

Proceedings of the 12<sup>th</sup> International Conference on  
Computational Fluid Dynamics in the Oil & Gas,  
Metallurgical and Process Industries

# Progress in Applied CFD – CFD2017



SINTEF Proceedings

Editors:

Jan Erik Olsen and Stein Tore Johansen

## **Progress in Applied CFD – CFD2017**

Proceedings of the 12<sup>th</sup> International Conference on Computational Fluid Dynamics  
in the Oil & Gas, Metallurgical and Process Industries

SINTEF Academic Press

SINTEF Proceedings no 2

Editors: Jan Erik Olsen and Stein Tore Johansen

**Progress in Applied CFD – CFD2017**

Selected papers from 10<sup>th</sup> International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries

Key words:

CFD, Flow, Modelling

Cover, illustration: Arun Kamath

ISSN 2387-4295 (online)

ISBN 978-82-536-1544-8 (pdf)

© Copyright SINTEF Academic Press 2017

The material in this publication is covered by the provisions of the Norwegian Copyright Act. Without any special agreement with SINTEF Academic Press, any copying and making available of the material is only allowed to the extent that this is permitted by law or allowed through an agreement with Kopinor, the Reproduction Rights Organisation for Norway. Any use contrary to legislation or an agreement may lead to a liability for damages and confiscation, and may be punished by fines or imprisonment

SINTEF Academic Press

Address:       Forskningsveien 3 B  
                  PO Box 124 Blindern  
                  N-0314 OSLO

Tel:             +47 73 59 30 00

Fax:            +47 22 96 55 08

[www.sintef.no/byggforsk](http://www.sintef.no/byggforsk)

[www.sintefbok.no](http://www.sintefbok.no)

**SINTEF Proceedings**

SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.

The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

## PREFACE

This book contains all manuscripts approved by the reviewers and the organizing committee of the 12th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries. The conference was hosted by SINTEF in Trondheim in May/June 2017 and is also known as CFD2017 for short. The conference series was initiated by CSIRO and Phil Schwarz in 1997. So far the conference has been alternating between CSIRO in Melbourne and SINTEF in Trondheim. The conferences focuses on the application of CFD in the oil and gas industries, metal production, mineral processing, power generation, chemicals and other process industries. In addition pragmatic modelling concepts and bio-mechanical applications have become an important part of the conference. The papers in this book demonstrate the current progress in applied CFD.

The conference papers undergo a review process involving two experts. Only papers accepted by the reviewers are included in the proceedings. 108 contributions were presented at the conference together with six keynote presentations. A majority of these contributions are presented by their manuscript in this collection (a few were granted to present without an accompanying manuscript).

The organizing committee would like to thank everyone who has helped with review of manuscripts, all those who helped to promote the conference and all authors who have submitted scientific contributions. We are also grateful for the support from the conference sponsors: ANSYS, SFI Metal Production and NanoSim.

Stein Tore Johansen & Jan Erik Olsen



Organizing committee:

Conference chairman: Prof. Stein Tore Johansen

Conference coordinator: Dr. Jan Erik Olsen

Dr. Bernhard Müller

Dr. Sigrid Karstad Dahl

Dr. Shahriar Amini

Dr. Ernst Meese

Dr. Josip Zoric

Dr. Jannike Solsvik

Dr. Peter Witt

Scientific committee:

Stein Tore Johansen, SINTEF/NTNU

Bernhard Müller, NTNU

Phil Schwarz, CSIRO

Akio Tomiyama, Kobe University

Hans Kuipers, Eindhoven University of Technology

Jinghai Li, Chinese Academy of Science

Markus Braun, Ansys

Simon Lo, CD-adapco

Patrick Segers, Universiteit Gent

Jiyuan Tu, RMIT

Jos Derksen, University of Aberdeen

Dmitry Eskin, Schlumberger-Doll Research

Pär Jönsson, KTH

Stefan Pirker, Johannes Kepler University

Josip Zoric, SINTEF

## CONTENTS

<b>PRAGMATIC MODELLING .....</b>	<b>9</b>
On pragmatism in industrial modeling. Part III: Application to operational drilling .....	11
CFD modeling of dynamic emulsion stability .....	23
Modelling of interaction between turbines and terrain wakes using pragmatic approach .....	29
<b>FLUIDIZED BED .....</b>	<b>37</b>
Simulation of chemical looping combustion process in a double looping fluidized bed reactor with cu-based oxygen carriers.....	39
Extremely fast simulations of heat transfer in fluidized beds.....	47
Mass transfer phenomena in fluidized beds with horizontally immersed membranes .....	53
A Two-Fluid model study of hydrogen production via water gas shift in fluidized bed membrane reactors .....	63
Effect of lift force on dense gas-fluidized beds of non-spherical particles .....	71
Experimental and numerical investigation of a bubbling dense gas-solid fluidized bed .....	81
Direct numerical simulation of the effective drag in gas-liquid-solid systems .....	89
A Lagrangian-Eulerian hybrid model for the simulation of direct reduction of iron ore in fluidized beds.....	97
High temperature fluidization - influence of inter-particle forces on fluidization behavior .....	107
Verification of filtered two fluid models for reactive gas-solid flows .....	115
<b>BIOMECHANICS.....</b>	<b>123</b>
A computational framework involving CFD and data mining tools for analyzing disease in carotid artery .....	125
Investigating the numerical parameter space for a stenosed patient-specific internal carotid artery model.....	133
Velocity profiles in a 2D model of the left ventricular outflow tract, pathological case study using PIV and CFD modeling.....	139
Oscillatory flow and mass transport in a coronary artery.....	147
Patient specific numerical simulation of flow in the human upper airways for assessing the effect of nasal surgery.....	153
CFD simulations of turbulent flow in the human upper airways .....	163
<b>OIL &amp; GAS APPLICATIONS .....</b>	<b>169</b>
Estimation of flow rates and parameters in two-phase stratified and slug flow by an ensemble Kalman filter .....	171
Direct numerical simulation of proppant transport in a narrow channel for hydraulic fracturing application .....	179
Multiphase direct numerical simulations (DNS) of oil-water flows through homogeneous porous rocks .....	185
CFD erosion modelling of blind tees .....	191
Shape factors inclusion in a one-dimensional, transient two-fluid model for stratified and slug flow simulations in pipes .....	201
Gas-liquid two-phase flow behavior in terrain-inclined pipelines for wet natural gas transportation .....	207

