management, School of Engineering, The University of Tokyo, Japan

Abstract

The discrete element method (DEM) and the moving particle semi-implicit (MPS) method are the most popular mesh-free particle methods in the discontinuum and continuum. This paper describes a state-of-the-art modeling on multi-phase flows using these mesh-free particle methods. Herein, a combinational model of the signed distance function (SDF) and immersed boundary method (IBM) is introduced for an arbitrary-shaped wall boundary in the DEM simulation. Practically, this model uses a simple operation to create the wall boundary. Although the SDF is a scalar field for the wall boundary of the DEM, it is useful for the wall boundary of the CFD through combination with the IBM. Validation tests are carried out to demonstrate the adequacy of the SDF/IBM wall boundary model. Regarding the mesh-free particle method for continuum, the phase change problem is one of the challenging topics, as the solid state is usually modeled by extremely high viscous fluid in the phase change simulation. The phase change simulation is shown to be efficiently performed through an implicit algorithm and a heat flux model in the MPS method. The adequacy of these models is verified by the numerical examples.

Keywords: mesh-free particle method, discrete element method, DEM-CFD method, signed distance functions, immersed boundary method, moving particle semi-implicit method

1. Introduction

The discrete element method (DEM) (Cundall and Strack, 1979), the smoothed particle hydrodynamics (SPH) (Monaghan, 1994) and the moving particle semi-implicit (MPS) method (Koshizuka and Oka, 1996) are well known as a mesh-free particle methods for simulations of multi-phase flows. Although they require different targets, these methods are capable of easily simulating the violent motion of the media. The DEM, for instance, has been employed for simulating discontinuum as in the case of granular media and is known to inevitably introduce inter-particle force. Consequently, the DEM can introduce cohesive forces, such as van der Waals force (Sakai et al., 2012) and liquid bridge force (Dhenge et al., 2013; Tsunazawa et al., 2016; Chan and Washino, 2018; Schmelzle and Nirschl, 2018; Chan and Washino, 2018; Sun and Sakai, 2018; Sakai et al., 2019). Very recently, modeling of the particle shape has been extensively studied in the DEM (Zhou et al., 2011; Govender et al., 2019; Kafashan et al., 2019; Nie et al., 2019; Shrestha et al., 2019; Yang et al., 2019). In contrast, the continuum media have been simulated using the SPH and the MPS methods, which have been consistently employed in the computation of free surface fluid flows, such as droplet impingement (Xiong et al., 2010, 2011), free surface fluid flow through porous media (Sun et al., 2019), discharge of highly viscous fluid flow (Sun et al., 2012), and wave migration (Shibata et al., 2011). Hence, the mesh-free particle methods make it possible to simulate complex multi-phase flow systems. From the background, this article highlights a discussion on the recent progress of these mesh-free particle methods from the numerical simulation viewpoint of multi-phase flows.

Regarding the multi-phase flows involving solid particles, the DEM–CFD method (Tsuji et al., 1993) has been widely employed, e.g., a fluidized bed (Sakai et al., 2010, 2014; Yue et al., 2019), a pneumatic conveyor (Klinzing, 2018), and a particle filtration technology (Kanaoka, 2019). With this method, the local volume average technique (Anderson and Jackson, 1967) has been introduced into the governing equations for the fluid phase. Moreover, staggered grids are usually employed in the fluid flow calculation, where the grid size is sufficiently larger.

† Received 24 June 2019; Accepted 18 August 2019
J-STAGE Advance published online 12 October 2019
1,2 7-3-1 Hongo, Bunkyo-ku, Tokyo 113-8656, Japan
* Corresponding author; Mikio Sakai;
E-mail: mikio_sakai@n.t.u-tokyo.ac.jp
TEL: +81-3-5842-6977  FAX: +81-3-5842-6977

Copyright © 2019 The Authors. Published by Hosokawa Powder Technology Foundation. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/).
Phase change is one of the significantly important and challenging problems in numerical modeling of particle technology, e.g., additive manufacturing (Steuben et al., 2016; Russell et al., 2018) and thermo-mechanical behavior of powder (Krok et al., 2016). The MPS method can simulate phase change at ease, and hence has been applied to various phase change problems (Chen et al., 2014; Duan et al., 2019). In the modeling, enthalpy has been introduced briefly, and then the combined SDF–IBM method has been proposed to solve this problem. In this model, the normal component of the contact force is derived from the elastic potential energy; as such, this model enables the particle-wall interaction to retain the energy conservation in a non-dissipative system.

The SDF, which is denoted by $\phi$, is given by

$$\phi(x) = s(x)d(x),$$

where $d$ and $s$ are, respectively, the distance from the nearest wall boundary, and the sign that indicates the inside or outside of the calculation domain. The normal vector in the particle-wall interaction $\mathbf{n}_{pw}$ and normal compact of overlap $\delta_{pwu}$ are given by

$$\mathbf{n}_{pw} = \frac{\nabla \phi}{|\nabla \phi|}$$

and

$$\delta_{pwu} = (\phi - r_p)\mathbf{n}_{pw},$$

where $r_p$ is the particle radius. Finally, in the new SDF, the normal component of the contact force between the particle and the wall is given by

$$F_{cn} = (-k\delta_{pwu} |\nabla \phi| - \eta v_n).$$

The tangential component of the contact force is obtained.
in the same way as the traditional DEM as

\[
F_{c} = \left\{ \begin{array}{ll}
-k \delta_{	ext{ps}} - \eta v_{n} & |F_{c}| < -\mu |F_{c}|
\\
-\mu \left[ \frac{F_{c}}{|F_{c}|} \right] v_{n} & |F_{c}| \geq -\mu |F_{c}|
\end{array} \right.
\]  

(5)

where \(v_{n}\) and \(\mu\) are the tangential components of relative velocity of the interacting particles and friction coefficient, respectively. Note that \(\phi\) is not necessary in the tangential component of the elastic force.

