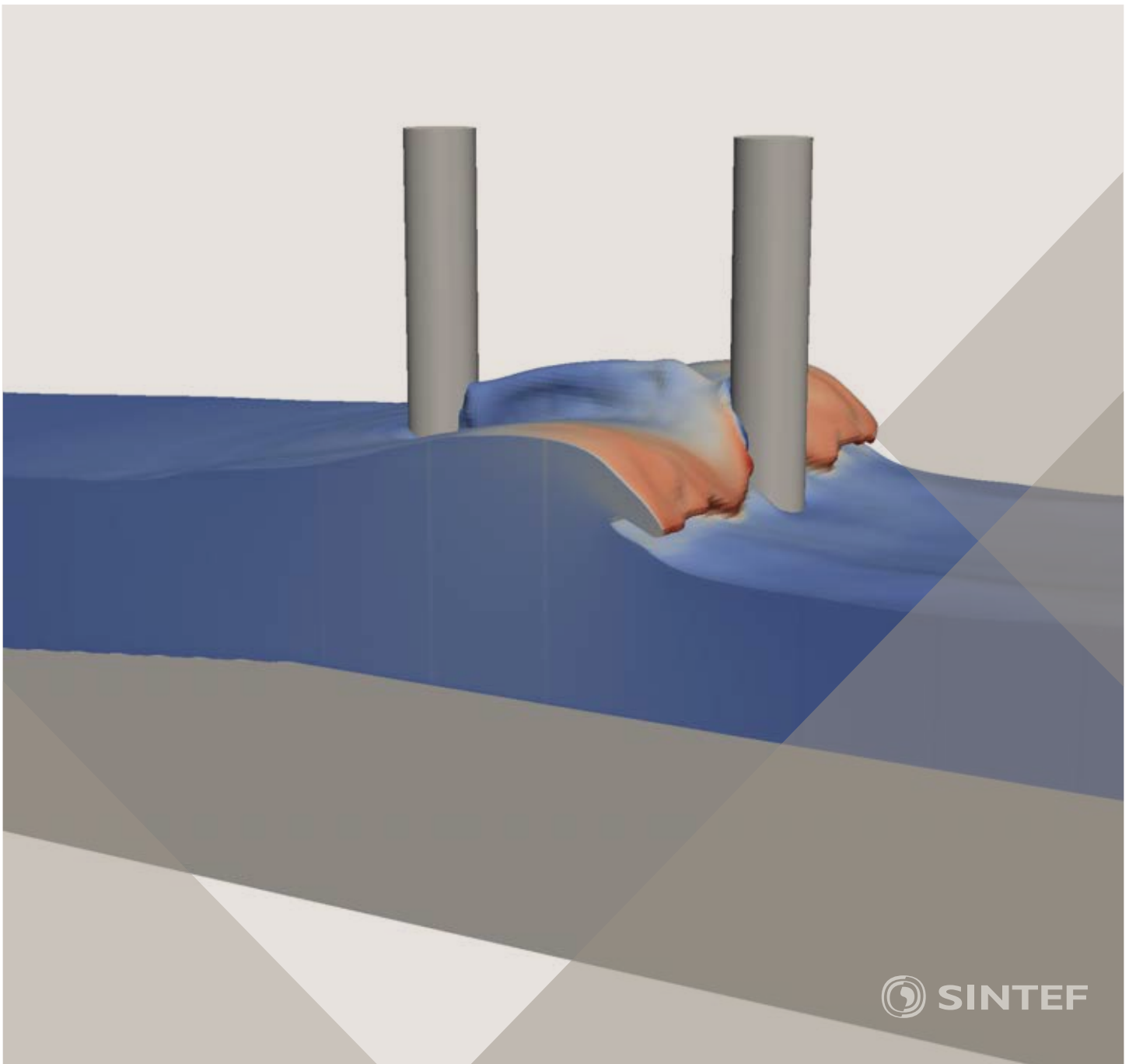


Proceedings of the 12<sup>th</sup> International Conference on  
Computational Fluid Dynamics in the Oil & Gas,  
Metallurgical and Process Industries

# 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 martensic 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 BASED STUDY OF DENSE MEDIUM DRUM SEPARATION PROCESSES

Arne EGGERS<sup>1\*</sup>, Wim DEWULF<sup>1†</sup>, Martine BAELMANS<sup>1‡</sup>, Maarten VANIERSCHOT<sup>1§</sup>

<sup>1</sup>KU Leuven Mechanical Engineering Department, 3000 Leuven, BELGIUM

\* E-mail: arne.eggerts@kuleuven.be

† E-mail: wim.dewulf@kuleuven.be

‡ E-mail: martine.baelmans@kuleuven.be

§ E-mail: maarten.vanierschot@kuleuven.be

### ABSTRACT

Dense Medium Drum (DMD) separation is applied particularly in the coal and recycling industries. Characteristic of the process is a separation based on density, driven by the buoyancy and gravitational forces acting on an object moving in a free surface flow. The fundamental phenomena occurring in a DMD have been widely investigated by (Napier-Munn, 1991). However, in contrary to other separation methods, such as the Dense Medium Cyclone which was investigated extensively by (Kuang *et al.*, 2014) and others, no Computational Fluid Dynamics (CFD) studies have been conducted for the DMD separation. Even recent studies, like (Meyer and Craig, 2015), use first order models which need to be calibrated with performance data of the investigated drum. Hence, important parameters like the flow-velocity and the detailed design of the drum are only taken into account indirectly.

