V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

NUMERICAL SIMULATION OF THE FLOW FIELD AND THE SEPARATION BEHAVIOR OF HYDROCYCLONES

Steffen Schütz, Gabriele Gorbach, Kathrin Kissling and Manfred Piesche

Institute of Mechanical Process Engineering (IMVT), University of Stuttgart, Germany

Böblinger Straße 72, D-70199 Stuttgart, Germany

Phone: (+49) 711 685-85209, Fax: (+49) 711 685-85390

E-Mail: schuetz@imvt.uni-stuttgart.de, kissling@imvt.uni-stuttgart.de

Key words: Hydrocyclone, Computational Fluid Dynamics, Turbulence Models, Multiphase Flow Models, Population Balances, Free Boundaries

Abstract. *Hydrocyclones are often used in solid-liquid, gas-liquid and liquid-liquid separation processes. The separation effect depends on the resulting centrifugal forces in the hydrocyclone and on the density difference of the phases which are to be separated. The numerical simulation of hydrocyclones based on computational fluid dynamics (CFD) is a challenging task as many different physical effects and a complex flow field determine the flow and the separation behavior. This presentation gives an overview on the basic aspects of hydrocyclone modelling and simulation. The influences of the numerical grid, boundary conditions, turbulence models and multiphase flow models on the simulation strategy and results are discussed. A closer look is done on operating conditions which are coupled with particle coalescence and break-up and with the formation of a gas core within a hydrocyclone. Population balances and models for free boundaries are applied in these cases. The state of the art in literature is summarized and further tasks regarding hydrocyclone simulations are discussed.*

1 INTRODUCTION

The separation of dispersed solid particles, liquid droplets or gas bubbles, generally denoted as particles, from a continuous liquid phase is an essential unit operation in many fields of mechanical separation technology. Typical apparatuses used are filters, centrifuges and hydrocyclones. Whereas a high energy input is necessary using centrifuges at high rotational speed, hydrocyclones work more economically as the only amount of energy which has to be supplied is to overcome the pressure drop. A further advantage of hydrocyclones is their high operational reliability as they are simple in construction without any moving parts. Additionally the case of varying operational conditions, for example with unsteady volume flow rates, high separation efficiency can be achieved. When the volume flow rate is large, hydrocyclones can be set in parallel which is called a multicyclone battery. The first patent on hydrocyclones was granted to Bretney in 1891 [7].

Contrary to gas cyclones there is no complete separation of the continuous and the disperse phases. The feed flow (volume flow rate $\dot{V}_{in}$) is conducted through the mostly tangential inlet into the cyclone. The particles with the higher specific weight are transported to the cyclone wall due to the centrifugal field in the vortex flow. The separated particles are withdrawn at the underflow with a certain portion of the continuous phase (volume flow rate $\dot{V}_u$). The clarified liquid phase (which perhaps contains gas bubbles) leaves the hydrocyclone by the overflow (volume flow rate $\dot{V}_o$) through the vortex finder. In Figure 1, the geometry of a common hydrocyclone is sketched with the main fluid streams.

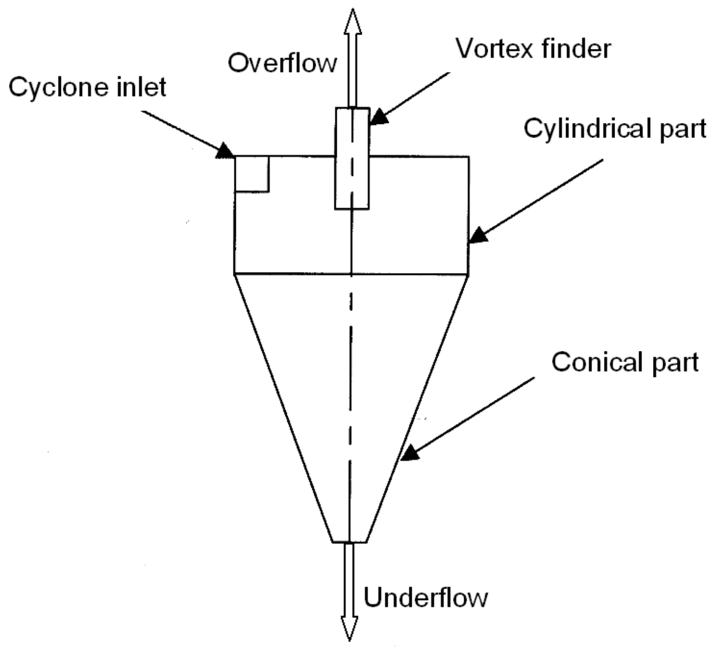

Figure 1: Basic design of a hydrocyclone.

The volume split k is defined as

$$k = \frac{\dot{V}_o}{\dot{V}_{in}} = 1 - \frac{\dot{V}_u}{\dot{V}_{in}}$$

and represents an important operational parameter.

Concerning the characterization of the separation behavior, the separation efficiency (or grade efficiency) is used in particle separation processes. The separation efficiency $T(d_p)$ with respect to particles of a certain size d_p is defined as

$$T(d_p) = \frac{\dot{m}_{p,u}(d_p)}{\dot{m}_{p,in}(d_p)}$$

with the mass flow rate $\dot{m}_{p,in}(d_p)$ of particles of size d_p in the feed flow and the particle mass flow rate $\dot{m}_{p,u}(d_p)$ at the underflow. As by this definition, a complete separation can be achieved by simply increasing the underflow without any real physical separation effect, another definition for the separation efficiency in a hydrocyclone is commonly used, the reduced efficiency $T'(d_p)$:

$$T'(d_p) = \frac{\frac{\dot{m}_{p,u}(d_p)}{\dot{m}_{p,in}(d_p)} - (1-k)}{k}$$

In this formula, the impact of the underflow is corrected by the volume split k. Therefore the real physical separation behavior is described by this definition.

2 COMPUTATIONAL GRID AND MODEL DIMENSIONS

The simulation of cyclone separators can be realized with two- and three-dimensional geometrical models. The application of a two-dimensional model implies the assumption of a rotationally symmetric flow field so local and temporal inhomogeneities are often damped out. The advantages of two-dimensional models are the lower effort in grid definition and the drastically reduced simulation time.

As discussed in the previous section, hydrocyclones are often equipped with a tangential inlet and the resulting flow field within the cyclone is three-dimensional when the feed stream hits the circulating flow within the cyclone. This effect cannot be described with a two-dimensional model and one has to deal with the problem how the feed flow is modeled with a boundary condition applying a two-dimensional model. One possible model approach is to assume a rotationally symmetric inlet around the whole cylindrical part of the hydrocyclone so that the inlet cross section is the same as for the real tangential inlet. In this case the inlet velocity remains the same as in reality but the height of the inlet in the simulation model is drastically reduced as the equivalence of the inlet cross sections in reality and in the rotationally symmetric model is required. However our own simulation runs have proven that this kind of boundary condition falsifies the flow field and the prediction of the pressure drop and the separation efficiency.

The other model approach is to assume the feed inlet with the same height in the rotationally symmetric model as in reality. This results in a bigger inlet cross section around the cylindrical part of the hydrocyclone than the real one. To keep the volume flow rate constant the inlet velocity must be reduced according to the bigger inlet cross section in the two-dimensional geometric model. With this simulation model results for the pressure drop and the separation behavior could be obtained which coincide well with experimental data.

The most prestigious way in hydrocyclone modeling is the definition of a three-dimensional model. The layout of the computational grid is more complex than for the two-dimensional models. However a three-dimensional model allows a precise description of the flow field close to the inlet region and the resolution of unsteady flow effects within the cyclone, e. g. an oscillating air or liquid core despite of steady-state boundary conditions.

