Improving of gas-liquid mass transfer in a Stirred Tank Bioreactor: A CFD Approach

Lilibeth Caridad Niño López, Mariana Peñuela, Germán Ricardo Gelves-Zambrano*
Bioprocess Group, Department of Chemical Engineering, University of Antioquia, Carrera 53 No. 61-30. Medellin, Colombia.
Corresponding author

Abstract
In this study CFD simulations (Computational Fluid Dynamics) were developed in order to study the gas-liquid mass transfer in a 1000 L spin filter bioreactor configured with a conventional Spin filter. Based on CFD results, three new geometries are proposed for improving gas liquid mass-transfer: (Geometry I) a micro-sparger and a curved bottom Spin filter with lateral axial blades, (Geometry II) a diffuser type spider and a curved bottom Spin filter with lateral axial blades and (Geometry III), a hybrid model with axial and radial blades. The numerical results based on typical animal culturing conditions are simulated and $k_{L}a$ mass transfer was estimated to be 61 percent higher than the conventional flat bottom spin filter when the Geometry II was analyzed in CFD. Similar enhancements were found for the other two new geometries (I and III). Motivated by these results, the new spin filter device (Geometry II) is qualified as a very promising tool for animal cell culture applications which are characterized by the constrain of achieving relatively high mass transfer conditions and avoiding cellular damage.

Keywords: bioreactor, scale up, multiple reference frame (MRF), population balance model (PBM), spin filter, computational fluid dynamics (CFD).

Introduction
Supplying adequate oxygen levels in aerobic cell culturing is a common problem in fermentation technology. This problem is increased in high cell density bioreactors due to oxygen transfer rate limitations affecting cell growth and productivity. Such limitations are common in perfusion cell cultures leading to anoxic process, cellular damage and therefore loss of cell viability [1]. In continuous cell culturing with cell retention (perfusion cell bioreactors) nutritional environment can be maintained approximately constant. Due to continuous nutrient supply and removal of drained metabolized culture medium, volumetric biomass productivity can considerably increase compared to batch or fed-batch mode [1]. However oxygen requirements are proportional to cell density, which means that gas-liquid mass would be a limitant parameter for high density cell culture applications.

Additionally, the loss of complete mixing conditions with increasing scale could degenerate gradients leading to a departure from optimal conditions found in laboratory scale. Scale-up strategies are based on physical criteria, derived from similitude theory [2,3]. Their selection and application is often empirically driven, rather motivated by successful applications in the past then by valid predictions for the unknown scale-up issue. Contrarily, there has been rapid progress in the modeling of large-scale performance by the application of computational fluid dynamics (CFD). Successful examples are giving covering the range from agitation systems, including models from single phase rotating impellers systems [4,5] to the use of population balance models [6,7]. The main purpose of these models is the knowledge of the flow fields in conventional stirred impellers related to mass transfer profiles [8].

One of the most common devices for perfusion processes is the Spin-Filter, which is used for animal and plant cell cultures in continuous production processes. As a consequence of biochemical engineering enhancements, perfusion bioreactors incorporate a rotating filter device into a stirred tank reactor. Previous CFD works using these devices have reported dead zones and gradients that significantly would affect the gas-liquid mass transfer and process productivity [9]. The latter is a reason why this device is unattractive to be operated on a large scale applications. That is why based on CFD simulations of gas-liquid mass transfer in a conventional spin filter, this research evaluated hydrodynamics by focusing on main parameters associated to the device geometry which can affect air dispersion and oxygen transfer. The latter was the main criterion for setting out the re-design of the conventional spin filter by proposing three new geometries (strategies) that can enhance the $k_{L}a$ gas-liquid mass transfer, air dispersion, Kolmogorov length scales, sauter bubble diameter and power input.

It is found that numerical results approaches come to the conclusion that Geometry II significantly increases $k_{L}a$ values at typical cell culture conditions installed. Similar enhancements were found for the other two new geometries (I and III).

