HYDRODYNAMICS OF DENSE HYDRATE SUSPENSIONS IN PRECIPITATORS

Esquerre, V.[1, 2], Barrué, H.[1], Bertrand, J.[1], Cristol, B.[2], Begon, G.[3], Xuereb, C.[1]
[1]*Laboratoire de Génie Chimique, Toulouse, France*
[2]*Aluminium Pechiney, Gardanne, France*
[3]*Robin Industries, France*

Abstract

In the last twenty years, precipitator technology has significantly evolved. Mixing of the suspension of hydrate and Bayer liquor was originally mainly ensured by forced circulation in a draft tube, with small diameter and fast propellers. Nowadays, solid suspension is often carried out with large diameter impellers rotating at low speed.

The present study aims at developing a process analysis methodology, based on numerical simulations (CFD), that is able to predict the flow patterns and the solid distribution in precipitators. A two fluid model with Eulerian approach has been chosen and checked using measurements of solid content distribution carried out on pilot facility.

A batch mode has been first investigated to evaluate the ability of the agitator system alone. This operating mode can be encountered if the vessel must be isolated due to process or operation constraints. The influence of rotational speed, mean solid concentration and particle diameter on homogeneity has been studied. The effect of PSD on sedimentation and risk of solid accumulation in the bottom part of precipitator has been assessed. In the case of high solid content slurries, the positive effect of solid concentration increase on the homogeneity in the vessel has been quantitatively determined.

Experimental and modelling investigation of the continuous mode confirms and quantifies the positive effect of slurry flow through a series of precipitators on homogenisation. This effect can be taken into account for the global design of precipitation facility.

CFD has shown to successfully predict the performances of a mixing system in high concentrated media, and consequently it can be used to optimise mixer design. A higher performance and less power-demanding mixing system is thus proposed.

1. Introduction

Precipitation of alumina hydrate is clearly a critical step in the Bayer process. Series of precipitators operating at different solid contents (from less than 200 to more than 800 g/L) in a continuous mode provide the alumina output.

In the last twenty years, precipitator technology has significantly evolved. Tank capacity has increased from 1500 m^3 to 4500 m^3, and the flat bottom has replaced the conical bottom. Mixing of the suspension was originally ensured by forced circulation in a draft tube with small diameter and fast propellers. Nowadays, solid suspension is often carried out with large diameter impellers rotating at low speed.

Physical and chemical parameters of crystallization are left aside here. Only agitation-related parameters, which fix the quality of homogenisation, are investigated.

The power involved in agitation is important. It has to be sufficient to keep the solids in suspension and must be able to handle process upsets, such as variations in concentration or in transfer flow rate.

Problems encountered are for instance "stalling" and "overflowing" (a variation in solid concentration can imply difficulties in transferring the suspension from one tank to another with the dip tube, thus resulting in "overflowing" of a lesser concentrated suspension).

Clearly, the vessel performance is closely related to the solid phase distribution. The study aims at developing a process analysis methodology, based on numerical simulations, that is able to give the flow patterns and the solid distributions in precipitators, for various operating conditions.

A batch mode and a continuous one are investigated, and different agitator configurations are tested in search of an improvement of the mixing system, which is also evaluated by the power requirement.

2. Description and Validation of the Numerical Model

In this paragraph, the experimental techniques are first described. They have been used to validate the numerical model, presented later.

2.1 *Experimental devices*

The pilot technology is similar to the one of the 4500 m^3 industrial precipitator. This is a flat-bottomed tank with a multi-stage impeller system, provided with two baffles.

The two experimental devices used for model building and validation are described below.

— 4.28 m^3 pilot vessel has been used to check modelling accuracy. Figure 1 shows the vessel geometry, and table 1 reports all dimensions. The flow regime is fully turbulent.

Aluminium hydroxide particles used for pilot experiments are 50 µm in mean diameter, and are suspended in water. The diameter has been calculated to keep the same terminal settling velocity in water as in the aluminate liquor, where the mean diameter is 95 µm.

The mixing system tested on the pilot plant will be noted F. It is composed of three axial flow propellers of Robin Industries HPM05, with two blades (D=0.91m), pumping downwards, and of an off-bottom turbine of Robin Industries TPM, with four 20°-pitched blades (D=1.03m). (Figure 2) The industrial agitator system has five or six impeller stages, depending on the H/T ratio.

Another configuration (noted S) has been considered. Four baffles are now located in the lower part of the tank, and the bottom turbine is different. It is a Robin Industries TPM/HPM, smaller in diameter than F, with four blades (D=0.71m).

The continuous mode is experimentally performed with an external recirculation, which reproduces the transfer from one tank to the following one (flow rate, keeping the same inner tube velocity: 15 m^3/h).

On the pilot plant, sampling allows experimental measurements of concentrations, and heights of interfaces are visually observed. Torque measurements lead to the power consumption.

— A laboratory vessel, similar in geometry (tank diameter T=0.19 m) with the pilot vessel, has been used for Laser Doppler Velocimetry measurements. Data were used for impeller representation (see below), and model validation in single-phase flow.

