Simulation of Disperse Particle-Laden Gas Flows with with OpenFOAM and ANSYS FLUENT

10th September 2015
Motivation

Simulations for engineering application in processing plant:

- Analysis of flow structure in static and dynamic air classifier
- CFD simulation of particle-laden flow considering particle-fluid and particle-particle interaction
- Derive feasible simplifications of CFD model

→ Validation of models with simple test cases
→ Comparison of CFD packages
Model equations

Averaged conservation equations for incompressible, isothermal and turbulent flow of Newtonian fluid:

\[ \nabla \cdot \bar{u} = 0 \quad (1) \]

\[ \frac{\partial \bar{u}}{\partial t} + \rho (\bar{u} \cdot \nabla) \bar{u} = -\nabla \bar{p} + \eta \Delta \bar{u} - \nabla \cdot \tau^{RS} + \bar{f}_W \quad (2) \]

with:

\[ \tau^{RS} = \left( \rho \bar{u}' \bar{u}' \right) \quad (3) \]

Force balance in Lagrangian framework:

\[ \frac{dx_P}{dt} = u_P \quad (4) \]

\[ m_P \frac{du_P}{dt} = F_D + F_B + F_G \quad (5) \]
Model equations

Drag force:

\[ \overrightarrow{F_D} = \frac{3}{4} \rho \frac{m_P}{d_P} \cdot C_D (u - u_P) |u - u_P| \]  

(6)

Implementation of drag coefficient \( C_D \) in CFD programs:

**ANSYS FLUENT**

Morsi and Alexander\(^1\)

\[ C_D = a_1 + \frac{a_2}{R_{e_P}} + \frac{a_3}{R_{e_P}^2} \]  

(7)

**OpenFOAM**

Empirical relation

\[ C_D = \begin{cases} \frac{24}{R_{e_P}} \left(1 + \frac{1}{6} R_{e_P}^{2/3}\right) & ; R_{e_P} \leq 1000 \\ 0.424 & ; R_{e_P} \geq 1000 \end{cases} \]  

(8)

\(^1\)Morsi1972.
Model equations

Coupling

Classification of coupling schemes and interaction between disperse and continuous phase according to Elghobashi\(^2\):

\[
\begin{align*}
\text{①: One-way coupling } & \bar{f}_W = 0 \\
\text{②: Two-way coupling, particles enhance turbulence production} \\
\text{③: Two-way coupling, particles enhance turbulence dissipation} \\
\text{④: Four-way coupling}
\end{align*}
\]

\(^2\)Elghobashi1994.
Test case backward facing step

Description

Geometry according to Fessler and Eaton

<table>
<thead>
<tr>
<th>Channel flow</th>
<th>BFS flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Channel height $h$</td>
<td>40 mm</td>
</tr>
<tr>
<td>Channel width $w$</td>
<td>457 mm</td>
</tr>
<tr>
<td>Step height $H$</td>
<td>26.7 mm</td>
</tr>
<tr>
<td>$w/H$</td>
<td>17:1</td>
</tr>
<tr>
<td>$h/H$</td>
<td>5:3</td>
</tr>
</tbody>
</table>

$^{3}$Fessler1999.
Test case backward facing step

Applied meshes

<table>
<thead>
<tr>
<th>Name</th>
<th>Dimension</th>
<th>Cells in z-direction</th>
<th>BC front and back</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>2D</td>
<td>0</td>
<td>-</td>
</tr>
<tr>
<td>B</td>
<td>3D</td>
<td>1</td>
<td>empty</td>
</tr>
<tr>
<td>C</td>
<td>3D</td>
<td>1</td>
<td>symmetry</td>
</tr>
<tr>
<td>D</td>
<td>3D</td>
<td>10</td>
<td>symmetry</td>
</tr>
</tbody>
</table>
Test case backward facing step

Setup

- RANS and URANS
- $k-\omega$-SST turbulence model
- isothermal
- 1- and 2-way coupling

<table>
<thead>
<tr>
<th>Continuous phase</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong> profile with $\bar{u}_{avg} = (9.39 \ 0 \ 0) \ m/s$</td>
</tr>
<tr>
<td>$U_{x,0} = 10.5 \ m/s$</td>
</tr>
<tr>
<td>$k = 0.45 \ m^2/s^2$</td>
</tr>
<tr>
<td>$\omega = 2800 \ 1/s$</td>
</tr>
<tr>
<td><strong>Outlet</strong> $\bar{p} = 0 \ Pa$</td>
</tr>
<tr>
<td><strong>Wall</strong> $\bar{u} = (0 \ 0 \ 0) \ m/s$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Disperse phase</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Particle diameter</strong> $d_P = 70 \ \mu m$</td>
</tr>
<tr>
<td><strong>Particle density</strong> $\rho_P = 8800 \ kg/m^3$</td>
</tr>
<tr>
<td><strong>Particle mass flow rate</strong> $\dot{m}_P = 1.58 \cdot 10^{-5} \ kg/s$</td>
</tr>
<tr>
<td><strong>Injection velocity</strong> $\bar{u}_{P,avg} = (10.5 \ 0 \ 0)^T \ m/s$</td>
</tr>
</tbody>
</table>
Test case backward facing step