This paper reports a detailed CFD analysis of the flow in a generic DMD separation process. The study comprises, a general understanding of the flow field and an analysis of the impact of different process parameters. Furthermore, the model predicts the actual separation performance of the DMD at different working points. To the authors' knowledge, this is the first study which conducts a CFD analysis of a DMD separation process. Therefore, findings concerning the flow field and its influence on the separation efficiency will be reported on. Moreover, the separation model can be used to derive the coefficients for highly used first order models without the need of experimental data. Like this, the early design phase of DMD separation processes can improve immensely.

**Keywords:** CFD, Process industries, separation, free surface flow

### NOMENCLATURE

#### Greek Symbols

$\varepsilon$	Relative error, [-]
$\eta$	Separation efficiency, [-]
$\mu$	Kinematic viscosity, [ $m^2/s$ ]
$\rho$	Mass density, [ $kg/m^3$ ]
$\phi$	Medium volume fraction, [-]
$\omega$	Turbulent Frequency, [ $1/s$ ]

#### Latin Symbols

$C_D$	Drag Coefficient, [-]
$d$	Diameter, [ $m$ ].
$dir$	Direction, [-].
$F$	Force, [ $N$ ].
$F_s$	Safety coefficient for the GCI, [-].

$g$	Gravitational acceleration, [ $m/s^2$ ].
$GCI$	Grid convergence index, [-].
$k$	Turbulent Kinetic Energy, [ $m^2/s^2$ ].
$m$	Mass, [ $kg$ ].
$n$	Number of objects, [-].
$p$	Pressure, [ $Pa$ ].
$po$	Order of grid convergence, [-].
$r$	Grid refinement factor, [-].
$Re$	Reynolds number, [-].
$t$	Time, [ $s$ ].
$T$	Viscous stress tensor, [ $N/m^2$ ].
$u$	Velocity, [ $m/s$ ].
$V$	Volume, [ $m^3$ ].
$x$	Location, [ $m$ ].

#### Sub/superscripts

$B$	Buoyancy,
$c$	Carrier fluid,
$cg$	Coarse grid,
$D$	Drag,
$eff$	Effective.
$i$	Index i.
$j$	Index j.
$p$	Particle or Object.
$turb$	Turbulent.
$r$	Relative.

### INTRODUCTION

In the environmental action plan for the circular economy, the European Union sets ambitious targets for recycling rates. This makes the development and optimization of current recycling processes mandatory. Dense Medium Drums (DMDs) are density separation machines frequently used in the coal and the recycling industry. In general the medium is a suspension of a high density powder with water. As illustrated in figure 1, objects with a density lower than the medium density float and leave the drum on the surface through the light fraction outlet. Other objects sink and are extracted from the drum by collectors or spirals which are driven by the rotation of the drum. Medium is supplied through inlet tubes and a small angle of inclination assures a constant stream towards the light fraction outlet of the drum. Although the coal industry applies DMD separation already for several decades, the approaches of modeling the process are restricted to first order models which get tuned by measurement data (Napier-Munn, 1991). These models predict for instance the behavior of the Wemco Drum, taking

different diameters of the separation objects into account (P.J. Baguley and T.J. Napier-Munn, 1996). The modeling of sub-processes like the DMD is accompanied by macroscopic analyses of the complete process in a plant (King, 1999). Furthermore, a recent study improves the modeling approaches by proposing a method, based on solving a system of differential equations, designed to be applied in the control of DMDs (Meyer and Craig, 2015). None of these studies directly investigates the flow in the drum and its influence on the separation process. This can only be achieved by utilizing experimental techniques or Computational Fluid Dynamics (CFD). DMDs for the application of plastic separation have been investigated experimentally (Dodbiba *et al.*, 2002). However, none of these studies applies CFD. CFD enables the possibility to base models for the design of DMDs on the flow simulation. This makes an optimization of the actual process during the design phase possible.