A number of validation tests have proven the adequacy of the SDF model, specifically, for a ribbon mixer (Basinskas and Sakai, 2016b), a pot blender (Basinskas and Sakai, 2016a), a twin-screw kneader (Sakai et al., 2015), and powder die-filling (Tsunazawa et al., 2015).

2.2 Wall boundary model for the CFD

Some CFD techniques have been proposed for modeling arbitrary-shaped wall boundaries, one of which adopts an unstructured grid (Adam et al., 2016; R. Hu et al., 2019), where meshes are generated along the boundary. Although this technique is well established, it poses a problem on the mesh size, namely, the mesh size becomes extremely small when highly complicated boundaries are modeled. In addition, treatment of a moving wall is extremely difficult.

The IBM can be employed to overcome these problems. With the IBM, the boundary profiles are projected to the non-conforming computational grids. As such, Cartesian grids can be used even with the existence of complicated walls. The IBM eliminates the necessity of mesh generation along the boundary. Thus, the IBM is more efficient than boundary-fitted meshes. Fig. 1 shows schematically the boundary-fitted coordination system and the IBM.

Incidentally, the IBM falls under two categories (Lu et al., 2016). One is the so-called “continuous forcing” (Peskin, 2002) approach, in which control points are distributed over the boundary surface to modify the velocity field. The other approach is called “direct forcing” (Kajishima et al., 2001; X. Sun and Sakai, 2016; Udono and Sakai, 2017), in which the boundary profile is projected directly to the fluid grid. Moreover, transfer of wall information to the computational grid plays a dominant role, for which an efficient method of projection is required.

The numerical technique combining the IBM with the SDF geometrical representation has been proposed (Xiaosong Sun and Sakai, 2016), with an objective of obtaining the local boundary profile easily. In the multiphase flows, the SDF provides a unified description of wall boundary for both the solid and fluid phases, defining superior computational efficiency. The numerical algorithm is shown briefly as follows.

Governing equations for the fluid phase include the continuity and Navier–Stokes equations, where the local volume average technique is introduced. These equations are expressed as follows:

\[
\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon \mathbf{u}) = 0,
\]  

(6)

and

\[
\frac{\partial (\varepsilon \rho \mathbf{u})}{\partial t} + \nabla \cdot (\varepsilon \rho \mathbf{u} \mathbf{u}) = -\varepsilon \mathbf{F} + \mathbf{f} + \nabla \cdot (\varepsilon \mathbf{f}) + \varepsilon \rho \mathbf{g},
\]  

(7)

where \(\varepsilon, \mathbf{u}, \rho, \mathbf{f}, \mathbf{F}, \tau, \) and \(\mathbf{g}\) are, respectively, the void fraction, fluid velocity, fluid density, fluid pressure, solid–fluid momentum exchange term, viscous stress, and gravitational acceleration. Combined Ergun (Ergun, 1952) and Wen–Yu (Wen and Yu, 1966) equation is often used in the evaluation of the drag force acting on a computational particle.

The IBM can simulate the solid–fluid interaction and even utilizes structured grids, for which the object size should be sufficiently larger than the mesh size. With the IBM, the volume-weighted average velocity is used to analyze the interaction between the fluid and the solid object. The volume-weighted average velocity \(\mathbf{U}\) is calculated by

\[
\mathbf{U} = (1 - \alpha) \mathbf{u} + \alpha \mathbf{u}_{\text{obj}}.
\]  

(8)

In Eq.(8) \(\alpha\) and \(\mathbf{u}_{\text{obj}}\) indicate the local volume fraction (LVF) of the object and the object velocity. Here \(\alpha\) can be determined efficiently by counting the saved points of the
method (Yao et al., 2018) was proposed for investigating wall boundary are precisely modeled. Simulated if solid–fluid interaction, gas flow, and moving plies that solid particle behavior can be preciselying on these particles is exceptionally high. This fact im-
is small but also their density. Hence, the drag force act-
ceutical particles are concerned, not only the particle size
sively focused in pharmaceutics. As far as the pharma-
tical particles are involved, not only the particle size

$$\frac{\partial(qpU)}{\partial t} + \nabla \cdot (qpUU) = -\varepsilon \nabla p + \mathbf{f}_s + \nabla \cdot (\varepsilon \mathbf{e}) + \varepsilon g + F_{IB},$$

where $F_{IB}$ is the correction term in the IBM. The force term $F_{IB}$ is given by

$$F_{IB} = \frac{qp(u_{obj} - U)}{\Delta t},$$

Very recently, the authors have shown that fluid flow might not be calculated suitably when the IBM is used as the wall boundary. This problem can be solved by considering the density scaling coefficient (Sun and Sakai, 2017), which is given by

$$\rho_{scale} = \begin{cases} \gamma \rho & (\phi < 0) \\ \rho & \text{otherwise} \end{cases},$$

where $\gamma$ is the density scaling factor. Actually, the density is scaled in the pressure Poisson equation, where the grids are filled by the object. Eventually, the pressure Poisson equation is be given by

$$\nabla \cdot \left( \frac{\rho_{scale}}{\rho} \nabla p^{n+1} \right) = \frac{1}{\Delta t} \left( \frac{\rho^{n+1} - \rho^n}{\Delta t} + \nabla \cdot (\varepsilon^{n+1} U^{n+1}) \right),$$

with the density scaling factor being introduced. In Eq. (12), superscript $n+1$ and $n$ are updated iteration number and present iteration number.