Although several works have dealt with the understanding flow field and particle dynamics of conventional spin filter devices, geometry enhancements for improving gas-liquid mass transfer phenomena are not studied so far. To the authors knowledge this is the first time that a novel spin filter device is proposed from CFD simulations based on gas-liquid hydrodynamics. So that this new device can be useful for overcoming large-scale bioprocess limitations. Hence it is the motivation of this work to simulate gas-liquid hydrodynamics in a Spin Filter bioreactor using a CFD approach for improving gas-liquid mass transfer.
Materials and Methods

Bioreactor setup:
A Spin Filter stirred tank bioreactor (New Brunswick Celligen 310) with 10 L working volume was used for dimensioning a 1000 L (T = 0.43 m) simulated conventional spin filter bioreactor. The gas is supplied through a cylinder microsparger. The operating conditions chosen were defined by typical settings used for culturing animal cells: Average \( V_{\text{tip}} \): 1.8 m/s, aeration rate: 0.08 vvm. All systems were tested applying the same operational conditions. Based on CFD results, three new geometries are proposed for improving gas liquid mass-transfer (Fig. 1). The properties used in the primary phase (water) are: \( \rho_L \): 998.2 kgm\(^{-3}\); \( \mu_L \): 0.001 kgm\(^{-1}\)s\(^{-1}\); \( \sigma \): 0.07 Nm\(^{-1}\). Secondary phase properties (air) are: \( \rho_G \): 1.225 kgm\(^{-3}\); \( \mu_G \): 1.789 \( \times \) 10\(^{-5}\) kgm\(^{-1}\)s\(^{-1}\)[10].

Computational Fluid Dynamic Model: Multiphase Flow Equations:
The gas and liquid phase are treated as interpenetrating continua and conservation of mass and momentum equations are solved for each phase. The conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases [11,12].

The Eulerian model is the most complex multiphase model in ANSYS FLUENT 13.0. It solves a system of n-momentum and continuity equations for each phase. The coupling is achieved through pressure and interfacial exchange coefficients. The mass conservation equation for each phase is shown (Eq. 1):

\[
\frac{\partial}{\partial t} (\rho_i \alpha_i) + \nabla \cdot (\rho_i \alpha_i \vec{U}_i) = 0
\]

Where \( \rho_i \), \( \alpha_i \) and \( \vec{U}_i \) represent the density, volume fraction and mean velocity, respectively, of phase \( i \) (L or G). It is assumed that the liquid phase and the gaseous phase share space proportional to their volume, such that their volume fractions sum up to unity in the cell domain (Eq. 2):

\[
\alpha_L + \alpha_L = 1.0
\]

The momentum equation for phase \( i \) is described below (Eq. 3):

\[
\frac{\partial}{\partial t} (\rho_i \alpha_i \vec{U}_i) + \nabla \cdot (\rho_i \alpha_i \vec{U}_i \vec{U}_i) = \vec{a}_p + \nabla \cdot \vec{t}_{\text{effi}} + \vec{f}_L + \alpha_i \vec{g}
\]

\( \vec{g} \)is the pressure shared by both phases and \( \vec{f}_L \) represents the interfacial momentum exchange. The \( f_L \) term represents the Coriolis and centrifugal forces expressed in the MRF (Multiple Reference Frame) model for rotating flows and is represented as (Eq. 4):

\[
\vec{f}_L = -2 \alpha_i \rho_L \vec{N} \times \vec{U}_L - \alpha_i \rho_L \vec{N} \times (\vec{N} \times \vec{r})
\]

\( \vec{N} \) is the angular velocity, \( \vec{r} \) is the position vector. The Reynolds stress tensor \( \vec{t}_{\text{effi}} \) (Eq. 5) is related to the mean velocity gradients through the Boussinesq hypothesis [11];

\[
\vec{t}_{\text{effi}} = \alpha_i (\mu_{\text{lam},i} + \mu_{\text{t},i}) \left( \nabla \vec{U}_i + (\nabla \vec{U}_i)^T \right) - \frac{2}{3} \alpha_i (\rho_i k + (\mu_{\text{lam},i} + \mu_{\text{t},i}) \vec{r} \cdot \vec{U}_i) \vec{I}
\]

\( \mu_{\text{lam},i} \) is the molecular viscosity of phase \( i \), \( \vec{I} \) is the strain tensor.

