TECHNISCHE UNIVERSITÄT
BERGAKADEMIE FREIBERG
The University of Resources. Since 1765.

Franziska Greifzu, Rüdiger Schwarze
Institute of Mechanics and Fluid Dynamics
Chair for Numerical Modeling of Granular Flows
Lampadiusstraße 4
09599 Freiberg

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

10th September 2015

Halle (Saale)

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

$\rightarrow$ Validation of models with simple test cases
$\rightarrow$ Comparison of CFD packages

Model equations

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

$$\nabla \cdot \underline{\bar{u}} = 0 \tag{1}$$

$$\frac{\partial \underline{\bar{u}}}{\partial t} + \varrho \left(\underline{\bar{u}} \cdot \nabla \right) \underline{\bar{u}} = -\nabla \bar{p} + \eta \Delta \underline{\bar{u}} - \nabla \cdot \underline{\underline{\tau}}^{\mathrm{RS}} + \underline{\bar{f}}_W \tag{2}$$

$$\text{with: } \underline{\underline{\tau}}^{\mathrm{RS}} = \left(\varrho \, \overline{\underline{u}' \underline{u}'} \right) \tag{3}$$

Force balance in Lagrangian framework:

$$\frac{d\underline{x}_{\mathrm{P}}}{dt} = \underline{u}_{\mathrm{P}} \tag{4}$$

$$m_{\mathrm{P}} \frac{d\underline{u}_{\mathrm{P}}}{dt} = \underline{F}_{\mathrm{D}} + \underline{F}_{\mathrm{B}} + \underline{F}_{\mathrm{G}} \tag{5}$$

Model equations

Drag force:

$$\underline{F}_{\mathrm{D}} = \frac{3}{4}\frac{\varrho}{\varrho_P}\frac{m_P}{d_P} \cdot C_{\mathrm{D}} \left(\underline{u} - \underline{u}_{\mathrm{P}}\right) \left|\underline{u} - \underline{u}_{\mathrm{P}}\right| \tag{6}$$

Implementation of drag coefficient C_{D} in CFD programs:

ANSYS FLUENT
Morsi and Alexander[1]

$$C_{\mathrm{D}} = a_1 + \frac{a_2}{Re_{\mathrm{P}}} + \frac{a_3}{Re_{\mathrm{P}}^2} \tag{7}$$

OpenFOAM
Empirical relation

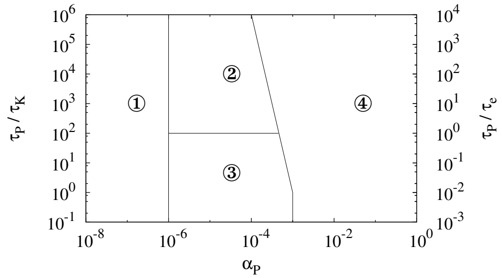
$$C_{\mathrm{D}} = \begin{cases} \frac{24}{Re_{\mathrm{P}}}\left(1 + \frac{1}{6} Re_{\mathrm{P}}^{2/3}\right) & ; Re_{\mathrm{P}} \leq 1000 \\ 0.424 & ; Re_{\mathrm{P}} \geq 1000 \end{cases} \tag{8}$$

[1] **Morsi1972**.

Model equations

Coupling

Classification of coupling schemes and interaction between disperse and continuous phase according to Elghobashi[2]:



①: One-way coupling $\underline{\bar{f}}_W = 0$

②: Two-way coupling, particles enhance turbulence production

③: Two-way coupling, particles enhance turbulence dissipation

④: Four-way coupling

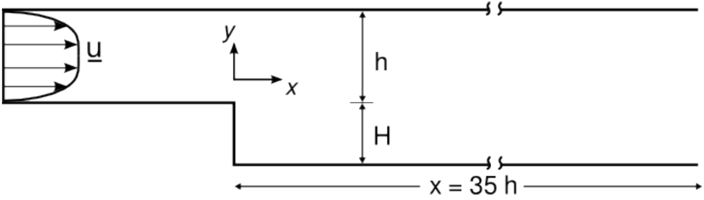
(② – ④: $\underline{\bar{f}}_W \neq 0$)

[2] Elghobashi1994.

Test case backward facing step

Description

Geometry according to Fessler and Eaton[3]



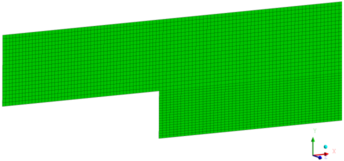
Channel flow		BFS flow	
Channel height h	40 mm	Step height H	26.7 mm
Channel width w	457 mm	w/H	17:1
		h/H	5:3

[3] Fessler1999.