The mesh definition should be done with some care. Figure 2 shows a horizontal cut through the calculation mesh in the upper part of a hydrocyclone with a tangential inlet duct [17, 45]. The regular structure of the grid cells is obvious. Especially close to the walls and in the inlet region a local grid refinement is advantageous to resolve the steep gradients in the state variables and to reduce the numerically induced dispersion. With this refined mesh, the particle transport behavior close to the cyclone walls is resolved well as a particle is assumed to be separated from the continuous phase when striking the wall. To avoid a numerical disturbance at the cyclone inlet, there is a smooth transition between the meshes in the tangential inlet duct and in the cyclone volume.

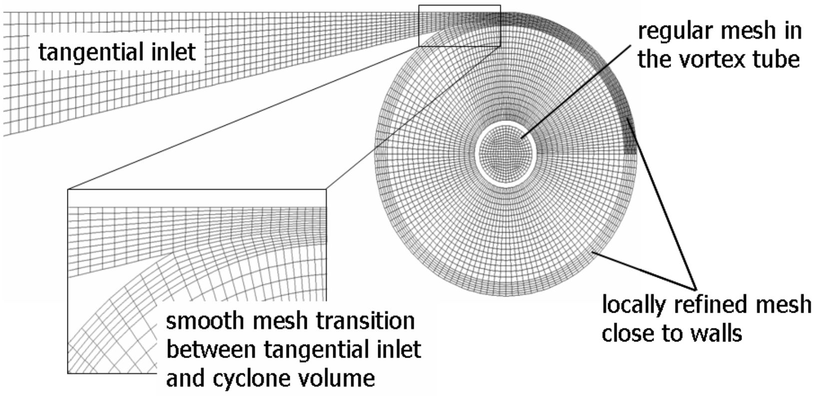

Figure 2: Cross-section of the simulation grid in the upper part of a hydrocyclone with the tangential inlet duct and with an adapted cell structure.

In Figure 3 an example for the sectional grid representation of the cyclone volume is given. This sectional representation allows the definition of regular and homogeneous grids for all hydrocyclone parts. The inlet part, the cyclone volume and the third part along the cyclone axis with the vortex finder are connected by the definition of interfaces on the contact surfaces. When the simulation grids from both sides are non-conformal at the interfaces interpolation techniques between the different node points are applied for the calculation of the state variables.

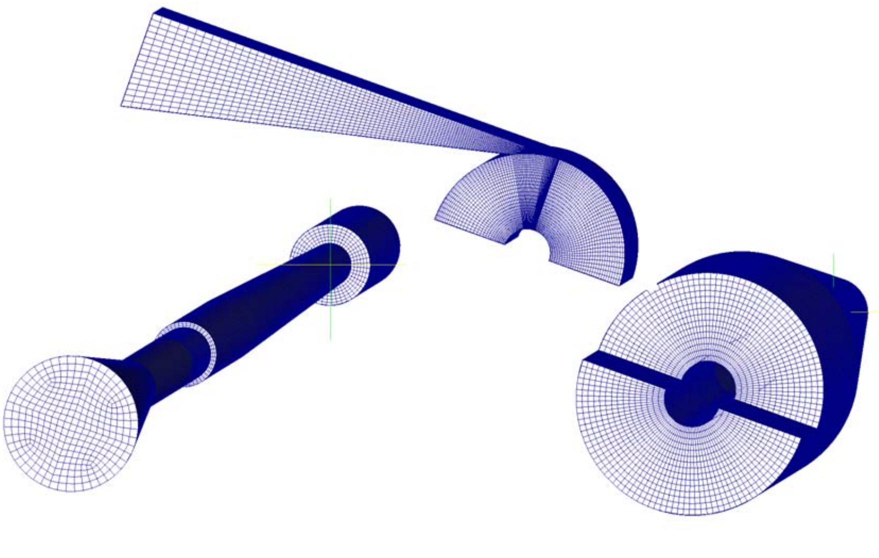

Figure 3: Sectional, three-dimensional representation of a simulation mesh for a hydrocyclone.

3 MATHEMATICAL SIMULATION MODELS

The mathematical description of the flow field, the particle transport and the separation behavior of a hydrocyclone comprises a complex set of model equations. The flow field of the continuous phase is described by the basic transport equations of fluid mechanics. Turbulence modeling is an essential topic with computational fluid dynamics in hydrocyclones as a broad range of turbulence models is available. Depending on the physical assumptions underlying these models some are quite suitable for hydrocyclone modeling whereas others should not be applied in this case. As hydrocyclones are used for the separation of different phases multiphase modeling is another core topic in this application. Various multiphase models are discussed with their ranges of application.

3.1 Basic Transport Equations

The flow field of the continuous phase within a hydrocyclone is described by the basic equations of fluid dynamics [3, 44]. The numerical solution of the mass and momentum conservation equations provide the pressure and the velocity dependent on position and time. Very often the flow field within a hydrocyclone is time-dependent with an oscillating behavior despite of the steady-state boundary conditions. The details of the time-dependent flow can only be resolved with sufficiently small time steps. The characteristic dimensionless number to capture time-dependent flow processes correctly is the Courant number Co. It is defined

$$Co = \frac{v\Delta t}{\Delta x}$$

with the advection velocity v, the numerical time step Δt and the grid cell length Δx. The Courant number gives the ratio between the path length of a single fluid element within one time step and the characteristic length of a cell in the numerical grid. For a good resolution of time-dependent effects $Co<1$ should be guaranteed at each time step and in each grid cell. When this requirement is fulfilled single fluid elements do not jump over single grid cells during the numerical solution procedure.

Usually the flow in hydrocyclones (and in gas cyclones) is assumed to be isothermal and the thermal energy balance is not necessary to be solved. Assuming an incompressible fluid phase with Newtonian behavior the viscosity is constant and neither a thermal state equation nor a rheological state equation is required. The mass and the momentum equations to be solved are the Navier-Stokes equations.

3.2 Turbulence Modeling

Turbulence effects are basically described by the Navier-Stokes equations, too. However the direct numerical simulation (DNS) of turbulence from the discretized Navier-Stokes equations requires a fine grid resolution with cell dimensions of a few micrometers. This cannot be applied to the simulation of real hydrocyclones as the data storage and the solution effort for the model equations cannot even be treated with the best supercomputers. So the turbulence transport effects must be described by turbulence models.

Applying the Reynolds- (time-)averaging procedure on the basic transport equations the Reynolds stresses are introduced as additionally unknown variables. The Reynolds stresses describe the increased momentum transport in turbulent flows and consider the anisotropic character of turbulence. The symmetric Reynolds stress tensor consists of six independent components altogether. Due to the swirling flow field within a hydrocyclone and due to the steep velocity and pressure gradients in radial direction the turbulence structure is highly anisotropic and turbulence shows a vector character. Turbulence models describe the local and time-dependent behavior of Reynolds stresses [19].

Due to its wide spread and its high numerical stability the standard k-ε-model [23] is sometimes used for the hydrocyclone simulation. Two transport equations for the turbulent kinetic energy k and for the dissipation rate ε of turbulent kinetic energy are solved. The k-ε-model is based on the principle of turbulent viscosity according to Boussinesq and therefore on the assumption of a locally isotropic turbulence structure which is not realistic for the strong vorticity in hydrocyclones. As simulation results, which rely on the standard k-ε-model, often deviate from experimental data the model parameters of the standard k-ε-model are sometimes adapted. However the physical character of the standard k-ε-model is then lost.

A more suitable two-equation turbulence model for the hydrocyclone simulation is the RNG-k-ε-model which is derived from the Renormalization Group Theory by Yakhot and Orszag [54]. Simulation results based on this turbulence model correlate well with experimental data with regard to velocity and pressure drop. Though the RNG-k-ε-model is a particular version of the standard k-ε-model locally different velocity gradients are considered explicitly in the terms of turbulence production and dissipation. The simulation quality is improved when the flow field close to the solid walls is described by a two-layer wall model.