<b>NUMERICS, METHODS &amp; CODE DEVELOPMENT .....</b>	<b>213</b>
Innovative computing for industrially-relevant multiphase flows .....	215
Development of GPU parallel multiphase flow solver for turbulent slurry flows in cyclone.....	223
Immersed boundary method for the compressible Navier–Stokes equations using high order summation-by-parts difference operators .....	233
Direct numerical simulation of coupled heat and mass transfer in fluid-solid systems .....	243
A simulation concept for generic simulation of multi-material flow, using staggered Cartesian grids.....	253
A cartesian cut-cell method, based on formal volume averaging of mass, momentum equations.....	265
SOFT: a framework for semantic interoperability of scientific software .....	273
 <b>POPULATION BALANCE .....</b>	 <b>279</b>
Combined multifluid-population balance method for polydisperse multiphase flows .....	281
A multifluid-PBE model for a slurry bubble column with bubble size dependent velocity, weight fractions and temperature.....	285
CFD simulation of the droplet size distribution of liquid-liquid emulsions in stirred tank reactors .....	295
Towards a CFD model for boiling flows: validation of QMOM predictions with TOPFLOW experiments .....	301
Numerical simulations of turbulent liquid-liquid dispersions with quadrature-based moment methods.....	309
Simulation of dispersion of immiscible fluids in a turbulent couette flow .....	317
Simulation of gas-liquid flows in separators - a Lagrangian approach.....	325
CFD modelling to predict mass transfer in pulsed sieve plate extraction columns .....	335
 <b>BREAKUP &amp; COALESCENCE .....</b>	 <b>343</b>
Experimental and numerical study on single droplet breakage in turbulent flow .....	345
Improved collision modelling for liquid metal droplets in a copper slag cleaning process .....	355
Modelling of bubble dynamics in slag during its hot stage engineering.....	365
Controlled coalescence with local front reconstruction method .....	373
 <b>BUBBLY FLOWS .....</b>	 <b>381</b>
Modelling of fluid dynamics, mass transfer and chemical reaction in bubbly flows .....	383
Stochastic DSMC model for large scale dense bubbly flows.....	391
On the surfacing mechanism of bubble plumes from subsea gas release.....	399
Bubble generated turbulence in two fluid simulation of bubbly flow .....	405
 <b>HEAT TRANSFER .....</b>	 <b>413</b>
CFD-simulation of boiling in a heated pipe including flow pattern transitions using a multi-field concept .....	415
The pear-shaped fate of an ice melting front .....	423
Flow dynamics studies for flexible operation of continuous casters (flow flex cc).....	431
An Euler-Euler model for gas-liquid flows in a coil wound heat exchanger.....	441
 <b>NON-NEWTONIAN FLOWS.....</b>	 <b>449</b>
Viscoelastic flow simulations in disordered porous media .....	451
Tire rubber extrudate swell simulation and verification with experiments .....	459
Front-tracking simulations of bubbles rising in non-Newtonian fluids.....	469
A 2D sediment bed morphodynamics model for turbulent, non-Newtonian, particle-loaded flows.....	479

<b>METALLURGICAL APPLICATIONS.....</b>	<b>491</b>
Experimental modelling of metallurgical processes .....	493
State of the art: macroscopic modelling approaches for the description of multiphysics phenomena within the electroslag remelting process .....	499
LES-VOF simulation of turbulent interfacial flow in the continuous casting mold .....	507
CFD-DEM modelling of blast furnace tapping .....	515
Multiphase flow modelling of furnace tapholes .....	521
Numerical predictions of the shape and size of the raceway zone in a blast furnace.....	531
Modelling and measurements in the aluminium industry - Where are the obstacles? .....	541
Modelling of chemical reactions in metallurgical processes.....	549
Using CFD analysis to optimise top submerged lance furnace geometries .....	555
Numerical analysis of the temperature distribution in a martensitic stainless steel strip during hardening.....	565
Validation of a rapid slag viscosity measurement by CFD.....	575
Solidification modeling with user defined function in ANSYS Fluent.....	583
Cleaning of polycyclic aromatic hydrocarbons (PAH) obtained from ferroalloys plant.....	587
Granular flow described by fictitious fluids: a suitable methodology for process simulations .....	593
A multiscale numerical approach of the dripping slag in the coke bed zone of a pilot scale Si-Mn furnace.....	599
<b>INDUSTRIAL APPLICATIONS .....</b>	<b>605</b>
Use of CFD as a design tool for a phosphoric acid plant cooling pond .....	607
Numerical evaluation of co-firing solid recovered fuel with petroleum coke in a cement rotary kiln: Influence of fuel moisture .....	613
Experimental and CFD investigation of fractal distributor on a novel plate and frame ion-exchanger .....	621
<b>COMBUSTION .....</b>	<b>631</b>
CFD modeling of a commercial-size circle-draft biomass gasifier.....	633
Numerical study of coal particle gasification up to Reynolds numbers of 1000.....	641
Modelling combustion of pulverized coal and alternative carbon materials in the blast furnace raceway .....	647
Combustion chamber scaling for energy recovery from furnace process gas: waste to value .....	657
<b>PACKED BED.....</b>	<b>665</b>
Comparison of particle-resolved direct numerical simulation and 1D modelling of catalytic reactions in a packed bed .....	667
Numerical investigation of particle types influence on packed bed adsorber behaviour .....	675
CFD based study of dense medium drum separation processes .....	683
A multi-domain 1D particle-reactor model for packed bed reactor applications.....	689
<b>SPECIES TRANSPORT &amp; INTERFACES .....</b>	<b>699</b>
Modelling and numerical simulation of surface active species transport - reaction in welding processes .....	701
Multiscale approach to fully resolved boundary layers using adaptive grids.....	709
Implementation, demonstration and validation of a user-defined wall function for direct precipitation fouling in Ansys Fluent.....	717



