Scaling method of CFD-DEM simulations for gas-solid flows in risers

Citation for published version (APA):

Document license:
CC BY

DOI:
10.1016/j.cesx.2019.100054

Document status and date:
Published: 01/02/2020

Document Version:
Publisher’s PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:
• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher’s website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.

Download date: 24. Apr. 2020
Scaling method of CFD-DEM simulations for gas-solid flows in risers

L. Mu a, K.A. Buista, J.A.M. Kuipers a, N.G. Deen b

a Eindhoven University of Technology, Dept. Chemical Engineering and Chemistry, Multiphase Reactors Group, PO Box 513, 5600 MB Eindhoven, the Netherlands
b Eindhoven University of Technology, Dept. Mechanical Engineering, Power & Flow Group, PO Box 513, 5600 MB Eindhoven, the Netherlands

ARTICLE INFO

Article history:
Received 10 May 2019
Received in revised form 13 December 2019
Accepted 21 December 2019

Keywords:
Scaling method
CFD riser
CFD-DEM
Hydrodynamics
Clusters

ABSTRACT

In this paper a scaling method is proposed for scaling down the prohibitively large number of particles in CFD-DEM simulations for modeling large systems such as circulating fluidized beds. Both the gas and the particle properties are scaled in this method, and a detailed comparison among alternative mapping strategies is performed by scaling both the computational grid size and the riser depth. A series of CFD-DEM simulations has been performed for a pseudo-2D CFB riser to enable a detailed comparison with experimental data. By applying the scaling method, the hydrodynamic flow behavior could be well predicted and cluster characteristics, such as cluster velocity and cluster holdups agreed well with the experimental data. For a full validation of the scaling method, four mapping conditions with different ratios of the grid size and particle volume and of modified ratio of riser depth to particle size were analyzed. The results show that in addition to hydrodynamic scaling of the particle and fluid properties, scaling of the dimensions for the interphase mapping is also necessary.

© 2019 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/).

1. Introduction

Gas-solid fluidization in riser reactors has received substantial interest as it is widely encountered in numerous industrial processes such as base chemical production, biomass gasification and catalytic cracking (Khan et al., 2014; Shah et al., 2016; Gómez-Barea and Leckner, 2010). The performance of risers has been extensively investigated over the past decades to acquire an in-depth insight of their hydrodynamics (Geldart and Rhodes, 1986; Tsuo and Gidaspow, 1990; Horio and Kuroki, 1994). Risers are usually operated at high superficial velocities under fast fluidization conditions. These systems exhibit a so-called “core annulus” flow structure, which is characterized by a dilute solids up-flow in the core of the riser, and a dense down-flow close to the walls (Geldart and Rhodes, 1986; Tsuo and Gidaspow, 1990; Horio and Kuroki, 1994; Bolton and Davidson, 1988; Hartge et al., 1988). The dense regions typically contain particle clusters, where the gas permeability is reduced, which negatively impacts the performance of risers as a chemical reactor. CFD-DEM simulations have the major advantage of having very good predictive capabilities. However in most circulating fluidized beds, the number of particles is in the order of $10^8$-$10^{12}$ which is the biggest bottleneck of these simulations. Such particle numbers would quickly lead to prohibitively long simulation times (months-years). Considering the computational cost, it is desirable to develop an appropriate scaling method to substantially reduce the number of particles by increasing the particle size whilst maintaining the capability to capture main flow features.

For gas-solid flows, Andrews and O’Rourke (1996) proposed the so-called “Multi-Phase Particle-In-Cell” (MP-PIC) approach, which is a parcel-approach and has been widely applied to granular flows (O’Rourke et al., 2009; O’Rourke and Snider, 2010; Snider, 2001). Instead of tracking collisions between particles directly, the MP-PIC employs a simple “particle pressure” model to prevent particles from becoming closely-packed. Patankar and Joseph (2001) explored an Euler-Lagrangian numerical simulation (LNS) scheme for particulate flow, and showed that the parcel approach is able to capture the basic flow features. But this scheme is strictly applicable when collisions do not play a dominant role in the flow behavior, meaning dilute flow. Sakai and Koshizuka (2009) developed a coarse-grained model, which considers the drag force and the contact force. Their model could simulate the 3D plug flow in a horizontal pipeline system accurately but is limited to systems with a small number of calculated particles. The similar particle assembly (SPA) model proposed for large-scale discrete element method simulation was validated in the work of Mokhtar et al. (2012). In their model, particles with similar physical or chemical properties are represented by a single particle. The influence of coarse graining was studied in literature (Bierwisch et al., 2009) and revealed that bulk volume fractions are independent of grain size provided the underlying forces scale quadratically with the grain diameter. Note that the latter study was restricted to dilute