An adequate and physically founded turbulence model for the simulation of the flow field of the continuous phase in hydrocyclones is the Reynolds stress model (RSM). With this model six transport differential equations for the six different components of the Reynolds stress tensor and a further equation for the dissipation rate ε of the turbulent kinetic energy are solved. The solution accuracy is very high and the anisotropic character of turbulence in hydrocyclones is resolved well. As the single terms in the Reynolds stress transport equations, e. g. for the redistribution of the turbulence intensity due to pressure fluctuations, have to be approached by further models there exists a number of different Reynolds stress models. However the computational effort for the numerical solution of the Reynolds stress transport equations is high and the convergence behavior within an iterative solution procedure is sometimes bad. The convergence can be stabilized by starting the simulation with a RNG-k-ε-model and switching to the Reynolds stress model after some time steps. A simplification is possible when the Reynolds stresses are calculated explicitly from algebraic equations (Algebraic Stress Model ASM) without the need to solve transport differential equations.

The large eddy simulation (LES) technique became more interesting in the last years with increasing computational power. The turbulence structures in the inertia and in the transition regime of the turbulence spectrum are resolved by the direct numerical simulation on a fine mesh. Therefore anisotropic turbulence structures and time-dependent flow effects can be calculated with an LES model accurately. Turbulent transport phenomena on the smallest length scales below the mesh size are modeled using a sub-grid-scale (SGS) model [48]. The computational effort for a LES based turbulence calculation is very high. Further improvements with LES models can be done when wall-influenced flows are described more accurately.

Figure 4 gives a numerical result for the flow field in a hydrocyclone with the three velocity components in tangential, axial and radial direction (from left to the right). Within this simulation the hydrocyclone geometry Z2 according to Table 1 was investigated. In this case the volume flow rate is 0.8 m^3/h with a volume split k=0.9. The simulation run was performed for a time-dependent flow with steady state boundary conditions. The Reynolds stress model was used as turbulence model [17]. The simulation results show an oscillating behavior of the state variables despite of the steady state boundary conditions. The commercial CFD software tool FLUENT™ was applied for the numerical solution of the hydrocyclone model.

Table 1: Geometric data for the hydrocyclones which are used for the numerical simulations in this text.

Geometric parameter	Hydrocyclone Z1	Hydrocyclone Z2
Total hydrocyclone height	312 mm	135 mm
Cylindrical diameter	50 mm	25 mm
Cone length	200 mm	82 mm
Vortex finder diameter	10 mm	8.5 mm

A simple procedure to judge the quality of simulation results is the direct comparison to experimental data. The pressure difference between the cyclone inlet and the cyclone outlet at the vortex finder is easy to investigate experimentally. From our experience we can deduce that the separation behavior of a hydrocyclone is well predicted when the pressure drop is calculated in good accordance with experimental data.

In Figure 5 a comparison is given for the pressure drop based on CFD results with 2D- and 3D geometric models, experimental investigations and analytical calculation results with semi-empirical models from Braun [5] and Meißner [29]. The volume split and the volume flow rates were varied. The underlying hydrocyclone corresponds to the geometric data of Z2 according to Table 1.

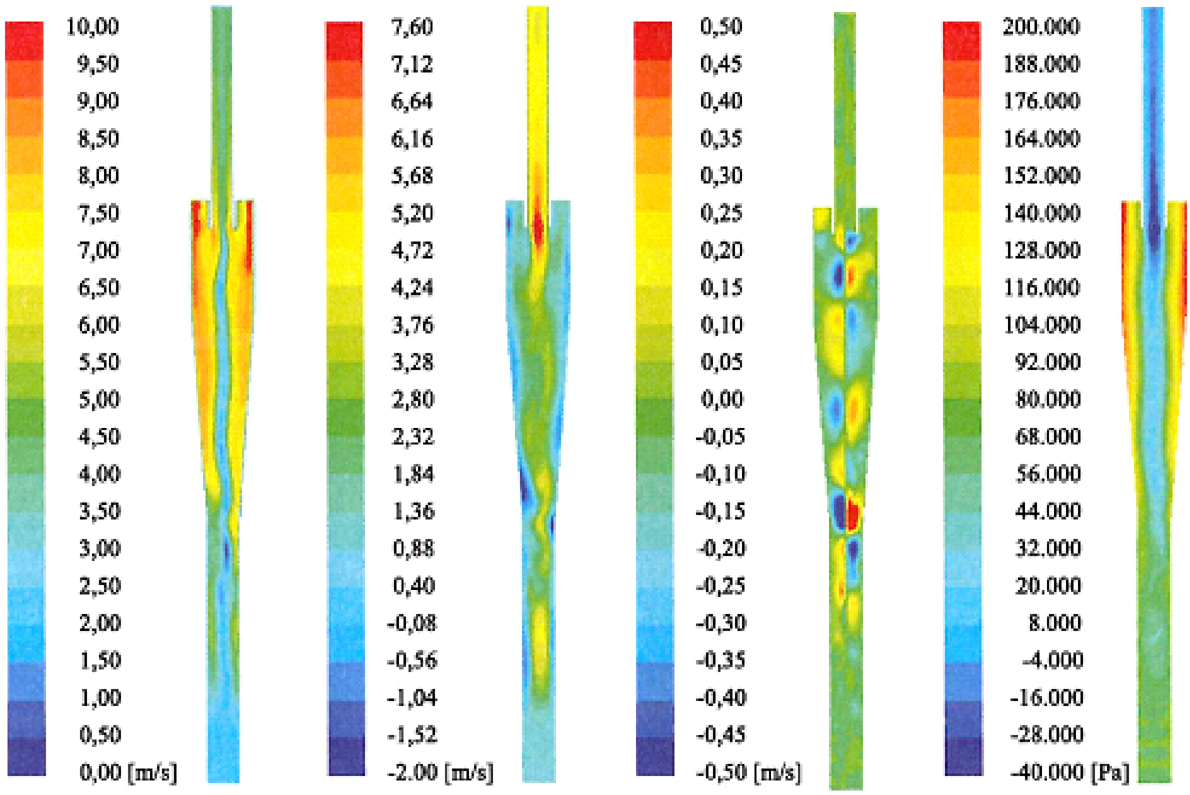

Figure 4: Time-dependent, oscillating flow field within the hydrocyclone Z2 according to Table 1 with steady-state boundary conditions. Volume flow rate 0.8 m^3/h, volume split k = 0.9. From left to right: tangential, axial and radial velocity and pressure field [17].

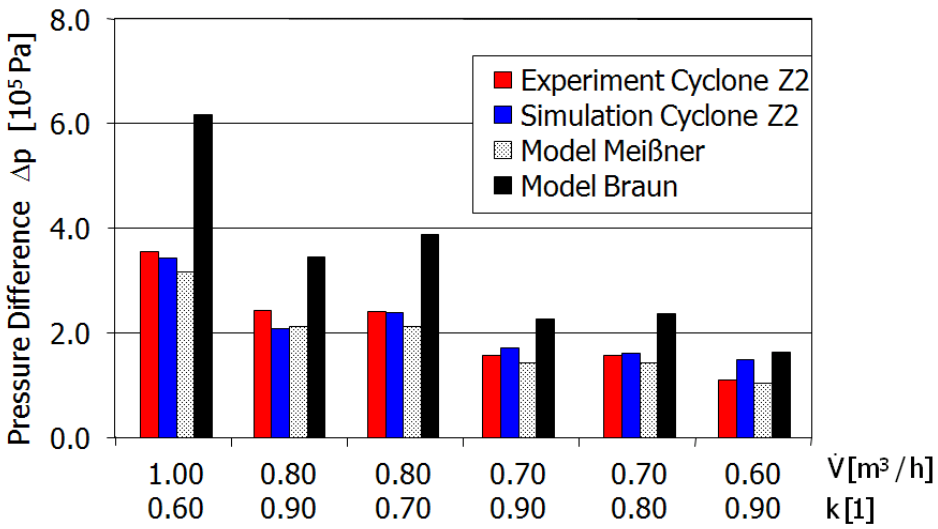

Figure 5: Pressure difference between inlet and overflow of the hydrocyclone Z2 according to Table 1. Comparison of CFD simulation results and results from semi-empirical design models for different operating conditions.