2.3 Powder flow involving airflow in die-filling

As a typical gas–solid flow system with a moving wall boundary, the powder die-filling is a preferable example. To date, fundamental studies of die-filling (Zakhvatayeva et al., 2018; Baserinia and Sinka, 2019) have been extensively focused in pharmaceutics. As far as the pharmaceutcal particles are concerned, not only the particle size is small but also their density. Hence, the drag force acting on these particles is exceptionally high. This fact implies that solid particle behavior can be precisely simulated if solid–fluid interaction, gas flow, and moving wall boundary are precisely modeled.

In the authors’ group, the Advanced DEM–CFD method (Yao et al., 2018) was proposed for investigating the effect of air flow on the particle filling state in die-filling. In the Advanced DEM-CFD method, combination of the SDF and the IBM was introduced into the DEM-CFD method. The schematic diagram of the die-filling system is shown in Fig. 2; here, the shoe was moved by 0.10 m/s toward the left side. In this study, calculations and experiments were performed under fair conditions.

The target powder was NONPAREIL-108® (FREUND CORPORATION). The particle size and the particle density were, respectively, 200 µm and 1,379 kg/m³, and the number of the calculated particles was 150,000. A combination of the SDF and IBM was used for the wall boundary model for the DEM and the CFD. Fig. 3 displays the schematic diagram of the die-filling system. Fig. 3(a) is the scalar field of the SDF, where the sign of the calculation domain and inside of the objects respectively became positive and negative. The SDF was created independently to each object. The interval of the SDF was usually one-quarter of the particle size. Fig. 3(b) designates the LVF for the IBM. When the object was fulfilled in a CFD grid, the value of the LVF of the gas phase, namely, $1 - \alpha$, became zero. By calculating the values of the LVF, interactions between the object and fluid could be simulated through the IBM. In order to show the influence of airflow on the flow of the solid particles, two kinds of simulations were performed: a DEM simulation for a single granular flow and a DEM–CFD simulation for a gas–solid flow. The calculation results are presented in Fig. 4. Die-filling simulation by the DEM–CFD method, where air flow was taken into consideration, demonstrated the appearance of a clearance on top of the powder bed, as illustrated in Fig. 4(a). On the contrary, the powder was fulfilled when the DEM was simulated without air flow.

Actually, the powder bed shape obtained from the DEM–CFD simulation exhibited an excellent agreement with that of the experiment. Hence, consideration of the air flow was shown as an indispensable factor for the simulation of the die-filling in pharmaceutical powders.
2.4 Industrial fluidized bed

A fluidized bed is extensively investigated by numerical simulation, where the DEM–CFD method is widely employed. At present, application of the DEM–CFD method to industrial fluidized beds (Sakai, 2016; C. Hu et al., 2019; Ge et al., 2019; Stroh et al., 2019) is among the important research topics. Not the shape of industrial fluidized beds is always simple, though rectangular systems have been employed in the existing fluidized bed simulations. Nonetheless, the structure of actual fluidized beds might be extremely complex, as tubes might have been inserted, a slope might have been set at the bottom, or they must have been equipped with cyclone. As a matter of course, number of the solid particles becomes huge in the industrial systems. In this sense, besides modeling of the calculation domain shape, the number of the computational particles becomes a critical problem as well. The authors’ group attempted to solve the problem with the limit of the calculated particles by developing a coarse graining DEM (Sakai and Koshizuka, 2009; Sakai et al., 2010; Sakai et al., 2012b; Sun et al., 2014; Takabatake et al., 2018), in which largely modeled particles represent the group of the original particles where total energy between them is assumed to agree. The adequacy of such approach has been shown through verification and validation tests, in which the combined SDF–IBM was employed in the numerical simulations for the fluidized bed. Very recently, the compatibility of the coarse graining DEM and the wall boundary model of SDF/IBM has been proven through validation tests (Mori, Wu and Sakai, 2019), whose results are addressed in this paper.

