NUMERICAL SIMULATION OF HYDROCYCLONES FOR CELL SEPARATION

R. A. MEDRONHO1, J. SCHUETZE2 and W.-D. DECKWER3

1 Escola de Química, Univ. Federal do Rio de Janeiro, CT, Bl. E, 21949-900 Rio de Janeiro-RJ, Brazil
medronho@ufrj.br
2 Fluent Germany, Hindenburgstrasse 36, D-64295 Darmstadt, Germany
jos@fluent.de
3 Technische Universität Braunschweig, GFB-German Research Centre for Biotechnology-TU-BCE,
Mascheroder Weg 1, D-38124 Braunschweig, Germany
wdd@gbf.de

Abstract — Numerical simulation of hydrocyclones aiming at investigating the separation of microorganisms and mammalian cells was performed using Computational Fluid Dynamics (CFD). The turbulence model used in the 2d-axisymmetric calculations was the Reynolds Stress Model (RSM), in order to take into account the high swirl effects that occur in this type of equipment, which induce anisotropic turbulence. The Volume of Fluid Model (VOF) was used to account for the gas/liquid interface. In all calculations, a cylindrical air core, running the whole length of the cyclone, appeared naturally as a consequence of a low pressure region that developed along the central axis. The separation of Escherichia coli, Saccharomyces cerevisiae and mammalian cells (BHK-21) using Bradley hydrocyclones was studied. According to the present work, Bradley hydrocyclones with diameters down to 10 mm cannot efficiently separate microorganisms, but the separation of mammalian cells with predicted efficiencies as high as 90% can be achieved.

Keywords — CFD, hydrocyclones, separation, microorganisms, mammalian cells

I. INTRODUCTION

Hydrocyclones are very simple devices (Fig. 1). In spite of this, research work is still in progress aiming at understanding the complex flow inside this apparatus or at developing new applications.

Regarding flow field, the high swirl effects that occur in hydrocyclones induce anisotropic turbulence, and this is the reason why the $k-\varepsilon$ turbulence model is not capable of properly describing the flow in hydrocyclones (He et al., 1999).

Regarding new applications, the use of hydrocyclones for yeast separation either from fermentation broths or from yeast-water suspensions have being investigated by some authors (Rickwood et al., 1992, Yuan et al., 1996a and 1996b, and Cilliers and Harrison, 1997). Even using hydrocyclones as small as 10 mm diameter, these authors could not obtain high separation efficiencies coupled with high underflow concentrations.

Figure 1. Perspective view of a hydrocyclone showing the fluid flow inside the equipment (a) and a schematic view showing its geometrical variables (b) whose definitions can be found in the item III.A.

A possible new application of hydrocyclones is their use in perfusion culture of mammalian cells. Cell retention in perfusion cultures is normally performed using centrifugation, cross flow microfiltration, spin-filtration, gravitational sedimentation, and ultrasonic separation. Unfortunately, all of these separation processes have specific problems (Castilho and Medronho, 2002). The use of hydrocyclones would increase process reliability, since their maintenance are virtually non-existent, and their service lifetime for this application would be extremely large.

The aim of the present work is to investigate, through the use of computational fluid dynamics (CFD), if hydrocyclones are capable of separating microorganisms and mammalian cells from a culture medium.

II. MATHEMATICAL MODELS

A. The Reynolds Stress Model (RSM)

Equations (1) and (2) are the time-smoothed equations of continuity and momentum transport, respectively (Bird et al., 2001; Launder, 1989).
\( \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j) = 0 \), \( \text{(1)} \)

\[
\rho \frac{D U_i}{D t} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + \\
+ \frac{\partial}{\partial x_j} \left( -\rho \mu_{ij} U_j \right) + \rho g_i , \quad \text{(2)}
\]

where \( \rho \) and \( \mu \) are the density and viscosity of the liquid, \( U_i \) and \( u_i \) are the \( x \) components of the mean fluid velocity and the fluctuating fluid velocity, \( p \) is pressure, \( g \) is gravity acceleration, and \( \rho \mu_{ij} \) are the components of the turbulent moment flux, known as “Reynolds stresses”.