2.2 *Description of the modelling technique*

The calculations are performed using the Fluent V.4.3 commercial package. The three-dimensional turbulent flow field is numerically simulated by a finite-volume method on a body-fitted coordinate grid. Considering the symmetry of the system, only half of the vessel is represented. A three-dimensional grid in the cylindrical coordinate system is used to calculate the single phase flow (33000 grid nodes: ($\theta \times r \times z$: 22 × 23 × 65)) and for the continuous mode. The two-phase flow patterns in the batch mode are obtained with a 2D-grid, after gaining confidence in this description.

In the continuous mode, the transfer tube is simulated with an inner tube.

A no-slip boundary condition is imposed on the solid walls, because of the non-elasticity of the fluid. The free surface is modelled with a zero axial velocity and a zero gradient option for other velocities. The vessel axis is specified as a symmetry axis and two cyclic vertical planes (periodic bounds) are imposed on the domain extremity.

The solution of the flow pattern is considered converged when the residuals sum is less than 10^{-3}.

Different points needed to perform the simulation are detailed below:

Impeller representation: Impeller Boundary Conditions (IBC) method

The multiphase model is not available with unstructured grid in the Fluent unstructured V.4 used in this work. It is thus resorted to the IBC method, which avoids direct modelling of the agitator.

Experimental work is then necessary to determine the boundary conditions to impose on the periphery of the volume which represents the agitator. The experimental velocities are measured with the Laser Doppler Velocimetry system for a single phase flow in the laboratory vessel (T=0.19 m). The velocity components and the turbulent characteristics (k and ε) are imposed in the discharged flow of the impeller and on the side. The turbulence kinetic energy k is a direct data of LDV experiments. It is computed with the root mean square (rms) of fluctuating velocity, $v_{i,rms}$:

$$k = \frac{1}{2}\left(v_{r,rms}{}^2 + v_{\theta,rms}{}^2 + v_{z,rms}{}^2\right) \qquad (1)$$

The energy dissipation rate ε is calculated by:

$$\varepsilon = A\frac{k^{3/2}}{D} \qquad (2)$$

The proportional coefficient A is a function of the geometry system and the impeller type. For example, with one propeller A=6, and in multi-stage stirred vessels A=2.4 (Baudou, 1997). For the industrial geometry studied in this work, the coefficient A is not known. This coefficient is a parameter of the simulation.

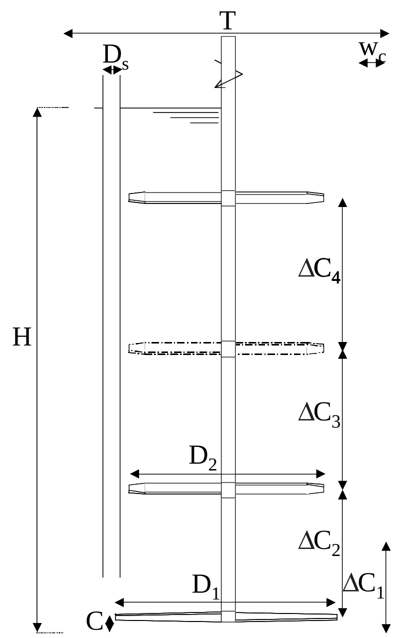


Figure 1 — Stirred vessel geometry

a) turbine F

b) turbine S

Figure 2 — Off-bottom agitators

Table 1 — Pilot geometry (dimensions in m.)

T	H	D_1	D_2	D_s	ΔC_1	C	ΔC_2	ΔC_3	ΔC_4	w_c
1.5	2.424	1.026	0.907	0.094	0.434	0.075	0.588	0.639	0.703	0.126

The IBC method has the disadvantage of requiring experimental measurements to carry out the simulation. Furthermore, it tends to under-predict the energy dissipation, since the energy dissipated in the impeller swept volume is not calculated (Kresta & Wood, 1991).

Physical models

❖ Multiphase model: the Eulerian model

In our case of dense solid-liquid suspension, the Eulerian approach is developed. The coupling between the hydrodynamics of the solid and the liquid phase is thus taken into account. The multiphase flow is complex and still requires simplified approaches. Each phase is assumed to coexist at every point in space in the form of interpenetrating continua. The conservative equations are written for each phase of the multiphase flow. Mass exchange between the phases is not considered due to the slow kinetics of crystal growth relative to the time scale of the system.

In order to couple the two momentum balances, a representation of interphase force is required. This interphase force between the two phases includes only the drag force. The added mass force, the Basset force and the lift force are assumed to be negligible relative to the drag force in the case of solid-liquid suspension (cf Barrué et al., 2001, for a more detailed description).

❖ Turbulence model

The standard k-ε turbulence model (Launder & Spalding, 1974) is used to calculate the turbulence kinetic energy k and its dissipation rate ε, in the single phase simulation. This model is the most widely tested and results are generally considered as reliable with a short calculation time.

The turbulence modelling in multiphase systems is extremely complex due to the large number of terms to be modelled in the momentum equations. The dispersed turbulence model has been used. The influence of interparticle collisions on turbulence is here supposed negligible; the influence of the liquid phase turbulence prevails in the random motion of the solid phase. This hypothesis is justified in a stirred vessel since multiphase flow is dominated by the liquid phase flow due to the impeller action. The domain must not have large regions where the volume fraction for the secondary phase approaches its maximum value. Consequently, the solid phase cannot be totally settled when calculation is started. Turbulent predictions for the continuous phase are obtained using the standard k-ε model supplemented with extra terms that include the interphase turbulent momentum transfer.