The modeling of a DMD based on CFD can be divided into two different major problems: The simulation of the free surface flow and the modeling of separation objects. Free surface flow simulation make currently mainly use of two models. The level-set method tracks the surface, directly storing its position in a distance field (Stanley Osher and James A. Sethian, 1988). Hence, a sharp interface is conserved. The second method makes use of the volume of fluid (VOF) method (C. W. Hirt and B.D. Nichols, 1981). One of the phases is marked using a coloring function which is advected passively with the local flow field. The interface is not necessarily sharp. To estimate its exact position an interface reconstruction, based on the distribution of the color function, is needed.

The modeling of the objects in flows can be characterized by its description of the object domain. Objects which are considered to be large compared to the grid cell size have to be resolved and simulated. These methods are physically accurate, on the one hand, but computationally expensive, on the other (G. Houzeaux *et al.*, 2010). Smaller objects, often referred to as particles, can be described as point mass (Dehbi, 2008). This makes the modeling of huge amounts of objects with reasonable computational costs possible.

In the present research a generic DMD will be simulated. The free surface is modeled using the VOF method (Omno Ubbink, 1997). Although the separation objects are considered to be large, the model describes them as point masses. To prevent nonphysical flow behavior due to the large point masses a one-way coupling, which neglects the influence of the objects onto the flow, is applied. Thus, the separation performance of the generic drum in dependency of different flow parameters can be estimated.

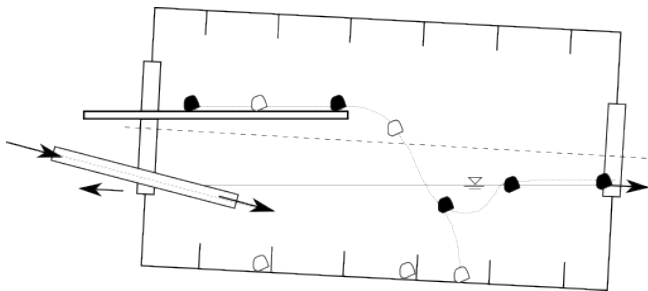


Figure 1: Sketch of a generic DMD.

## MODEL DESCRIPTION

The model has been implemented into the existing framework of the OpenFOAM toolbox. Hence, all equations and models described below are solved using this toolbox. The incompressible Reynolds averaged Navier-Stokes (RANS) equations are solved,

$$\frac{\partial \bar{u}_i}{\partial x_j} = 0, \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \frac{1}{\rho} T_{ij} - \overline{u'_i u'_j} \right) + g_i. \quad (2)$$

As these equations on their own are not closed, a  $k-\omega$  SST model is applied to model the Reynolds stress tensor and the turbulent viscosity. This model solves an extra set of equations for the turbulent kinetic energy and the turbulent frequency. A detailed description of it can be found in the literature (Menter, 1994). A finite volume scheme with a first order upwind interpolation was used for the spatial discretization. For the time derivation an Euler forward scheme has been applied.

In the investigated generic drum geometry the medium is a suspension out of Ferro Silicon powder and water. The model assumes that the powder and the water is in perfect suspension at any place and time and hence behaves as one continuum with constant fluid dynamical properties. The VOF method models the free surface between the air phase and the medium phase in the drum (Omno Ubbink, 1997). A color function  $\phi$  is utilized which indicates the volume proportion of the medium phase at a certain location of the drum.  $\phi = 1$  in the medium phase,  $\phi = 0$  in the air phase and  $0 < \phi < 1$  in the interface area. The color function advects passively with local flow field,

$$\frac{\partial \phi}{\partial t} + \bar{u}_j \frac{\partial \phi}{\partial x_j} = 0. \quad (3)$$

## Object Tracking

The Lagrangian equations of motion describe the movement of the objects in the flow,

$$\frac{dx_{p,i}}{dt} = u_{p,i}, \quad (4)$$

$$\frac{du_{p,i}}{dt} = \frac{1}{m_p} (F_{D,i} + F_{B,i}). \quad (5)$$

The summed forces in equation 5 consist out of two contributions: the drag force and the buoyancy force. Equation 5 is integrated using a first order explicit Euler scheme. The forces are modeled as described in the following.

### Buoyancy Force

The difference in density between the objects and carrier fluids introduces a buoyancy force which drives the separation of the objects. The buoyancy force can be calculated,

$$F_{B,i} = \rho_p \cdot V_p \cdot g_i \cdot \left( 1 - \frac{\rho_c}{\rho_p} \right). \quad (6)$$

### Drag Force

The drag force is modeled based on a drag coefficient,

$$F_{D,i} = \frac{1}{2} \cdot C_D(Re_p) \cdot |u_{r,i}| \cdot u_{r,i} \cdot A_p. \quad (7)$$

Here  $u_{r,i}$  is the relative velocity between the local flow field and the object.  $Re_p$  is defined as the particle Reynolds number,

$$Re_p = \frac{|u_{r,i}| \cdot d_p}{\mu} \quad (8)$$

The drag coefficient of a sphere can be estimated,

$$C_D(Re) = \begin{cases} 0.424, & \text{if } Re \geq 1000 \\ 24(1 + 0.15 \cdot Re^{0.678}), & \text{otherwise} \end{cases} \quad (9)$$