E-mail address: N.G.Deen@tue.nl (N.G. Deen)

https://doi.org/10.1016/j.cesx.2019.100054
2590-1400 © 2019 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/).
granular systems where only binary collisions occur. Whether the scaling is still valid in the intermediate to dense regimes is not clear. Alternatively, Liu and Xu (2009) have modeled CFBs by modeling the gas-rich lean phase and separately modeling the solid-rich cluster phase using an equation for the cluster motion based on Newton’s law. Additional models for the collision, coalescence and breakup of clusters were derived.

Filtered two-fluid models (TFM) have been constructed for coarse-grid simulation (Parmentier et al., 2012; Milioli et al., 2013; Gao et al., 2018; Schneiderbauer and Pirker, 2014). Gao et al. (2018) conducted a comparative evaluation of several model settings to assess the effect of mesoscale solid stress in a coarse-grid TFM simulation of gas-solid fluidized beds of Geldart A particles over a broad range of fluidization regimes. Various researchers have explored proper scaling rules for DEM simulations (Radl et al., 2011; Li, 1994; Radl et al., 2012; Link et al., 2009). To keep the contact forces constant, Radl et al. (2011) derived a scaling law for a linear-spring dashpot interaction model that enables tracking of clouds of particles through DEM-based simulation of scaled pseudo-particles. However this work did not consider the gas phase. In relatively coarse simulations the consideration of the gas-particle interactions were handled with the Energy Minimization and Multiscale Analysis (EMMS) (Li, 1994). EMMS allows to model a coarse grained gas-particle flow, accounting for sub grid structures or clusters. This is needed when the assumption of a uniform porosity inside the grids is no longer valid. A sophisticated filtered drag model was established in literature (Radl et al., 2012) for use in coarse-grid Euler-Lagrangian simulations, highlighting the significant effect of particle clustering on the average slip velocity between particles and fluid and indicated how this clustering can be accounted for in unresolved EL-based simulations. The unclosed terms in the filtered model could also be constructed through a filtering operation of fine-grid resolved CFD-DEM simulations (Ozel et al., 2017, 2016). Ozel et al. (2017) performed Euler-Lagrangian simulations of gas-solid flows in periodic domains to study the effective drag force model to be used in coarse-grained EL and filtered EE models. A dynamic scale-similarity approach was used to model the drift velocity but the predictability of that model is not entirely satisfactory. One scaling law that could apply for the flow regime was constructed by Glicksman et al. (1994) by keeping the key dimensionless parameters constant, including the ratio of the inertial and viscous forces. This scaling law could be extended to model bed-to-wall heat transfer. A somewhat similar approach for the gas-particle interaction was proposed by Liu et al. (2013), where both the Reynolds and Archimedes numbers are scaled. Additional scaling parameters however are still required, to account for proper scaling of the particle contact forces A combination of hydrodynamic scaling as with Liu et al. (2013) and the particle contact forces as in the work of Radl et al. (2011) is expected to provide better results.

The clustering of particles in circulating fluidized beds continues to be a fundamental issue in gas-particle hydrodynamics and has been found to be significant in risers (O’Brien and Syamlal, 1993; Guenther and Breault, 2007). A lot of effort has been dedicated to the characterisation and quantification of clusters (Lackermeier et al., 2001; Cocco et al., 2010; Shaffer et al., 2013). However, discussions about how to scale cluster phenomena is still lacking.