2.3 *Validation of the modelling technique*

Single-phase flow pattern

The single-phase flow pattern has been first simulated. This step aims at determining the flow induced by the impeller system, in conditions similar to complete and homogeneous suspension.

The numerical flow prediction for the laboratory vessel has been validated here through experimental velocity measurements. Figure 3 compares numerical and experimental velocity vectors in the vertical plane located between two baffles. The agreement is satisfactory. The main difference is located near the free surface, where numerical axial velocities are slightly higher.

The main difficulty in the simulation lies in correctly describing the turbulence field. Values of the energy dissipation rate, imposed as boundary conditions on the impellers, have a great influence on the turbulence representation. Hence, the proportionality coefficient A, used to calculate ε with experimental data (equation 2), is critical. In this system, a value of A=1 has shown to lead to a correct numerical prediction of the turbulence.

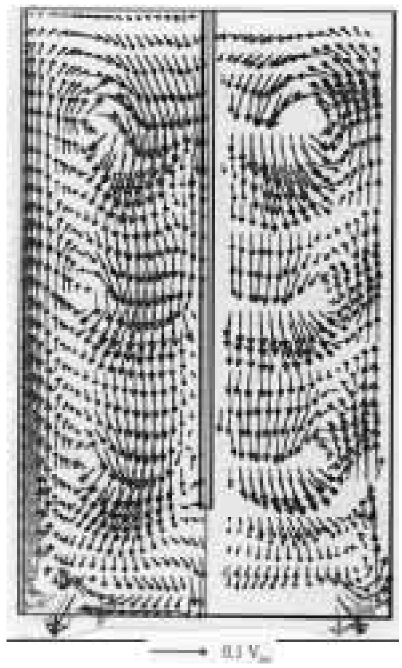


Figure 3 — Velocity vectors in a vertical plane located between two baffles; left: numerical; right: experimental

Two-phase flow pattern

The two-phase flow pattern is calculated with a dynamic simulation. The boundary conditions fixed around the impellers are the same as those for the single-phase flow. The calculation is initiated with an homogeneous suspension and zero-velocities. It is performed up to the steady state, when the axial solid concentration profile is constant.

In the flow patterns obtained, the difference between liquid and solid velocities cannot be observed since the slip velocity is of the same order as the terminal settling velocity (Gosman et al., 1992), which is much lower than liquid velocities observed (ratio of 1000).

Solid distributions are reported in figure 4 for the configuration F in the batch mode. Numerically obtained solid distributions are superposed with experimental results. The agreement is satisfactory. Experimentally measured heights of interfaces compare also well with numerical values.

3. Process parameters impact

The ability of the agitator system in different operating conditions has been evaluated in the batch mode. This operating mode can be encountered if the vessel must be isolated due to process or operation constraints.

3.1 *Rotational speed influence*

Figure 5 shows the liquid flow patterns and the solid volume fraction fields for a mean solid concentration SC=630 g/L, and two rotational speeds (N=60 rpm, and N=40 rpm).

For N=60 rpm (figure 5-a), the liquid flow pattern is quite similar to that of the single-phase. One large circulation loop is created, and the system shows little compartmentalisation. The concentration is homogeneous in the whole volume. Only near the free surface, the solid volume fraction is low and a clear interface is created. The maximum concentration in the bottom is 0.4.

For N=40 rpm (figure 5-b), the flow pattern in the vessel is modified by the presence of the solid phase.

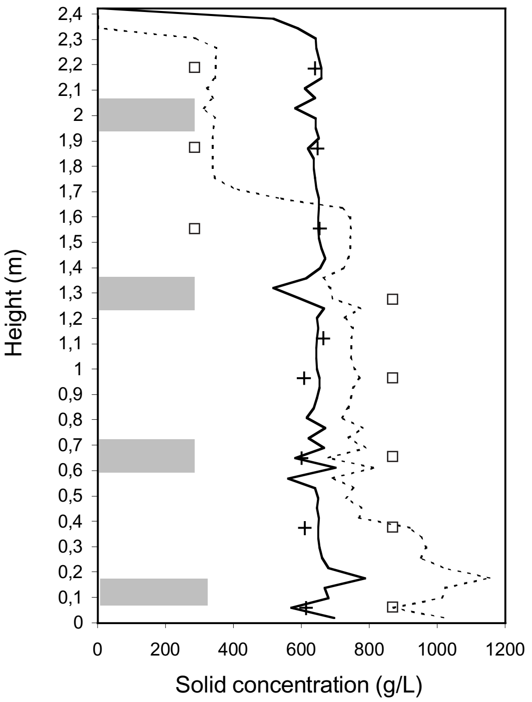


Figure 4 — Numerical and experimental solid profiles; Batch mode, configuration F, SC=630 g/L;
Numerical: ___N=60rpm; - - - N=40rpm
Experimental: + N=60rpm ; □ N=40 rpm

Compartments are created. The upper propeller generates one independent circulation loop. A second loop developed around the two other propellers, and a third loop is generated by the off-bottom turbine. Moreover, the concentration in the vessel is not homogeneous. Four zones are distinguished. Near the free surface, the solid volume fraction is zero. It reaches a value of 0.14 in the upper propeller zone. In the zone of the two intermediate propellers, the solid volume fraction is equal to the mean value, 0.26. And in the last zone, near the bottom, it is very high, with a maximum value of 0.6.