<b>FREE SURFACE FLOW &amp; WAVES .....</b>	<b>727</b>
Unresolved CFD-DEM in environmental engineering: submarine slope stability and other applications.....	729
Influence of the upstream cylinder and wave breaking point on the breaking wave forces on the downstream cylinder .....	735
Recent developments for the computation of the necessary submergence of pump intakes with free surfaces .....	743
Parallel multiphase flow software for solving the Navier-Stokes equations .....	752
 <b>PARTICLE METHODS .....</b>	 <b>759</b>
A numerical approach to model aggregate restructuring in shear flow using DEM in Lattice-Boltzmann simulations .....	761
Adaptive coarse-graining for large-scale DEM simulations.....	773
Novel efficient hybrid-DEM collision integration scheme.....	779
Implementing the kinetic theory of granular flows into the Lagrangian dense discrete phase model.....	785
Importance of the different fluid forces on particle dispersion in fluid phase resonance mixers .....	791
Large scale modelling of bubble formation and growth in a supersaturated liquid.....	798
 <b>FUNDAMENTAL FLUID DYNAMICS .....</b>	 <b>807</b>
Flow past a yawed cylinder of finite length using a fictitious domain method .....	809
A numerical evaluation of the effect of the electro-magnetic force on bubble flow in aluminium smelting process.....	819
A DNS study of droplet spreading and penetration on a porous medium.....	825
From linear to nonlinear: Transient growth in confined magnetohydrodynamic flows.....	831

## CFD-SIMULATION OF BOILING IN A HEATED PIPE INCLUDING FLOW PATTERN TRANSITIONS USING A MULTI-FIELD CONCEPT

Thomas HÖHNE\*, Eckhard KREPPER, Dirk LUCAS

Helmholtz-Zentrum Dresden-Rossendorf (HZDR) - Institute of Fluid Dynamics  
 P.O.Box 510119, D-01314 Dresden, GERMANY

\* E-mail: [t.hoehne@hzdr.de](mailto:t.hoehne@hzdr.de)

### ABSTRACT

The paper presents the extension of the GENTOP model for phase transfer and discusses the sub-models used. Boiling flow inside a wall heated vertical pipe is simulated by a multi-field CFD approach. Sub-cooled water enters the pipe from the lower end and heats up first in the near wall region leading to the generation of small bubbles. Further along the pipe larger and larger bubbles are generated by coalescence and evaporation. This leads to transitions of the two-phase flow patterns from bubbly to churn-turbulent and annular flow. The CFD simulation bases on the recently developed GGeneralized TwO Phase flow (GENTOP) concept. It is a multi-field model using the Euler-Euler approach. It allows the consideration of different local flow morphologies including transitions between them. Small steam bubbles are handled as dispersed phases while the interface of large gas structures is statistically resolved. The GENTOP sub-models and the Wall Boiling Model need a constant improvement and separate, intensive validation effort using CFD grade experiments.

**Keywords:** multi-phase, boiling, GENTOP, multi-scale, CFD

### NOMENCLATURE

$\alpha$	Void fraction
$\rho_k$	Density of the phase-k
$C_D$	Drag coefficient
$C_{clust}$	constant
$d_{lg}$	Interfacial length scale, i.e., the mean particle diameter for the particle model
$GasC$	potentially continuous gaseous phase
$GasD$	disperse gaseous phase
$\sigma$	Surface tension coefficient
$C_{cd}$	Interface drag force for the FAD turbulence dispersion model
$\Delta_\alpha$	Transition range width
$f_b$	blending function for bubble regime
$f_d$	blending function for droplet regime
$M_{kj}^i$	Interfacial momentum transfer per unit time between the fields k and j
$\mu$	Molecular (dynamic) viscosity

$\Psi_{surf}$	Blending Function
$\phi_{sf}$	Interface blending function
$\nu_t$	Kinematic eddy viscosity
$\dot{m}$	Evaporation mass transfer rate per unit wall area
Nu	Nusselt number
$h_{g,sat}$	Specific enthalpy of saturated vapor
$h_l$	Specific enthalpy of sub-cooled liquid
$c_{lg}^{(h)}$	Volumetric heat transfer coefficient
$\lambda$	Thermal conductivity
$T_s$	Interfacial temperature
$\tau_{dg \rightarrow cg}$	Time constant
$h_l$	Heat transfer coefficient for liquid
$h_g$	Heat transfer coefficient for gas

### 1. INTRODUCTION

Two-phase flows can be found in various industrial applications: nuclear power plants, processing industries, heat transfer systems, transport systems, and of course also in nature in general (ocean waves, river flooding).

Various classifications of two-phase flows exist and they are mainly based on flow morphologies. Such classifications are often difficult to make since the interface structure changes occur continuously. One of the two-phase flow classifications is divided in three major groups and several subgroups - flow regimes:

- Stratified flows (film flow, annular flow, horizontal stratified and jet flow),
- Mixed or transitional flows (cap, slug, churn-turbulent flow, bubbly annular flow, droplet annular flow and bubbly-droplet annular flow),

- Dispersed flows (bubbly flow, droplet flow and flow with solid particles).

Much progress has been achieved in establishing models to describe various multiphase flow phenomena using Computational Fluid Dynamics (CFD).

The GENTOP-concept (Hänsch et al., 2012) enables to consider such transitions in a consistent way as coalescence and breakup processes. The potential of this concept was demonstrated in Hänsch et al. (2012) and Hänsch et al. (2014) for adiabatic flows without heat and mass transfer. In this paper the GENTOP concept is applied to simulate boiling effects in a vertical pipe where transitions from bubbly flow to churn turbulent and then annular flow are involved.

Boiling is a process in which heat transfer causes liquid evaporation. Flow boiling refers to a boiling process when the fluid is imposed by a forced flow. It can be classified as saturated boiling and subcooled boiling. In the saturated boiling, the bulk temperature of the fluid is as equal as its saturation temperature, in the subcooled boiling regime the bulk temperature of the fluid is less than its saturation temperature. Due to latent heat transport, boiling heat transfer plays a very important role in wide number of applications in many technological and industrial areas including nuclear reactor cooling systems, car cooling and refrigeration systems.