A schematic diagram of the fluidized bed system is illustrated in Fig. 5. In the fluidized bed, four tubes were inserted. Gas was injected at 0.2 m/s from the bottom side; glass beads were used for the powder and gas for air. The coarse graining DEM was employed in this study; hence, the number of computational particles could be drastically reduced than the actual one. Namely, the coarse grain ratio was 5.0 for 250,000 calculated particles. Actually, the number of solid particles reached the order of seven digits, and hence, calculation could not be performed without the coarse graining DEM on a single PC. Hence, application of the coarse graining DEM makes it possible to perform large-scale gas–solid systems with a smaller number of the calculated particles, as compared to using the actual number. The SDF and the IBM were employed as the wall boundary model, as depicted in Fig. 6. In Fig. 6(a), the calculation domain was expressed as positive in the SDF. LVF of the gas phase \( (1 - \alpha) \) was zero inside of the object in the IBM, as illustrated in Fig. 6(b).
Fig. 7 designates snapshots obtained from the validation tests. Note the excellent agreement between the calculated and experimental results. Specifically, bubble behavior and spatial location of the particles in the simulation and in the experiment were in good agreement. Besides, the pressure drop quantitatively agreed in the computation and the experiment. Thus, the coarse grain DEM was shown capable of simulating the macroscopic behavior of the original gas–solid flow system, although the particle size was larger than the actual one. This is because the total energy was modeled to agree between the coarse grain particle and a group of the original particles. In fact, the calculation has been done prior to the experimentation. Surprisingly, the agreement could be obtained without any necessary adjustment in the calculation.

3. Mesh-free particle method for a phase change problem

Modeling the phase change between the solid and liquid phases is one of the challenging problems in the CFD. The mesh-free particle method is a promising approach to simulate the phase change, and has been applied into various systems, such as additive manufacturing (Russell et al., 2018), welding (Wang et al., 2018), and nuclear severe accidents (Duan et al., 2019). During a phase change, the solid and liquid states are mixed, thus, such state is regarded as a solid–liquid flow in a broad sense. In the simulation of the phase change by the mesh-free methods, the efficient calculation for highly viscous fluid flow, as well as supplying heat flux on the free surface, is essential. The authors’ group has developed certain models for the highly viscous fluid flow and the heat flux on the fluid surface, which are described herein.

3.1 The MPS method for phase change simulation

The original MPS method (Koshizuka and Oka, 1996) has been developed for an incompressible fluid flow, where the pressure Poisson equation is calculated implicitly. Recently, weakly incompressible algorithm has been introduced into the MPS method by calculating the pressure explicitly. Such method is referred to as E-MPS method. Here, the kernel functions are the same as the original ones. In general understanding, the accuracy of the E-MPS method is quite similar to that of the original MPS method, as both have been employed even in the solid–liquid flow states in chemical engineering (Sakai et al., 2012; Yamada and Sakai, 2013; Sun et al., 2014). Modeling of the phase change between the solid state and
The governing equations for phase change simulation by the MPS method include a Lagrangian description of the continuity equation, the Navier–Stokes equation, and energy conservation equation,

\[
\frac{D \rho}{Dt} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = 0,
\]

and

\[
\frac{D \mathbf{u}}{Dt} = -\frac{1}{\rho} \mathbf{V} p + \frac{\mu_f}{\rho} \mathbf{V}^2 \mathbf{u} + \mathbf{g} + \frac{1}{\rho} \sigma \kappa \delta \mathbf{n},
\]

where \( \mu_f, \sigma, \kappa, \delta, m, h, \lambda, \) and \( Q \) are, respectively, fluid viscosity, coefficient of surface tension, curvature, delta function, normal vector, enthalpy, heat conductivity and heat source. In Eq. (15), the heat flux can be given as the volumetric heat source,

\[
\dot{Q} = \mathbf{V} \cdot \mathbf{q},
\]

with \( \mathbf{q} \) as the heat flux.

Moreover, in the MPS method, the pressure and viscous terms in the governing equations are modeled by a gradient model and a Laplacian model. Regarding fluid particle \( i \), these models are given by

\[
\{\nabla \Phi_i\} = \frac{2D}{\Omega \rho^0} \sum_{j \neq i} \left[ \Phi_j - \Phi_i \right] w(r_j - r_i),
\]

where \( D, n^0, r, \) and \( \Omega \) are, respectively, dimension, initial particle number density, position vector, and Laplacian constant in the MPS method. In these models, weight functions are used to give smoothed interpolation for quantities through the interaction between fluid particles. In order to perform a phase change, the enthalpy is introduced as,

\[
h_s = \rho C_p T_m,
\]

\[
h_l = h_s + \rho L,
\]

where \( h_s, C_p, T_m, h_l, \) and \( L \) are, respectively, enthalpy of the solid phase, heat capacity, melting temperature, enthalpy of the solid phase, and latent heat. Incidentally, solid state is simulated as exceptionally highly fluid expressed by Arrhenius viscosity formula.

When the viscous term is calculated implicitly, updated fluid velocity \( \mathbf{u}^{k+1} \) can be obtained by the expressions,

\[
\frac{\mathbf{u} - \mathbf{u}^k}{\Delta t} = \mathbf{g} - \frac{1}{\rho} \mathbf{V} p,
\]

\[
\frac{\mathbf{u}^{k+1} - \mathbf{u}^k}{\Delta t} = \frac{\mu_f}{\rho} \mathbf{V}^2 \mathbf{u}^{k+1}.
\]

A detailed discussion on the MPS method is presented in the original papers (Koshizuka and Oka, 1996).

### 3.2 Investigation on the calculation efficiency of a highly viscous fluid

First, implicit calculation of the viscous term is shown to be efficiently performed in the simulation of a discharged fluid flow in a glass melter.