Flow pattern compartments are related to concentration compartments and to accompanying differences in apparent density and viscosity.

The large influence of the rotational speed on the solid distribution has been numerically demonstrated. A 40 rpm rotational speed is too low for the crystalliser to run correctly. The high concentration close to the bottom results in an increased power consumption and an impaired precipitation process.

3.2 *Solid concentration influence*

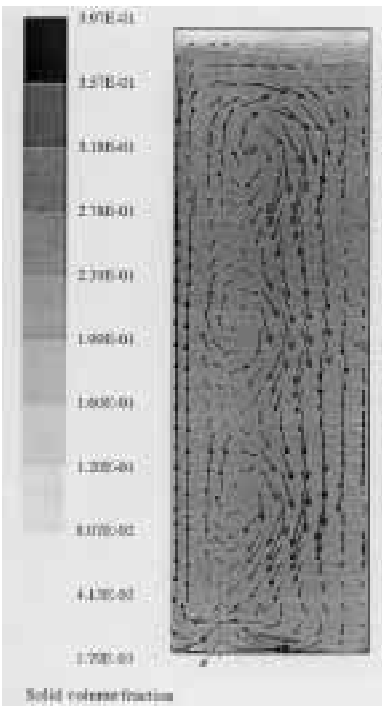
The solid volume fraction fields are compared in figure 6-a for solid concentrations of 630 g/L and 870 g/L, at a 40 rpm rotational speed. The solid distribution observed with a 630 g/L solids content is heterogeneous, while it is homogeneous at a higher solid concentration of 870 g/L. (At a rotational speed of 60 rpm, the two different concentrations lead to the same homogeneous solid distributions.)

In the case of dense solids suspension, the increase in concentration enhances the homogeneity in the vessel. Few experimental results are reported in the literature to support these findings. Nienow (1985) refers to the work reported by Einenkel and Mersmann (1977) who showed that above around 17% by volume of solids, no further increase in the just-suspended rotational speed should be found.

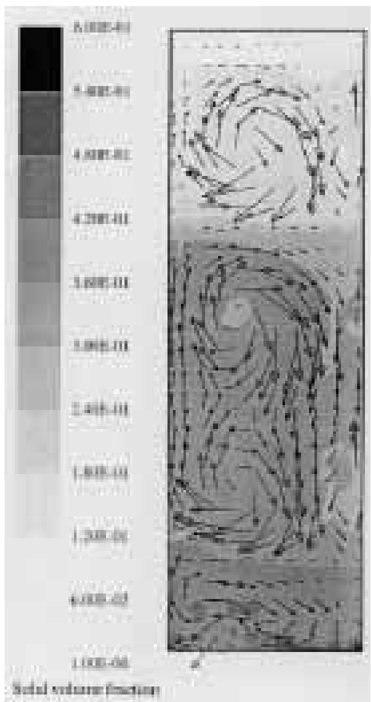
3.3 *Mean diameter influence*

The particle size is numerically characterized by a mean diameter. The influence of the mean particle diameter d on the homogeneity is studied, in the case where SC=630 g/L and N=40rpm. Three values are tested: 30, 50 and 70 µm. The corresponding solid distributions are reported in figure 6-b.

For a 50 µm mean diameter, four zones with different concentrations are observed. They correspond with different circulation loops.