Thus, in order to fully understand and predict the boiling phenomenon, the high gas volume fractions must be taken into account. Realizing this need, the GENTOP concept was utilized and further developed for flows with heat and mass transfer in this paper. It allows the modelling for bubbles smaller than the grid size and tracking the interface of large continuous bubbles (larger than the grid size). Thus, it is like a combination of Euler–Euler two fluid modeling and interface tracking techniques. It has been further advanced and validated for churn turbulent flow regimes (Montoya, 2014).

The concept has not yet applied to the situation involving transitions from bubbly flows to churn turbulent and then annular flows. This paper presents a simulation of a generic boiling phenomenon in a vertical pipe with the help of the GENTOP concept in ANSYS-CFX, where important new models have been discussed and applied.

## 2. CFD SIMULATION OF GAS-LIQUID TWO PHASE FLOWS

### 2.1 The generalized two phase flow (GENTOP) concept

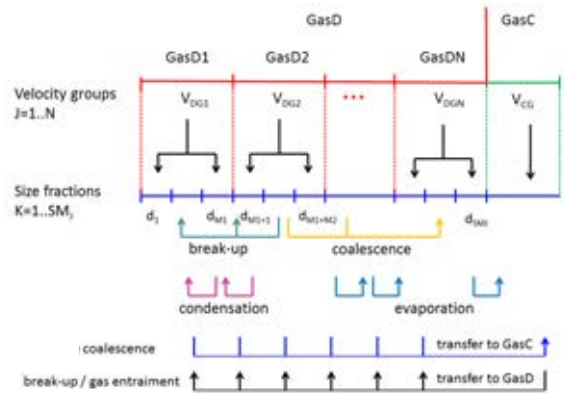


Figure 1 Scheme of the extended GENTOP model including phase transfer

The Generalized Two Phase flow (GENTOP) concept is based on a multi-field two-fluid approach. The flow is represented by a continuous liquid phase l, one or several poly-dispersed gas phases GasD and a continuous gas phase GasC.

The dispersed gas GasD is modelled in the framework of the inhomogeneous Multiple Size Group (iMUSIG) -approach to deal with different bubble size groups and associated velocity fields (Krepper et al. 2008). Within the poly-dispersed gas phases, transfers between different bubble size groups due to coalescence- and breakup as well as due to condensation and evaporation are taken into account by appropriate models.

The GENTOP concept has been developed as an extension of the inhomogeneous Multiple Size Group (iMUSIG) by adding a potentially continuous gas phase GasC which is included within the MUSIG framework. (Fig. 1). This last velocity group represents all gas structures which are larger than an equivalent spherical bubble diameter,  $d_{(dg,max)}$ . The interactions between GasC and the liquid phase are handled in a similar way like in the AIAD-concept (Höhne et al. 2011). This includes the blending for bubbly flow, interface and droplet regions allowing to apply e.g. for a low volume fraction of GasC closures for bubbly flow. For this reason it is called potentially continuous phase.

In the actual paper the GENTOP concept is extended by mass and heat transfer (Fig. 1).

## 2.2 Turbulence modeling

In terms of turbulence treatment, the dispersed phase zero equation is used for the dispersed gaseous phases, while the SST  $k-\omega$  approach is used for the liquid phase. One of the advantages of the  $k-\omega$  model over the  $k-\varepsilon$  is the treatment when in low Reynolds numbers for a position close to the wall. The effect of bubbles on the liquid turbulence is considered by additional source terms (Rzehak and Krepper, 2013).

## 2.3 Modeling of momentum transfer between the dispersed phases and liquid

Due to the averaging of the conservation equations all information on the interface is lost, but has to be reintroduced by the use of closure relations. The closure laws objective is to account for the mass and momentum transfer between the different fields and phases while providing the functional form expected from the interfacial forces. The present models are limited by the need of local condition dependent coefficients, derived from the fact that the closure laws have been developed for ideal bubbly flow and are now being applied to churn-turbulent flow and slug conditions.

Rzehak et al. (2015) have tested and successfully validated a number of poly-dispersed closure laws for Euler-Euler calculations and set up a so called Baseline Model for multiphase poly-dispersed bubbly flows (Table 1).

The total momentum exchange between dispersed gas and continuous liquid phase can be expressed as the superposition of several component forces (see Eq. 1).

$$\mathbf{M}_k^i = \mathbf{M}_k^D + \mathbf{M}_k^{VM} + \mathbf{M}_k^{TD} + \mathbf{M}_k^L + \mathbf{M}_k^W \quad (1)$$

In the baseline model (Rzehak et al., 2015) the drag force  $\mathbf{M}_k^D$  is calculated according to Ishii and Zuber (1979).

**Table 1:** Baseline model (Rzehak et al., 2015) for poly-dispersed flows used in GENTOP

Model	Name
Drag coefficient ( $C_{D,k}$ ),	Ishii and Zuber (1979)
Interfacial lift force	Tomiyama (2002)
Turbulent dispersion force	Burns (2004)
Wall lubrication force	Hosokawa (2002)

## 2.4 Handling of the potentially continuous phase GasC

### 2.4.1 Interface detection

To resolve the interface of continuous gas structures, the interface has to be localized. This is based on an appropriate blending function  $\Psi_{surf}$  (Gauß and Porombka, 2015). It bases on the volume fraction and its gradient and is designed in a generalized form capable for later applications describing not only bubble regions but also droplet regions. It replaces the blending taken from the AIAD model (Höhne, 2014) which was combined with a volume fraction based interface function in the original GENTOP concept of Hänsch et al. (2012).