In this system, kinematic viscosity, fluid density, and surface tension coefficient were, respectively, 0.03 m²/s, 2500 kg/m³, and 0.3 N/m. The highly viscous fluid was flown by only the gravitational force. Fig. 8 illustrates the snapshots obtained from a simulation of a discharged flow of the highly viscous fluid in a glass melter (elapsed time: 5.0 s, 10 s, 15 s and 20 s). Here, the volume flow rate in the simulation agreed well with the experiment, in the steady state. Subsequently, the acceleration due to the implicit model was investigated based on the maximum value of the time step. Specifically, the time step ratio of the Courant condition to diffusion number was roughly \( 10^{-4} \), which implies that the computation could be accelerated up to \( 10^4 \) times faster than the original viscous calculation model in this system. Thus, the efficiency of the implicit calculation algorithm was proven in the highly viscous fluid flow system.
3.3 Adequacy of the heat flux model in a mesh-free particle method

Subsequently, the adequacy of the heat flux model is mentioned in the MPS simulation. In the MPS method, instead of the heat flux, the temperature is usually given (the Dirichlet boundary condition) at the free surface due to ease. On the contrary, heat balance cannot be kept if the heat flux (the Neumann boundary condition) is not given. As shown in Eq. (16), the heat flux can be introduced into the mesh-free particle method as heat source. This attempt has been done very recently by the authors (Takabatake et al., 2016).

The adequacy of the heat flux model was examined via heat transfer simulation in a simple system (Fig. 9).

As boundary conditions, heat flux was 25,000 W/m² at the top of the rectangle, while the temperature at the bottom was 300 K. Regarding the physical properties, the density, kinetic viscosity, specific heat, and thermal conductivity were, respectively, 1,000 kg/m³, 1.0 × 10⁻³ m²/s, 10 J/(kg K), and 50 W/(m K). Temperature distribution in the calculation and theoretical results were later compared.

The calculation result is shown in Fig. 10. Temperature increased gradually from the initial value (300 K). The temperature distribution achieved a steady state in 20 s and was found to quantitatively agree with the theoretical results. This result implies that the heat flux can be given even in the mesh-free particle method, where the free surface is violently moved.

This aside, a phase change simulation was performed through the supplied heat flux. Two types of materials, namely, Material-1 and Material-2, were used in the calculations. The density, kinetic viscosity, specific heat, thermal conductivity, latent heat and melting temperature of Material-1 were, respectively, 1,000 kg/m³, 1.0 × 10⁻³ m²/s, 100 J/kg K, 50 W/m K, 10,000 K, and 1.0 × 10⁴ J/kg. Material-2 had the same density, kinetic viscosity, and latent heat, while the specific heat, thermal conductivity, and melting temperature were, respectively, 10 J/kg K, 500 W/m K, and 320 K. As far as the boundary conditions were concerned, the initial heat flux was 100,000 W/m² on the upper side and the temperature was 300 K at the bottom side.

Snapshots obtained from the phase change calculation are shown in Fig. 11. Due to the heat flux, the upper side was molten at the beginning. Solidification occurred as the molten fluid approached the floor, as the floor temperature was kept at 300 K. Hence, simulation of the phase change was demonstrated by the MPS method.
4. Conclusions

This paper described a cutting-edge modeling on simulations for multi-phase flows using the DEM and the MPS methods.

Relative to the DEM, a brand-new wall boundary model was introduced, namely, a combinational model of the SDF with the IBM, which was shown to be useful in generating an arbitrary shape wall boundary in the DEM–CFD simulation. Such new model is advantageous in the sense that an arbitrary shape wall boundary could be created by a simple operation, and thus, it might be standardized in the future DEM–CFD simulations.

Regarding the mesh-free particle method for continuum such as the MPS method, the phase change problem has generally been a challenging topic. An implicit calculation method, along with the heat flux model, was developed for efficiently simulating the phase change. The usefulness of such heat transfer modeling was demonstrated in this paper. Hereafter, it is expected to contribute to the numerical simulation of a thermo-mechanical problem.

Acknowledgments

Parts of this study were financially supported by JSPS KAKENHI Grant Numbers 17H02825 and 17KK0110.

Nomenclature

DEM discrete element method
CFD computational fluid dynamics
MPS moving particle semi-implicit
SDF signed distance function
IBM immersed boundary method
CPU central process unit
GPU graphical processing unit
D dimension number (–)
$F_{Cu}$ normal component of contact force (N)
$F_{Ct}$ tangential component of contact force (N)
$F_{IB}$ correction term in the IBM (N/m$^3$)
$f_s$ solid–fluid momentum exchange term (N/m$^3$)
g gravitational acceleration (m/s$^2$)
$h_s$ enthalpy of solid phase (J/m$^3$)
$h_l$ enthalpy of liquid phase (J/m$^3$)
k spring constant (N/m)
$L$ latent heat (J/kg)
n normal vector (–)
n$\text{pu}$ normal vector in the particle-wall interaction (–)
$n^0$ initial particle number density (–)
p fluid pressure (Pa)
$Q$ heat source (W/m$^3$)
r distance between two fluid particles (m)
r$_p$ particle radius (m)
$\Delta t$ time step (s)
$U$ volume-weighted average velocity (m/s)
$u_{\text{obj}}$ object velocity (m/s)
$v_s$ relative velocity of the interacting particles (m/s)
$\alpha$ local volume fraction of a object (–)
$\delta_{pw}$ displacement in particle-wall interaction (m)
$\delta$ delta function (–)
$\varepsilon$ void fraction (–)
$\phi$ signed distance function (m)
$\gamma$ density scaling factor (kg/s)
$\eta$ damping coefficient (–)
$\kappa$ curvature (m$^{-1}$)
$\lambda$ heat conductivity (W/m K)
$\mu_t$ viscosity (Pa s)
$\mu_s$ friction coefficient of a solid particle (–)
$\Omega$ Laplacian constant in the MPS method (–)
$\rho$ fluid density (kg/m$^3$)
$\sigma$ coefficient of surface tension (N/m)
$\tau$ viscous stress (N/m$^2$)
References