A model for the Reynolds stresses is then needed in order to close Eqn. (2). The \( k-\varepsilon \) model is widely used for this purpose. The problem with this model is that it assumes isotropic turbulence and this condition appears not to hold for the flow inside a hydrocyclone. The RSM abandons the condition of isotropic eddy-viscosity hypothesis and closes the Reynolds-averaged Navier-Stokes equation by solving transport equations for the individual Reynolds stresses, together with an equation for the turbulence energy dissipation rate \( \varepsilon \). This means that 4 additional transport equations are required for simple two-dimensional flow calculations, and 7 for two-dimensional flows with swirl, as well as three-dimensional flows (Fluent, 1998). The RSM deals with the effects of streamline curvature, swirl, and rapid changes in the strain rate in a more rigorous manner than the \( k-\varepsilon \) model does. Therefore it has more potential to give good predictions for the complex flow field inside a hydrocyclone. Since the RSM is a more complex model, it requires on average 50-60% more CPU time per iteration and 15-20% more memory than the \( k-\varepsilon \) model (Fluent, 1998).

For a turbulent flow, Eqn. (3) can describe the transport of the Reynolds stress tensor (Lauder, 1989; Fluent, 1998).

\[
\frac{\partial}{\partial t} \left( \rho \mu_{ij} U_j \right) + C_{ij} = D_{ij}^T + D_{ij}^F + P_{ij} + G_{ij} + \phi_{ij} + \psi_{ij} + \mu_{ij} , \quad \text{(3)}
\]

where \( C_{ij} \) is convection, \( \phi_{ij} \) is turbulent diffusion, \( \psi_{ij} \) is molecular diffusion, \( P_{ij} \) is stress production, \( \phi_{ij} \) is buoyancy production, \( \phi \) is pressure strain, \( \varepsilon_{ij} \) is dissipation, and \( \mu_{ij} \) is production by system rotation. These quantities are given by Eqns. (4) to (11), respectively.

\[
C_{ij} = \frac{\partial}{\partial x_k} \left( \rho U_k u_i u_j \right) , \quad \text{(4)}
\]

\[
D_{ij}^T = \frac{\partial}{\partial x_k} \left( \rho \mu_{ij} u_k u_j + \mu \left( \frac{\partial U_i}{\partial x_k} + \frac{\partial U_k}{\partial x_i} \right) \right) , \quad \text{(5)}
\]

\[
D_{ij}^F = \frac{\partial}{\partial x_k} \left( \mu_{ij} u_k u_j + \rho \left( \frac{\partial U_i}{\partial x_k} + \frac{\partial U_k}{\partial x_i} \right) \right) , \quad \text{(6)}
\]

\[
P_{ij} = -\rho \left( \frac{\partial U_i}{\partial x_k} + \frac{\partial U_k}{\partial x_i} \right) \\
G_{ij} = \left( f_{ij} u_i + f_{ij} u_j \right) \\
\phi_{ij} = \mu \left( \frac{\partial U_i}{\partial x_k} + \frac{\partial U_k}{\partial x_i} \right) \\
\psi_{ij} = -2 \mu \left( \frac{\partial U_i}{\partial x_k} \frac{\partial u_j}{\partial x_k} \right) , \quad \text{(9)}
\]

\[
F_{ij} = -2 \mu \left( \mu_{ij} + \phi_{ij} \right) , \quad \text{(10)}
\]

\[
\mu_{ij} = C_\mu \rho \frac{k^{1/2}}{\ell_\mu} , \quad \text{(12)}
\]

\[
\varepsilon = \frac{k^{3/2}}{\ell_\varepsilon} , \quad \text{(13)}
\]

where \( C_\mu \) is an empirical constant, and \( \ell_\mu \) and \( \ell_\varepsilon \) are the characteristic length scales of turbulent viscosity and energy dissipation, respectively (Eqns. 14 and 15).