The objective of this paper is to obtain an appropriate scaling method to scale down the vast number of solid particles in large systems. We employ the CFD-DEM code (OpenFOAM-CFDEM-LIGGGHTS) to obtain computational data of a pseudo-2D riser reactor, and perform a full comparison with experimental data (Carlos Varas et al., 2017) to study the hydrodynamics and associated cluster characteristics. The axial and horizontal profiles of time-averaged solids volume fraction and solids mass flux are analyzed to study the hydrodynamic behavior. The solids holdup and spatial velocity distribution of clusters are discussed to study the cluster phenomena. The present paper is organized as follows: in Section 2 we present the mathematical model, collision model and scaling method. The fluid and particle parameters and mapping conditions are described in Section 3. In Section 4, the scaled results are presented. Finally, the main conclusions are summarized in Section 5.

2. Numerical methodology

2.1. Description of the gas phase

The gas flow is modeled by the volume-averaged Navier-Stokes equations:

\[
\frac{\partial (\rho u_g)}{\partial t} + \nabla \cdot (\rho u_g u_g) = 0
\]  

(1)

\[
\frac{\partial (\rho u_g u_g)}{\partial t} + \nabla \cdot (\rho u_g u_g u_g) = -\nabla \rho P_g - \nabla \cdot (\mu \nabla u_g) - S_g + \varepsilon \rho u_g g
\]  

(2)
where the source term \( S_p \) is defined as:

\[
S_p = \frac{1}{\Delta V} \sum_{i=0}^{N_p} \frac{\beta V_p}{1 - \bar{V}_d}(\mathbf{u}_p - \mathbf{v}_p) \delta(\mathbf{r} - \mathbf{r}_p) dV \tag{3}
\]

where \( \Delta V \) is the volume of the computational cell. The distribution-function \( \delta \) distributes the reaction force of the particles exerted on the gas phase to the velocity nodes on the Eulerian grid. To calculate the interphase momentum exchange coefficient \( \beta \) we employed the Beestra drag model (Beestra et al., 2007):

\[
F_{drag} = \frac{\beta d_p^2}{\mu} = 10 \frac{1 - \varepsilon}{\varepsilon^2} + \varepsilon^2 \left( 1 + 1.5 \sqrt{1 - \varepsilon} \right) + \frac{0.413 \Re_p}{24\varepsilon^2} \left[ \frac{1}{1 + \Re_p^{(1-\varepsilon)}} + 8.4 \Re_p^{0.343} \right]
\]

with:

\[
\Re_p = \frac{\varepsilon \rho_v |\mathbf{u}_p - \mathbf{v}_p| d_p}{\mu g}
\]

where \( \Re_p \) is the particle Reynolds number.

2.2. Description of the particle phase

The soft-sphere Discrete Particle Model (DPM) used in this work was firstly proposed by Cundall and Strack (1979) and first employed in a gas-fluidized system by Tsuji et al. (1993). The particle behavior is governed by Newtonian equations of motion:

\[
m_p \frac{d \mathbf{v}_p}{dt} = -V_p \nabla P + \frac{\beta V_p}{1 - \bar{V}_d} (\mathbf{u}_p - \mathbf{v}_p) + m_p g + \sum_{j \neq i} (F_{nij} + F_{tij}) \tag{6}
\]

\[
l_p \frac{d \theta_p}{dt} = \tau_p \tag{7}
\]

where \( \tau_p \) represents the torque and \( I_p \) the moment of inertia.

2.3. Collision parameters

Particles interact with their nearest neighbors via contact forces. The particle collisional forces are computed by means of the linear spring-dashpot soft sphere model (Cundall and Strack, 1979). To solve Eqs. (6) and (7), five parameters need to be specified: normal and tangential spring stiffness \( k_n \) and \( k_t \), normal and tangential damping coefficient \( \eta_n \) and \( \eta_t \), and the friction coefficient \( \mu_p \). In this model, the normal component of the contact force between two particles at \( i \) and \( j \) can be calculated by

\[
F_{g,n} = -k_n \delta_n n_{ij} - \eta_n \dot{n}_{ij} \tag{8}
\]

By combining this force expression with Newton’s equation of motion we obtain:

\[
m_{eff} \frac{d^2 \delta}{dt^2} = -k_n \delta_n - \eta_n \frac{d \delta}{dt} \tag{9}
\]

where \( k_n \) is the normal spring stiffness, \( \eta_n \) the normal damping coefficient, \( \delta_n \) the overlap of the two particles involved in the collision and \( m_{eff} \) the reduced mass of normal linear spring-dashpot system.