Adam A., Pavlidis D., Percival J.R., Salinas P., Xie Z., Fang F., Pain C.C., Mugggeridge A.H., Jackson M.D., Higher-order conservative interpolation between control-volume meshes: Application to advection and multiphase flow problems with dynamic mesh adaptivity, Journal of Computational Physics., 321 (2016) 512–531. https://doi.org/10.1016/j.jcp.2016.05.058

Anderson T.B., Jackson R., Fluid mechanical description of fluidized beds. Equations of motion, Industrial and Engineering Chemistry Research, 6 (1967) 527–539. https://doi.org/10.1021/i160024a007

Basinskas G., Sakai M., Numerical study of the mixing efficiency of a ribbon mixer using the discrete element method, Powder Technology, 287 (2016a) 380–394. https://doi.org/10.1016/j.powtec.2015.10.017

Basinskas G., Sakai M., Numerical study of the mixing efficiency of a batch mixer using the discrete element method, Powder Technology, 301 (2016b) 815–829. https://doi.org/10.1016/j.powtec.2016.07.017

Chan E.L., Washino K., Coarse grain model for DEM simulation of dense and dynamic particle flow with liquid bridge forces, Chemical Engineering Research and Design, 132 (2018) 1060–1069. https://doi.org/10.1016/j.cherd.2017.12.033

Chen R., Oka Y., Li G., Matsuura T., Numerical investigation on melt freezing behavior in a tube by MPS method, Nuclear Engineering and Design, 273 (2014) 440–448. https://doi.org/10.1016/j.nucengdes.2014.03.049

Cundall P.A., Strack O.D.L., A discrete numerical model for granular assemblies, Geotechnique, 29 (1979) 47–65. https://doi.org/10.1680/geot.1979.29.1.47

Dhenge R.M., Washino K., Cartwright J.J., Houns-low M.J., Salman A.D., Twin screw granulation using conveying screws: Effects of viscosity of granulation liquids and flow of powders, Powder Technology, 238 (2013) 77–90. https://doi.org/10.1016/j.powtec.2012.05.045

Duan G., Yamaji A., Koshizuka S., A novel multiphase MPS algorithm for modeling crust formation by highly viscous fluid for simulating corium spreading, Nuclear Engineering and Design, 343 (2019) 218–231. https://doi.org/10.1016/j.nucengdes.2019.01.005

Ergun S., Fluid flow through packed columns. Chemical Engineering Progress Symposium Series, 48 (1952) 89–94.

Ge W., Chang Q., Li C., Wang J., Multiscale structures in particle – fluid systems: Characterization, modeling, and simulation, Chemical Engineering Science, 198 (2019) 198–223. https://doi.org/10.1016/j.ces.2018.12.037

Govender N., Wilke D.N., Wu C.Y., Rajamani R., Khinast J., Glasser B.J., Large-scale GPU based DEM modeling of mixing using irregularly shaped particles, Advanced Powder Technology, 29 (2018) 2476–2490. https://doi.org/10.1016/j.apt.2018.06.028

Hu C., Luo K., Wang S., Sun L., Fan J., Influences of operating parameters on the fluidized bed coal gasification process: A coarse-grained CFD-DEM study, Chemical Engineering Science, 195 (2019) 693–706. https://doi.org/10.1016/j.ces.2018.10.015

Hu R., Fang F., Salinas P., Pain C.C., Domingo N.D.S., Mark O., Advances in water resources numerical simulation of floods from multiple sources using an adaptive anisotropic unstructured mesh method, Advances in Water Resources, 123 (2019) 173–188. https://doi.org/10.1016/j.advwatres.2018.11.011

Kafashan J., Wiącek J., Abd N., Jieqing R., Two-dimensional particle shapes modelling for DEM simulations in engineering: A review, Granular Matter, (2019) 21:80. https://doi.org/10.1007/s10035-019-0935-1

Kajishima T., Takiguichi S., Hamasaki H., Miyake Y., Turbulence structure of particle-laden flow in a vertical plane channel due to vortex shedding, JSME International Journal Series B, 44 (2001) 526–535. https://doi.org/10.1299/jsmeb.44.526

Kanaoka C., Fine particle filtration technology using fiber as dust collection medium, KONA Powder and Particle Journal, 36 (2019) 88–113. https://doi.org/10.14356/kona.2019006

Klinzing G.E., Historical review of pneumatic conveying, KONA Powder and Particle Journal, 35 (2018) 150–159. https://doi.org/10.14356/kona.2018010

Koshizuka S., Oka Y., Moving-particle semi-implicit method for fragmentation of incompressible fluid, Nuclear Science and Engineering, 123 (1996) 421–434. https://doi.org/10.13182/NSE96-A24205