Interfacial Momentum Exchange:
The most important interphase force is the drag force acting on the bubbles. This force (Eq. 6) depends on friction, pressure, cohesion, and other hydrodynamic effects [13].

\[
R_L = -R_G = K (\vec{U}_G - \vec{U}_L)
\]

\( K \) is the exchange coefficient of liquid and gaseous phases and is determined by (Eq. 7):

\[
K = \frac{3}{4} \rho_L \alpha_L \alpha_G \frac{C_D}{d} |\vec{U}_G - \vec{U}_L|
\]

dis the bubble diameter, and the drag coefficient \( C_D \) is defined as a function of Reynolds number (Eq. 8):

\[
R_e_p = \frac{\rho_L |\vec{U}_G - \vec{U}_L| d}{\mu_L}
\]

To calculate the drag coefficient the standard correlation [14] was applied (Eq. 9):

\[
C_D \begin{cases} 
24 (1 + 0.15 R e_p^{0.687}) & , R e_p \leq 1000 \\
0.44 , R e_p > 1000 
\end{cases}
\]

Equations for the turbulence model
The dispersed turbulence \( k - \varepsilon \) model can be considered as the multiphase standard turbulence approach. It represents the extension of the single phase \( k - \varepsilon \) model and is used when the secondary phase concentrations are diluted on primary phase. \( k \) and \( \varepsilon \) equations describing this model are as follows (Eqs. 10 and 11):

\[
\frac{\partial}{\partial t} (\rho_L \alpha_L k_L) + \nabla \cdot (\rho_L \alpha_L \vec{U}_L k_L) = \vec{a}_p \frac{\mu_k}{\sigma_k} \nabla \vec{U}_L + \alpha_L G_{k,L} - \alpha_L \rho_L \varepsilon L + \alpha_L \rho_L \Pi_{k,L}
\]

\[
\frac{\partial}{\partial t} (\rho_L \alpha_L \varepsilon_L) + \nabla \cdot (\rho_L \alpha_L \vec{U}_L \varepsilon_L) = \nabla \cdot \left( \alpha_L \frac{\mu_k}{\sigma_{\varepsilon}} \nabla \varepsilon_L \right) + \alpha_L G_{\varepsilon,L} - \alpha_L \rho_L \varepsilon_L + \alpha_L \rho_L \Pi_{\varepsilon,L}
\]

In these equations, \( G_{k,L} \) represents the generation of turbulent kinetic energy, \( k_L \) of the liquid phase due to mean velocity gradients, \( \varepsilon_L \) is the turbulent dissipation energy, \( \Pi_{k,L} \) and \( \Pi_{\varepsilon,L} \) represent the influence of the dispersed phase in the continuous phase, respectively [15].

The turbulent viscosity \( \mu_{t,L} \) is calculated from (Eq. 12):
The values of the constants used in this experiment were $1.44$, $1.92$, $0.09$, $1.00$ and $1.30$. $\sigma_k$ and $\sigma_s$ represent turbulent Prandtl number for $k$ and $\varepsilon$, respectively [15].

Population balance model for bubble distributions

The discrete method [16,17] is used herein to solve the population balance equations. The bubble population is discretized into a finite number of intervals of bubble diameters. The population balance equations (Eq. 13) for different bubble classes can be written as [18,19]:

$$\frac{\partial}{\partial t} \left( \rho \frac{n_i}{\Gamma_i} \right) + \nabla \cdot (\rho \nu \nabla n_i) = \rho \left( \Gamma_{bi} - \Gamma_{bi} + \Gamma_{bi} - \Gamma_{bi} \right) \tag{13}$$

Where $n_i$ is the number of bubbles per class $i$, $\Gamma_{bi}$ and $\Gamma_{bi}$ are birth rates due to coalescence and breakage, respectively, $\Gamma_{bi}$ and $\Gamma_{bi}$ are the death rates.

