Numerical modeling of bubbly flow in square column

Giorgio Besagni, Gaël Guédon, Fabio Inzoli

giorgio.besagni@polimi.it
Department of Energy - Politecnico di Milano
www.cfdlab.polimi.it
Outline

Part 1. Introduction

Part 2. Case study
1. Benchmark
2. CFD model settings

Part 3. Numerical investigation
1. Drag force
2. Lift force
3. Turbulent dispersion
4. Wall force
5. Virtual mass

Part 4. Conclusion
The Eulerian-Eulerian framework and its issues

Numerical simulations are used for studying two phase flow reactors and two approaches are mostly employed: the Euler–Lagrange and the Euler–Euler.

The Euler-Euler approach describes the motion of the two-phase mixture in a macroscopic sense and is preferred for industrial applications. This approach has three main issues*:

1. interphases forces modeling
2. turbulence modeling
3. bubble diameter**

Each model is suitable for a certain application, operating condition and flow pattern and has to be calibrated and validated using experimental data. There is not a unique model for every flow condition.

A CMFD model should be calibrated over literature benchmarks before being used to study a real reactor or a full scale facility.

This work is focused on the Eulerian-Eulerian approach and deals with the interfacial closures.

---


Part 2. Case study

1. Benchmark

The Deen et al.* experimental set-up consists of a square column filled with distilled water. A distributor plate composed of 49 holes with diameter of 1 mm was placed in the middle of the base of the column. The flow is dominated by the energetic, large-scale structures in the core of the flow, with wall effects having a smaller impact on the overall flow field. The experimental data consist in PIV measurements of liquid and gas axial velocity (mean and fluctuations) at different:

- **Column heights**
  \[ y/W = 2.17, 1.67 \text{ and } 0.83 \text{ at } z/W = 1/2 \]

- **Transversal positions:**
  \[ z/W = 1/2, 1/4, 1/8 \text{ and } 1/16 \text{ at } y/W = 2.17 \]

1. CFD model: settings

In this study, the commercial code ANSYS Fluent Release 14.5.7 has been used. Simulations are carried out with a 3D pressure-based solver using a second order implicit unsteady formulation.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure-Velocity coupling</td>
<td>Phase coupled SIMPLE</td>
</tr>
<tr>
<td>Mesh</td>
<td>10125 cells 10 x 10 x 10 mm³</td>
</tr>
<tr>
<td>Numerical settings</td>
<td>Temporal discret. Second order Euler implicit.</td>
</tr>
<tr>
<td></td>
<td>The gradients. The least squares cell based method.</td>
</tr>
<tr>
<td></td>
<td>Advection terms. A third order accurate MUSCL scheme.</td>
</tr>
<tr>
<td>Milelli condition</td>
<td>Satisfied</td>
</tr>
<tr>
<td>(d_{\text{bubble}})</td>
<td>4 mm</td>
</tr>
<tr>
<td>Fluid settings</td>
<td>Despite the air phase has a slightly varying density from the bottom to the top of the column, both fluids are considered as incompressible</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>(k-\varepsilon) Standard Per Phase</td>
</tr>
<tr>
<td>Outlet</td>
<td>Degassing boundary condition</td>
</tr>
<tr>
<td>(y^+_{\text{first node, wall}})</td>
<td>120 – Standard Wall functions</td>
</tr>
<tr>
<td>Time step</td>
<td>0.005 s (sensitivity analysis on four time step). An estimate of the time step size can be obtained via the CFL number.</td>
</tr>
</tbody>
</table>

\[ v_{g,in} = v_{g,s} WD / \alpha_g A_{in} \]

A superficial gas velocity of 0.0049 m/s leads to \(v_{g,in} = 0.12\) m/s for the employed grid spacing.
2. CFD model: closures investigated

\[ \frac{\partial}{\partial t} (\alpha_k \rho_k \vec{u}_k) + \nabla \cdot \alpha_k < \rho_k \vec{u}_k \vec{u}_k >_k = -\alpha_k \nabla P_k + \nabla (\alpha_k \tau_k) + \alpha_k \rho_k g + M_{I,K} \]

