Multi-fluid models for industrial gas-liquid flows

Dirk Lucas

Helmholtz-Zentrum Dresden - Rossendorf e.V.
Institute of Fluid Dynamics
Bautzner Landstr. 400
01328 Dresden
Germany
The Helmholtz-Zentrum Dresden-Rossendorf

Member of the **Helmholtz Association of German Research Centers** (since 01/2011)

**Foundation** 01.01.1992 (e.V.)

**Employees** **1,100** from **50 nations**

**Research** at eight institutes, in three research fields, and with five large-scale facilities

- ELBE – Center for High-Power Radiation Sources with Free-Electron-Laser and the High-Intensity Laser Draco
- Dresden High Magnetic Field Laboratory (HLD)
- Ion Beam Center (IBC)
- PET Center
- TOPFLOW Facility

**Research Sites** **Dresden**, Freiberg, Leipzig, Grenoble, Schenefeld
Experimental Thermofluidodynamics
- complex two-phase-flows in energy- and process engineering
- measurement techniques

Computational Fluid Dynamics (CFD)
- numerical simulation of two-phase- and MHD flows

Magnetohydrodynamics (MHD)
- electrically conducting fluids: liquid metals, semiconductors, electrolytes
- contactless control by magnetic fields
- measurement techniques

Transport processes at interfaces
- multiscale transport phenomena at interfaces, e.g. particle-bubble attachment in flotation processes
- instabilities of / at interfaces
Multiphase flows

Different phases may be involved: gas-liquid, liquid-liquid, gas-solid, liquid-solid, three-phase flows,…

In this lecture the focus is on gas-liquid flows – deformable interfaces of sizes with different scales.

Different flow morphologies are involved in many practical applications and transitions between them may occur, e.g.

• bubble entrainment by plunging jets, waves, free vortex flow,…
• droplet generation in breaking jets, from wave crests,…
• generation of large gas structures by coalescence, …

Model frameworks at HZDR:  
- iMUSIG for poly-disperse flows  
- AIAD for large interfaces  
- GENTOP for morphology transitions
Outline

MULTI-FLUID MODEL

BASELINE MODEL CONCEPT

POLYDISPERSE BUBBLY FLOWS
• iMUSIG
• closures
• validation

SEGREGATED FLOWS
• AIAD
• validation

MORPHOLOGY TRANSITIONS
• GENTOP
• demonstration cases

CONCLUSIONS
**Euler-Euler model for two-phase CFD**

**Resolved Interface Model:**
- at each position
- either gas or liquid

**Two Fluid Model:**
- both gas and liquid everywhere with certain probability

→ averaging → information on interface lost

**need to add closure models**
Typical problems for multi-fluid models

- uncountable options for closure models including tuning parameters
- post-test simulations with a CFD setup only for one or few experiments
- mainly “result-oriented” selection and tuning of closure models

→ finally often “good agreement with experimental data”, but
- may be based on compensating errors → no transferability
- often claimed: “parameter A agrees well with experimental data, but B shows deviations”
  → the simulation does not reflect the local phenomena correctly!
- limited consideration of BPGs
- experimental error often not known
- local flow characteristics versus global ones

→ limited predictive capabilities
→ consolidation of Euler-Euler-CFD required
Closure models have to reflect the local phenomena!!!

- even in globally total different flows similar local phenomena have to be considered
- Local characteristics in bubbly flows:
  - liquid shear rate
  - liquid turbulence
  - gas volume fraction
  - bubble size distribution

→ The same closure models should be applicable in a certain range of validity of local flow parameter

→ Baseline model concept to improve predictive capabilities

Lucas et al., NED 299(2016)2-11
Baseline model concept

Baseline model – as far as possible:
• includes sub-models for all potentially relevant flow phenomena
• uses physical based closure models
• has all model parameter fixed

Simulate experiments with different flow configurations under consideration of BPGs