The interface blending function is defined as

$$\psi_{FS} = \varphi_{sf}(f_b - f_d) \quad (2)$$

which is equal to zero for at a interphase boundary. Additionally, it provides information about the morphology:

$$\psi_{FS} = \begin{cases} 1 & \text{bubble region} \\ 0 & \text{interface} \\ -1 & \text{droplet region} \end{cases}$$

In the actual application only the bubble region and the interface region are of interest. The blending functions for the potentially continuous-phase bubble regime  $f_b$  and droplet regime  $f_d$  are given by:

$$f_b = \frac{1}{2} \left[ 1 + \cos \left( \pi \frac{\tilde{\alpha}^G - (\alpha_{b,crit} - \Delta_\alpha)}{2\Delta_\alpha} \right) \right] \quad (3)$$

$$f_d = \frac{1}{2} \left[ 1 + \cos \left( \pi \frac{\tilde{\alpha}^L - (\alpha_{d,crit} - \Delta_\alpha)}{2\Delta_\alpha} \right) \right] \quad (4)$$

The interface blending function is given by:

$$\varphi_{sf} = \frac{1}{2} \left[ 1 + \cos \left( \pi \frac{\nabla \tilde{\alpha}^c - (\nabla \alpha_{crit} - \Delta_{\nabla \alpha})}{2\Delta_{\nabla \alpha}} \right) \right] \quad (5)$$

### 2.4.2 Complete coalescence

During the calculation low fractions of dispersed gas in the region of mainly continuous gas might arise. To solve this unphysical situation a special coalescence method for complete gaseous mass transfer was established and is now included in the concept in order to replace the coalescence due to

the averaged coalescence models when the critical void fraction is reached. The coalescence rate is turning all the remained dispersed gas, within a specific grid cell, into continuous gas. The complete coalescence is turned off inside the interface in order to allow coalescence and breakup at those positions. The mass transfer is defined by:

$$S_{GasD \rightarrow GasC} = (1 - |\Psi|) \rho_{dg} \alpha_{dg} / \tau_{dg \rightarrow cg} \quad (6)$$

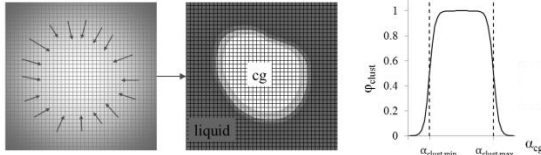
Where  $\tau_{dg \rightarrow cg} = \Delta t$  is a time constant that regulates how fast the mechanism occurs in consistency with the numerical scheme.

### 2.4.3 Clustering force

The clustering force (Figure 2) allows the transition from the dispersed towards the continuous gas phases using an aggregative effect within the volume fraction of the continuous gas. Modeling using an Eulerian approach will produce smearing of the volume fraction by numerical diffusion, thus this force produces interface stabilizing effects.

This force is additional interfacial force acting exclusively between the continuous gas and the liquid phase and is included in the interfacial momentum transfer. This force acts proportionally to the gradient of the volume fraction of the liquid as given in the following eq. (7)

$$M_{cg}^{clust} = \max(\psi_{clust}, 0) c_{clust} \rho_l \nabla \alpha_l \quad (7)$$



**Figure 2** Detail of a continuous gas liquid interface, and the blending function for a filtered interface (from Hänsch et al., 2012)

As soon as the specific critical void fraction of continuous gas is reached, this force will create regions of continuous gas volume fraction by inducing aggregation on the continuous gas phase volume fraction until a complete formation of gas structures is reached. The force acts outside the interface region, agglomerating the gas, and blends out as soon as the critical gradient of volume fraction appears, completely disappearing as soon as a fully formed interface occurs ( $\Psi_{surf} = 0$ ).

The clustering force disappears within the continuous structure. A constant value of  $c_{clust} = 1$  is recommended for the GENTOP application.

### 2.4.4 Interfacial momentum transfer

The Algebraic Interfacial Area Density (AIAD) model, shown in Höhne et al. (2014), allows

detection of morphological form of two phase flow and is able to distinguish between bubbles, droplets and the interface through a corresponding switching via a blending function of each correlation from one object pair to another.

Based on  $\Psi_{surf}$  (blending function), formulations for interfacial area density and drag are defined as in eqs. (8) and (9),

$$A_{GasC} = (1 - |\Psi_{surf}|) A_{fs} + a_{sign} |\Psi_{surf}| A_b + (1 - a_{sign}) |\Psi_{surf}| A_d \quad (8)$$

$$C_{D, GasC} = (1 - |\Psi_{surf}|) C_{D, fs} + a_{sign} |\Psi_{surf}| C_{D, b} + (1 - a_{sign}) |\Psi_{surf}| C_{D, d} \quad (9)$$

## 2.5 Phase change model for GasD and GasC

For the simulation of boiling, the thermal phase change model has been used for the disperse gas phase (GasD) and liquid pair and the continuous gas phase (GasC) and liquid pair.

In our case of heat transfer between liquid and gas, the use of overall heat transfer coefficient is not sufficient to model the interphase heat transfer process. This model considers separate heat transfer process on each side of the phase interface. This is achieved by using two heat transfer coefficients defined on each side of the phase interface.

The sensible heat flux to liquid from the interface is given as:

$$q_l = h_l (T_s - T_l) \quad (10)$$

Similarly, the sensible heat flux to gas from the interface:

$$q_g = h_g (T_s - T_g) \quad (11)$$

The fluid specific Nusselt number is given by:

$$Nu_l = \frac{h_l d_{lg}}{\lambda_l} \quad (12)$$

For spherical bubbles the Ranz Marshall correlation can be applied to calculate the Nusselt number. In the present simulation the Ranz Marshall (1952) correlation was used for the the disperse gas phase (GasD) and liquid pair. The Hughes and Duffey (1991) model uses the surface renewal theory and is used for the potentially continuous gas phase (GasC) and liquid pair.

The wall boiling model is only activated for the disperse gas phase (GasD) and liquid pair. Initially, water is below its saturation temperature. Water becomes supersaturated locally, leading to the formation of bubbles. The bubbles will start

departing and before the formation of next bubble, some of heat will go in superheating the water. This process is known as quenching. In regions of the wall not affected by bubble growth, wall heat transfer to the water is described by single phase convective heat transfer.

In the actual paper the wall boiling heat flux partitioning model developed at RPI (Kurul, 1991) and implemented in CFX with its basic submodels and parameters is applied. In the present paper the basic framework of the GENTOP concept is in the focus of interest. A detailed discussion of the aspects of wall boiling can be found in Krepper et al. 2013.