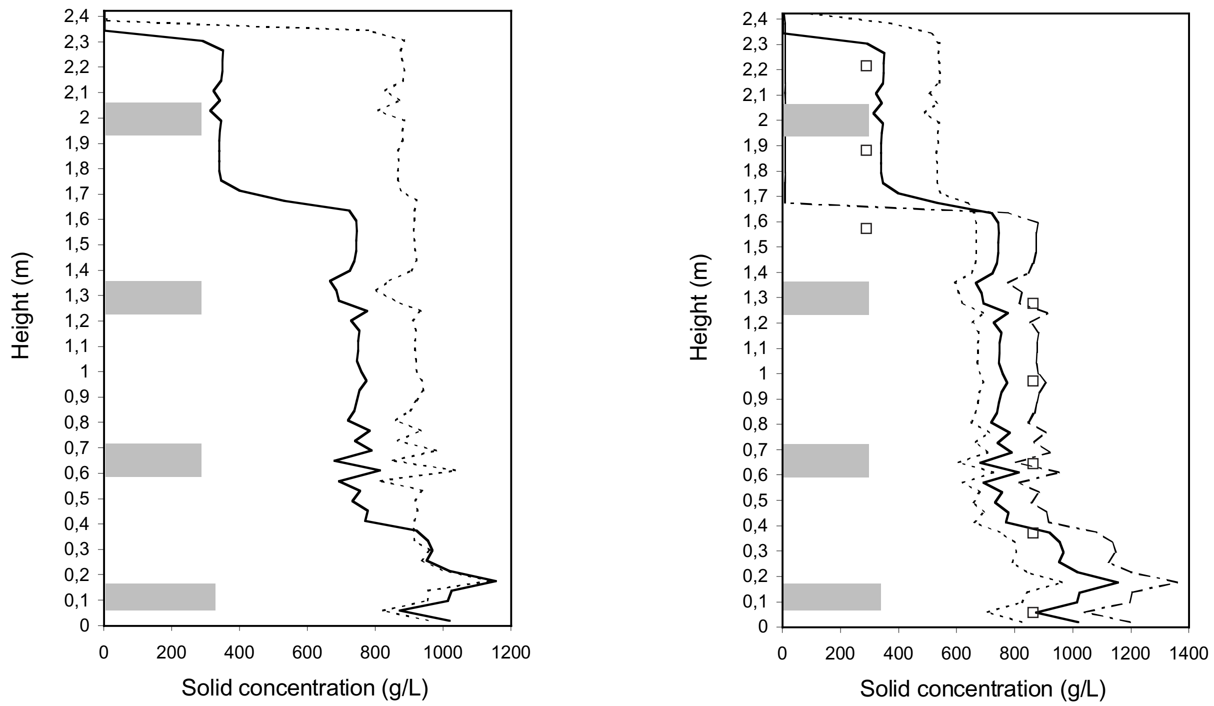


a) N=60rpm



b) N=40rpm

Figure 5 — Flow pattern and solid volume fraction field — Influence of the rotational speed; Batch mode, configuration F, SC=630 g/L; a) N=60 rpm, b) N=40 rpm



a) Influence of the mean solid concentration

b) Influence of the mean particle diameter

Figure 6 — Solid concentration axial profile; Batch mode, configuration F , N=40 rpm;
a) ____ SC=630 g/L - - - - SC=870 g/L;
b) SC=630 g/L; - - - d=30 µm; ____ d=50 µm; - - --- d=70 µm; □ experimental values

When the mean diameter in increased to 70 µm, sedimentation is more important. The region, where the solid volume fraction is zero, is larger and reaches the upper propeller zone. Moreover, the bottom solid volume fraction has its maximum value in a wider zone.

When the mean diameter is 30 µm, homogenisation is acceptable. At the top of the vessel, the volume fraction is not zero, it is low in a small vertical zone. In the bottom, the volume fraction does not reach its maximum value.

The experimental profile, with a particle size distribution, is also visible on figure 6-b: the value of 50 µm for the mean diameter can correctly predict the behaviour of the solid.

These results illustrate the sensitivity of homogeneity on the particle size. The enhanced sedimentation with larger particles, and consequently the higher risk of accumulation in the bottom part of the vessel has been demonstrated. When the bottom concentration is higher, power requirements are more important, and stalling can occur.

4. Precipitation technology

The developed model has been performed to compare various technologies for the precipitation process.

4.1 *Continuous versus batch mode*

For the continuous mode, rather low rotational speeds have been tested. Without transfer, they lead to the formation of a solid concentration gradient in the vessel. The comparison between the two operating modes is illustrated with the configuration F.

When N=40 rpm and SC=630 g/L, three main circulation loops are observed, each corresponding to the propellers. The vessel is yet rather homogeneous (figure 7), with a minimum value of solid volume fraction (0.04), which is higher than in the batch mode. The continuous mode improves homogenisation. The same operating conditions in the batch mode lead indeed to a bad homogeneity (see comparison on figure 8).

If the rotational speed is reduced to 30 rpm in the continuous mode, the solid volume fraction in the precipitator changes slightly (figure 7). The concentration is slightly higher in the bottom part, but no serious sedimentation or at least no clear interface is noted on the top surface. The homogeneity is correct, but the accumulation of solids in the bottom causes the concentration in the inner tube to significantly increase (750 g/L with N=40 rpm vs 900 g/L with N=30 rpm). This constitutes the main operating constraint and air injection in the transfer tube is therefore required.

4.2 *Agitator system design*

4.2.a Bottom impeller design

The off-bottom turbine S is compared with the agitator F, on the basis of suspension homogeneity.

In the batch mode, when SC=630 g/l and N=60 rpm, solid concentration is rather homogeneous with the configuration S, as with the configuration F. Concentration profiles for the two configurations are compared in figure 9-a. The concentration near the free surface is the same. In the bottom part, the peak in concentration for configuration F does not exist for configuration S. The bottom impeller S is of higher performance.

When N=40 rpm, no compartmentalisation in flow pattern is observed with S. Solid concentration is quite homogeneous, but an interface is now present. The comparison of configuration F with S is shown in figure 9-b. The configuration S, while eliminating compartmentalisation, makes it possible to reduce the operating rotational speed.