\[
\ell_\mu = C_{\ell_\mu} \left[ 1 - \exp \left( -\frac{Re_y}{A_\mu} \right) \right] , \quad \text{(14)}
\]

\[
\ell_\varepsilon = C_{\ell_\varepsilon} \left[ 1 - \exp \left( -\frac{Re_y}{A_\varepsilon} \right) \right] , \quad \text{(15)}
\]
where \( c, A_x \) and \( A_c \) are constants, \( y \) is the normal distance from the wall at the cell centre, and \( Re_y \) is the turbulent Reynolds number, given by:

\[
Re_y = \frac{y k^{1/2} \rho}{\mu} .
\]

(16)

**C. The Volume of Fluid Model (VOF)**

The VOF model is a physically rigorous yet efficient grid technique for numerically treating free boundaries between immiscible fluid phases. It uses a function \( F \) defined in such a way that its value is unity at any point occupied by one of the fluids and zero otherwise. The mean value of \( F \) in a computational cell represents the fractional volume of the cell occupied by this fluid. \( F \) values between zero and one indicate that the computational cell must contain an interface (Hirt and Nichols, 1981).

The VOF model solves a single set of momentum equations that are shared by the fluid phases, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain as a separate conserved quantity. The time dependent transport of \( F \) is governed by the following equation:

\[
\frac{\partial P}{\partial t} + U x \frac{\partial P}{\partial x_i} = 0 .
\]

(17)

The fields for all variables and properties are shared by the phases, and represent volume-averaged values in computational cells that contain fractional volumes of more than one fluid phase. For instance, in a multiphase system the volume-fraction-averaged viscosity in each cell is given by:

\[
\mu = \sum_i F_i \mu_i ,
\]

(18)

where \( F_i \) is the fractional volume of the cell occupied by fluid i, and \( \sum F_i = 1 \).

**III. MATERIALS AND METHODS**

**A. The Hydrocyclone, the grid, and the models**

Numerical simulations were performed using FLUENT 5.1, a CFD tool from Fluent Inc. The simulated hydrocyclone geometry obeys Bradley's recommended proportions (Bradley, 1965), and its internal dimensions (see Fig. 1b) are \( D_i = 10.0 \) mm, \( D_n = 1.4 \) mm, \( D_0 = 2.0 \) mm, and \( D_s = 1.0 \) mm for the cylindrical part, inlet, overflow and underflow diameters, respectively; \( \ell = 3.3 \) mm, \( L_1 = 5.0 \) mm, and \( L_2 = 62.1 \) mm for vortex finder, cylindrical part, and hydrocyclone lengths, respectively; and \( \theta = 9^\circ \) as cone angle. Three two-dimensional grids representing the Bradley hydrocyclone were built with 42295, 68271 and 80116 quadrilateral cells, respectively. Any further increase in cell number did not result in any change in the predicted velocity profiles, indicating that the finest grid was sufficiently fine for mesh-independent flow predictions.

The Reynolds stress turbulence model (RSM) was used to calculate the axisymmetric swirling flow. The volume of fluid (VOF) model was employed to predict the gas/liquid interface, and the two-layer zonal model was used to resolve the viscosity-affected near-wall region of the turbulent flow, with the constants suggested by Chen and Patel (1988):

\[
c = \kappa \mu^{-3/4}, \quad A_x = 70, \quad \alpha = 2c, \quad \beta = 0.09 .
\]

(19)

The partial differential equations for mass conservation and the transport of momentum, Reynolds stresses, and turbulent energy dissipation rate (\( \varepsilon \)) were solved using FLUENT’s segregated solver. This is based on the finite-volume method, which turns the differential formulation of all conservation equations into a control-volume based balancing of diffusive and advective fluxes. The latter are driven by fluid pressure, while the former are intensified by turbulence as described by the applied turbulence model. Velocity components in the axial, radial, and circumferential directions were treated. As all variation of the flow field in circumferential direction was neglected, the simulation could be performed in two dimensions. The solver accounts for all additional terms resulting from the projection of an axisymmetric geometry and flow field into two dimensions. All simulations were performed in a time-independent manner (steady state assumption). Density and dynamic (laminar) viscosity of the liquid were assumed to be constant, which corresponds to an isothermal approach. Pressure-velocity coupling between continuity and momentum equations was achieved using the “semi-implicit method for pressure linked equations”, known as SIMPLE-algorithm (Patankar and Spalding, 1972; Van Doormaal and Raithby, 1984). For pressure discretisation, the “pressure staggering option” (PRESTO) scheme was applied. This is a numerical approach that resembles the “staggered grid method”, described by Patankar (1980). It is particularly suited for the simulation of flow fields involving steep pressure gradients (Fluent, 1998). For the discretisation of all other conservation equations, upwind discretisation was used.