Equation 7 computes the forces acting on an object based on the averaged flow field from the solution of the RANS equations. As stated in previous studies one of the major influences on the separation process are the turbulent fluctuations of the flow which effects the objects in a stochastic way (P.J. Baguley and T.J. Napier-Munn, 1996). The computed averaged flow does not include any information about these fluctuations. However, the k- $\omega$  SST turbulence model conserves turbulent quantities, namely the turbulent kinetic energy and the turbulent frequency, which can be used to model the influence of the fluctuation statistically. Here, this is done using a popular Eddy Interaction Model (EIM) (Gosman and Ioannides, 1983). Without taking the turbulent fluctuations into account, objects with a density higher than the medium will get sorted as heavy fraction and all others as light fraction, if the drum is sufficiently large and the velocities small.

If an object enters the domain the forces are summed up and equation 5 is integrated. However, before the drag force is computed the turbulent fluctuations are added to the local velocity to calculate the effective local velocity,

$$u_{eff,i} = u_i + u_{turb,i} \quad (10)$$

The magnitude of the turbulent fluctuations is sampled from a Gaussian normal distribution, with an expectation value of 0 and a standard deviation of 1, and scaled with the expected magnitude,

$$|u_{turb,i}| = \sqrt{\frac{2k}{3}} \cdot Gauss(0, 1) \quad (11)$$

Turbulence models, as the applied k- $\omega$  SST model, utilizes an eddy viscosity hypothesis. As such, it assumes homogeneous turbulence. Hence, the same assumption is applied for the turbulent dispersion modeling leading to randomly sampled turbulent fluctuation,

$$dir_{turb,i} = \begin{bmatrix} rnd() \\ rnd() \\ rnd() \end{bmatrix} \text{ with } |dir_{turb,i}| = 1. \quad (12)$$

The turbulent fluctuation velocity can now be estimated,

$$u_{turb,i} = |u_{turb,i}| \cdot dir_{turb,i} \quad (13)$$

This process samples a new turbulent velocity when equation 5 has been integrated for the duration of one turbulent time scale,

$$t_{turb} = \frac{1}{\omega} \quad (14)$$

### Coupling

Although the objects in this model are considered to be large, they are still modeled as spherical point masses. As such a drag coefficient can model the influence from the flow onto the object and the simulation of large amounts of objects

is enabled. Several models exist to account for the influence from the objects onto the flow. However none of them can cope with large objects experiencing directional forces. Hence, this model utilizes, in a first pragmatic approach, a one-way coupling.

This might be a too strong assumption for a accurate simulation of the flow in the drum. However, the model is designed to deliver a reliable prediction of the separation efficiency of a DMD. In this context the assumption might be accurate enough. The algorithm updates the positions of the objects at first and then solves the flow and turbulence equations.

### Separation Performance

In classical separation modeling first order models generate separation curves. These curves describe the separation number, the probability that an object gets separated as heavy or light in dependency of the object density or its theoretical terminal velocity.

To achieve these curves based on a CFD calculation the paths of a huge amount of objects have to be tracked. The number of objects leaving the drum through the light and the heavy fraction outlet can be used to calculate the separation efficiency,

$$\eta_{heavy} = \frac{n_{heavy\ outlet}}{n_{heavy\ outlet} + n_{light\ outlet}}, \quad (15)$$

$$\eta_{light} = \frac{n_{light\ outlet}}{n_{heavy\ outlet} + n_{light\ outlet}} \quad (16)$$

For these equations it is assumed that all objects are supposed to be heavy fraction, in case of equation 15, or light fraction, in case of equation 16, independently from their density and the medium density. The efficiency is assumed to be converged if the statistical error is below 0.1%.

### RESULTS

In this study a generic drum geometry is simulated and its performance is predicted for different densities and object diameters. For the calculation of the separation efficiency it is assumed that all objects are supposed to be sorted as heavy fraction. Hence, equation 15 is utilized. Based on these results separation curves for different object diameters are computed. Furthermore, a grid study is conducted and the medium flow field is investigated.