3.3 Boundary Conditions

The numerical solution of the model equations requires boundary conditions which specify the state variables at the inlet and at the outlet cross sections. A standard boundary condition is the mass flow rate or, equivalent for incompressible fluids, the volume flow rate or the corresponding mean fluid velocity at the inlet cross section. At the outlet cross sections (hydrocyclone underflow and overflow) either pressure values are predefined and the volume split k is calculated correspondingly or the volume split is predefined and a Neumann zero-gradient boundary condition is given for the state variables. Directly at solid walls the adhesion condition is valid.

The radial profile of the tangential velocity close to solid walls is described by wall models. Due to the cyclone swirl and the anisotropic turbulence intensity a non-equilibrium wall-model is more suitable than the standard equilibrium wall model. Another possibility is the definition of a very fine mesh close to the walls and a direct numerical simulation of the velocity boundary layer profile. With an approach like this particle-wall-interactions with respect to particle-wall-collisions, secondary particle transport effects due to dynamic lift forces perpendicular to the wall in the near-wall shear flow or the particle deposition on walls can be calculated with high accuracy. For the prediction of the separation efficiency of a hydrocyclone transport effects influenced by solid walls are very important.

4 SIMULATION MODELS FOR MULTIPHASE FLOWS

The simulation of a multiphase flow can be realized with a range of different model approaches whose choice is dependent on the properties of the multiphase system. Two main distinctive features are commonly used to characterize multiphase flow systems: the particle concentration and the occurrence of free interfaces.

4.1 Euler-Lagrange model for low particle concentrations

The multiphase flow within a hydrocyclone is often characterized by low concentrations of solid particles, droplets or gas bubbles with a maximum volumetric concentration of a few per cent. In these cases the Euler-Lagrange model is usually applied. The denomination of the Euler-Lagrange model is based on the different points of view by which the transport phenomena in the continuous fluid phase and the discrete particle phase are described. The calculation of the time-dependent or the steady state flow field of the continuous phase is performed from the viewpoint of an outstanding, quiescent Eulerian observer. In contrast, the simulation of the transport of single particles is defined from the viewpoint of a Lagrangian observer who is moving with a particle [50].

The basic idea of the Euler-Lagrange model is the concept of a volumeless mass point. Each real particle independent of its size or shape is assumed as a mass point losing its individual particle properties. The particle trajectory within a flow field is calculated from a force balance based on Newton's law. For deriving the force balance for arbitrary, non-spherical particles, the impact of the real shape can be considered with appropriate model parameters.

As the particles are not resolved volumetrically, the pressure and the viscous forces acting from a fluid phase on a particle's surface cannot be determined directly from the flow field around the particle by a surface integration. In fact one needs empirical or semi-empirical models to calculate the relevant forces affecting the particle transport.

Further wall boundary conditions have to be determined to calculate the particle separation in the hydrocyclone. A common wall boundary condition is to assume a particle separation when a particle hits the inner cyclone wall in the cylindrical or conical part. Then the trajectory calculation is stopped and the subsequent transport of a single particle towards the underflow is not regarded any more. Further forces acting on a single particle like electric or magnetic forces, dynamic forces from the flow field (Saffman and Magnus forces) or the added mass force in strongly accelerated fluids are modeled separately.

Two examples for the predictive capacity of an Euler-Lagrange model with respect to the grade efficiency curve of a hydrocyclone are given in Figure 6. The hydrocyclone geometries Z1 and Z2 correspond to the geometric data in Table 1. The good accordance between simulated and experimental data is obvious. Also the particle cut sizes calculated from analytical design models (Bradley [4], Meißner [29], Braun [5]) are included.

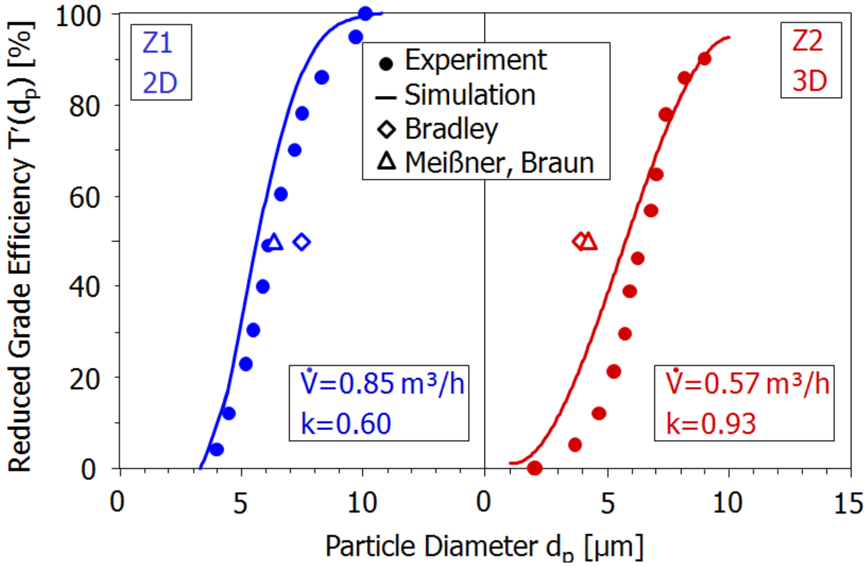

Figure 6: Grade efficiency curves for the hydrocyclones Z1 and Z2, dependent on the particle size. The solid lines and the dot symbols result from own simulation runs and experiments, the rhomb and the triangular symbols are calculated particle cut sizes from the semi-empirical models according to Bradley [4] and Meißner [29] and Braun [5].

4.2 Euler-Euler model for increased particle concentrations

With an increasing particle concentration, roughly speaking 5-10 volume-% or more, the tracking of single particles becomes very time-consuming and, additionally, particle-particle interactions have to be considered. Regarding, for instance, a large number of closely spaced particles, especially in the underflow of a hydrocyclone, the idea comes up to describe the particulate phase in terms of a continuous medium. Following up the matter leads to the description of motion of the fluid and the particles as though they were interpenetrating continua. Both phases, the fluid and the dispersed phase, are described from the viewpoint of an outstanding observer, which is the definition of the Eulerian approach. The advantages of the Eulerian consideration of multiphase flow can be derived from the motivation of introducing the model: A method exists to calculate multiphase flows at increased volumetric concentrations within an acceptable effort of time. Within these mixtures the “macroscopic” large-scale motions are obtained at the expense of averaging out the “microscopic” details. An averaging procedure of the basic transport equations can be done with respect to the volume of a single phase or to the total flow volume. This means that the detailed information about the motion of one particle or the interaction of one phase with another gets lost, which is clearly a disadvantage. Also the information on the dynamic behavior of phase interfaces is not preserved [56].

The simulation-based prediction of a grade efficiency curve of a hydrocyclone applying an Euler-Euler model is possible if the particle phase is subdivided into several discrete particle size classes. Each class can be regarded as a single Euler phase. The mass flow rate of each size class which leaves the hydrocyclone through the underflow is assumed to be separated.

Figure 7 gives a comparison for grade efficiency curves calculated on the base of an Euler-Lagrange and an Euler-Euler model. The solid concentration at the cyclone inlet is 5 vol-% [17]. The cyclone model is two-dimensional and the cyclone is charged with a volume flow rate of 2.1 m^3/h with a volume split k=0.9. In this case the results from the Euler-Euler model are in good accordance with the experimental data. The application of the Euler-Lagrange model overpredicts the separation efficiency of the hydrocyclone.

4.3 Multiphase Mixture Model

The multiphase mixture model is a simplified Euler-Euler approach. This model describes the multiphase flow with a single momentum balance with volumetrically phase-averaged fluid properties. For a multiphase system with N phases (one continuous phase and N-1 particle phases) N-1 phase related continuity equations are defined. The relevant state variable is the volume fraction α for each phase which is determined by these continuity equations. Further an overall phase averaged continuity equation is applied. The distribution of each phase is described by the local and time dependent volume fraction of each phase.