Identify most severe shortcomings

Figure out a better model for these particular aspects

Update the baseline model

if successful

Demonstrate the overall improvement

if not successful

Lucas et al., NED 299(2016)2-11
Baseline model for poly-disperse bubbly flow

- Closure is a very complex problem
- Interactions depend on bubble size

… without claiming completeness
Baseline model for poly-disperse bubbly flow

Bubble forces (momentum transfer) strongly depend on bubble size which may lead to a separation of small and large bubbles → in general more than one velocity field for gas required!

\[
Lift-Force \ (\text{Tomiyama} \ 2002) \\
\vec{F}_{LIFT} = -C_L \rho_L (\vec{V}_G - \vec{V}_L) \times rot\vec{V}_L
\]

Krepper et al. NED 235(2005)597-611
Lucas et al., CES 62(2007)4146-4157
Baseline model for poly-disperse bubbly flow

Inhomogeneous MUSIG

The inhomogeneous MUSIG-Modell allows to consider a small number of velocity groups and a larger number of size groups.

Radial volume fraction profiles and bubble size distributions for air-water flow

Experiment standard new model

Rzehak et al., MMPE(2014)
Baseline model for adiabatic poly-dispersed bubbly flow

### Bubble Forces*
- drag \([\text{1979}_\text{Ishii}]\)
- (shear) lift \([\text{2002}_\text{Tomiyama}]\)
- wall (lift) \([\text{2002}_\text{Hosokawa}]\)
- turbulent dispersion \([\text{2004}_\text{Burns}]\)
- virtual mass \(C_{VM} = \frac{1}{2}\)

### Turbulence
- liquid phase only
- SST model + source terms for \(k\) and \(\varepsilon/\omega\) (obtained from DNS)

\[ S_{k}^{RANS} = \min(0.18 \cdot Re_p^{0.23}, 1) \cdot F_{Drag} |u_L - u_G| \]
\[ S_{\omega}^{RANS} = \frac{1}{C_{\mu_k}} S_{\varepsilon} - \frac{\omega}{k} S_k, \text{ with } S_{\varepsilon} = \frac{0.3 C_D}{C_{\omega}} S_k, \tau = \frac{d_p}{U_r} \]

### Coalescence
\[
\Gamma(d_i, d_j) = \frac{\alpha_{\max}}{\alpha_{\max} - \alpha_r} \left( \Gamma_{\text{turb}} + \Gamma_{\text{eddy}} + \Gamma_{\text{buoy}} + \Gamma_{\text{shear}} + \Gamma_{\text{wake}} \right)
\]

*largely based on experiments with single bubbles in laminar flows!!!

### Breakup
\[
\Omega(d_i, d_j) = \sum \frac{u_{s,i}}{d_i}
\]

---

Rzehak et al., CET 38(2015)1972
Liao et al., CES 122(2015)336
Baseline model for poly-disperse bubbly flow

Validation strategy

- from most simple to complicated cases
- try to separate effects as far as possible

1. adiabatic fixed mono-disperse (small bubbles)
2. adiabatic fixed poly-dispersed (small and large bubbles, but no significant change of the bubble size distribution)
3. adiabatic poly-dispersed with bubble coalescence & breakup
4. flows with phase transfer (evaporation, condensation, chemical reaction)

Problem: CFD-grade experimental databases should at least contain:

- gas volume fraction distributions,
- bubble size distributions,
- velocity fields,
- turbulent fluctuation velocities.
# Baseline model for poly-disperse bubbly flow