### Geometry

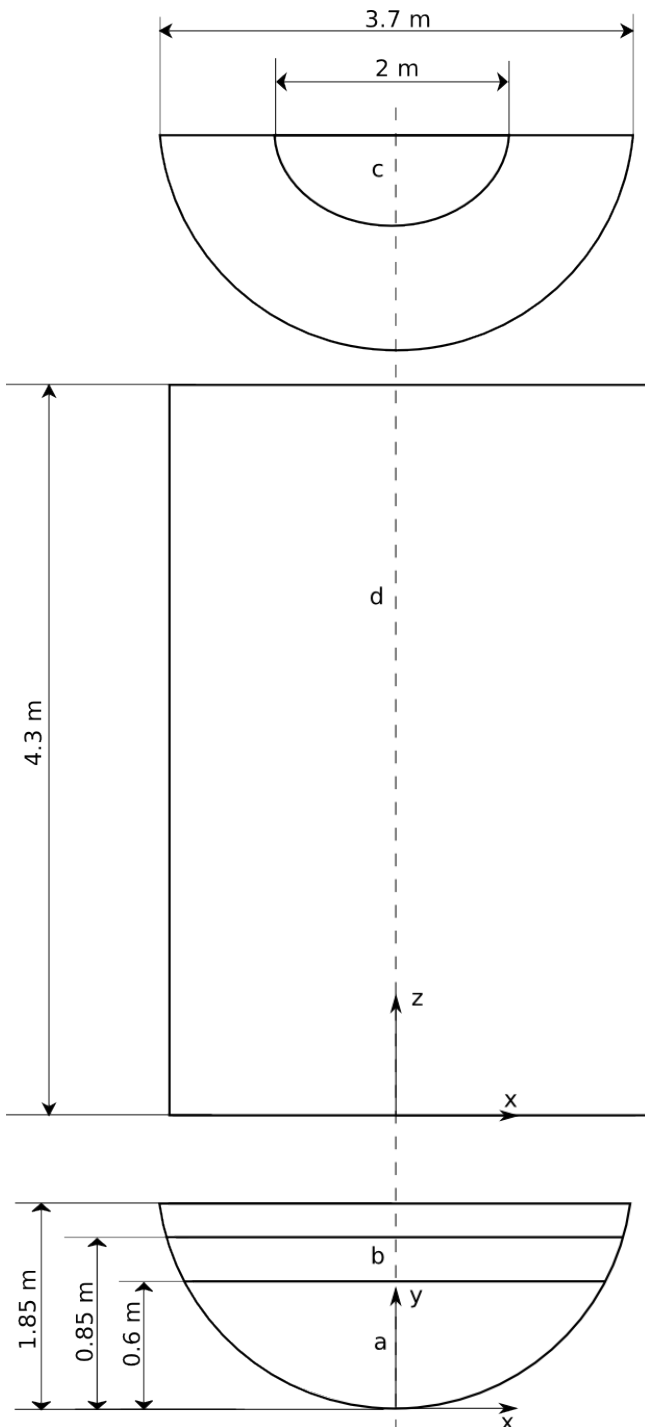
Major dimensions and a general overview on the geometry are given in figure 2. It is a simplified, generic version of a real applied drum. No collectors for the heavy fraction objects are taken into account and the drum is not rotating. The separation zone of the drum is long enough to ensure a sufficient residence time. The whole lower part of the inlet side, in figure 2 marked as a and b, is defined as medium inlet. Here medium enters the drum with a low velocity generating a slow and steady inflow. Separation objects are released through the middle part of the inflow section, in figure 2 marked as b. The complete curved wall of the drum, in figure 2 marked as d, is defined as heavy fraction outlet. Through this outlet only separation objects but no medium can leave the drum. On the outlet side of the drum the higher light fraction outlet is located, in figure 2 marked as c. This outlet allows medium and objects to leave the drum.

### Computational Grid

The computational grid, which is shown in figure 3, consists of 344640 cells. This structured grid is used for performance predictions. For the grid study a second grid with 849216 grid cells has been used.

### Grid Study

For the grid study two grids are used. At first the two flow fields of the medium phase are compared and then their influence onto the separation efficiency is assessed.



**Figure 2:** Sketch of the computational domain. With views on the light fraction outlet (c) (top), on the heavy fraction outlet (d) (middle), on the water inlet (a) and on the water objects mixture inlet (b) (bottom).

### Medium Flow Field

In figure 4 the medium velocity profiles in the symmetry plane of the drum ( $x=0$ ) are compared. After the medium enters the drum a steady flow field with a maximum velocity of approximately  $u_z = 0.25 \frac{m}{s}$  develops. In this area the solutions of both grids align very well and can be considered grid independent.

However, this changes as the medium passes through the drum and gets influenced by the outlet of the drum. Compared to the fine grid, the coarse grid underpredicts the outlet vortex and hence the back flow area in the lower part of the drum is smaller. Furthermore, the fine grid predicts a slightly higher peak velocity at the outlet.