Krok A., Garcia-tri P., Peciar M., Wu C.-Y., Finite element analysis of thermomechanical behaviour of powders during tabletting, Chemical Engineering Research and Design, 110 (2016) 141–151. https://doi.org/10.1016/j.cherd.2016.03.019

Lu L., Xu J., Ge W., Gao G., Jiang Y., Zhao M., Liu X., Li J., Computer virtual experiment on fluidized beds using a coarse-grained discrete particle method—EMMS-DPM, Chemical Engineering Science, 155 (2016) 314–337. https://doi.org/10.1016/j.ces.2016.08.013

Monaghan J.J., Simulating free surface flows with SPH, Journal of Computational Physics, 110 (1994) 399–406. https://doi.org/10.1006/jcph.1994.1034

Mori Y., Wu C.-Y., Sakai M., Validation study on a scaling law model of the DEM in industrial gas-solid flows, Powder Technology, 343 (2019) 101–112. https://doi.org/10.1016/j.powtec.2018.11.015

Nie Z., Zhu Y., Wang X., Gong J., Investigating the effects of Fourier-based particle shape on the shear behaviors of rockfill material via DEM, Granular Matter, 21 (2019) 22. https://doi.org/10.1007/s10035-019-0875-9
Shigeto Y., Sakai M., Arbitrary-shaped wall boundary modeling

Sakai M., Shibata K., Koshizuka S., Tanizawa K., Transparent

Schmelzle S., Nirschl H., DEM simulations

Sakai M., Yamada Y., Shigeto Y., Shibata K., Kawasaki V.M.,

Sakai M., Takahashi H., Pain C.C., Latham J.-P., Xiang J., Study

Sakai M., Koshizuka S., Large-scale discrete element modeling

Sakai M., Abe M., Shigeto Y., Mizutani S., Takahashi H., Viré

Percival J.R., Xiang J., Pain C.C., Verification and validation

do coarse grain model of the DEM in a bubbling fluidized bed, Chemical Engineering Journal, 244 (2014) 33–43. https://doi.org/10.1016/j.cej.2014.01.029

Kazuya T., Kotaro T., Hatano K., Minakuchi A., Why do wet-particles adhere to a high-speed roll in a three-roll mill? Physics of Fluids, 31 (2019) 033302. https://doi.org/10.1063/1.5085693

Koshizuka S., Large-scale discrete element modeling in pneumatic conveying, Chemical Engineering Science, 64 (2009) 533–539. https://doi.org/10.1016/j.ces.2008.10.003

Shigeto Y., Basinskas G., Hosokawa A., Fuji M., Discrete element simulation for the evaluation of solid mixing in an industrial blender, Chemical Engineering Journal, 279 (2015) 821–839. https://doi.org/10.1016/j.cej.2015.04.130

Sun X., Sakai M., A liquid bridge model for spherical particles applicable to asymmetric configurations, Chemical Engineering Science, 182 (2018) 28–43. https://doi.org/10.1016/j.ces.2018.02.034

Sun X., Sakai M., Immersed boundary method with artificial density in pressure equation for modeling flows confined by wall boundaries, Journal of Chemical Engineering of Japan, 50 (2017) 161–169. https://doi.org/10.1252/jcej.16we115

Sun X., Sakai M., Direct numerical simulation of gas-solid-liquid flows with capillary effects: An application to liquid bridge forces between spherical particles, Physical Review E, 94 (2016) 063301. https://doi.org/10.1103/PhysRevE.94.063301

Sun Xiasong, Sakai M., Numerical simulation of two-phase flows in complex geometries by using the volume-of-fluid/immersed-boundary method, Chemical Engineering Science, 139 (2016) 221–240. https://doi.org/10.1016/j.ces.2015.09.031

Sun X., Sakai M., Sakai M.-T., Yamada Y., A Lagrangian–Lagrangian coupled method for three-dimensional solid-liquid flows involving free surfaces in a rotating cylindrical tank, Chemical Engineering Journal, 246 (2014) 122–141. https://doi.org/10.1016/j.cej.2014.02.049

Sun X., Sakai M., Shibata K., Tochigi Y., Fujiwara H., Numerical modeling on the discharged fluid flow from a glass melting by a Lagrangian approach, Nuclear Engineering and Design, 248 (2012) 14–21. https://doi.org/10.1016/j.nucengdes.2012.04.004

Sun X., Sakai M., Sun M., Takabatake K., Pain C.C., Numerical simulation of free surface fluid flows through porous media by using the explicit MPS method, Transport in Porous Media, 127 (2019) 7–33. https://doi.org/10.1007/s11242-018-1178-z

Takabatake K., Moriy Y., Khinist J.G., Sakai M., Numerical investigation of a coarse-grain discrete element method in solid mixing in a spouted bed, Chemical Engineering Journal, 346 (2018) 416–426. https://doi.org/10.1016/j.cej.2018.04.015

Takabatake K., Sun X., Sakai M., Pavlidis D., Xiang J., Pain C.C., Numerical study on a heat transfer model in a Lagrangian fluid dynamics simulation, International Jour-
Tsuji Y., Kawaguchi T., Tanaka T., Discrete particle simulation of two-dimensional fluidized bed, Powder Technology, 77 (1993) 79–87. https://doi.org/10.1016/0032-5910(93)85010-7