<table>
<thead>
<tr>
<th>Overview on validation</th>
<th>Configuration</th>
<th>Geometry</th>
<th>Media</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Upward pipe flow</td>
<td>DN25~200mm</td>
<td>Air-water</td>
</tr>
<tr>
<td></td>
<td>Downward pipe flow</td>
<td>DN50mm</td>
<td>Air-water</td>
</tr>
<tr>
<td></td>
<td>Inclined pipe flow</td>
<td>DN50mm</td>
<td>Air-water</td>
</tr>
<tr>
<td></td>
<td>Vertical pipe with obstacles</td>
<td>DN200mm</td>
<td>Air-water</td>
</tr>
<tr>
<td></td>
<td>Bubble column</td>
<td>Rectangular, cylindrical</td>
<td>Air-water, N₂-NaOH solution, CO₂-water</td>
</tr>
<tr>
<td></td>
<td>Bubble plume</td>
<td>Rectangular</td>
<td>Air-water, Argon-GaInSn</td>
</tr>
<tr>
<td></td>
<td>Airlift</td>
<td>Rectangular, cylindrical</td>
<td>Air-water</td>
</tr>
<tr>
<td></td>
<td>Static mixer</td>
<td>DN80mm</td>
<td>Air-water</td>
</tr>
<tr>
<td></td>
<td>Stirred tank</td>
<td>DN~200mm</td>
<td>Air-water</td>
</tr>
<tr>
<td>Phase change</td>
<td>Condensing pipe flow</td>
<td>DN~200mm</td>
<td>Steam-water</td>
</tr>
<tr>
<td></td>
<td>Evaporating pipe flow</td>
<td>DN~200mm</td>
<td>Steam-water</td>
</tr>
<tr>
<td></td>
<td>Sub-cooled wall boiling</td>
<td>5×5 Bundle, DN19.2mm</td>
<td>Steam-water R12</td>
</tr>
<tr>
<td></td>
<td>Reaction</td>
<td>bubble column</td>
<td>CO2 / NaOH solution</td>
</tr>
</tbody>
</table>
Baseline model for poly-disperse bubbly flow

**Round bubble column**
(Mudde et al., Ind.Eng.Chem.Res. 2009)

- Gas volume fraction
- Measurement: 0.6m
- Inlet: 0.0m
- Water level: 1.3m

**Upwards pipe flow**
(Liu, ICMF 1998)

- Gas volume fraction
- Liquid velocity

**Counter-Current pipe flow**
(HZDR)

- Gas volume fraction
- Gas and liquid velocity

**Rectangular bubble column**
(Mohd Akbar et al., MST 2012)

- Turbulent fluctuations
MULTI-FLUID MODEL

BASELINE MODEL CONCEPT

POLYDISPERSE BUBBLY FLOWS
  • iMUSIG
  • closures
  • validation

SEGREGATED FLOWS
  • AIAD
  • validation

MORPHOLOGY TRANSITIONS
  • GENTOP
  • demonstration

CONCLUSIONS
Baseline model for segregated flows

Algebraic Interfacial Area Density Model (AIAD)

- allows the detection of the local morphology and the corresponding switching between closure models
- provides models for IAD, $C_D$ and other parameter for full range $0 \leq \alpha \leq 1$
- free surface drag depends on shear stresses
- permits to introduce modified turbulence parameters at the free surface
- Blending functions basing on volume fraction and volume fraction gradient allow to switch between different closure models

\[
\begin{array}{ccc}
  a_G = 1 & \rightarrow & \text{Droplet region} \\
  a_G = 0 & \rightarrow & \text{Bubble region}
\end{array}
\]

- $\alpha_G$ high
- $\alpha_G$ medium
- $\alpha_G$ small

\[
\begin{aligned}
  C_{D,D} & ; & A_D \\
  C_{D,FS} & ; & A_{FS} \\
  C_{D,B} & ; & A_B
\end{aligned}
\]

Porombka and Höhne CES 134(2015) 348-359
Baseline model for segregated flows

Algebraic Interfacial Area Density Model (AIAD)

Blending functions

$$\psi_{surf} = \varphi_{surf} (f_b - f_d)$$

$$f_b = \frac{1}{2} \left[ 1 + \cos \left( \frac{\alpha^G - (\alpha_{b,crit} - \delta_\alpha)}{2\delta_\alpha} \right) \right]$$

$$f_d = \frac{1}{2} \left[ 1 + \cos \left( \frac{\alpha^L - (\alpha_{d,crit} - \delta_\alpha)}{2\delta_\alpha} \right) \right]$$

$$\varphi_{surf} = \frac{1}{2} \left[ 1 + \cos \left( \frac{\nabla \alpha^c - (\nabla \alpha_{crit} - \delta_\nabla)}{2\delta_\nabla} \right) \right]$$