### Separation Efficiency

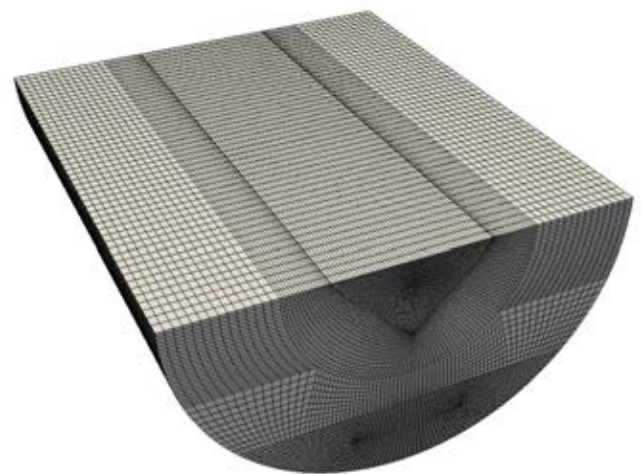
The grid convergence of the separation efficiency is assessed using the grid convergence index (GCI) (P. J. Roache, 1994),

$$GCI_{cg} = \frac{F_s \cdot |\epsilon| \cdot r^{po}}{r^{po} - 1} \quad (17)$$

The grid refinement factor is  $r = 3$ , the safety coefficient is  $F_s = 3$  and the order of grid convergence is assumed to be  $p = 2$ . The results are summarized in table 1. Although the medium flow field differs between the two grids, the GCI for the separation efficiency remains low. Hence, the separation efficiency can be examined using the coarsest grid. The GCI drops with increasing object density as the buoyancy forces increase while the drag force remains constant. The grid study has been conducted only for objects with a diameter of  $d = 0.005$ . It is expected that the grid influence is bigger for smaller objects as in these cases the grid independent buoyancy force is less important compared to the drag force.

### Medium Flow Field

The medium is entering the drum through the lower part of the inlet side, which is visible in figure 2 (a and b). The inlet velocity is defined to be  $u_{in} = 0.1 \frac{m}{s}$ . The combination of a large inlet area and a low velocity generates a slow and steady flow in the drum.



**Figure 3:** The computational grid used for the performance prediction.

**Table 1:** GCI and error in mean for objects of diameter  $d = 0.005$  and different densities.

Density ( $kg/m^3$ )	GCI (%)	error in mean
900	0.0	1.348e-05
990	2.531	5.07e-05
1000	1.635	0.0002
1020	0.329	0.0003
1030	0.075	0.0002

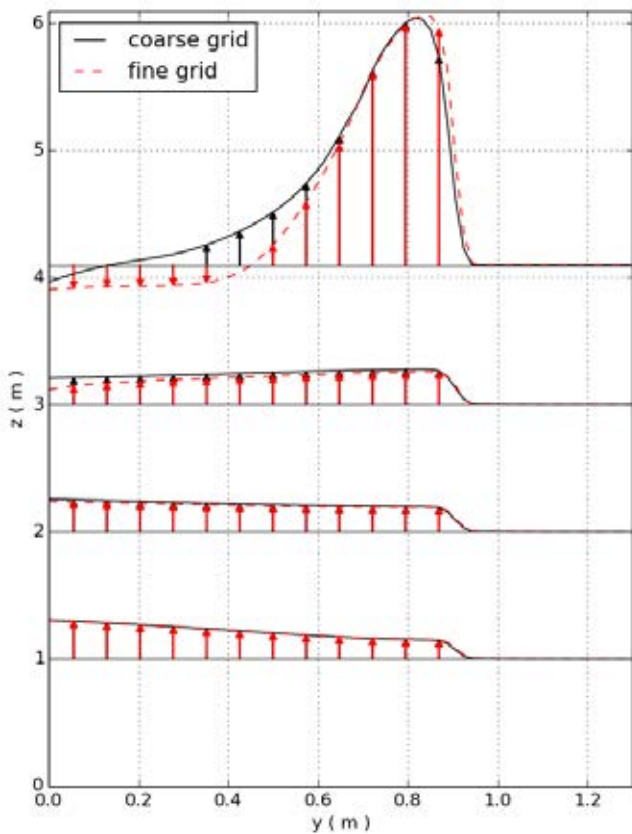
The smaller outlet drives an acceleration of the medium. As shown in figure 5, a vortex is created under the outlet. Separation objects which do not leave the drum through the light fraction outlet get dragged towards the heavy fraction outlet by this vortex. The acceleration is strongest in the symmetry plane ( $x=0$ ) and gets less in the side planes of the drum. The same trend is observed for the outlet vortex.