The terms of breakage and coalescence are (Eqs. 14-17):

$$\Gamma_{bi} = \frac{1}{2} \int_0^\infty a(v' - v,v)n(v',v,t)n(v,v',t)dv' \tag{14}$$

$$\Gamma_{bi} = \int_{D_{bi}} \rho g(v')\beta(v'v)n(v',t)dv' \tag{15}$$

$$\Gamma_{bi} = \rho g(v)n(v,t) \tag{16}$$

$$a(v,v')$$ is the coalescence rate between bubbles of size $v$ and $v'$; $g(v)$ is the breakup rate of bubbles of size $v$; $\beta(v'v)$ is the breakup frequency of bubble $v'$ and $\beta(v'v)$ is the probability density function of bubbles broken from the volume $v'$ in a bubble of volume $v$.

The bubble coalescence is modeled by considering the bubble collision due to turbulence, buoyancy and laminar shear. The coalescence rate (Eq. 21) is defined as the product of the collision frequency $a(v,v')$ and coalescence probability $p_{\text{ag}}(v_i, v_j)$ and is defined as [20]:

$$a(v,v') = \omega_{\text{ag}}(v_i, v_j)p_{\text{ag}}(v_i, v_j) \tag{21}$$

The collision frequency is defined as (Eq. 22):

$$\omega_{\text{ag}}(v_i, v_j) = \frac{\pi}{4} (d_i^2 + d_j^2) \eta_i \eta_j \bar{u}_{ij} \tag{22}$$

Where $\bar{u}_{ij}$ is the characteristic collision velocity of two bubbles with diameter $d_i$ and $d_j$ and bubble density $n_i$ and $n_j$.

The sum of the volume fractions of each group of bubbles is equal to the volume fraction of the dispersed phase (Eq. 25):

$$\sum_i \alpha_i = \alpha_G \tag{25}$$

The volume fraction of each group sizes is expressed in terms of the total fraction of the dispersed phase (Eq. 26):

$$f_i = \frac{\alpha_i}{\alpha_G} \tag{26}$$

Therefore, including Eq. (26) in Eq. (13), is (Eq. 27):

$$\frac{\partial}{\partial t} \left( \rho \frac{n_i}{\Gamma_i} \right) + \nabla \cdot (\rho \nu \nabla n_i) = \rho \left( \Gamma_{bi} - \Gamma_{bi} + \Gamma_{bi} - \Gamma_{bi} \right) \tag{27}$$

The Sauter diameter $d_{32}$ is used to fit the population balance equations with the bioreactor hydrodynamics (Eq. 28):

$$d_{32} = \frac{\sum_i n_i d_i^3}{\sum_i n_i d_i^2} \tag{28}$$

Mesh processing and numerical technique

Three geometries based on CFD simulations of the conventional device were used (Figure 1): Geometry I (Figure 2), a spin filter with curved bottom and centered axial blades; Geometry II (Figure 3), a spin filter with curved bottom, centered axial blades and diffuser type spider and Geometry III (Figure 4), Spin filter with curved bottom, centered axial blades, diffuser type spider, auxiliary radial blades and modified pitch blade (separated blades). For each geometry, a fine 3D mesh is composed for hybrid cells with 1000 computational k cells [9].

The finite volume technique implemented in the CFD code Ansys Fluent 13.0 Software was used to convert the Navier-Stokes equations into algebraic equations which can be solved numerically. Tank walls, stirrer surfaces and baffles are treated with no slip conditions and standard wall functions.
Figure 1. Geometry for the Conventional Spin filter device.