---

Baseline model for segregated flows

Wall-like damping procedures at the free surface in k-ω turbulence model

Source terms to mimic the effect of this boundary condition near the free surface

\[ S_{D,i} = f_{FS} \cdot \beta \cdot \rho_i \left( B \cdot \frac{6 \mu_i}{\beta \rho_i \Delta n^2} \right)^2 \]

Höhne and Mehlhoop, IJMF 62(2014)1-16

Sub-grid wave turbulence

- Waves created by Kelvin-Helmholtz instabilities that are smaller than the grid size are modelled
- Influence on the turbulence kinetic energy of the liquid side can be significantly large
- Brocchini and Peregrine (2001) try to quantify this in the L-q diagram
Counter-Current Flow Limitation (CCFL) in a hot leg of a Pressurized Water Reactor

- Good qualitative and quantitative agreement with the experiment
- Typical flow structures are well reflected by the AIAD model
- Simulations gives insights into instantaneous local flow structures and waves

Experiment and simulation using AIAD

\[
J_i^* = j_i \cdot \sqrt{\frac{1}{g \cdot L} \cdot \frac{\rho_i}{\rho_L - \rho_G}}
\]

CCFL characteristics
MULTI-FLUID MODEL

BASELINE MODEL CONCEPT

POLYDISPERSE BUBBLY FLOWS
  • iMUSIG
  • closures
  • validation

SEGREGATED FLOWS
  • AIAD
  • validation

MORPHOLOGY TRANSITIONS
  • GENTOP
  • demonstration

CONCLUSIONS
Examples of multi-scale flows

←
ultrafast X-ray tomography of HZDR
Experimental Thermal Fluid Dynamics

→
X-LIMMCAST of HZDR Magnetohydrodynamics
Morphology transitions – How to model?

- Full resolution of all interfaces is only feasible for very small simulation domains.
- For medium and large industrial scales: Euler/Euler or Euler/Lagrange
  - Resolved and non-resolved parts according to the mesh size
  - Similar to LES turbulence modelling
- Distinction between continuous and dispersed is related to the mesh size
- More than 2 fields (sets of conservation equations) required
- Transitions between continuous and dispersed morphologies by coalescence and breakup models
- Interaction with turbulence – turbulent scales
- Interfacial transfers strongly depend on the size and structure of each single interface
**GENTOP – basic idea**

- **GENerated TwO Phase Flow model**
- presently only three fields (no phase for drops)
- $d_{dg,\text{max}} = z \Delta x$ with $1 < z < 10$, $\Delta x$ mesh cell size

<table>
<thead>
<tr>
<th>Phase</th>
<th>gas dg</th>
<th>gas cg</th>
<th>liquid l</th>
</tr>
</thead>
<tbody>
<tr>
<td>Morphology</td>
<td>polydispersed</td>
<td>continuous</td>
<td>continuous</td>
</tr>
<tr>
<td>Model</td>
<td>Multiple Size Group (MUSIG) -approach</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- $d_b > d_{dg,\text{max}}$
- resolved interface

---

Hänsch et al., Int. J. Multiphase Flow 47(2012)171-182
Implementation in ANSYS-CFX

Extension of inhomogeneous MUSIG-framework (one or more dispersed gas phases) by a potentially continuous gas field.

velocity groups $J=1\ldots N$

size fractions $K=1\ldots \sum M_J$

New models for coalescence and breakup

transfer into cg

breakup to dg
Implementation in OpenFOAM

- **phase**
- **morphology**
- **velocity fields**
- **framework**

**gas**
- polydisperse
- polydisperse
- continuous

**coalescence/breakup/growth**

inhomogeneous class method

Coalescence

Number density

e.g. 6mm migration toward

mesh dependent

near wall region pipe center
The continuous gas phase should be a hybrid phase allowing beside statistically resolved gas-liquid interfaces also regions where no interface can be formed (bubbly flow or drop regions).