In the continuous mode, the same conclusions are drawn. Comparison between configurations F and S with N=40 rpm and SC=630 g/L is illustrated on figure 10. The turbine S avoids solid accumulating on the bottom. It is interesting to note that the 30 rpm rotational speed is acceptable with the configuration S (although a limiting value for a correct running), while it is not with the configuration

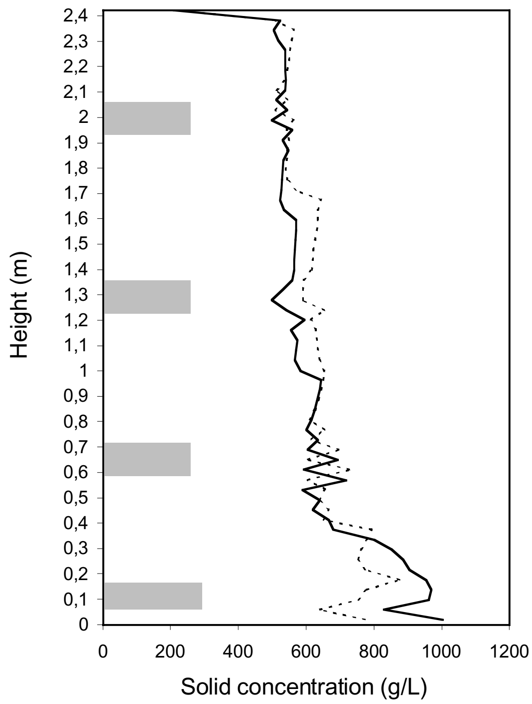


Figure 7 — Solid concentration axial profile;
Continuous mode, configuration F, SC=630 g/L ;
- - - - N=40 rpm ____ N=30 rpm

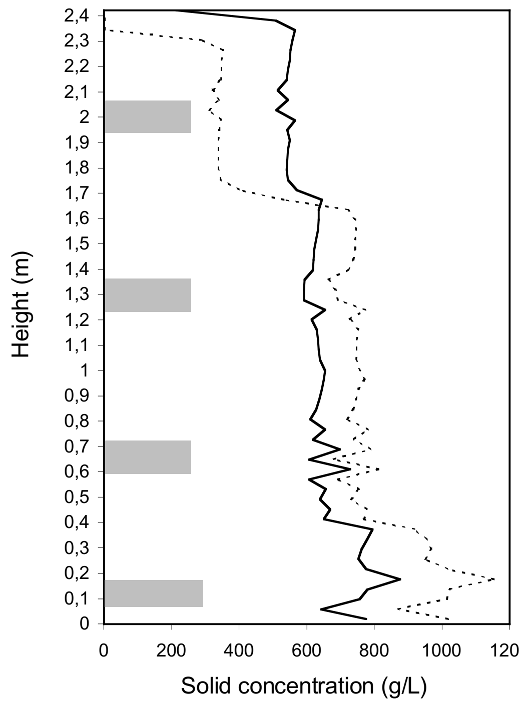


Figure 8 — Solid concentration axial profile;
Configuration F, SC=630 g/L; N=40 rpm;
- - - - batch mode ____ continuous mode