Figure 2. Geometry I, a micro-sparger and a curved bottom Spin filter with lateral axial blades.

Figure 3. Geometry II, a diffuser type spider and a curved bottom Spin filter with lateral axial blades.

Figure 4. Geometry III, a hybrid model with axial and radial blades.
The gas flow rate at the cylinder micro-sparger and spider were defined via inlet-velocity type boundary condition with gas volume fraction equal to unity. MRF model was applied for the impeller (Pitched Blade) and Spin Filter device. PC SIMPLE algorithm was used for solving the partial differential equations. The second order upwind scheme was applied for the spatial terms. It was assumed that the solution converges when the scaled residuals remain with values smaller than $10^{-5}$ and when the pseudo-regime for hold-up is reached [21,22]. The size of the bubbles was set by the discretization scheme based on the geometric ratio [21]. The size of the bubbles for the Spin Filter Bioreactor was assumed from 75 to 3500 µm in diameter. 13 size classes were used to discretize the population balance equations [22], who analyzed hydrodynamics of a propeller type stirrer and found satisfactory results using the same classifiers. Power input $P$ is calculated using torque $M$ of the impeller (in ANSYS FLUENT called the Moment Center) [6]:

$$
P = 2\pi N_i M
$$

(29)

Theoretical power input was calculated according to Zhu, et al [23]:

$$
P = \rho N_i^3 D_i^5
$$

(30)

Results and Discussion
The main objective of this work was to evaluate different geometries using CFD simulations in order to improve gas liquid mass transfer and reduce dead zones in a large scale spin filter. Based on the CFD results from a conventional spin filter, main parameters associated to equipment geometry were evaluated. The latter was the main criterion for setting out the re-design of the conventional spin filter by proposing three new geometries (strategies) that can enhance the gas-liquid mass transfer, air dispersion, Kolmogorov length scales, sauter bubble diameter and power input. In Figure 5 it is noted that air dispersion is too low on the bottom and top of bioreactor, with only local dispersion in the area close to impeller blades and micro-diffuser. The latter could be caused by the design of conventional flat bottom of the spin filter that interferes with the air distribution to other areas of the bioreactor, generating consequently low mixing velocities in the lateral area surrounding the spin filter [9]. Evidently the formation of heterogeneous environments that may affect the growth of a cell culture is unavoidable in a conventional spin filter. This is reflected by the high values (obtained from CFD) of bubble sizes affecting the hydrodynamics (Fig. 9-12), as explained later. To overcome this problem, three new geometry strategies have been proposed in this research: The first (Geometry I) consists of modifying the flat bottom of the spin filter by a curved bottom. Additionally four auxiliary blades are proposed, located in the sidewalls of the spin filter in order to achieve better dispersion of smaller air bubbles at the top of the bioreactor. For the second strategy (Geometry II) a micro-diffuser spider type with a curved bottom and four auxiliary blades is proposed. The latter to allow better air dispersion and smaller bubbles from the bottom of bioreactor. As a third alternative (Geometry III ) a hybrid system is proposed. It can generate a pattern of axial and radial flow configured with axial and radial blades in order to maximize oxygen transfer rate based on the reports of Mirro and Voll[24], in which they state that axial and radial flows combinations provide better overall mixing and creates a higher $k_a$ oxygen mass transfer rate than that of unidirectional marine blade impellers. It is observed by CFD air volume fraction (Fig. 5-8), a considerable improvement in air dispersion using geometry I compared to conventional geometry. This was, due to centrifugal forces generated by the axial blades of the spin filter that allow a better air distribution in the upper zone of the bioreactor although improvements are minor in the bottom of the tank. However, in Geometries II and III is observed that this limitation in the bottom of the device is overcome at using a micro-diffuser spider type since air dispersion improves considerably. Similar results were found by Gelves[25], who reported a noticeable substantial increase of the oxygen transfer using rotating micro-spargers compared to a ring-sparger.