→ Implementation in ANSYS-CFX by combining the iMUSIG and AIAD models
  • AIAD allows a blending for such regions depending on volume fraction and volume fraction gradient criteria
  • Additional cluster force supports formation of interfaces
  • Due to technical limitations in CFX the cluster force was also used for interface sharpening

→ Implementation in OpenFOAM
  • similar blending like in AIAD foreseen
  • interface compression scheme instead of cluster force
  • some ideas to do the clustering on a more consistent way

Bubble column with strong coalescence
Minimum resolution required

- one fluid approaches: at least 25 – 30 cells per bubble diameter required
- in the two-fluid approach clearly less cells are sufficient

Volume fraction of a rising bubble
\[ d_B = 9 \text{ mm} \]

Bubble entrainment is a special kind of bubbly breakup (breakup of the continuous phase into disperse phase).

- should occur in the interface region (blending – previous slide)
- smeared interface shows advantages for this process
- however, well defined thickness seems to be required (or a normalization of the entrainment rate by interface thickness)

Disperse gas should be zero in regions of the gas side of the interface.

- complete coalescence
GENTOP-concept – surface tension

- Surface tension force according to Brackbill
  \[ \vec{F}_\sigma = -\sigma \kappa \vec{n} \]

- curvature
  \[ \kappa = \nabla \cdot \vec{n} = \nabla \cdot \frac{\nabla \alpha_l}{|\nabla \alpha_l|} \]

- normal to the interface
  \[ \vec{n} = \frac{\nabla \alpha_l}{|\nabla \alpha_l|} \]

- In GENTOP this formulation is blended within the free surface region detection by the function \( \Psi_{\text{surf}} \)
  \[ \vec{F}_\sigma = -(1 - |\Psi_{\text{surf}}|) \sigma \kappa \vec{n} \]

\[ \text{Brackbill J.U. et al. 1992, J. Comp. Phys. 100, 335-354} \]
Demonstration cases

**Impinging jet**

- $c_g \rightarrow d_g$
- according to experiments in test basin
- $v_0 = 1.7 \text{ m/s}$
- $d_j = 16 \text{ mm}$
- $L_j = 10 \text{ mm}$

- two-dimensional simulation
- equidistant grid with $\Delta x=2\text{mm}$
- 37,750 elements

Experimental data by


from Hänsch et al., Int. J. Multiphase Flow 47(2012)171-182
Demonstration cases

Impinging jet

- Interfacial transfer dg / liquid:
  - Lift Force: Tomiyama
  - Turbulent Dispersion: Favre Averaged Drag Model
  - Turbulence model: Dispersed Phase Zero Equation / SST
  - Drag Force: Ishii Zuber

- Interfacial transfer cg / liquid as shown before

<table>
<thead>
<tr>
<th>$v_g$</th>
<th>dg</th>
<th>cg</th>
</tr>
</thead>
<tbody>
<tr>
<td>Morphology</td>
<td>polydispersed</td>
<td>continuous</td>
</tr>
<tr>
<td>Size Groups</td>
<td>BS1</td>
<td>BS2</td>
</tr>
<tr>
<td>$d_i$ [mm]</td>
<td>1.5</td>
<td>2.5</td>
</tr>
</tbody>
</table>
CFX demonstration case: Plunging jet (1/2)

continuous to disperse

time 0.67 [ s ]

cg volume fraction

dg volume fraction

Dr. Dirk Lucas | Head of Computational Fluid Dynamics division | Institute of Fluid Dynamics
Impinging jet

from Hänsch et al., Int. J. Multiphase Flow 47(2012)171-182
Demonstration cases

**Impinging jet**

- polydispersed gas phase emerges due to breakup models
- air entrainment after pinch-off of gas structures
- quasi-steady state reached after 3s
- mass transfer depends on ratio $F_B/F_C$