Besides the phase averaged velocity which is used in the phase averaged balance equations the so-called drift velocity of particles of a certain particle size class is defined as the difference between the absolute velocity of the disperse phase and the phase mean velocity. Further the relative velocity for each phase is used which describes the difference between the absolute velocity of a particle phase and the absolute velocity of the continuous phase. This relative velocity is usually calculated from an algebraic equation and the model approach is called algebraic slip model. The separation of single particles from the continuous phase in a hydrocyclone is characterized by this relative velocity which is the sedimentation velocity of the particles in the centrifugal field of the hydrocyclone.

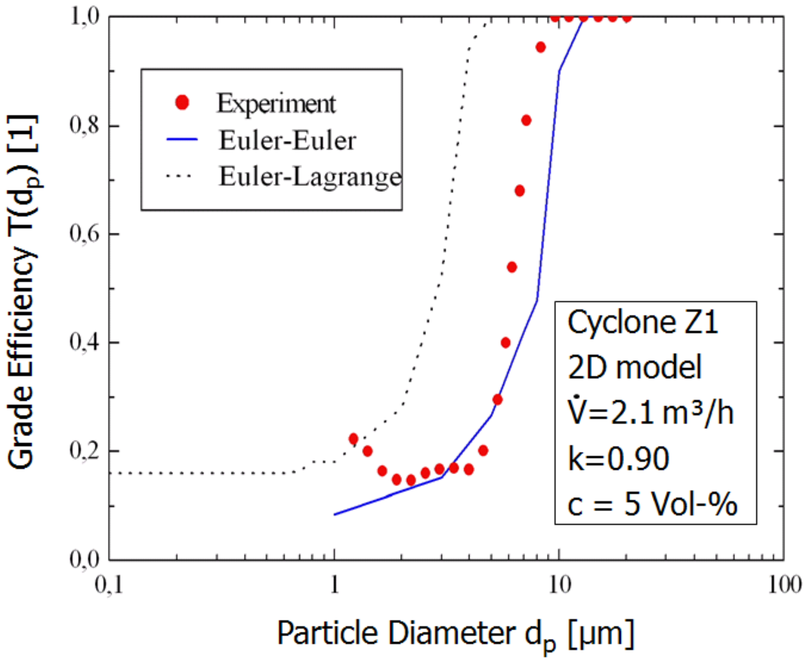

Figure 7: Comparison of experimental and simulation (Euler-Euler and Euler-Lagrange model) results for the grade efficiency curve of a hydrocyclone separation process with a solid particle concentration of 5 vol-% at the cyclone inlet.

4.4 Models for free Boundaries

Very often, the operating conditions in hydrocyclones are characterized by the appearance of an oscillating gas core with a free boundary along the cyclone axis. Usually the gas core arises with an increased pressure drop and with a direct connection of the cyclone volume with the ambient air atmosphere through the underflow or through the overflow. The dynamic behavior of free boundaries is not described by the aforementioned multiphase models. One of the most popular simulation techniques for free boundaries is the Volume-of-Fluid (VoF) method which was developed by Hirt and Nichols [20]. This method is based on a kind of multiphase mixture model. A single phase-averaged momentum balance and mass balance are solved as well as a continuity equation for N-1 phases when the multiphase system comprises N phases. If the volume fraction of one of the phases is one in a single cell of the computational mesh this cell contains no interface. If the volume fractions of two immiscible phases within a single cell are greater than zero and lower than one, then there is an interface within this cell.

The additional force in such a cell caused by the interfacial tension is treated as a source term in the phase-averaged momentum equation. For the formulation of this interfacial tension force the continuous surface force model (CSF) according to Brackbill is often used [6]. The surface tension force itself is obviously a surface force. To convert this surface force into a volumetric force as it is used in the momentum equation the Gauss divergence theorem is applied.

Apart from the formulation of the interfacial tension, we need to lay emphasis on the calculation of the convective and molecular fluxes across the faces of the single grid cells. Cells which are completely filled with one fluid can be treated as single-phase flow systems, so standard numerical schemes can be applied. The concentration therefore is laid on the cells which are located at the interface. Special numerical schemes are required which are important for the second step in the simulation of free boundaries – the reconstruction of the phase interface.

The historical interpolation schemes are the Donor-Acceptor scheme, introduced by Hirt and Nichols [20] and the Geometric-Reconstruction scheme. The Donor-Acceptor scheme is especially applied if numerical diffusion at the interface needs to be suppressed. The most commonly used interpolation scheme is the Geometric-Reconstruction scheme. The basic assumption is the stepwise linear approximation of the free boundary. In a first step the relative position of the interface is calculated by taking the volume fractions of the present phases and their local derivatives. The fluxes through the single cells are computed in a second step, as a function of the geometrical shape of the interface and the normal and tangential velocity components to the interface. Depending on all the previous information the new volume fraction in the cell is calculated and the interface is reconstructed. The Geometric-Reconstruction scheme is very stable and calculates sharp interface shapes, only for very high discrepancies between the properties of the phases, the interface affects to smear.

For those cases CICSAM (Compressive Interface Capturing scheme for Arbitrary Meshes) is applied, which was published by Ubbink [53]. In this algorithm the interface is tracked by solving a scalar transport equation. At the free boundary the scalar function shows a discontinuity, while being continuous in all other cases. A further interface reconstruction method is the Inter-Gamma-Differencing scheme which was proposed by Jasak and Weller [22]. The numerically induced smearing of the interface is suppressed by introducing an additional compression term into the continuity equation.

5 PARTICLE TRANSPORT EFFECTS

The modeling approaches discussed in the previous chapter describe the basic transport phenomena in multiphase flows. Regarding single particles in a multiphase flow more effects have to be considered to describe a detailed particle transport especially in a mechanical separation device. The main effects are summarized in Figure 8.

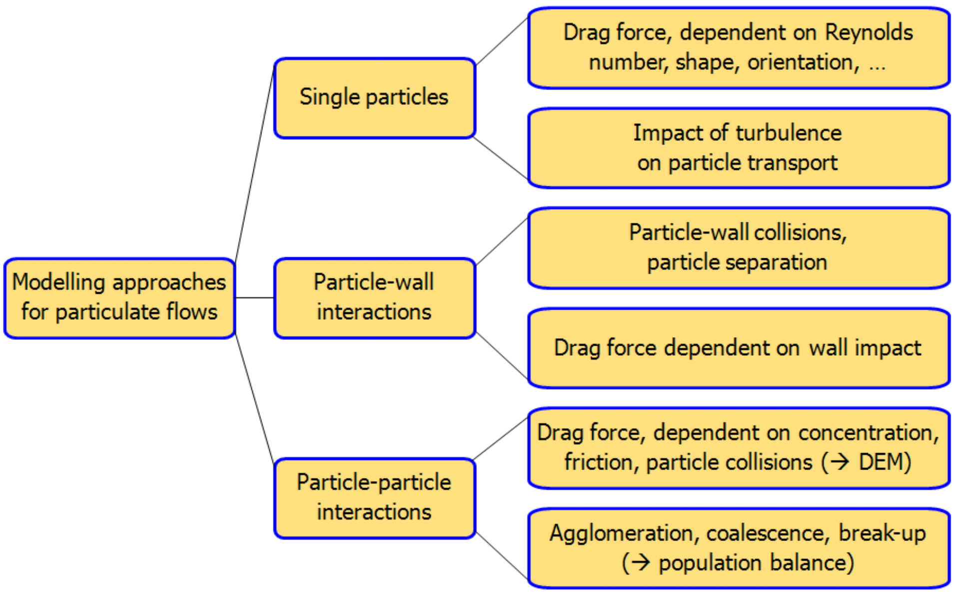

Figure 8: Transport effects with single particles.