The damping coefficient is determined by:

\[
\eta_n = 2 \sqrt{k_n m_{eff}} \tag{10}
\]

A similar procedure can be applied to the tangential spring-dashpot system. So the tangential damping coefficient is defined as:

\[
\eta_t = \frac{-2 \ln \left( m_{eff} k_t \right)}{\sqrt{\pi^2 + \ln^2 \varepsilon_t}} \tag{11}
\]

where \( m_{eff} \) is the reduced mass of the tangential spring-dashpot system. The definitions of these two reduced masses are as follows:

\[
m_{eff} = \frac{m_a m_b}{m_a + m_b} \tag{12}
\]

\[
m_{eff} = 2 \frac{m}{m + \bar{m}} \tag{13}
\]

The relation between the normal and tangential spring stiffness is given by:

\[
k_n = \frac{2 \pi^2 + (\ln \varepsilon_t)^2}{\eta_n} \tag{14}
\]

where \( \varepsilon_n \) and \( \varepsilon_t \) are the normal and tangential coefficient of restitution respectively, which are empirically determined. \( k_n \) is chosen such that the maximum overlap between two colliding particles in the simulation is smaller than 1% of the particle radius.

2.4. Scaling method

In order to reduce the number of discrete particles, the equations of motion and the collisional behavior of the particles will be scaled. The scaling rules presented in this section are chosen such that the particle size can be scaled in order to reduce the number of simulated particles drastically, whilst maintaining the same hydrodynamic behavior. The starting point of the analysis is the force balance of the particles. Here we consider the vertical component (z) of the force balance introduced earlier in Eq. (6).

For the sake of the discussion, we will write it in a compact form, i.e. without complete expression of all forces:

\[
m \frac{d v_z}{dt} = F_{g,z} + F_{p,z} + F_{D,z} + \sum_{j \neq i} (F_{n,zij} + F_{t,zij}) \tag{15}
\]

where the terms on the right hand side respectively represent forces due to gravity, far field pressure, drag and normal and tangential inter-particle contact. For simplicity we will subsequently drop the sub-script \( z \) in the subsequent equations.

The combined gravity and far field pressure forces can be described by:

\[
F_g + F_p = V_g (\rho_s - \rho_f) \tag{16}
\]

Eq. (15) can be re-rewritten by dividing all terms by \( F_g \):

\[
\frac{1}{g} \frac{d v_z}{dt} = \frac{F_g + F_p + F_D}{F_g} + \sum_{j \neq i} \left( \frac{F_{n,zij} + F_{t,zij}}{F_g} \right) \tag{17}
\]

In order to obtain the same hydrodynamic behavior each of the four dimensionless groups on the right hand side should be kept constant. This means that if we change the particle diameter (while keeping the overall solids volume the same), other parameters should be changed in such a manner that the respective dimensionless groups remain constant. We will now consider each of the four terms (groups) indicated in Eq. (17).

2.4.1. The ratio of buoyancy and gravity force (group I)

The first group implies that the ratio between the pressure and gravity forces should remain constant. This ratio can be simplified as:
The contact parameters dimensionless groups spring stiffness and damping coefficient, which are contained in the scales with the particle diameter and that the change in overlap scaling.

4.2. The ratio of drag and gravity force (group II)

The second group implies that the ratio between the drag force and net gravity force should remain constant. This ratio can be expressed in terms of the particle Reynolds number Re, the Archimedes number Ar and the local void fraction ε:

\[
\mu = \frac{\rho_d - \rho_f}{\rho_f} = \frac{\rho_d - \rho_f}{\rho_f} = \frac{\rho_d - \rho_f}{\rho_f}
\]

when Re, ε and N\text{ar} are kept constant, the proper scaling would be obtained.