The assumption of a two-dimensional axisymmetric swirling flow implies that the feed has to be equally distributed at the suspension entrance. As used by other authors (He et al., 1999; Hsieh and Rajamani, 1991; Hargreaves and Silvester, 1990; Brayshaw, 1990), a full 360° inlet ring was assumed in the simulations. It is assumed that the fluid entering a hydrocyclone through a tangential inlet pipe quickly distributes itself around the cylindrical section. Therefore, the 360° inlet ring is expected to be a good approximation of this condition (Hsieh and Rajamani, 1991). It allows otherwise
accurate simulations in fine two-dimensional grids. The vastly increasing demand in computer memory and time for three-dimensional simulations would enforce the use of much coarser meshes, which eventually would not allow any mesh-independent fluid flow simulation.

B. The Boundary Conditions
Uniform velocity boundary conditions were used at the inlet, based on a flow rate of 28 cm$^3$s$^{-1}$. This flow rate was chosen because it lies approximately in the middle of the usual flow rate range quoted in the literature for 10 mm hydrocyclones. The inlet radial velocity was obtained dividing the flow rate by the area of the 360° inlet ring; the swirl velocity at the inlet was assumed to be the average velocity at the actual inlet pipe, and the inlet axial velocity was assumed to be zero. Uniform turbulent kinetic energy $k$ and turbulent dissipation rate $\varepsilon$ were calculated at the inlet using Eqns. (20) and (21) (Fluent, 1998).

\[
k = 1.5(U_{\text{inlet}})^2, \tag{20}
\]

\[
\varepsilon = C_{\mu}k^{1.5} \varepsilon, \tag{21}
\]

where $U_{\text{inlet}}$ is the average fluid velocity at the inlet, $I$ is the turbulence intensity given by Eqn. (22), and $\ell$ is the turbulence length scale ($\ell$ was assumed to be 7% of the inlet diameter $D$, which is approximately the maximum value of the turbulence length scale in fully-developed turbulent flow in pipes).

\[
I = 0.16 \left( \frac{D U_{\text{inlet}} \rho}{\mu} \right)^{-1/8}, \tag{22}
\]

where the quantity between brackets is the Reynolds number based on $D$ and the average fluid velocity at the inlet $U_{\text{inlet}}$.

To allow for the simulation to predict the flow split between overflow and underflow, pressure boundary conditions were chosen at both outlets. Hence, atmospheric pressure was set at the hydrocyclone outlets. Air was chosen as backflow if the calculated outlet pressure fell below the atmospheric pressure.

C. The Particle Tracking
After calculating the flow field, the chosen microorganisms or mammalian cells were introduced. FLUENT can simulate a discrete phase in a Lagrangian formulation and, as the trajectory of any particle is computed, the program keeps track of the momentum exchange between the particle and the surrounding continuous phase. After calculating the trajectories of many particles, the resulting momentum exchange terms were incorporated in a subsequent continuous phase calculation. By alternating calculations of the continuous phase and discrete phase flows, a converged solution for the two-way coupled multi-phase flow could be achieved. The trajectories of the discrete phase were calculated using the following equation of motion for a single particle:

\[
\frac{dv_i}{dt} = F_D(U_i - v_i) + \frac{(\rho_s - \rho)}{\rho_s}g_i + F_i, \tag{23}
\]

where $v$ is the particle velocity, $F_D(U_i - v_i)$ is the drag force per unit particle mass ($F_D$ is given by Eqn. 24), and $F_i$ are other forces acting on the particle, such as lift and Brownian forces ($F_i$ was assumed to be negligible in the present work).

\[
F_D = \frac{3\mu C_D}{\rho_s d^2} \left( \frac{dU + \varepsilon}{\mu} \right), \tag{24}
\]

where $C_D$ is the drag coefficient (evaluated through the use of the equation proposed by Haider and Levenspiel, 1989), and the term between brackets is the Reynolds number for the particle.