The first step for the calculation of the transport and separation process of particles in a hydrocyclone is the definition of the particle size or the size distribution and the spatial distribution of the particles across the cyclone inlet. When an Euler-Euler model is used the volumetric concentration of the particles or of each particle size class must be known. Usually the particles are assumed to move without slip relative to the continuous phase at the hydrocyclone inlet.

Single particles are characterized by their transport behavior. The common Euler-Lagrange approach is originally valid for particles of any shape. The effect of the particle geometry is considered with the drag coefficient in the calculation of the drag force. However, regarding especially the separation process of particles close to the hydrocyclone walls the wall impact on the particle transport is important. A possibility to consider this effect is the calculation of drag forces for particles with wall impact by a volumetric particle resolution. This procedure was developed at the IMVT for some years [44, 46].

The particle transport is influenced by particle-particle and particle-wall collisions. When these collisions are described mathematically material parameters must be known e. g. the restitution coefficient to classify a collision as plastic or elastic. Further one should know whether particle deformation occurs or not which results in a different transport behavior of a particle after a collision.

Within hydrocyclones the transport of single particles is influenced by turbulence effects depending on the ratio of the characteristic time scale of turbulent oscillations and the relaxation time scale of single particles. So especially the transport of small particles with a size of a few microns and a low relaxation time is influenced by turbulence [51]. This stochastic process is described by particle turbulence models, e. g. the discrete random walk model (DRW), and the oscillating velocity of a particle is added to the mean advection velocity.

Another effect is the superposition of the separation process with particle coalescence and particle break-up. These processes occur either in solid-liquid separation with strong viscous or momentum forces or in liquid-liquid and gas-liquid separation. In these cases droplet and bubble break-up and coalescence determine the separation behavior. To consider these effects in a simulation model for hydrocyclones, the basic equations of fluid mechanics must be amended by population balances [21, 42, 17, 45].

Population balances are number balance equations which describe the number of particles of particular properties, e. g. of a specific size, dependent on time and position. Population models are defined with characteristic source and sink terms which describe the gain or loss of particles of a particular size through coalescence and break-up. With population balances different discrete particle size classes are defined and the population balances look equivalent to the continuity equations for single phases or discrete particle size classes in multiphase flow models. According to this equivalence population balance equations and multiphase flow models can be coupled: A change in the number of particles of a certain size is equivalent to a change in the volume fraction of the corresponding particle size class. So the increase and the loss of particles due to coalescence and break-up result in a local and time-dependent volume fraction of each regarded size class. Therefore the source and sink terms of population balances are also source and sink terms in single phase continuity equations.

In Figure 9 two results of simulation runs based on the coupling of CFD simulation and population balances are shown. The underlying fluid system consists of water droplets dispersed in diesel fuel. The water droplets should be separated from the diesel fuel in a hydrocyclone. The so-called real droplet size distribution is the volume flow rate weighted sum of the droplet size distributions at the cyclone underflow and at the cyclone overflow. It was defined to characterize the effects of droplet coalescence or break-up [30]. The black asterix symbols in Figure 9 indicate the water droplet size distribution at the cyclone inlet with a mean diameter of 300 microns. The real size distribution in the left diagram, shown in blue, was obtained with a conventional hydrocyclone with one tangential inlet. One can see that the break-up of the water droplets dominates as the real droplet size distribution is shifted towards smaller droplets compared to the droplet size distribution at the inlet. The experimental results (dot symbols) and the simulation results (solid line) show good accordance [17, 45].

The results in the right part of Figure 9 were obtained from a hydrocyclone with two tangential inlets opposed with an angle of 180 degrees [49]. The droplet size distribution from the inlet is shifted towards bigger droplets which can be separated more easily. In this case the droplet coalescence dominates the droplet break-up as the inertial and viscous forces in the cylindrical part of the hydrocyclone are drastically reduced with the modification using two inlets.

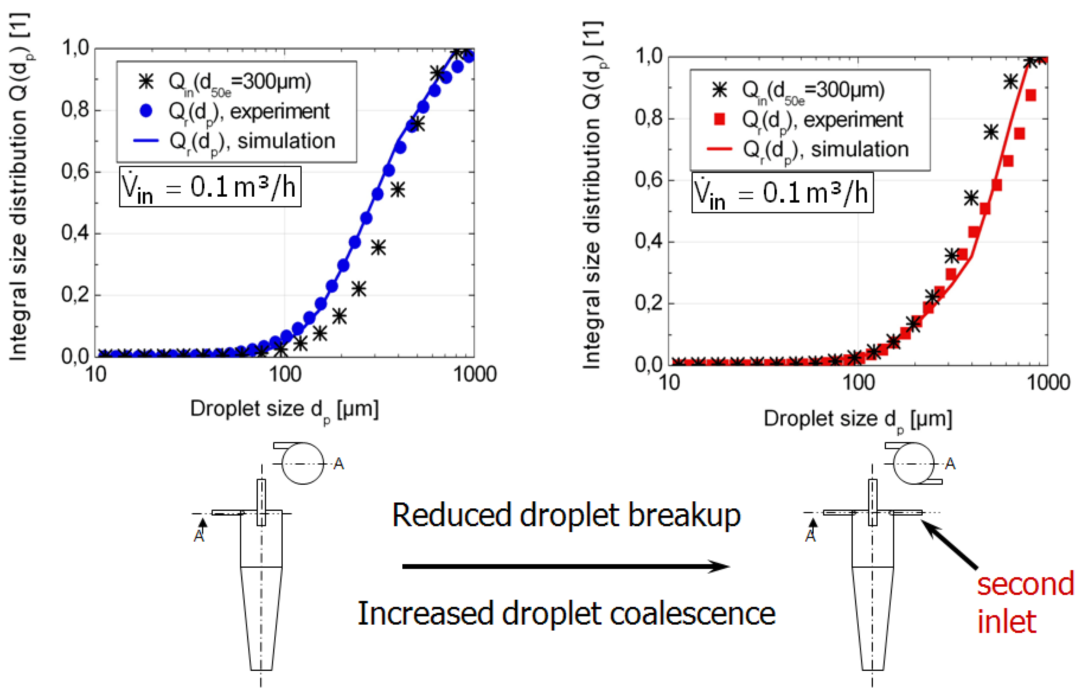

Figure 9: Real droplet size distributions in a hydrocyclone with net droplet break-up (left) and net droplet coalescence (right).

6 NUMERICAL SOLUTION ALGORITHMS

The numerical solution of the basic equations of fluid dynamics is mostly performed through the application of the Finite Volume method [41]. The computational domain is discretized with small but finite volume elements, the cells. The balance equations are solved for each element in an integral manner. This is done by integration of the transport differential equations with respect to each single cell volume and by applying the Gauss theorem to these volume integrals. With this procedure the convective and molecular transport terms are replaced by flux terms. This procedure guarantees the preservation of the conservation principle. The state variables at the center of each cell are representative for the whole cell.

The molecular and convective fluxes across the cell faces are calculated with an approximation for the state variables on the cell faces, as they are known at the cell centers. Common approximation schemes are of first (UPWIND scheme) or higher (UPWIND, Central Differencing, Power Law, etc. scheme) order accuracy. The choice of the approximation scheme determines the accuracy and the stability of the numerical solution [16].

For the solution of the transport equations themselves, segregated solvers are applied. The continuity and the momentum equations are solved with a pressure-correction algorithm. In a first step the pressure field is estimated and a first approximation of the velocity field is calculated which does not fulfill the continuity equation. Through an iterative procedure the pressure and the velocity fields are corrected. Subsequently other scalar transport equations (e. g. the energy balance or turbulence model equations) are solved [16].