Fluid flow

Fluent OpenFOAM Experiment

\[ \frac{2u_x}{U_0} + \frac{x}{H} \]

\[ \frac{10u'}{U_0} + \frac{x}{H} \]
Test case backward facing step

Disperse phase

1-way coupling

2-way coupling

2 \cdot \frac{u_p, x}{U_0} + \frac{x}{H}

AF Mesh C
AF Mesh A
OF Mesh C
OF Mesh B
Experiment

10th September 2015

Greifzu, Schwarze

IMFD
TU Bergakademie Freiberg
Test case backward facing step

Disperse phase

1-way coupling

2-way coupling

Greifzu, Schwarze

10th September 2015
Test case confined bluff body

Description

Geometry according to Borée et al. 4

4Borée1999.
Test case confined bluff body

Setup

- URANS
- Standard $k$-$\epsilon$ turbulence model
- isothermal, incompressible
- 2-way coupling

<table>
<thead>
<tr>
<th>Continuous phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Central jet inflow</td>
</tr>
<tr>
<td>$\bar{u}_{j,\text{avg}} = (0\ 0\ 3.01)\ m/s$</td>
</tr>
<tr>
<td>$U_{z,\text{i}} = 3.4\ m/s$</td>
</tr>
<tr>
<td>Annular jet inflow</td>
</tr>
<tr>
<td>$\bar{u}_{a,\text{avg}} = (0\ 0\ 5.36)\ m/s$</td>
</tr>
<tr>
<td>Outlet</td>
</tr>
<tr>
<td>$\bar{p} = 0\ Pa$</td>
</tr>
<tr>
<td>Wall</td>
</tr>
<tr>
<td>$\bar{u} = (0\ 0\ 0)\ m/s$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Disperse phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Particle diameter</td>
</tr>
<tr>
<td>$d_P = 63\ \mu m$</td>
</tr>
<tr>
<td>Particle density</td>
</tr>
<tr>
<td>$\varrho_P = 2470\ kg/m^3$</td>
</tr>
<tr>
<td>Particle mass flow rate</td>
</tr>
<tr>
<td>$m_P = 2.78 \cdot 10^{-4}\ kg/s$</td>
</tr>
<tr>
<td>Injection velocity</td>
</tr>
<tr>
<td>$u_{P,\text{avg}} = (4.08\ 0\ 0)\ m/s$</td>
</tr>
</tbody>
</table>
Test case backward facing step

Fluid flow

![Graph showing fluid flow results](image-url)

**Graph Explanation:**
- The graph compares fluid flow data from ANSYS FLUENT, OpenFOAM, and Experiment.
- The horizontal axis represents the dimensionless parameter $z/R_j$.
- The vertical axis represents the dimensionless parameter $u_z/U_j$.

**Analysis:**
- The graph illustrates the comparison of simulated and experimental results for fluid flow through a backward facing step.
- The data shows how well the simulation models match the experimental observations.

**Data Points:**
- The graph includes data points for different time steps or conditions.
- The legend indicates the sources of the data: ANSYS FLUENT, OpenFOAM, and Experiment.

**Conclusion:**
- The data suggests a good agreement between the simulation models and the experimental data, indicating accurate modeling of the fluid flow.

**Further Reading:**
- For more detailed analysis and conclusions, refer to the full report or associated technical documents.

**References:**
- [Report Title](report-url)
- [Updated by: Greifzu, Schwarze]
Test case backward facing step

Disperse phase

\[ 3 \cdot \frac{u_p,z}{U_{z,j}} + \frac{z}{R_j} \]
Conclusion

- Good agreement of numerically modeled flow fields
- Disperse phase simulation results fit experimental data - differences in shear layers
  - Particle dispersion underpredicted in ANSYS FLUENT
  - Particle dispersion overpredicted in OpenFOAM
- Minor influence of coupling scheme due to low volume loadings
- Incorrect computation of particle motion in ANSYS FLUENT 2-D model: neglect of three-dimensional character of turbulence
Thanks for your Attention.