4.2.3. The ratio of normal contact force and gravity force (group III)

The third group implies that the ratio between the normal contact force and the gravity force remains constant. The normal contact force depends on the particle-particle overlap during contact and can be expressed as a combination of a linear spring-dashpot (Hooke model) (Hoornans et al., 1996):

\[
F_n = \frac{\kappa_n}{\rho_d} R E f(R e, \varepsilon) = \frac{\kappa_n}{\rho_d} f(R e, \varepsilon), \quad \mu = \frac{\gamma_n v_0 d^3}{\rho_d d g}
\]

where \(\gamma_n\) and \(\gamma_n\) represent the normal spring stiffness and damping coefficient, which are contained in the dimensionless groups \(N_{Kn}\) and \(N_{Kn}^{*}\). It is desirable that the overlap scales with the particle diameter and that the change in overlap scales with the impact velocity. These two criteria imply that the dimensionless groups \(N_{Kn}\) and \(N_{Kn}^{*}\) should be kept constant in the scaling.

4.2.4. The ratio of normal and tangential contact forces (group IV)

The fourth group implies that the ratio between the normal and tangential contact forces remains constant. This depends on the applied contact model, but is usually guaranteed, as both forces depend on the particle diameter and mass in the same way.

4.2.5. Scaling rules

The following dimensionless group should be kept constant when scaling the particle diameter:

\[
\mu, \quad Re = \frac{\mu \rho_d u_{ref} d}{\mu}, \quad Ar = \frac{d^3 (\rho_d - \rho_f) \rho_d g}{\mu^2}, \quad N_{Kn} = \frac{k_n}{\rho_d d g}, \quad N_{Kn}^{*} = \frac{\gamma_n v_0}{\rho_d d g}
\]

where \(\mu\) and \(\mu\) represent the normal and tangential contact forces, respectively.

4.3. Simulation conditions

3.1. Simulation setup

To test the derived scaling rules, a pseudo-2D riser reactor with dimensions 1570 \times 70 \times 6 mm (height \times width \times depth) was simulated in our study. This riser was defined to serve as the base case for the series of CFD-DEM simulations. In the riser reactor (see Fig. 1a), solid particles and gas flow co-currently upwards to the top of the riser, where the gas and particles are separated: gas will leave the system whilst particles are fed back into the riser from the bottom inlet region. Note that during insertion particle overlap was not accepted. No-slip boundary conditions were applied at all vertical walls.

The particles are initially placed in a random position in the bottom section of the riser up to a height of 0.25 m. In the base case, there are 50,000 particles positioned in this domain, where the gas superficial velocity was set at 5.55 m/s for the base case. This is high enough to ensure proper particle circulation in the CFB system, yet it is low enough to produce clusters. The (unresolved) gas-solid interactions were represented by the Beetsstra drag force correlation (Beetsstra et al., 2007), whereas the collision parameters correspond to properties of glass beads previously reported by Hoornans et al. (1996). Further details of simulation settings in the base case are specified in Table 1 (column K = 1).

3.2. Simulation settings

To test the scaling, variations were made with respect to the base case, using three different scaling factors, i.e. \(K = 0.8, 1.25\) and 2. The scaled parameters were scaled according to Eq. (24) and are listed in Table 1 along with the other key parameters.

A very important aspect to realize when scaling the particle size, is that the ratio between the particle size and other important spatial scales changes. In particular this involves the ratio of volumes of the computational grid cell and the particle (AV/\(V_p\), see Eq. (3)) and the ratio of the shallow depth of the riser.
and the particle diameter ($d_p/D$). To check the importance of these two parameters, four scenarios of the scaled simulations were considered:

- Condition a, base case, i.e. no scaling of grid size or riser depth,
- Condition b, scaling of grid size and no scaling of riser depth,
- Condition c, no scaling of grid size, but scaling of riser depth,
- Condition d, scaling both grid size and riser depth.

In Fig. 1b and c the top views of the base case (Condition a) and cases of scaled riser depth (Conditions c and d) are shown respectively. The scaled diameter is $D_1 = K D_0$, where $D_0$ is the (unscaled) experimental riser depth. The scaled mapping parameters are specified in Table 2. Considering the fact that the grid number must be integer, in conditions b and d the volume ratio $\Delta V/V_p$ is kept almost constant. For all simulations, the last 10 s of the total 20 s were post-processed to obtain the time-averaged data. The computational cost was found to scale roughly linearly with the number of particles and computational cells.