The multiphase models can be coupled with the solution procedure in different manners. Regarding the Euler-Lagrange approach the particle trajectories are calculated on the base of the flow field of the continuous phase. With low particle concentrations the reaction of the particle transport on the flow field can be neglected. This is the so-called one-way coupling. Within a two-way coupling the reaction of the particles on the flow field of the continuous phase is considered. This means that the iteration procedure within the pressure-correction algorithm should only be applied until a certain degree of convergence. The particle trajectories and especially the drag forces of the particles are calculated. These drag forces are considered in the source terms of the momentum equation of the continuous phase and the corresponding velocity field is solved again by the pressure-correction technique. All equations for the continuous and the disperse phase must be solved till a pre-defined accuracy is reached.

Regarding the Euler-Euler approach, the corresponding transport equations for each phase must be included in the solution algorithm like the pressure-correction algorithm. Regarding models for free boundaries the transport equations are included like scalar transport equations which are solved after the application of the pressure-correction algorithm. However the numerical solution of the model equations in the VoF method and the reconstruction of the free boundary require additional specific algorithms.

7 LITERATURE SURVEY

In the reference list a range of papers is summarized dealing with the numerical simulation of hydrocyclones based on CFD [1, 2, 8, 9, 10, 11, 12, 13, 14, 15, 18, 24, 25, 26, 27, 28, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 43, 47, 52, 55]. Some investigations are of pure numerical character, others often deal with the comparison of hydrocyclones with various geometries with respect to pressure difference and separation behavior. Regarding the cyclone geometry tangential cyclones with a straight or spiral inlet duct are considered. Some papers deal with the time-dependent behavior of the flow field within a hydrocyclone.

For the simulation of the particle separation usually low particle concentrations are assumed. The reaction of the particle phase on the continuous phase is mostly neglected and the model equations are solved with a one-way-coupling technique. Some very detailed models consider particle-particle and particle-wall collisions and the interaction between particulate and continuous phase at increased particle concentrations.

The list of cited references constitutes an extraction from the multitude of published work on the simulation of hydrocyclone separators and makes no claim on completeness. Above all, papers are cited which were published within the last decade and which represent the state-of-the-art regarding the simulation models and the computational techniques. A survey on further research work dealing with the CFD based simulation of hydrocyclones, also from former years, was provided by Nowakowski and Doby [39].

In the main part of the scientific papers on hydrocyclone simulation and in the own work of the authors the commercial CFD software FLUENT™ was used. First simulation runs for hydrocyclones applying the OpenFOAM® software are performed actually.

8 SUMMARY AND OUTLOOK

This survey describes the various aspects which should be considered with the CFD-based simulation of hydrocyclones. It can be seen easily that the simulation of hydrocyclones is an ambitious task despite of the simple functional principle of these apparatuses. The combination of a vortex flow and the transport of dispersed particles is a challenging task with respect to the mathematical-physical models and the numerical solution strategies. Due to the increasing computational power during the last years some substantial progress was made.

An important task for the future time is the enhancement of the mathematical-physical models which describe particle-particle and particle-wall interactions for an improved prediction of particle separation processes. On the one hand this is important for the simulation of the transport behavior of small particles which are entrained by bigger particles and for the relative transport of particles with different physical properties as the particle shape or the particle density. Further research work is also to be done regarding separation processes with particle-particle interactions due to coalescence, agglomeration and break-up. Considering hydrocyclones, these effects occur especially in the regions of a dense multiphase flow, e. g. at the cyclone underflow.

Regarding the operational behavior of hydrocyclones with a free gas core modeling and numerical solution methods have to be improved and adapted to predict the separation characteristics under these operating conditions. These methods for free boundaries can also be applied to the investigation of the flow patterns at the cyclone underflow. Till now, there is almost no numerical model to capture the different flow regimes like roping, spraying or the formation of cords at the cyclone underflow. The reaction of the flow regime at the underflow is important to improve the separation efficiency of hydrocyclones.

Another separation phenomenon with hydrocyclones that cannot be explained sufficiently till now is the so-called fish-hook effect. This effect describes the increase of the grade efficiency curve with reduced particle sizes. It can be seen for example in the curve for the experimentally determined grade efficiency in Figure 7. A possible explanation is an agglomeration of particles close to the hydrocyclone underflow and the inclusion of smaller particles in the agglomerates. Another reason could be the pure entrainment of small particles by the bigger ones when leaving the hydrocyclone through the underflow. Detailed simulation models regarding particle-particle interactions more precisely could help to elucidate this phenomenon.

9 REFERENCES

[1] Bhaskar, K. U.; Murthy, Y. R.; Raju, M. R.; Tiwari, S.; Srivastava, J. K.; Ramakrishnan, N.: CFD simulation and experimental validation studies on hydrocyclone. Minerals Engineering 20 (2007), 60-71.

[2] Bhaskar, K. U.; Murthy, Y. R.; Ramakrishnan, N.; Srivastava, J. K.; Sarkar, S.; Kumar, V.: CFD validation for flyash particle classification in hydrocyclones. Minerals Engineering 20 (2007), 290-302.

[3] Bird, R. B.; Stewart, W. E.; Lightfoot, E. N.: Transport Phenomena. New York: John Wiley & Sons (1960).

[4] Bradley, D.: The hydrocyclone. London: Pergamon Press Ltd. (1965).

[5] Braun, T.: Theoretische und experimentelle Untersuchungen des Einflusses der Feststoffkonzentration und der Partikelgrößenverteilung auf das Trennverhalten von Hydrozyklonen. PhD Thesis Technical University Braunschweig, Germany (1989).

[6] Brackbill, J. U.; Kothe, D. B.; Zemach, C.: A continuum method for modeling surface tension. Journal of Computational Physics 100 (1992), 335-354.

[7] Bretney, E.: Water purifier. US Patent 543 105 (1891).

[8] Chu, K. W.; Wang, B.; Yu, A. B.; Vince, A.; Barnett, G. D.; Batnett, P. J.: CFD-DEM study of the effect of particle density distribution on the multiphase flow and performance of dense medium cyclone. Minerals Engineering 22 (2009), 893-909.

[9] Chu, K. W.; Wang, B.; Yu, A. B.; Vince, A.: CFD-DEM modeling of multiphase flow in dense medium cyclone. Powder Technology 193 (2009), 235-247.

[10] Cortés, C.; Gil, A.: Modeling the gas and particle flow inside cyclone separators. Progress in Energy and Combustion Science 33 (2007), 409-452.

[11] Cullivan, J. C.; Williams, R. A.; Cross, C. R.: Understanding the hydrocyclone separator through computational fluid dynamics. Transactions of the Institution of Chemical Engineers A 81 (2003), 455-466.

[12] Cullivan, J. C.; Dyakowski, T.; Williams, R. A.; Cross, C. R.: New understanding of a hydrocyclone flow field and separation mechanism from computational fluid dynamics. Minerals Engineering 17 (2004), 651-660.

[13] Delfos, R.; Murphy, S.; Stanbridge, D.; Olujic, Z.; Jansens, P. J.: A design tool for optimising axial-liquid-liquid hydrocyclones. Minerals Engineering 17 (2004), 721-731.

[14] Delgadillo, J. A.; Rajamani, R. K.: A comparative study of three turbulence closure models for the hydrocyclone problem. International Journal of Mineral Processing 77 (2005), 217-230.

[15] Delgadillo, J. A.; Rajamani, R. K.: Exploration of hydrocyclone designs using computational fluid dynamics. International Journal of Mineral Processing 84 (2007), 252-261.

[16] Ferziger, J. H.; Peric, M.: Computational methods for fluid dynamics. Berlin: Springer Verlag (2002).

[17] Gorbach, G.: Modellierung von Mehrphasenströmungen am Beispiel von Hydrozyklonen zur Auftrennung von Suspensionen und Emulsionen. PhD Thesis, Institute of Mechanical Process Engineering IMVT, University of Stuttgart (2007).

[18] Gupta, R.; Kaulaskar, M. D.; Kumar, V.; Sripriya, R.; Meikap, B. C.; Chakraborty, S.: Studies on the understanding mechanism of air core and vortex formation in a hydrocyclone. Chemical Engineering Journal 144 (2008), 153-166.