**Performance Prediction**

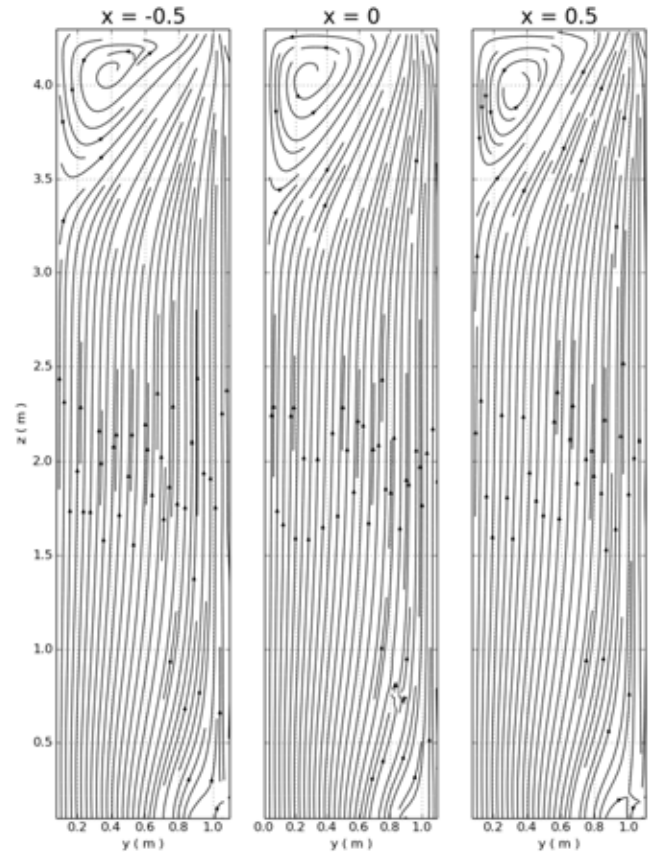
For the prediction of the separation performance of the generic geometry, depending on the flow and object properties, multiple hundred thousand of objects have to be traced on their path through the drum.

Figure 6 shows a typical convergence curve of the separation efficiency. The efficiency, which is calculated based on equation 15, starts with a value of one, as separation objects enter the drum over its full width. Hence, a small number of objects enter the computational domain close to the heavy fraction outlet and an even smaller amount gets dragged through it. As objects need approximately 15 seconds to pass the drum, no objects leave through the light fraction outlet at this point and an efficiency of one is computed. However, it drops

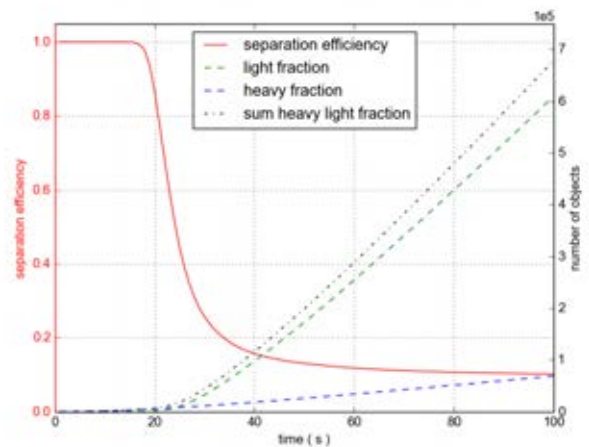
as soon as objects reach the light fraction outlet. Once the outflow of objects through the heavy and light fraction outlet stabilizes at a constant value, the computed separation efficiency will converge to a statistically constant value as well. As no object-object interactions and no back-coupling from the objects onto the flow have been taken into account, the mass flow of separation objects does not influence the predicted efficiency. The case shown in figure 6 reaches convergence approximately after 80 seconds and more than 500000 tracked objects. A simulation like this predicts a separation efficiency for a specified object density and diameter. Hence,



**Figure 4:** Medium velocity profiles in the symmetry plane ( $x=0$ ) of the drum on the fine and the coarse mesh.



**Figure 5:** Streamlines in three different planes in the medium zone of the drum. The outlet is in the top ( $z=4.3$ ) and the inlet in the bottom ( $z=0$ ) of the figure.



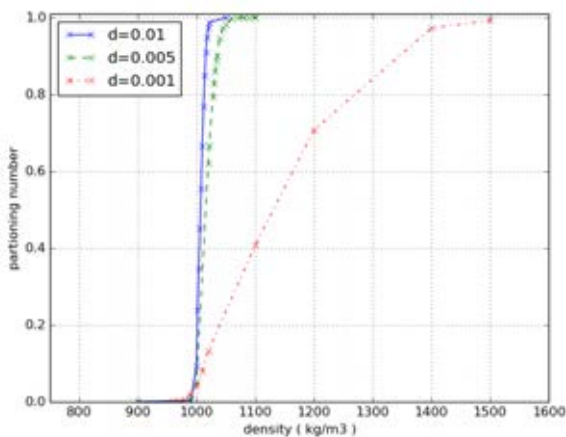
**Figure 6:** Computed separation efficiency and number of objects, which left the drum, over simulated time.

it delivers one point in a separation curve. If simulations are conducted for different densities and diameters the whole separation curve can be predicted.