Drag forces are important in a stirred gas-liquid system due to the influence of air dispersion in bioreactor. The drag coefficient is an a-dimensional parameter that measures these impacts. For this reason these values were calculated from CFD using equations (8 and 9). Results obtained for drag coefficients calculated for each geometries were: 0.55 (conventional device); 0.70 (Geometry I); 1.52 (Geometry II) and 1.96 (Geometry III), which explains the large differences found in the air volume fraction.

![Figure 5. Volume fraction [-] contours calculated for Conventional Spin filter device.](image-url)
Figure 6. Volume fraction [-] contours calculated for the Geometry I, a micro-sparger and a curved bottom Spin filter with lateral axial blades.

Figure 7. Volume fraction [-] contours calculated for the Geometry II, a diffuser type spider and a curved bottom Spin filter with lateral axial blades.

Size distribution results from physical interactions imposed on uprising bubbles and considers mechanisms such as coalescence and breakage. These phenomena together with hydrodynamics, trigger the formation of dissolved oxygen and trace gases gradients of which can affect productivity in aerobic plant cell cultures. For that reason, when breakage phenomenon prevails, bubble size decreases, the interfacial area for mass transfer therefore is increased and consequently, the oxygen transfer is promoted. On the other hand, large bubbles generated by coalescence are characterized by higher rising velocities and short residence times. Thereby, bubbles cannot be uniformly dispersed for the agitation system. The most relevant parameter for analyzing these hydrodynamic mechanisms is the Sauter mean diameter. It can be interpreted as an integral parameter, summarizing all individual impacts affecting the bubble sizes and their distribution in one single value [8]. It is estimated from CFD based hydrodynamics and population balance models.

In Figures (9-12) the contours for the Sauter diameter is observed in each evaluated system. The values for Sauter diameter were 5000 µm in most of the conventional spin filter type bioreactor (Fig. 9), indicating the prevalence of the coalescence phenomena generated by poor mixing velocities as a consequence of the flat bottom design. Therefore, generation of these areas of low gas-liquid mass transfer and local mixtures, are a major disadvantage for applications of spin filter perfusion at large scale. This phenomenon was significantly exceeded by modifying the geometry (Fig. 10).
using auxiliary blades and a curved bottom Spin Filter (Geometry I). It is observed a 60% bubble size reduction in the upper zone with bubble size of 2000 µm. However, limitations still continue appearing at the bottom of the device (large bubble sizes of 5000 µm) which are corrected with the second and third design strategies (Geometry II and III). In Figures 11 and 12 the effect of geometries II and III on Sauter mean diameter are showed. For Geometry II, bottom of the Spin Filter was changed, auxiliary blades and a 4 legs type micro-sparger were used and a hybrid system was setting for geometry III. It is observed that bubble diameter improved noticeable compared to the conventional device and values found in the bottom geometry I due to the uniformity of bubble dispersion around all zones of the bioreactor with overall values of ~ 2000 µm. The Latter is the consequence of the improved hydrodynamics which promotes the formation of uniform bubble sizes, generally caused by the mixing homogeneity [8].

Figure 9. Sauter diameter contours [m] calculated for the Conventional Spin filter.

Figure 10. Sauter diameter contours [m] calculated for the Geometry I, a micro-sparger and a curved bottom Spin filter with lateral axial blades.