### 4. Results and discussion

In this section the simulation results are presented, where first the overall flow patterns will be discussed. Subsequently, the solids volume fraction and mass flux distributions will be analyzed respectively, by making a comparison between simulation results and experimental data from literature (Carlos Varas et al., 2017). Finally, cluster characteristics will be discussed.

#### 4.1. Overall flow patterns

In this sub-section a group of full-field configurations will be shown to study the gas-particle flow behavior. Fig. 2 shows snap-
Table 2
Ratio in four different mapping cases. The parameters that are kept (almost) constant are indicated in boldface.

<table>
<thead>
<tr>
<th></th>
<th>K = 0.8</th>
<th>K = 1</th>
<th>K = 1.25</th>
<th>K = 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>ΔV/V_p</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>a</td>
<td>57.0</td>
<td>29.2</td>
<td>14.9</td>
<td>3.6</td>
</tr>
<tr>
<td>b</td>
<td>30.4</td>
<td>29.2</td>
<td>29.9</td>
<td>29.2</td>
</tr>
<tr>
<td>c</td>
<td>60.8</td>
<td>29.2</td>
<td>14.9</td>
<td>3.6</td>
</tr>
<tr>
<td>d</td>
<td>29.2</td>
<td>29.2</td>
<td>29.9</td>
<td>29.2</td>
</tr>
<tr>
<td>D/d_p</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>a</td>
<td>8.82</td>
<td>7.06</td>
<td>5.65</td>
<td>3.53</td>
</tr>
<tr>
<td>b</td>
<td>8.82</td>
<td>7.06</td>
<td>5.65</td>
<td>3.53</td>
</tr>
<tr>
<td>c</td>
<td>7.06</td>
<td>7.06</td>
<td>7.06</td>
<td>7.06</td>
</tr>
<tr>
<td>d</td>
<td>7.06</td>
<td>7.06</td>
<td>7.06</td>
<td>7.06</td>
</tr>
<tr>
<td>N_p (N_cell)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>a</td>
<td>97 656 (67 200)</td>
<td>50 000 (67 200)</td>
<td>25 600 (67 200)</td>
<td>6 250 (67 200)</td>
</tr>
<tr>
<td>b</td>
<td>97 656 (126 000)</td>
<td>50 000 (67 200)</td>
<td>25 600 (33 600)</td>
<td>6 250 (8 400)</td>
</tr>
<tr>
<td>c</td>
<td>78 125 (50 400)</td>
<td>50 000 (67 200)</td>
<td>32 000 (84 000)</td>
<td>12 500 (134 400)</td>
</tr>
<tr>
<td>d</td>
<td>78 125 (105 000)</td>
<td>50 000 (67 200)</td>
<td>32 000 (42 000)</td>
<td>12 500 (16 800)</td>
</tr>
</tbody>
</table>

Fig. 2. Instantaneous gas-solid flow pattern: Left to right: K = 0.8, K = 1, K = 1.25, K = 2.

Fig. 3. Snapshots of particle velocity of the middle region (dashes region in Fig. 2). Left to right: K = 0.8, K = 1, K = 1.25, K = 2.
shots of gas void fraction while applying four scaling factors to the base case (Condition a). It is noted that in the vertical direction there is similar tendency of gas volume fraction revealing a dense region close to the riser inlet and more dilute middle and top regions. This distribution can be explained by the nature of particle motion in circulating fluidized beds; particles are fed to the system at the bottom of the riser, resulting in more collisions between particles and consequently more particles residing in this region. In the horizontal direction, a so called “core-annulus” flow pattern prevails, which has a dilute core and dense annular regions. In the dense region particle clusters are observed with complex mutual interactions.

Fig. 4. Axial profiles of solids holdup under four scaling conditions.

Fig. 5. Cross-sectional profiles of solids volume fraction under different scaling condition. H = 0.3 m.
Fig. 3 shows a zoomed in section of the riser, which shows the cluster behavior in more detail. Clusters are formed in all four scaling scenarios, and appear to exhibit the same motion, moving downwards along the wall and upwards in the core region. When increasing the scaling factor $K$, it is noted that there are fewer particles with a larger diameter in the riser, which matches the definition of the scaling method. Clusters could be observed for all scaling scenarios but there appear to be less and smaller clusters with higher values of the scaling factors.