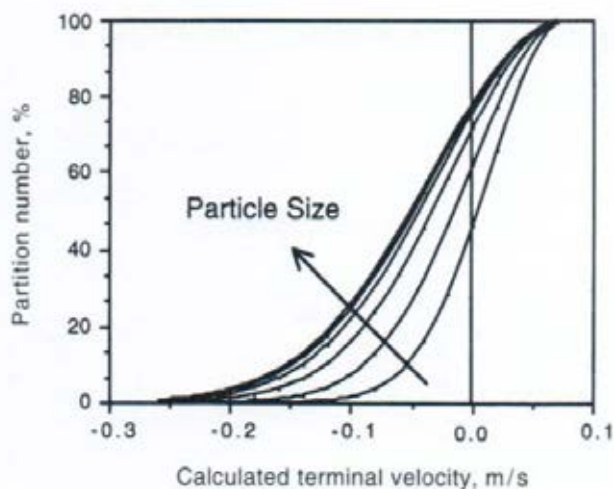
Figure 7 shows the predicted partitioning curves for the generic geometry of different object diameters. The typical S-shape and the influence of the object diameter size is recognizable in the same way as in figure 8. Furthermore, the generic drum slightly tends to separate objects as light fraction, although their density is higher than the medium density. This is due to the high medium mass flow which accelerates towards the outlet. However, particularly for the larger object diameters the generic drum has a high separation efficiency, which leads to a very sharp S-shape of the partitioning curve. For smaller objects the influence of the buoyancy force, which drives the separation, becomes less dominant compared to the drag force. Hence, the separation efficiency for these cases are much lower and the S-shape is more blurry.

## CONCLUSION

A CFD based model, which predicts the separation efficiency of a DMD has been proposed. It has been applied to a generic



**Figure 7:** Partitioning curves of the generic DMD for different object diameters.



**Figure 8:** Typical S-shape of a drum separation curve and the influence of different object diameters on it (P.J. Baguley and T.J. Napier-Munn, 1996).

and simplified drum geometry. As the CFD simulations deliver information on the general flow field in the drum, this has been investigated as well and key structures like the outlet vortex have been discussed.

The model reproduces the characteristic S-shape of separation curves for DMDs including its trend for objects with different size. Hence, in a next step a real DMD, for which geometrical data and partitioning curves are available, will be investigated. Thus, the performance of the model can be assessed. Furthermore, in real drums a suspension is used as medium. This enables sedimentation effects, which might influence the separation process.

## REFERENCES

- C. W. Hirt and B.D. Nichols (1981). "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries". *Journal of Computational Physics*, **39**, 201–225.
- DEHBI, A. (2008). "A CFD model for particle dispersion in turbulent boundary layer flows". *Nuclear Engineering and Design*, **238(3)**, 707–715.
- DODDIBA, G., HARUKI, N., SHIBAYAMA, A., MIYAZAKI, T. and FUJITA, T. (2002). "Combination of sink–float separation and flotation technique for purification of shredded PET-bottle from PE or PP flakes". *International Journal of Mineral Processing*, **65(1)**, 11–29.
- G. Houzeaux, C. Samaniego, H. Calmet, R. Aubry, M. Vazquez and P. Rem (2010). "Simulation of Magnetic Fluid Applied to Plastic Sorting". *The open Waste Management Journal*, **3**, 127–138.
- GOSMAN, A.D. and LOANNIDES, E. (1983). "Aspects of Computer Simulation of Liquid-Fueled Combustors". *Journal of Energy*, **7(6)**, 482–490.
- KING, R.P. (1999). "Practical Optimization Strategies for Coal-washing Plants". *Coal Preparation*, **20(1-2)**, 13–34.
- KUANG, S., QI, Z., YU, A., VINCE, A., BARNETT, G. and BARNETT, P. (2014). "CFD modeling and analysis of the multiphase flow and performance of dense medium cyclones". *Minerals Engineering*, **62**, 43–54.
- MENTER, F.R. (1994). "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA Journal*, **32(8)**, 1598–1605.
- MEYER, E. and CRAIG, I. (2015). "Dynamic model for a dense medium drum separator in coal beneficiation". *Minerals Engineering*, **77**, 78–85.
- Napier-Munn (1991). "Modelling and Simulating Dense Medium Separation Processes - A Progress Report". **4**, 329–346.
- Omno Ubbink (1997). *Numerical prediction of two fluid system with sharp interfaces*. Ph.D. thesis, Imperial College of Science, Technology & Science.
- P. J. Roache (1994). "Perspective: A Method for Uniform Reporting of Grid Refinement Studies". *Journal of Fluids Engineering*, **116**, 405–413.
- P.J. Baguley and T.J. Napier-Munn (1996). "Mathematical model of the dense-medium drum". **105(Jan-April)**, C1–C8.
- Stanley Osher and James A. Sethian (1988). "Fronts Propagating with Curvature-Dependent Speed: Algorithms Based on Hamilton-Jacobi Formulations". *Journal of Computational Physics*, **79**, 12–49.