D. The Efficiency Assessment
The ultimate aim of simulating hydrocyclones is to predict their performance, i.e. to predict the total efficiency $E_T$, the reduced total efficiency $E'T$, the reduced grade efficiency $G'$, and the reduced cut size $d'_{50}$.

The total efficiency $E_T$ is defined as the mass fraction of solids recovered in the underflow. The fraction of fluid that is discharged in the underflow is called flow ratio $R_f$. Since the fluid carries solid particles with it, some particles will be discharged into the underflow not due to the centrifugal action of the separator but due to bare entrainment. In spite of some controversy (Frachon and Cilliers, 1999; Nageswararao, 2000; Coelho and Medronho, 2001), this bypass is normally assumed to be equal to the flow ratio. Therefore, $R_f$ is the minimal efficiency at which a separator will operate even if no centrifugal action takes place.

The reduced total efficiency $E_T'$, also called centrifugal efficiency, is the separation efficiency taking into account only those particles that will be separated due to the centrifugal field. Hence, $E_T'$ does not consider the particles that are “separated” due to the flow ratio. The reduced total efficiency is defined by Eqn. (25).

\[
E_T' = \frac{E_T - R_f}{1 - R_f}. \tag{25}
\]

The reduced grade efficiency curve $G'$ gives the centrifugal separation efficiency for each particle size present in the feed suspension. The particle size that is separated with $G'=50\%$ is known as the reduced cut size $d'_{50}$. $G'$ is obtained from the grade efficiency $G$, which gives the actual separation efficiency for each particle size:

\[
G' = \frac{G - R_f}{1 - R_f}. \tag{26}
\]
The fluids used in the numerical simulations were water (\( \rho = 998 \text{ kg m}^{-3} \) and \( \mu = 0.001 \text{ Pa s} \)) and air (\( \rho = 1.225 \text{ kg m}^{-3} \) and \( \mu = 1.79 \times 10^{-5} \text{ Pa s} \)). The cells were *Escherichia coli*, *Saccharomyces cerevisiae* I (baking yeast), *Saccharomyces cerevisiae* II (brewing yeast), and mammalian cells BHK-21, whose experimental size distribution curves are given by Fig. 2. The computational code allowed the calculation of particle tracks for particles of different sizes simultaneously, and for that, the particle size distribution needed to be given as a Rosin-Rammler distribution (Rosin and Rammaler, 1933).

For calculating the total efficiency \( E_T \), a given mass flow rate of each of the aforementioned cells was injected into the hydrocyclone, and the particle trajectories were calculated. FLUENT could then calculate and report the mass flow rate of trapped and escaped particles, which were the particles reaching the underflow and overflow, respectively. Based on these mass flow rates, the total efficiency could be calculated. A similar procedure was adopted for grade efficiency calculations, but using mass flow rates of particles of the same size.

Based on the simulated values obtained for the total efficiencies, grade efficiencies and flow ratio, the reduced total efficiencies and the reduced grade efficiencies were calculated using Eqns. (25) and (26), respectively. The reduced cut size \( d'_{50} \) could then be obtained since \( G'(d'_{50})=0.5 \).

The workstation used for all simulations was a Compaq Alpha-server DS20, 500 MHz, 2GB RAM. The average CPU time consumed for each iteration was 4.7 s. Convergence was assumed to be reached when no further changes in the interesting quantities happened, and never before the residuals decreased to \( 10^{-3} \).