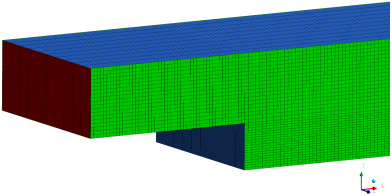
Test case backward facing step

Applied meshes

Mesh A



Mesh D



Name	Dimension	Cells in z-direction	BC front and back
A	2D	0	-
B	3D	1	empty
C	3D	1	symmetry
D	3D	10	symmetry

Test case backward facing step

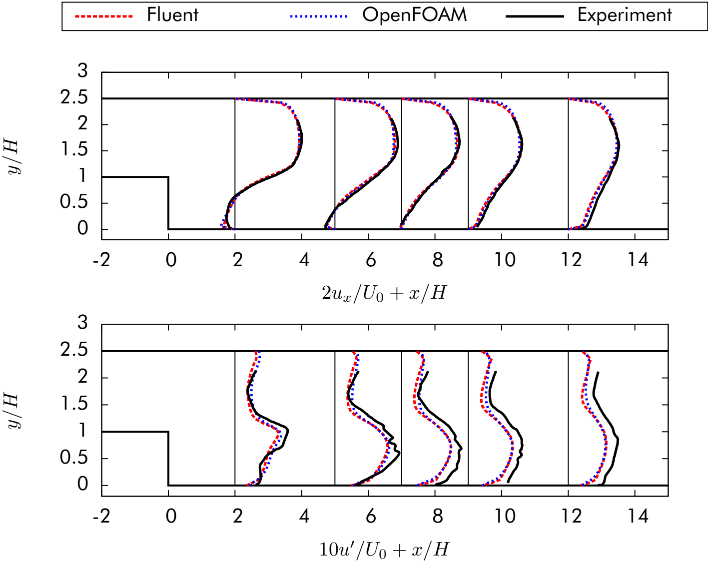
Setup

- RANS and URANS
- k-ω-SST turbulence model
- isothermal
- 1- and 2-way coupling

Continuous phase	
Inlet	profile with $\underline{\bar{u}}_{\text{avg}} = (9.39\ 0\ 0)\, m/s$
	$U_{x,0} = 10.5\, m/s$
	$k = 0.45\, m^2/s^2$
	$\omega = 2800\, 1/s$
Outlet	$\bar{p} = 0\, Pa$
Wall	$\underline{\bar{u}} = (0\ 0\ 0)\, m/s$
Disperse phase	
Particle diameter	$d_{\text{P}} = 70\, \mu m$
Particle density	$\varrho_{\text{P}} = 8800\, kg/m^3$
Particle mass flow rate	$\dot{m}_{\text{P}} = 1.58 \cdot 10^{-5}\, kg/s$
Injection velocity	$\underline{\bar{u}}_{\text{P,avg}} = (10.5\ 0\ 0)^T\, m/s$