### 3 DEMONSTRATION CASE OF A WALL HEATED TUBE

To illustrate the previous described concept a demonstration example of a vertical side wall heated tube is given. The tube has a length of 0.5 m and a diameter of 0.025 m. Water is considered at a pressure of 1 bar. At this pressure the saturation temperature amounts to 372 K. The initial temperature was set to a subcooling of 3 K. The temperature of the heated wall is set to a superheating of 13 K. The inlet velocity is 0.2 m/s.

#### 3.1 Geometry, mesh and general setup

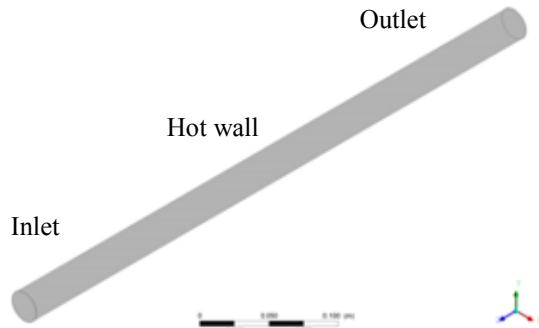


Figure 3 Pipe Geometry

The pipe is presented by a fully 3D geometry shown in Fig. 3 along with the name of the different zones (i.e., inlet, hot wall and outlet). The resulting mesh is made of approximately 127,300 hexahedral cells. A grid resolution study was conducted to ensure that convergence with respect to the spatial resolution has been achieved. A multiphase simulation was set up. Gas was described in the inhomogeneous poly-dispersed multiple size group (iMUSIG) framework by the dispersed gaseous phases GasD1 and GasD2 and the continuous gas phase GasC.

A total of four velocity fields, three gas and one for the continuous liquid were solved. Gas was assumed at saturation temperature. Properties of dry steam at saturation temperature have been taken from the steam tables. At the hot wall a wall boiling model generating GasD was applied. GasC then arise either by coalescence of GasD or by evaporation in the bulk. For the heat and mass transfer between gas and liquid in the bulk the implemented phase change models using the Ranz-Marshall correlation for the pair GasD/Liquid and the Hughes and Duffey (1991) model for the potentially continuous gas phase (GasC) and liquid pair were applied.

The following table 2 shows the numerical scheme used in the case:

Table 2 Solver setup

Advection scheme	Option	High Resolution
Transient scheme	Option $\Delta t$	Second Order Backward Euler 0.005 s
Convergence control	Timescale control Min./max. coeff. loops	Coefficient loops 4/50
Convergence criteria	Residual type Residual target	RMS 1e-04

#### 3.2 Overview of the settings and models used in the GENTOP framework

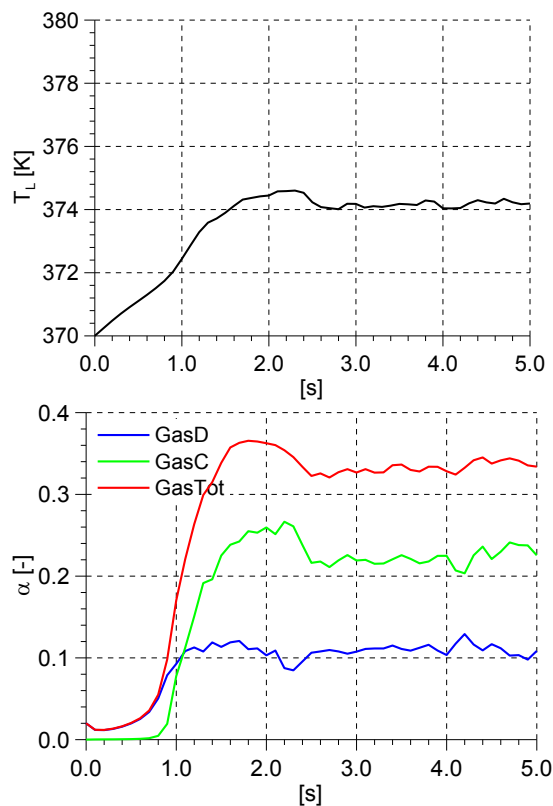
For the specified fluid water/steam at a pressure of 1 bar the critical bubble size, where the lift coefficient changes its sign, is found for  $d_B = 5.4$  mm. For GasD (dispersed gas) the iMUSIG model with 3 size fractions for GasD1 and 3 size fractions for GasD2 was applied. In this way the lift coefficient for GasD1 is clear different from the lift coefficient of GasD2. GasC was considered as last size fraction of the iMUSIG framework, to include it in the coalescence and fragmentation process. All gas structures equal or larger than 15 mm sphere equivalent diameter are assigned to GasC. The coalescence and breakup models according to Luo & Svendsen (1996) and Prince & Blanche (1990) with coefficients of  $F_B=0.01$  and  $F_C=2$  were applied. Momentum exchange between GasD and liquid was simulated considering all usually applied exchange terms for drag and non-drag forces were used. Concerning the drag between GasC and Liquid the formulation of AIAD was applied (Höhne 2014). The liquid phase was simulated as turbulent using the shear stress transport model. The influence of bubbles of GasD on the liquid

turbulence was considered. The exchange models were implemented using subdomains. Surface tension for the pair GasC and Liquid was implemented. Effects of numerical diffusion were compensated by an additional force, the Clustering force acting between GasC and Liquid to keep the interface between GasC and Liquid stable. The disappearance of unphysical fractions of dispersed gas in zones of prevailing GasC was enforced by complete coalescence. Concerning the turbulence of the liquid at the presence of an interface to GasC experiences with the AIAD model were used. Turbulence damping at the interface was considered and waves smaller as the grid resolution were treated as in the AIAD model.

### 1. RESULTS AND DISCUSSION

Figure 4 shows the time course of volume the averaged parameters in the whole flow domain. During the first 0.8 s only dispersed gas is generated by boiling (see Fig. 4b). After this time also continuous gas arises, mainly by coalescence of dispersed gas. After about 1 s the whole domain is heated up.