### IV. RESULTS AND DISCUSSION

The numerical simulation produced a water flow ratio of 25% and a pressure drop equal to 2.1 bar, when feeding the hydrocyclone with 28 cm\(^3\) s\(^{-1}\) of water. For comparison, a well tested semi-empirical model (Antunes and Medronho, 1992; Castilho and Medronho, 2000) for Bradley hydrocyclones was used to predict these two variables. The results were a water flow ratio of 20% and a pressure drop of 3.3 bar. In hydrocyclones, good predictions of water flow ratios are the most difficult task. The available correlations for this purpose can lead to errors as high as 200% (Coelho and Medronho, 1992). Therefore, the 20% difference between the two water flow ratio predictions is reasonable. Empirical and semi-empirical equations correlating flow rate and pressure drop, on the other hand, produce usually good results (Antunes and Medronho, 1992; Coelho and Medronho, 1992). The lower pressure drop found in the numerical simulation is possibly due to the simplification of the feed inlet to a full 360° ring. This simplification implies that the fluid is entering the hydrocyclone in a much more smooth way.

Table 1 shows the results of total efficiency, reduced total efficiency and reduced cut size for the separation of the different cells simulated in this work. According to these results, it is not possible to separate *Escherichia coli* with the studied hydrocyclone. As Bradley hydrocyclones are high efficiency cyclones (Castilho and Medronho, 2000), it is possible to infer that bacteria can not be separated with conventional hydrocyclones down to 10 mm diameter. Yeast can be separated only with very low centrifugal efficiencies. Therefore, the use of conventional hydrocyclones to separate yeast is supposed to be unfeasible. On the other hand, an efficiency of 90% was obtained for the studied mammalian cell. This indicates that the separation of mammalian cells is an accomplishable task.

---

**Figure 2. Size distribution (% undersize) of the cells used in the simulations.**
Table 2. A comparison between the experimental results found in the literature for separation of brewing and baking yeast with hydrocyclones and the simulated results obtained in this work.

<table>
<thead>
<tr>
<th>Yeast Hydrocyclone</th>
<th>This work</th>
<th>Cilliers &amp; Harrison (1997)</th>
<th>Cilliers &amp; Harrison (1997)</th>
<th>Yuan et al. (1996b)</th>
<th>Yuan et al. (1996a)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter (cm)</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Overflow diam. (cm)</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>2 &amp; 2.6</td>
<td>2 &amp; 2.6</td>
</tr>
<tr>
<td>Underflow diam. (cm)</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>2 &amp; 1</td>
<td>2 &amp; 1</td>
</tr>
<tr>
<td>Pressure drop (bar)</td>
<td>2.1</td>
<td>6.6</td>
<td>1-9</td>
<td>4</td>
<td>6 &amp; 4.8</td>
</tr>
<tr>
<td>Flow ratio (%)</td>
<td>25</td>
<td>-</td>
<td>18</td>
<td>30-85 &amp; 7-35</td>
<td>60 &amp; 10</td>
</tr>
<tr>
<td>Total Efficiency (%)</td>
<td>28 &amp; 46</td>
<td>22-38</td>
<td>18-33</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Reduced Total Eff. (%)</td>
<td>4 &amp; 28</td>
<td>-</td>
<td>-</td>
<td>10-38 &amp; 10-15</td>
<td>36 &amp; 13</td>
</tr>
</tbody>
</table>

diluted yeast suspension. For this flow rate, the simulated total efficiency found in the present work was 28%.

A typical feature of hydrocyclones operating with the outlets opened to the atmosphere is the natural appearance of an air core, cylindrical in shape, running from the top of the overflow pipe until the underflow orifice. In the present work, an air core was formed as a result of the numerical simulation. It is important to state that no user interference, such as air bubble injection (Pericleous and Rhodes, 1986), cylinder with slip wall conditions (Averous and Fuentes, 1997) or imposition of the air core into the solution (Dai et al., 1999; Dyakowsky and Wiliams, 1993) was used. The air core appeared naturally as a consequence of a low pressure region developed along the central axis during the calculations. Figure 3 shows the air core in three different sections of the hydrocyclone (see also Fig. 1 for better understanding). These three sections are the top part that includes the overflow pipe (Fig. 3.a), the central part (Fig. 3.b), and the bottom part that includes the underflow orifice (Fig. 3.c). The calculated air core had a diameter of 0.9 mm from the bottom of the vortex finder until around two thirds of the cyclone length. From the beginning of the last third part of the cyclone length until the apex, the air core started smoothly to decrease in size until it reached 0.4 mm at the underflow exit. As can be seen in Fig. 3.a, the calculated air core increased rapidly in size as it entered the overflow pipe, until it reached a stable value of 1.1 mm. The numerical calculations were also able to naturally simulate the typical shape of a spray discharge in the underflow, as can be seen in Fig. 3.c. It should be pointed out that the accuracy of the calculated size of the air core was not checked experimentally. For this purpose, a transparent hydrocyclone is being built and further investigations will be carried out.