4.2. Time-averaged solid volume fraction distribution

The axial profiles of time-averaged solids volume fraction are shown in Fig. 4. Every figure contains experimental data (Carlos Varas et al., 2017) and four sets of simulation results representing different scaling approaches. It is noted that both experimental and simulation results show the same tendency in the sense that dense regions exist in the bottom of the riser while the system becomes more dilute with increasing axial direction. This matches the observations from Fig. 2. In Fig. 4, each sub-figure represents a mapping condition. Even though they all maintain the same range and tendency, sub-figures (b) and (d) show a better agreement with experiment than (a) and (c). The grid cells cannot be too large because else the representation of the flow field is not captured and secondly the assumption of a uniform porosity does not hold. The grid cells cannot be too small either because the drag force correlations assume a porosity field and not discrete values of the two phases. Both case a, as well as case c show very similar behavior and difference when varying the scaling factor $K$. This suggest that the poor comparison of both cases with experiments is mostly related to their changes is grid size, and secondly, that the differences in case c cannot be attributed to the rise depth. $K = 0.8$ seems to only show satisfactory results under condition b. In both condition a and condition c this is related to the fact that the grid mapping is not consistent and the grid volume to particle volume ratio is different amongst the different scaling factors. In condition d this could be related to the particle wall conditions, which are altered by the riser depth. Which underlying mechanism is acting there is object of further study.

In Figs. 5 and 6, the cross-sectional profiles of solids volume fraction are displayed. The axial positions are 0.3 m and 1.2 m respectively in each figure, and each figure contains four sub-figures, which represents four different mapping conditions. In each sub-figure four simulation results using different scaling factors are compared with the experimental data. It can be seen that all simulation results show the same tendency as the experimental data. In the horizontal direction, the typical U-shaped profile is obtained. In the axial direction, the solids holdup decreases with increasing heights, which matches what we observed in Fig. 4. The U-shaped profiles possess a flatter solids holdup profiles at higher positions in the riser.

The profiles of solids holdup in the dense (bottom) region are shown in Fig. 5. All the mapping conditions are acceptable except condition c which is scaling riser depth while keeping the grid size constant. While the volume ratio of grid size and particle is different, the ratio of riser depth and particle diameter is constant. As the fact that the riser is a pseudo-2D bed with no-slip boundary condition for the front/back walls, when scaling the
Fig. 7. Cross-sectional profiles of solids mass flux under four scaling factor. $H = 0.3$ m.

Fig. 8. Cross-sectional profiles of solids mass flux under four scaling factor. $H = 1.2$ m.
riser depth, the bridge formation of gas-solid hydrodynamics would be changed. Furthermore conditions b and d are suitable for all cases which applied the closed scaling up of the particle diameters.

In Fig. 6 the solids holdup profiles in the dilute (top) region are displayed. It can be seen that at axial position \( H = 1.2 \) m the U-shaped profiles possess an asymmetric distribution for conditions b and d, which are cases having (almost) constant ratio of computational cell and particle volume. This is due to the fact that the curved one-sided lateral solids outlet has an apparent effect on the solids volume fraction in these two conditions.

4.3. Time-averaged solids mass flux profiles

The solids mass flux constitutes another important quantity and is computed for each computational cell from:

\[
G_s(t) = \rho_s \langle \phi(t) u_s(t) \rangle_{cell}
\]

where the product of the local solids volume fraction \( \phi = 1 - \varepsilon \) and solids velocity is time-averaged.

Figs. 7 and 8 show the simulated and experimental profiles of the solids mass flux at two axial positions \( 0.3 \) m and \( 1.2 \) m. The organization of these figures is the same as in Section 4.2. The experimental data and simulation results exhibit the same tendency of solids up-flow in the core region of the riser and a relatively high down-flow close to the riser walls. At higher axial coordinates, the solids mass flux distribution becomes relatively flat.

By comparing the numerical results for different scaling factors, it is observed that scaling conditions b and d produce a better performance than the other two approaches. In Fig. 8, the profiles of solids mass flux for \( K = 2 \) are shown. The distribution of solid mass flux for \( K = 2 \) becomes more uniform than the other cases.