In Figure 5 the cross sectional averaged values of liquid temperature (a) and gas volume fractions (b to c) dependent on the height  $z$  are shown.



**Figure 4** Time course of the averaged liquid temperature (up) and the volume fractions for dispersed and continuous gas (down)

Figure 6 presents gas volume fractions for dispersed gas (GasD), continuous gas (GASC) and the sum of both (GasTot) after a heating time of 3.0 s. During this time a steady state oscillating period is reached (compare Fig. 4b).

At the beginning of the heating up process mainly small bubbles occur near the wall. The wall boiling model releases bubbles having a diameter of about 1 mm. By the agglomerative effect of the cluster-force and using the principles of the GENTOP-concept it is possible to create continuous gas structures out of a dispersed gas phase as demonstrated in Fig. 6. After the wall boiling of small bubble sizes the domain with the smallest bubble size group the dispersed gas phase is characterized by an increase of mean bubble diameter due to the coalescence processes in the MUSIG-framework.

When the mass transfer to the continuous gas begins and the volume fraction of GasC exceeds the threshold value  $\alpha_{cg} > \alpha_{clust,min}$ , here set to 0.5, the cluster-force agglomerates the continuous volume fraction until the complete coalescence replaces the dispersed gas fractions and large gas structures are resolved.

They further coalesce to larger gas structures forming distorted cap-bubbles and larger slugs represented in the picture (Fig. 6). In grid cells where the continuous gas volume fraction stays below the threshold value  $\alpha_{cg} < \alpha_{clust,min}$  the gas is treated as a dispersed phase following the particle model formulations.

Close observation of the GasD and GasC / Liquid interface show that the flow regimes discussed in chapter 1 except the annular mist flow regime can be found in the simulation.

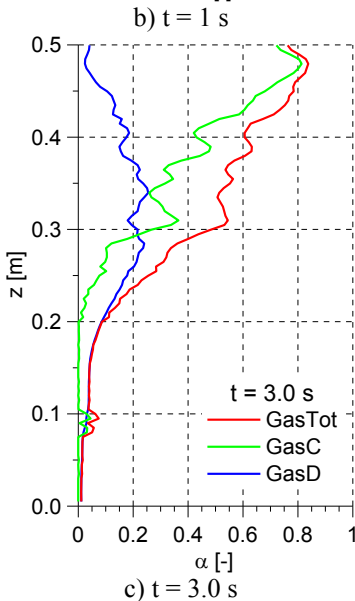
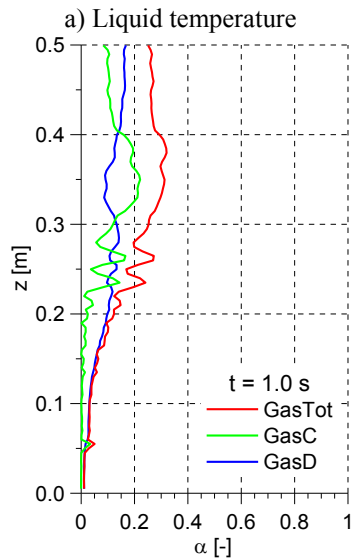
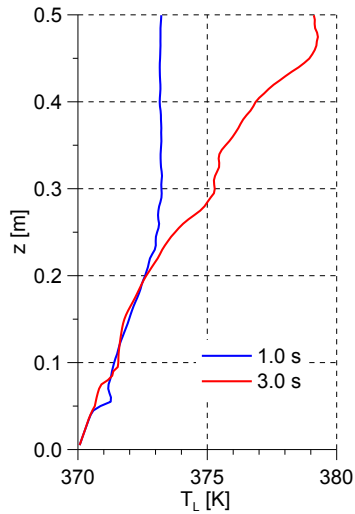
The bubble flow regime occurs at relatively low gas flow rates, for which the gas phase appears in the form of small bubbles in the lower part of the pipe.

Later Bubbly-slug flow is characterized by the presence of relatively large cap-shaped bubbles, which occupy nearly the entire pipe cross-section and flow alongside smaller, deformable bubbles.

The tendency to annular flow can be seen clearly. The churn turbulent flow appears to be highly chaotic and frothy and may seem to move upwards at some instants and downwards at other instants.

Also in the annular flow regime at the end of the pipe, one may notice the existence of a gas core and a relatively uniform annular liquid film on the pipe wall as well as liquid slugs. The annular film mostly moves upwards but occasionally may seem to pause. This pause occurs when a liquid slug fills the

local cross-section of the pipe, thus blocking the flow of gas in the core.



**Figure 5** Cross sectional averaged profiles for the liquid temperature (a) and the gas volume fractions for different times (b to c).

Shortly afterwards, however, the liquid slug gets penetrated by gas and the upward annular-type flow is resumed.

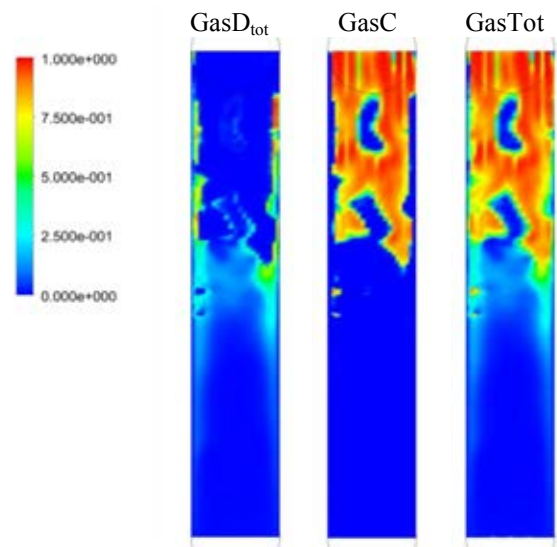
Fig. 7 represents essential GENTOP parameters at 2 s. The interface detection marks the identified interface. The cluster force is acting stabilizing the interface between GasC and Liquid. From the other side the surface tension force is acting in contradiction to the cluster force.