Figure 4 shows the reduced grade efficiency as a function of the normalised diameter (d/d'o). It can be seen that the predictions using CFD show a good agreement with the experimental points from Bradley (Bradley and Pulling, 1959) and with the curve based on experiments carried out in another work (Antunes and Medronho, 1992).

V. CONCLUSIONS

According to CFD simulations, a 10 mm Bradley hydrocyclone can separate mammalian cells with high efficiencies. However, yeast and bacteria are not supposed to be efficiently separated by Bradley hydrocyclones, even for cyclone diameters down to 10 mm.

![Figure 3. Details of the calculated air core for a 10 mm Bradley hydrocyclone with an underflow diameter of 1 mm, processing 28 cm3s-1 of water at 20°C.](image)

![Figure 4. Reduced grade efficiency for Bradley hydrocyclones showing the predictions using CFD, the original experimental points from Bradley (Bradley and Pulling, 1959) and the curve based on experiments carried out in another work (Antunes and Medronho, 1992).](image)
The computational code (FLUENT 5.1) used in the numerical simulations was capable of predicting reduced grade efficiencies curves with good accuracy. Experiments need to be carried out to confirm the air core diameter predicted by the numerical simulations.

**NOTATION**

- $A_c$: constant in Eqn. (15)
- $A_m$: constant in Eqn. (14)
- $C_D$: drag coefficient
- $c_r$: constant in Eqns. (14) and (15)
- $C_a$: Underflow volumetric concentration
- $C_a$: constant in Eqn. (12)
- $d$: particle diameter (m)
- $d_{s0}$: reduced cut size (m)
- $E_T$: total efficiency
- $E_T^r$: reduced total efficiency
- $f$: fluctuating body-force per unit volume (N m$^{-3}$)
- $F$: function defined in the volume of fluid model
- $F_D$: drag force per unit of particle mass divided by $(U_i-v_i)$ (s$^{-1}$)
- $F_i$: other forces in Eqn. (23) (kg m s$^{-2}$)
- $g$: gravity acceleration (m s$^{-2}$)
- $G$: grade efficiency
- $G^r$: reduced grade efficiency
- $I$: turbulence intensity
- $k$: turbulent kinetic energy (m$^2$ s$^{-2}$)
- $\ell$: turbulence length scale (m)
- $\ell_d$: length scale of dissipation (m)
- $\ell_s$: length scale of viscosity (m)
- $p$: pressure (Pa)
- $R_f$: flow ratio
- $Re_y$: turbulent Reynolds number
- $t$: time (s)
- $U$: streamwise fluid velocity (m s$^{-1}$)
- $u_i$: component of fluctuating fluid velocity (m s$^{-1}$)
- $u_i$: component of mean fluid velocity (m s$^{-1}$)
- $U_{inlet}^{av}$: average fluid velocity at the feed inlet (m s$^{-1}$)
- $v$: particle velocity (m s$^{-1}$)
- $v_i$: component of particle velocity (m s$^{-1}$)
- $x_i$: Cartesian direction coordinates (m)
- $y$: normal distance from the wall at the cell centre (m)
- $\varepsilon$: dissip. rate of turbulent kinetic energy (m$^2$ s$^{-3}$)
- $\mu$: liquid viscosity (Pa s)
- $\mu_e$: eddy viscosity (Pa s)
- $\rho$: liquid density (kg m$^{-3}$)
- $\rho_s$: solid density (kg m$^{-3}$)
- $\Omega$: angular rotation rate (s$^{-1}$)

**REFERENCES**


Received: January 6, 2003.
Accepted: July 25, 2003.
Recommended by Subject Editor Leonel Teixeira Pinto.