4.4. Cluster profiles

As previously mentioned, constant solids holdup thresholds are employed along the entire region of the pseudo-2D riser in order to detect and classify clusters. In this way, it is ensured that a uniform definition of clusters is used and that the quantified cluster-related properties are not influenced by a changing definition along the axial and cross-sectional directions of the riser. Clusters are defined as connected regions with local solids volume fractions larger than 0.2 everywhere that have a minimum area of \( 60 \) mm\(^2\) and a dense core with at least one grid with \( \phi_c > 0.4 \). In this work the detection of clusters was performed by a Matlab script. Fig. 9 displays the snapshots of particle velocity obtained from four scaling factor scenarios using mapping condition b. Clusters can be observed, with dense cores formed close to the wall and big dilute strands of particles that tend to move upwards. Comparing with Fig. 3, especially in snapshot of \( K = 2 \) the cluster phenomena becomes more apparent in Fig. 9. The represented clusters could be better captured when applying the scaling approach condition b than condition a.

To quantitatively study the cluster characteristics, profiles of cluster solids holdup are shown in Fig. 10. The mean solids volume fraction of each cluster is computed for the corresponding cells which are occupied by the identified cluster. Therefore the averaged cluster internal solids holdup is calculated from the following expression:

\[
\phi_{\text{cluster}} = \frac{\sum \phi_s}{\sum \text{N}_{\text{cell occupied}}}
\]

It is noted that for all the cases using different scaling factors and mapping conditions, the solids holdups of clusters are in a similar range with more clusters close to the walls whilst the clusters are sparsely distributed in the core region. This reveals that our scaling method captures the cluster phenomena quite well. It can also be seen that a higher scaling factor leads to less clusters in the whole system. Whilst the cluster numbers do not show an obvious difference among the applied scaling factor approaches in conditions b and d and the core-annulus distribution is also well captured. This indicates that the volume ratio of grid and particle has a significant influence on cluster solids holdup profiles.

In Fig. 11, profiles of cluster velocity are plotted for different scaling factors and mapping conditions. The typical symmetric core-annulus distribution is seen here, with mostly downwards flowing clusters near the wall whilst clusters in the core region are mostly moving upwards. This also matches with the observa-
tions from Fig. 9. Using different scaling factors, the distribution of cluster velocity is quite similar and in the annulus region there are more clusters moving downwards except for the cases $K = 2$. In the set of $K = 2$ cases, the velocity distribution of clusters becomes more symmetric near the walls. In conditions b and d, the cluster number is more uniform in comparison with other conditions, so is the cluster velocity. Hence for further studies on cluster analyses, keeping the volume ratio constant would be advisable.

<table>
<thead>
<tr>
<th>$K$ = 0.8</th>
<th>$K$ = 1</th>
<th>$K$ = 1.25</th>
<th>$K$ = 2</th>
</tr>
</thead>
</table>
| ![Cluster solids holdup vs centroid locations under different scaling factors and mapping conditions](image)

Fig. 10. Cluster solids holdup vs centroid locations under different scaling factors and mapping conditions.
5. Conclusions and recommendations

In this work, a scaling method is developed and validated in detail by performing extensive simulations of a pseudo-2D circulating fluidized bed riser. To maintain the same hydrodynamic behavior seven gas and particle properties were scaled, such as the particle diameter, the gas viscosity, gravity, normal spring stiffness etc. Besides scaling the gas and particle properties, the grid size and riser depth are also scaled by considering four scaled mapping conditions. The influence of the scaling method, the scaled grid size and riser depth on the fluidized riser hydrodynamics has been quantified.

The experimental and simulated solids volume fraction and mass flux profiles provide quantitative information about the performance of the scaling method on gas-particle flow behavior. Firstly, it is noted that the scaling method could well capture the

![Cluster velocity under different scaling factors and mapping conditions.](image)

Fig. 11. Cluster velocity under different scaling factors and mapping conditions.
Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgement

This work was supported by the Netherlands Center for Multiscale Catalytic Energy Conversion (MCEC), an NWO Gravitation program funded by the Ministry of Education, Culture and Science of the government of the Netherlands.

References