Tsunazawa Y., Fujihashi D., Fukui S., Sakai M., Tokoro C., Contact force model including the liquid-bridge force for wet-particle simulation using the discrete element method, Advanced Powder Technology, 27 (2016) 652–660. https://doi.org/10.1016/j.apt.2016.02.021

Tsunazawa Y., Shigeto Y., Tokoro C., Sakai M., Numerical simulation of industrial die filling using the discrete element method, Chemical Engineering Science, 138 (2015) 791–809. https://doi.org/10.1016/j.ces.2015.09.014

Udono H., Sakai M., A numerical study on dynamic inertial focusing of microparticles in a confined flow, Granular Matter, 19 (2017) 79. https://doi.org/10.1007/s10035-017-0758-x

Wang X., Shao M., Gao S., Gau J., Tang H., Jin H., Liu H., Numerical simulation of laser impact spot welding, Journal of Manufacturing Processes, 35 (2018) 396–406. https://doi.org/10.1016/j.jmapro.2018.08.028

Wen C.Y., Yu Y.H., Mechanics of fluidization, Chem. Eng. Progr. Symposium Series, 62 (1966) 100–111.

Xiong J., Koshizuka S., Sakai M., Investigation of droplet impingement onto wet walls based on simulation using particle method, Journal of Nuclear Science and Technology, 48 (2011) 145-153. https://doi.org/10.1080/18811248.2011.9711689

Xiong J., Koshizuka S., Sakai M., Numerical analysis of droplet impingement using the moving particle semi-implicit method, Journal of Nuclear Science and Technology, 47 (2010) 314–321. https://doi.org/10.3327/jnst.47.314

Yamada Y., Sakai M., Lagrangian–Lagrangian simulations of solid–liquid flows in a bead mill, Powder Technology, 239 (2013) 105–114. https://doi.org/10.1016/j.powtec.2013.01.030

Yang J., Guo Y., Buettner K.E., Curtis J.S., DEM investigation of shear flows of binary mixtures of non-spherical particles, Chemical Engineering Science, 202 (2019) 383–391. https://doi.org/10.1016/j.ces.2019.03.027

Yao H., Mori Y., Takabatake K., Sun X., Sakai M., Numerical investigation on the influence of air flow in a die filling process, Journal of the Taiwan Institute of Chemical Engineers, 90 (2018) 9–17. https://doi.org/10.1016/j.jtice.2017.11.031

Yokoi K., Numerical method for interaction between multiparticle and complex structures, Physical Review E, 72 (2005) 046713. https://doi.org/10.1103/PhysRevE.72.046713

Yue Y., Wang T., Sakai M., Shen Y., Particle-scale study of spout deflection in a flat-bottomed spout fluidized bed, Chemical Engineering Science, 205 (2019) 121–133. https://doi.org/10.1016/j.ces.2019.04.031

Zakhvatayeva A., Zhong W., Makroo H.A., Hare C., Wu C.Y., An experimental study of die filling of pharmaceutical powders using a rotary die filling system, International Journal of Pharmaceutics, 553 (2018) 84–96. https://doi.org/10.1016/j.ijpharm.2018.09.067

Zhou Z.Y., Pinson D., Zou R.P., Yu A.B., Discrete particle simulation of gas fluidization of ellipsoidal particles, Chemical Engineering Science, 66 (2011) 6128–6145. https://doi.org/10.1016/j.ces.2011.08.041
Authors’ Short Biographies

Mikio Sakai

Dr. Mikio Sakai is currently Associate Professor in Resilience Engineering Research Center in The University of Tokyo. He earned his Ph.D. degree from The University of Tokyo in 2006. Then, he became Assistant Professor in 2007 and Associate Professor in 2008 in the same university. He has been Visiting Reader at Imperial College London since 2016. He extensively studies modeling of granular flows, multi-phase flows and the heat transfer, besides the parallel computation techniques. He is a world-leading professor in computational granular dynamics, and hence, has delivered lots of invited lectures in conferences. He holds important posts in powder technology community, such as Director of Society of Powder Technology of Japan and Head of Simulation & Modeling Division in Association of Powder Process Industry and Engineering, JAPAN. At present, he is also Associate Editor of Chemical Engineering Science and Editor of Granular Matter.

Yuki Mori

Mr. Yuki Mori is currently a freshman PhD student in Department of Nuclear Engineering and Management, School of Engineering, The University of Tokyo, and is JSPS Research Fellow (DC1). He joined Prof Sakai’s group since 2016 and won Dean’s Award (Annual Best Student Award) upon completion of his masteral course in The University of Tokyo.

Xiaosong Sun

Dr. Xiaosong Sun earned his Ph.D. degree from The University of Tokyo in 2016. Then, he became Project Researcher in Resilience Engineering Research Center, School of Engineering, The University of Tokyo. He joined Prof Sakai’s group since 2011 and won Dean’s Award (Annual Best Student Award) upon completion of his masteral and doctorate courses in The University of Tokyo.

Kazuya Takabatake

Mr. Kazuya Takabatake is currently a third-year PhD student in Department of Nuclear Engineering and Management, School of Engineering, The University of Tokyo, and is JSPS Research Fellow (DC2) since 2018. He joined Prof Sakai’s group since 2015 and won Dean’s Award (Annual Best Student Award) upon completion of his masteral course in The University of Tokyo.