from Hänsch et al., Int. J. Multiphase Flow 47(2012)171-182
Demonstration cases

**Bubble column**

- $dg \rightarrow cg$
- Equidistant 3D-grid with $\Delta x=5\text{mm}$
- 40.000 elements
- Parabolic mass flow inlet of $dg$ with size group $k=1$
  - 5 bubble size classes
  - $d_{\text{min}} = 1\text{ mm}$, $d_{\text{max}} = 20\text{ mm}$
  - $d_{\text{dg,max}} = 15\text{ mm} \sim 3\Delta x$
  - High coalescence rate: $F_B / F_C = 0.1$

from Hänsch et al., Int. J. Multiphase Flow 47(2012)171-182
CFX demonstration case: Coalescing bubbly flow
disperse to continuous
Demonstration cases

**Bubble column**

- typical bubble chain following zig-zag path
- growing bubbles due to mass transfer from dg
- emerged interface can finally be detected

from Hänsch et al., Int. J. Multiphase Flow 47(2012)171-182
Demonstration cases

**Dam break**

- cg → dg → cg
- Koshizuka experiment
- quasi-2D grid with $\Delta x = 4\text{mm}$
- 146x146 elements
- monitor point for local pressure

- CFX 14.0
- 5 bubble size classes
- $d_{\text{min}} = 1\text{mm}, d_{\text{max}} = 11\text{mm}$
- $d_{\text{dg, max}} = 10\text{mm}$
- $F_B / F_C = 1.0$
- sub-grid wave turbulence

---

Demonstration cases

cg volume fraction

dg volume fraction

time 0.8 [ s ]
time 0.8 [ s ]
Demonstration cases

**Dam break**

- shapes of interfaces correspond with experimental images
- main flow characteristics are captured
Demonstration cases

**Dam break**

- rise of pressure at MP when water hits the obstacle
- good agreement with results from interface capturing methods
- pressure repulsion when water tongue hits the opposite wall

---

**pressure at monitor point**

![Diagram showing pressure at monitor point with time on the x-axis and pressure on the y-axis, with arrows indicating the leading edge hitting the obstacle and pressure repulsion.]
Demonstration cases

Churn-turbulent flow

MUSIG  TOPFLOW-Experiment  GENTOP
Slug flow

in a vertical pipe
OpenFOAM demonstration case: Bubble column

Resolved structures

Disperse
CFX demonstration case: Boiling pipe flow

- $L = 0.5 \text{ m}; \quad d = 0.025 \text{ m}$
- GasD1, GasD2: 0-9 mm, iMUSIG: 4,5
- GasD, GasC at $T_{\text{SAT}}$
- $P_{\text{Ref}} = 5.0 \text{ MPa}, \quad T_{\text{sat}} = 537.1 \text{ K}$
- $T_{\text{IN}} = T_{\text{sat}} - 3 \text{ K}$
- $T_{\text{Wall}} = T_{\text{sat}} + 10 \text{ K}$
- $V_{\text{IN}} = 0.2 \text{ m/s}$

- ca. 130,000 cells
CFX demonstration case: Boiling pipe flow
Conclusions

- Consolidation of multiphase CFD in the Euler-Euler framework required
  → Baseline concept to develop CFD-models with predictive capabilities.

- A baseline model for poly-disperse flows has been established basing on the iMUSIG approach and was applied for large number of different experimental data.

- For segregated flows the AIAD model allows the blending between regions of bubbly flow, (large) interface and droplet flow

- The innovative GENTOP concepts combines dispersed and segregated flows.
  - combines iMUSIG and AIAD – most general
  - aims to extent the range of applicabilitlty of CMFD
  - is still not mature
Thank you for your attention!

16th Multiphase Flow Conference & Short Course
November 13 - 16, 2018 Dresden

Announcement

16th Multiphase Flow Conference & Short Course: Simulation, Experiment and Application

13 – 16 November 2018, Dresden (HZDR)
www.hzdr.de/multiphase