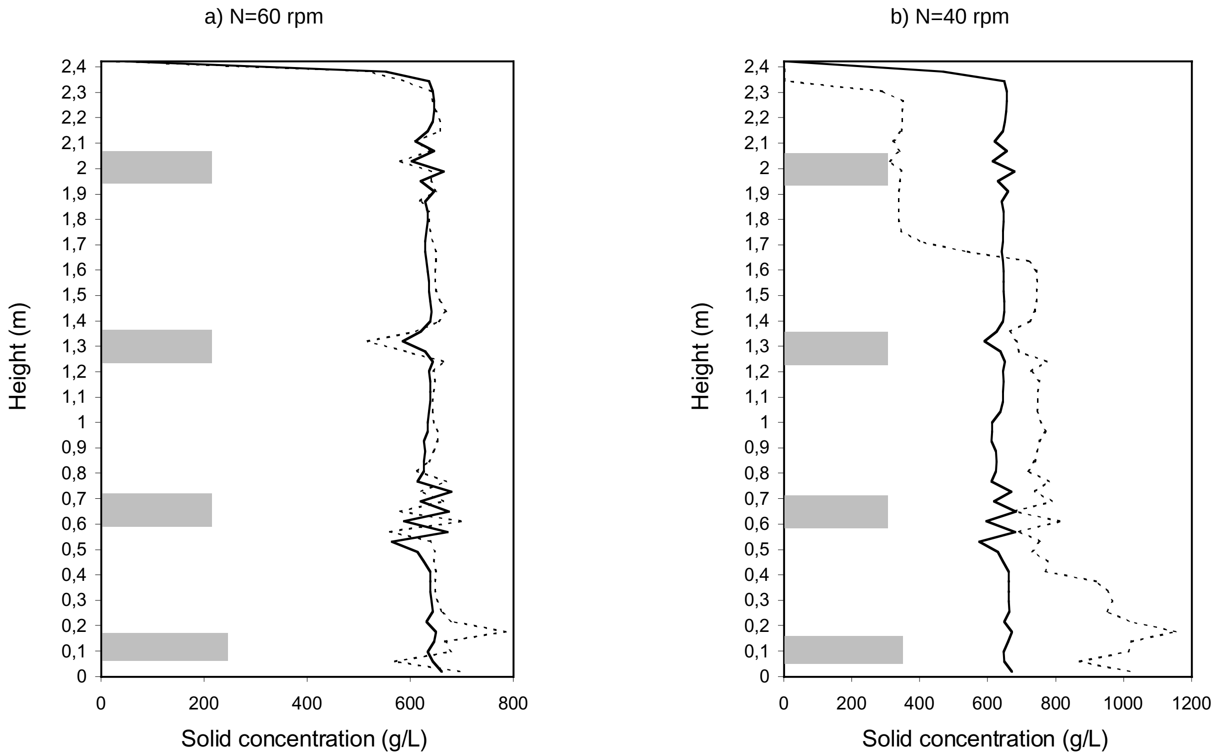


Figure 9 — Solid concentration axial profile — Comparison between configurations F and S;
Batch mode, SC=630 g/L; - - - - F ____ S

F: the concentration in the inner tube is 750 g/L with S and 950 g/L with F (figure 8).

In the determination of the just-suspended rotational speed, the agitator diameter is to a negative power (Zwietering, 1958). For the same agitator type, a larger diameter agitator is thus better able to suspend particles, by increasing pumping capacity. The bottom turbine S is shorter in diameter than F. But in spite of a reduced swept bottom area, the shape of its blades improves the pumping capacity of S (the pumping number for S, calculated from measurements on the laboratory vessel, is more than four times the pumping number for F). The off-bottom turbine S has better mixing capabilities, with a clearer improvement in difficult operating conditions.

4.2.b Impeller positioning on the shaft

An optimisation of the agitator system is proposed via the validated numerical tool. The off-bottom turbine S is used, and the two intermediate impellers are removed, since the "transfer" tube brings the suspension from the bottom to the top of the vessel. The off-bottom turbine ensures the particles to be swept across the bottom, and suspended. The new geometry is tested with N=40 rpm. The solid distribution is reported on figure 11. The homogenisation is correct. Even if restarting would be more difficult with this new configuration of only two agitators, the system has the advantage of using the transfer to circulate the suspension to the upper part of the vessel, with reduced power requirements.

5. Power consumption

Power consumption is a critical parameter to evaluate the performance of one system compared to another. An index of 100 is given to the case F with N=40rpm – SC=630g/L to compare values of the power consumption in the different systems studied (see table 2). (For the reference case, the numerical power consumption is 43% of the experimental one, but an index of 100 is given to both). Relative numerical values are thus reported, since the power is not predicted by the Impeller Boundary Conditions method. It can however be checked that the experimental evolution is well reproduced.

It has been shown in the batch mode that homogeneity is increased with a higher solid concentration. Consequently, working in higher concentrated media allows impeller rotation at lower speeds and reduction of power consumption, while reaching the same state of suspension.

Concerning the design, the agitator S appears to be less power-demanding than the agitator F, for identical rotational speeds and solid concentration values. Moreover, the fact that S can eliminate compartmentalisation enables the operating rotational speed to be reduced (see for SC=630 g/L, in the batch mode, F: N=60 rpm, and S: N=40 rpm lead to an equivalent homogeneity). The new geometrical configuration, with only two agitators, leads to a 40% reduction in the power consumption, compared to the configuration S, with a correct homogenisation.

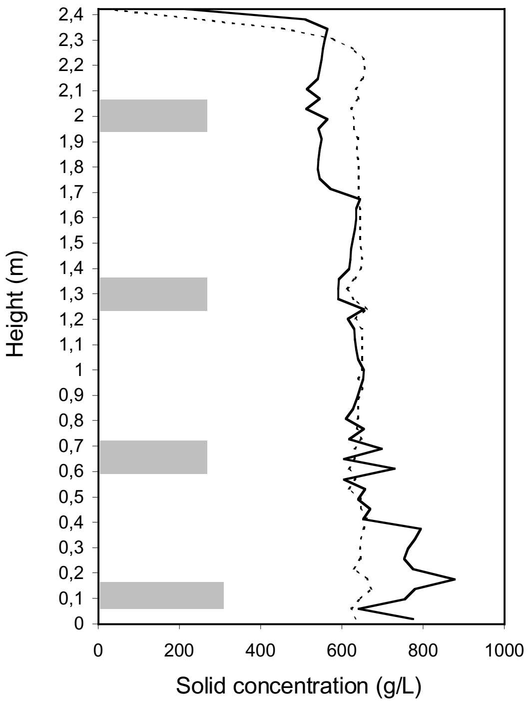


Figure 10 — Solid concentration axial profile; Continuous mode, SC=630 g/L, N=40 rpm; - - - - S ; ____ F