Test case backward facing step

Fluid flow

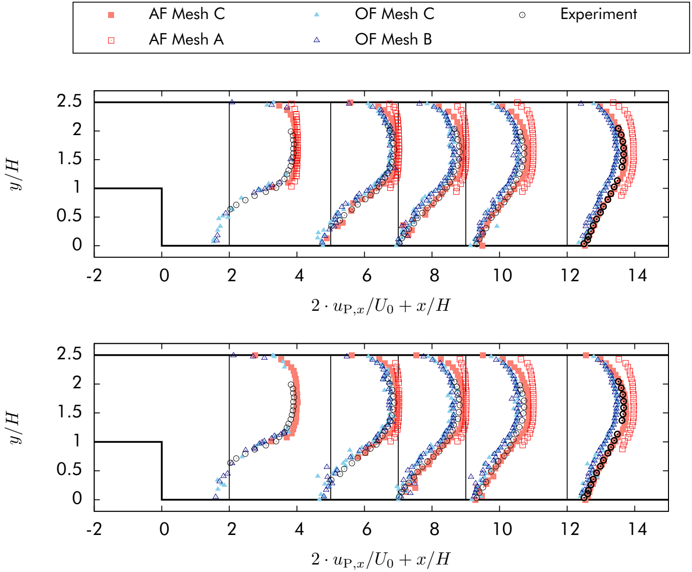


Test case backward facing step

Disperse phase

1-way coupling

2-way coupling

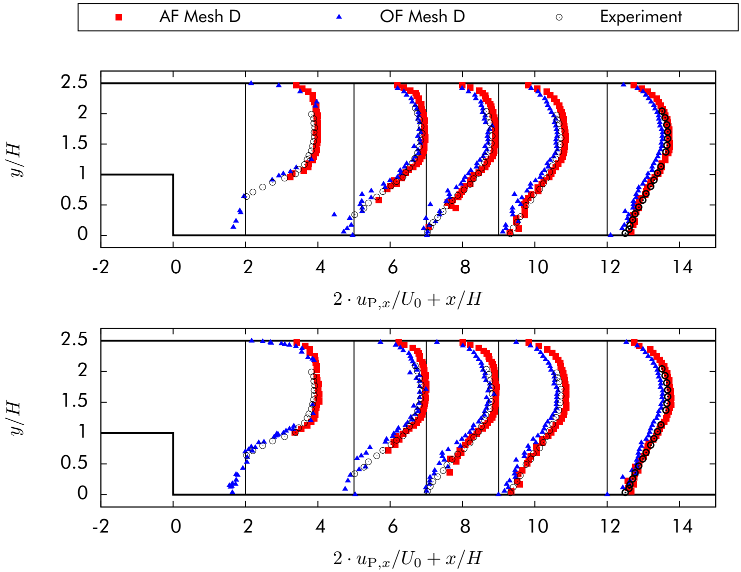


Test case backward facing step

Disperse phase

1-way coupling

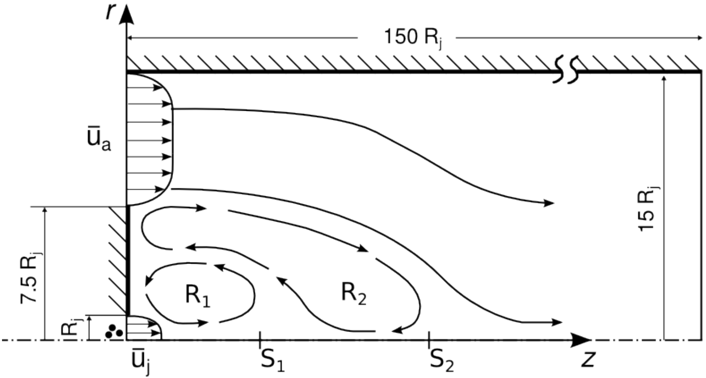
2-way coupling



Test case confined bluff body

Description

Geometry according to Borée et al.[4]



[4] Boree1999.

Test case confined bluff body

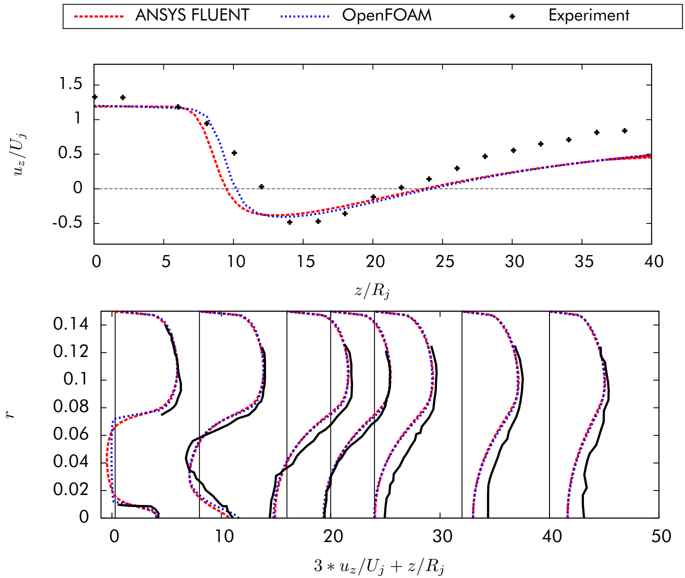
Setup

- URANS
- Standard k-ϵ turbulence model
- isothermal, incompressible
- 2-way coupling

Continuous phase	
Central jet inflow	$\underline{\bar{u}}_{\mathrm{j,avg}} = (0\ 0\ 3.01)\, m/s$
	$U_{z,\mathrm{j}} = 3.4\, m/s$
Annular jet inflow	$\underline{\bar{u}}_{\mathrm{a,avg}} = (0\ 0\ 5.36)\, m/s$
Outlet	$\bar{p} = 0\, Pa$
Wall	$\underline{\bar{u}} = (0\ 0\ 0)\, m/s$
Disperse phase	
Particle diameter	$d_{\mathrm{P}} = 63\, \mu m$
Particle density	$\varrho_{\mathrm{P}} = 2470\, kg/m^3$
Particle mass flow rate	$\dot{m}_{\mathrm{P}} = 2.78 \cdot 10^{-4}\, kg/s$
Injection velocity	$\underline{u}_{\mathrm{P,avg}} = (4.08\ 0\ 0)\, m/s$

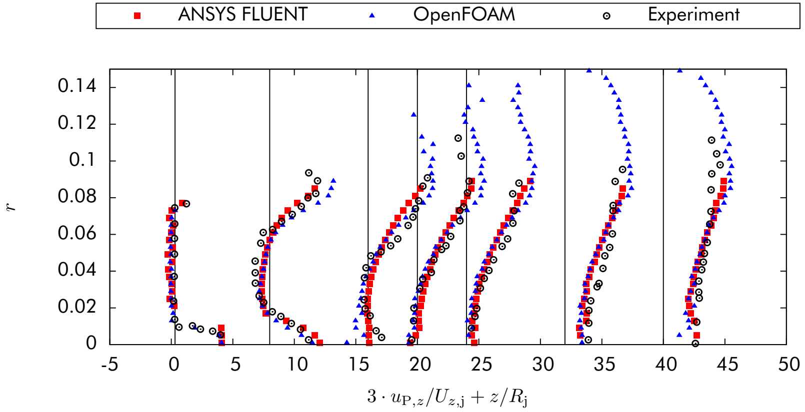
Test case backward facing step

Fluid flow



Test case backward facing step

Disperse phase



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.