Figure 11. Sauter diameter contours [m] calculated for the Geometry II, a diffuser type spider and a curved bottom Spin filter with lateral axial blades.
The cell size plays a major role related to sensitivity in operative hydrodynamic conditions in a bioreactor. This has been strongly related to the interaction of cells with small eddies [26]. The theory of interaction of these eddies with suspended particles in a stirred tank and its relationship with dissipation energy was published by Kolmogorov [27,28]. In general terms this theory postulates that damage will occur if the eddy size is similar or smaller than the cell. The most dangerous average sizes are in the range of 10 to 50 µm. So that the more affected systems could be animal cells (sizes from 10 to 30 µm) and plant cells (from 20 to 120 µm). In this study Kolmogorov length scale \( \lambda_k \) of 50 µm were found at the center of the pitch blade (Figure 13). For other modified geometries, the Kolmogorov length scale \( \lambda_k \) showed values between 52-67 µm. The smaller scales were found at the top of the bioreactor close to the auxiliary. These results indicate that the Kolmogorov length scale \( \lambda_k \) could not generate a lethal damage to animal cells for the evaluated geometries, due to the modifications performed in each system.
Figures 15-20 shows the contours of gas-liquid mass transfer (Figures 17-20) and higher values are observed for simulations of Geometry II and III compared to the conventional device. Based on these results it is possible to affirm that an inverse proportionality between the bubble and mass transfer is related. Similar findings regarding the bubble size have also been reported by Fayolle, et al [29] who found that a decrease in the bubble size of 10% induces an increase in the oxygen transfer coefficient by 15%. Inversely, increasing the bubble diameter by 10% induces a decrease in the oxygen transfer coefficient (11%).

With the proposed improvements by modifying the device geometry of the spin filter it can be said that the geometries I and II can overcome the limitations founded in simulations. The latter could help to expand its use to large scale applications by taking into account the changes proposed. However for industrial application purposes, it is important to analyze the power input in terms of the yield $P$ and values of 0.22 (conventional device); 0.08 (Geometry I); 1.14 (Geometry II) and 0.44 h/W (Geometry III) are calculated. The use of radial blades in geometry III (hybrid system) generates significant increases in power input [8]. Although $k_{l,a}$ values are maximized for Geometry III (Fig. 17-20), the observed yield of 0.44 h/W defines the geometry II as the best strategy for large-scale applications in terms of yield $k_{l,a}/P$ with a value of 1.14 h/W, equivalent to more than 3 times higher than that found in the hybrid system. Also power input values were calculated from CFD using equation (29) and compared with the theoretical values determined by equation (30) according to Zhu, et al [23]. The calculated values obtained using CFD were 82.69 (Conventional Device); 216.39 (Geometry I); 26.01 (Geometry II) and 73.51 W (Geometry III). The theoretical values obtained from equation (30) were 76.66 (Conventional Device); 239.63 (Geometry I); 29.95 (Geometry II) and 76.25 W (Geometry III). The good agreement found by comparing both calculations is qualified as an evidence for suitability of the CFD model.
Figure 17. $k_a$ mass transfer coefficient [1/h] calculated for the Conventional Spin filter.

Figure 18. $k_a$ mass transfer coefficient [1/h] calculated for the Geometry I, a micro-sparger and a curved bottom Spin filter with lateral axial blades.

Figure 19. $k_a$ mass transfer coefficient [1/h] calculated for the Geometry II, a diffuser type spider and a curved bottom Spin filter with lateral axial blades.

Figure 20. $k_a$ mass transfer coefficient [1/h] calculated for the Geometry III, a hybrid model with axial and radial blades.
Conclusions
By the study with CFD important parameters can be analyzed at 1000L spin filter bioreactor influenced by proposed geometries with the aim of improving some limitations. CFD is a promising tool for preliminary design analysis of new prototypes for $k_La$ mass transfer which depend on important parameters such as Drag coefficient, bubble size diameter, Kolmogorov length scales and mass transfer coefficient. In this study the effect of modifying the geometry on the gas-liquid mass transfer was evaluated by CFD and based on simulations it was possible to redesign the conventional prototype for improving the $k_La$ mass transfer in a bioreactor operated with a Spin filter. It was found that $k_La$ is significantly increased by modifying the flat bottom design of the conventional Spin Filter by a curved bottom, four auxiliary blades and using a micro-diffuser type Spider. This modifications proposed could help expand the use of spin filter devices on a large scale applications. Motivated by these results CFD simulations are qualified as a very promising not only for predicting gas-liquid hydrodynamics but also for finding design requirements that must be implemented to optimize an aerobic bioprocessing.

References