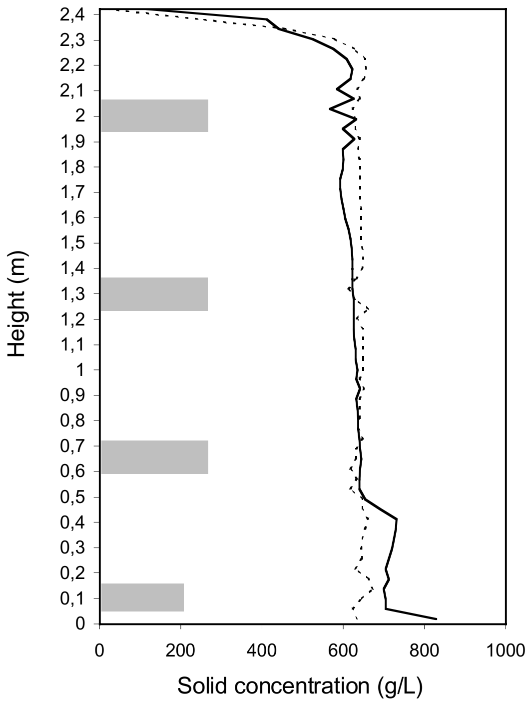


Figure 11 — Solid concentration axial profile; Continuous mode, SC=630 g/L, N=40 rpm; - - - - S ; ____ new configuration

Table 2 — Power consumption P in different configurations

mobile	SC (g/L)	Mode	N (rpm)	P exp.	P num.	solid distribution
F	630	Batch	40	100	100	heterogeneous
F	630	Batch	60	317	243	homogeneous
F	630	continuous	40		83	homogeneous
F	630	continuous	30		71	SC increases in the bottom: difficult siphon transfer
S	630	Batch	40		74	homogeneous
S	630	Batch	60		223	homogeneous
S	630	continuous	40		57	homogeneous
S	630	continuous	30		40	SC increases in the bottom, but acceptable
new config.	630	continuous	40		34	homogeneous
F	870	Batch	40	112	109	homogeneous
F	870	Batch	60	341	289	homogeneous

6. Conclusion

The Computational Fluid Dynamics has proved to be a useful tool in prediction of hydrodynamics and of solid distribution in multi-stage stirred crystallisers. The Eulerian approach has to be developed in case of high coupling between the hydrodynamics of the two phases, in dense solid-liquid suspension. It makes therefore possible to describe changes in flow patterns caused by a non-uniform solid phase distribution.

The influence of operating conditions has been investigated in a batch operating mode, and has shown the particular behaviour of these high concentrated media. The positive effect of solid concentration increase on homogeneity has thus been established. The rotational speed can therefore be reduced when a higher mean concentration is performed. Validation of this behaviour has been done in an industrial precipitator of 4500 m^3 (rotational speed N=7.45 rpm, power consumption P=45 kW) where a less homogeneous solid distribution has been observed with a lower solid concentration.

The increase in the off-bottom concentration due to a reduction in the rotational speed or to an increase in particle mean diameter has also been highlighted in the study. Difficult conditions for precipitation process or transfer from one tank to another can thus be quantitatively anticipated.

Investigation of the continuous mode shows an enhanced homogenisation by the slurry flow through a series of precipitators. The circulation of the suspension from the bottom to the upper surface, independently of the stirring system, significantly improves the homogenisation capabilities of the crystalliser. This suggests taking into account this effect for the global design of precipitation facility. For example, maintaining a forced circulation through a series of precipitators (pumping from the last tank to the first one) can be regarded as an advantageous solution, in case of interruption of pregnant liquor flow rate. In terms of capital expenditure, it can be cheaper than over design of agitators.

Large diameter impellers rotating at low speed have demonstrated to be little power-demanding with a high level of performance. Moreover, the low tip speed has the advantage of not wearing the impellers and of creating a low shearing rate, which avoids crystal breakage and promotes maximum crystal growth. Furthermore, the exploitation cost is cheap due to low power consumption and no maintenance.

Two agitator systems have been compared. The configuration called S, which differs from the other one by the off-bottom turbine, leads to better mixing capabilities, with a lesser power consumption. A system with only one two agitators: one at the top and one at the bottom (type S) is the least power-demanding design.

The CFD tool has shown to be able to investigate new configurations and their limits. It can be useful in enhancing operating conditions or in designing future facilities, with limited pilot plant study.

References

Barrué, H., Bertrand J., Cristol B. and Xuereb C. (2001). Eulerian simulation of dense solid-liquid suspension in multi-stage stirred vessel. *Journal of Chemical Engineering of Japan, Vol. 34, No. 5, 585–594.*

Baudou, C. (1997). Agitation par des systèmes axiaux simples ou multi-étagés. Obtention de l'hydrodynamique par vélocimétrie laser à effet Doppler, Ph D Thesis, INPT, Toulouse, France.

Gosman, A.D., Lekakou, C., Palitis, S., Issa R. I. and Looney, M.K. (1992). Multidimensional modelling of turbulent two-phase flows in stirred vessels. *AIChE J., 38, 1946–1956.*

Kresta, S.M. and Wood, P.E. (1991). Prediction of the three-dimensional turbulent flow in stirred tank. *AIChE J., 37, 448–460.*

Launder, B.E. and Spalding, D.B. (1974). The numerical computation of turbulent flows. *Comp. Meth. Appl. Mech. Engng., 3, 269–289.*

Nienow, A.W. (1985). The suspension of solid particles. In A.W. Nienow, M. F. Edwards & N. Harnby (Eds.), *Mixing in the process industries* (pp. 297–321). London: Butterworths.

Zwietering, T.N. (1958). Suspending of solid particles in liquid by agitators. *Chem. Eng. Sc., Vol. 8, 244–253.*