### SUMMARY AND FUTURE WORK

The GENTOP concept, which allows dealing with configurations involving dispersed and continuous interfacial structures, was coupled with a wall boiling model and extended to consider heat and mass transfer between gas and liquid in the bulk.

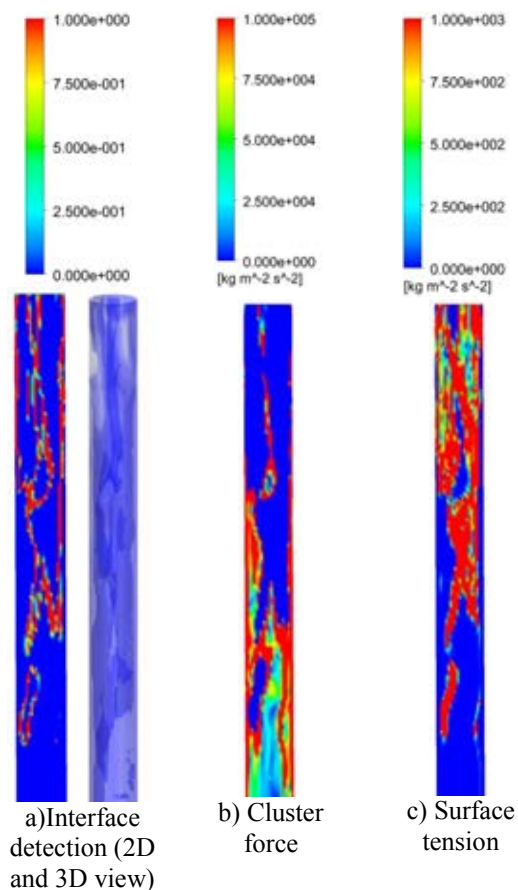
New model aspects of GENTOP were implemented and tested. Starting with a sub-cooled liquid in a hot pipe, bubbles (boiling) start to appear as soon as the liquid reaches its saturation temperature. Since the temperature of pipe wall is above the saturation temperature of the liquid, a series of flow regimes appear starting from bubbly flow, churn turbulent flow to annular flow. The simulation of the transitions between different flow regimes during boiling in a pipe is now feasible.

Next the demonstration case using the GENTOP-concept will follow experiments for a qualitative comparison of simulation results. The GENTOP sub-models and the Wall Boiling Model need a constant improvement and separate, intensive validation effort.



**Figure 6** Distribution of the gas volume fraction at 3.0 s (stretched height)





**Figure 7** Essential parameters of the GENTOP model framework at  $t=3$  s

## REFERENCES

- Burns, A.D., et al. The Favre averaged drag model for turbulent dispersion in Eulerian multi-phase flows. in 5th international conference on multiphase flow, ICMF. 2004.
- Gauß, F. and Porombka, P., Internal HZDR Communication, 2015
- Hänsch, S., et al., A multi-field two-fluid concept for transitions between different scales of interfacial structures. *International Journal of Multiphase Flow*, 2012. 47: p. 171-182.
- Hänsch, S.; Lucas, D.; Höhne, T.; Krepper, E., Application of a new concept for multi-scale interfacial structures to the dam-break case with an obstacle, *Nuclear Engineering and Design* 279(2014), 171-181
- Höhne, T. and J.-P. Mehlhoop, Validation of closure models for interfacial drag and turbulence in numerical simulations of horizontal stratified gas-liquid flows. *International Journal of Multiphase Flow*, 2014. 62(0): p. 1-16.
- Höhne, T.; Deendarlianto; Lucas, D., 2011. Numerical simulations of counter-current two-phase flow experiments in a PWR hot leg model using an area density model. *Int. J. Heat Fluid Flow* 32, Issue 5, 1047-1056.
- Hughes, E. D., Duffey, R. B. "Direct contact condensation and momentum-transfer in turbulent separated flows", *International Journal of Multiphase flow* 17, 599-619. (1991).
- Ishii, M.; Zuber, N. "Drag Coefficient and Relative Velocity in Bubbly, Droplet or Particle Flows". *AIChE J.*, 25: 843-855 (1979).
- Krepper, E., et al., The inhomogeneous MUSIG model for the simulation of poly-dispersed flows. *Nuclear Engineering and Design*, 2008. 238(7): p. 1690-1702.
- Krepper, E.; Rzehak, R.; Lifante, C.; Frank, T. 2013. CFD for subcooled flow boiling: Coupling wall boiling and population balance models, *Nuclear Engineering and Design* 255, 330-346
- Kurul, N. and M. Podowski. On the modeling of multidimensional effects in boiling channels. in *ANS Proceeding of the 27th National Heat Transfer Conference*. 1991.
- Luo, H. and H.F. Svendsen, Theoretical model for drop and bubble breakup in turbulent dispersions. *AIChE Journal-American Institute of Chemical Engineers*, 1996. 42(5): p. 1225-1233.
- Montoya, G., et al. Analysis and Applications of a Generalized Multi-Field Two-Fluid Approach for Treatment of Multi-Scale Interfacial Structures in High Void Fraction Regimes. in *Proc. Int. Congress on Adv. on Nucl. Power Plants. ICAPP2014-14230*, USA. 2014.
- Prince, M.J. and H.W. Blanch, Bubble coalescence and break-up in air-sparged bubble columns. *AIChE Journal*, 1990. 36(10): p. 1485-1499.
- Ranz, W. and W. Marshall, Evaporation from drops. *Chem. Eng. Prog.*, 1952. 48(3): p. 141-146
- Rzehak, R. & Krepper, E. CFD modeling of bubble-induced turbulence, *International Journal of Multiphase Flow*, 2013, 55, 138-155
- Rzehak, R.; Ziegenhein, T.; Liao, Y.; Kriebitzsch, S.; Krepper, E.; Lucas, D., Baseline model for simulation of bubbly flows, *Chemical Engineering & Technology* 38 (2015), 1972
- Tomiyama, A.; Sakoda, K.; Hayashi, K.; Sou, A.; Shimada, N.; Hosokawa, S., 2006. Modeling and Hybrid Simulation of Bubbly Flow. *Multiphase Sci. Tech.*, 18(1), 73-110.