M_{I,L} = -M_{I,G} = M_{D,L} + M_{L,L} + M_{VM,L} + M_{TD,L} + M_{WF,L}

a) Drag law. 8 formulation
   Tomiyama, Grace, Morsi and Alexander, Schiller and Naumann, Symmetric, Grevskot et al., Universal drag law and \( C_D = 0.44 \).

b) Lift law. 7 formulation + Lift/Drag interaction
   Tomiyama, Moraga et al., Saffman Mei, Legendre-Magnaudet models and \( C_L = 0.1, 0.2 \) and 0.5.

c) Wall lubrication force. 4 formulations
   Antal et al., Tomiyama, Hosokawa and Frank et al. models.

d) Turbulent dispersion force. 9 formulations
   Lopez de Bertodano, Simonin and Burtns et al. models.
   Each model is implemented with three different coefficients: \( C_{TD} = 0.1, 0.2 \) and 0.5.

e) Virtual mass.
   Implemented with a coefficient of \( C_{VM} = 0.5 \).

- The effect of the drag force on the flow pattern is higher than other closures.
- When the drag is used as the only interfacial force, simulation cannot predict flow phenomena.
- The addition of a lift law enhances prediction capability.
The axial velocity of the liquid phase can be qualitatively predicted, the axial gas velocity is always overpredicted. This may come from the fact that all the drag models used predict lower drag forces.

It seems that the lift force stabilise the flow: bubbles are more equally distributed over the column cross-section. This is due to the stabilising effect of a positive sign lift force.

**Mean axial velocity – Different column height**

- **Tomiyama**
  - Drag law
  - $y = 0.35\,\text{m}$
  - $z = 0.075\,\text{m}$

- **Moraga et al.**
- **Saffman and Mei**
- **Legendre-Magnaudet**
- **Deen (2000)**

*Axial liquid velocity [m/s] – x/D [-]*

- **Grace**
  - Drag law
  - $y = 0.35\,\text{m}$
  - $z = 0.075\,\text{m}$

- **Grace**
  - $y = 0.25\,\text{m}$
  - $z = 0.075\,\text{m}$

- **Grace**
  - $y = 0.125\,\text{m}$
  - $z = 0.075\,\text{m}$
The results improve also for the lower sections of the column.

The application of a turbulent dispersion model enhances the prediction also at the lower sections. For the case of the Lopez de Bertodano + $C_{TD} = 0.5$ model, the results were non-physical. For the case of the other models with $C_{TD} = 0.5$, axial velocity was greately under predicted.
The wall lubrication force enhances the predictions of the non-symmetric axial velocity profiles. The Antal and the Frank model perform better than the other models. The model of Hosokawa predicts a non-symmetric axial velocity profile opposite than the others. Compared to the other models, the Tomiyama model causes an overestimation of the axial liquid velocity profiles in the bottom and middle part of the column with respect to the case with turbulent dispersion force only.
The wall lubrication force enhances the predictions of the non-symmetric axial velocity profiles.

The non-symmetric velocity profile predicted is due to the prediction of liquid recirculation in the column. This may be due to the periodic bubble plume interaction (repulsion) with the walls.

The results are very similar to the case without wall force...

1. We have used a coarse mesh.
2. The constants of these models were developed and validated for bubbly flow in vertical pipe. A calibration for the case of bubble columns should be considered.
Part 3e. Virtual mass

- The addition of the virtual mass force leads to minor effects for the case of drag law only. The acceleration and deceleration of the liquid is restricted to small end regions of the column.
- For the case of lift law and wall force, the virtual mass effect is significant. Axial velocity profiles increase and become very similar in both cases. The effect of the virtual mass overwhelm the one of the other non-drag forces.
- The case of virtual mass with turbulent dispersion force is not presented because of convergence problems.
Conclusions

- The numerical results have been compared with experimental data from the literature.
- The nature of the drag coefficient is found to have a significant effect on the global hydrodynamics.
- The inclusion of a lift force is found to be necessary to obtain a local axial velocity distribution that is consistent with the experimental measurements.
- The inclusion of a turbulent dispersion force is found to improve the modelling accuracy.
- The wall lubrication force enhances the predictions of the recirculation pattern.
- The virtual mass force is also discussed.

Ongoing simulations

- Testing other interfacial closures
- Testing turbulence model closures

Future developments

- 1. Calibration of wall forces
- 2. Population balance model
- 3. LES implementation in Fluent
Thank you for the attention!