[19] Hanjalic, K.: Advanced turbulence closure models: A view of current status and future prospects. International Journal of Heat and Fluid Flow 15 (1994) 3, 178-203.

[20] Hirt, C. W.; Nichols, B. D.: Volume of fluid (VoF) method for the dynamics of free boundaries. Journal of Computational Physics 39 (1981), 201-225.

[21] Hulburt, H. M.; Katz, S. L.: Some problems in particle technology – A statistical mechanical formulation. Chemical Engineering Science 19 (1964) 8, 555-574.

[22] Jasak, H; Weller, H.: Interface tracking capabilities of the inter gamma differencing scheme. Technical Report, Imperial College London (1995).

[23] Launder, B. E.; Spalding, D. B.: Lectures in Mathematical Models of Turbulence. London: Academic Press (1972).

[24] Kraipech, W.; Nowakowski, A.; Dyakowski, T.; Suksangpanomrung, A.: An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone. Chemical Engineering Journal 111 (2005), 189-197.

[25] Kraipech, W.; Suksangpanomrung, A.; Nowakowski, A.: The simulation of the flow within a hydrocyclone operating with an air core and with an iserted metal rod. Chemical Engineering Journal 143 (2008), 51-61.

[26] Li, X. D.; Yan, J. H.; Cao, Y. C.; Ni, M. J.; Cen, K. F.: Numerical simulation of the effects of turbulence intensity and boundary layer on separation efficiency. Chemical Engineering Journal 95 (2003), 235-240.

[27] Mainza, A.; Narasimha, M.; Powell, M. S.; Holtham, P. N.; Brennan, M.: Study of flow behavior in a three-product cyclone using computational fluid dynamics. Minerals Engineering 19 (2006), 1048-1058.

[28] Mayer, G.; Lehmann, T.; Schütz, S.; Piesche, M.: Numerische Simulation des Betriebsverhaltens von Hydrozyklonen. Chemie-Ingenieur-Technik 75 (2003) 3, 227-232.

[29] Meißner, P.: Zur turbulenten Drehsenkenströmung im Zyklonabscheider. PhD Thesis, Technical University Karlsruhe, Germany (1977).

[30] Meyer, M.: Einfluß von Tropfengrößenverteilung und Konzentration der dispersen Phase auf die Flüssig/Flüssig-Trennung im Hydrozyklon. PhD Thesis, Technical University Braunschweig, Germany (2002).

[31] Narasimha, M.; Sripriya, R.; Banerjee, P. K.: CFD modelling of hydrocyclone – prediction of cut size. International Journal of Mineral Processing 75 (2005), 53-68.

[32] Narasimha, M.; Brennan, M.; Holtham, P. N.: Large eddy simulation of hydrocyclone – prediction of air-core diameter and shape. International Journal of Mineral Processing 80 (2006) 1, 1-14.

[33] Narasimha, M.; Brennan, M. S.; Holtham, P. N.: Numerical simulation of magnetite segregation in a dense medium cyclone. Minerals Engineering 19 (2006), 1034-1047.

[34] Narasimha, M.; Brennan, M.; Holtham, P. N.: Prediction of magnetite segregation in dense medium cyclone using computational fluid dynamics technique. International Journal of Mineral Processing 82 (2007) 1, 41-56.

[35] Narasimha, M.; Brennan, M. S.; Holtham, P. N.; Napier-Munn, T. J.: A comprehensive CFD model of dense medium cyclone performance. International Journal of Minerals Engineering 20 (2007), 414-426.

[36] Neesse, T.; Schneider, M.; Dueck, J.; Golyk, V.; Buntenbach, S.; Tiefel, H.: Hydrocyclone operation at the transition point rope/spray discharge. Minerals Engineering 17 (2004), 733-737.

[37] Neesse, T.; Dueck, J.: Dynamic modelling of the hydrocyclone. Minerals Engineering 20 (2007), 380-386.

[38] Nowakowski, A. F.; Kraipech, W.; Williams, R. A.; Dyakowski, T.: The hydrodynamics of a hydrocyclone based on a three-dimensional multi-continuum model. Chemical Engineering Journal 80 (2000), 275-282.

[39] Nowakowski, A. F.; Doby, M. J.: The numerical modelling of the flow in hydrocyclones. KONA Powder and Particle Journal 26 (2008), 66-80.

[40] Ovalle, E.; Araya, R.; Concha, F.: The role of wave propagation in hydrocyclone operations. I: An axisymmetric streamfunction formulation for a conical hydrocyclone. Chemical Engineering Journal 111 (2005), 205-211.

[41] Patankar, S. V.: Numerical heat transfer and fluid flow. Hemisphere Publishing Corporation (1980).

[42] Ramkrishna, D.: Population balances – Theory and application to particulate systems in engineering. San Diego: Academic Press (2000).

[43] Schütz, S.; Mayer, G.; Bierdel, M.; Piesche, M.: Investigations on the flow and separation behavior of hydrocyclones using computational fluid dynamics. International Journal of Mineral Processing 73 (2004) 2-4, 229-237.

[44] Schütz, S.; Kissling, K.; Schilling, M.; Seyfert, C.: Numerical simulation of multiphase flow systems in Martin, S. and Williams, J. R. (eds.): Multiphase flow research. New York: Nova Science Publishers, Inc. (2009), 1-146.

[45] Schütz, S.; Gorbach, G.; Piesche, M.: Modeling fluid behavior and droplet interactions during liquid-liquid separation in hydrocyclones. Chemical Engineering Science 64 (2009), 3935-3952.

[46] Schilling, M.: Transport processes of volumetrically resolved discrete particles using dynamic adaptive grids. Proceedings V European Conference on Computational Fluid Dynamics ECCOMAS CFD Lisbon, Portugal (2010).

[47] Slack, M. D.; Del Porte, S.; Engelman, M. S.: Designing automated computational fluid dynamics modelling tools for hydrocyclone design. Minerals Engineering 17 (2004), 705-711.

[48] Smagorinski, J.; General circulation experiments with the primitive equations – I. The basic experiment. Monthly Weather Review 91 (1963), 99-164.

[49] Smyth, I. C.; Thew, M. T.; Colman, D. A.: The effect of split ratio on heavy dispersion liquid-liquid separation in hydrocyclones. Proc. 2nd International Conference on Hydrocyclones (1984), 177-190, Bath, UK.

[50] Sommerfeld, M.: Modellierung und numerische Berechnung von partikelbeladenen turbulenten Strömungen mit Hilfe des Euler/Lagrange Verfahrens, Habilitationsschrift, Universität Erlangen-Nürnberg (1996).

[51] Sommerfeld, M.; Ho, C. A.: Numerical calculations of particle transport in turbulent wall bounded flows. Powder Technology 131 (2003) 1, 1-6.

[52] Stanbridge, D.; Swanborn, R.; Heijckers, C. P.; Olujic, Z.: Validation of a CFD model of a novel recycle axial flow cyclone for droplets removal from gas streams. Proceedings European Symposium on Computer Aided Process Engineering 10 (2000), 391-396.

[53] Ubbink, O.: Numerical prediction of two fluid systems with sharp interfaces. PhD Thesis, Imperial College of Science, Technology and Medicine, London (1997).

[54] Yakhot, V.; Orszag, S. A.: Renormalization group and local order in strong turbulence. Nuclear Physics B (Proc. Suppl.) 2 (1987), 417-440.

[55] Yoshida, H.; Norimoto, U.; Fukui, K.: Effect of blade rotation on particle classification performance of hydro-cyclones. Powder Technology 164 (2006), 103-110.

[56] Zhu, C.; Fan, L. S.: Multiphase flow: gas/solid in Johnson, R. W. (ed.): The handbook of fluid dynamics. Heidelberg: CRC Press LLC/Springer-Verlag (1998), 18-1 - 18-48.