COMPUTATIONAL FLUID DYNAMICS MODELLING OF A RAPIDORR 444 CLARIFIER

[1]SD PEACOCK, [2]SB DAVIS, [3]KA GOVENDER, [4]K MOODLEY & [5]CJ BROUCKAERT

[1]Tongaat-Hulett Sugar Ltd, Private Bag 3, Glenashley, 4022

[2]Sugar Milling Research Institute, University of Natal, Durban, 4041

[3]Engen Refinery, PO Box 956, Durban, 4000

[4]Feltex, PO Box 13330, Jacobs, 4052

[5]School of Chemical Engineering, University of Natal, Durban 4041

Abstract

The multi-tray Rapidorr 444 clarifier is widely used in the southern African sugar industry for the clarification of mixed juice. An interesting phenomenon associated with these clarifiers is their ability to perform satisfactorily at much higher juice throughputs than those for which they were originally designed. A sugar industry project was carried out to generate a computational fluid dynamics (CFD) model of a single compartment of a Rapidorr 444 clarifier. The model was used to study the fluid flow patterns occurring within the compartment, leading to a better understanding of the behaviour of these vessels. Additionally, the model was used to investigate the effects of simple structural changes on the performance of the clarifier, leading to a recommendation for the placement of a simple baffle within the vessel which could lead to enhanced clarification in the factory. Unfortunately, this recommendation proved to be impractical to implement on a factory scale. Further work on the project in the coming year will focus on the development of a three-dimensional model of the clarifier, which will allow for more flexibility in the modelling of modifications to the clarifier and yield a greater degree of confidence in the results.

Introduction

Computational fluid dynamics (CFD) modelling has found application as a useful tool, primarily for design purposes, in many industries. Within the sugar industry, the Sugar Research Institute in Australia has made use of CFD modelling in many of its research projects over the past few years, one of the most notable applications being the design of the SRI short retention time clarifier (Steindl, 1996).

In order to investigate the use of CFD modelling in the South African sugar industry, the Sugar Milling Research Institute (SMRI) undertook a collaborative project in 1998, involving the SMRI, the sugar industry and the School of Chemical Engineering of the University of Natal, Durban. The project undertook to model one compartment of a Rapidorr 444 clarifier, and to recommend modifications to enhance clarifier performance, based on the results of the CFD modelling.

CFD Modelling

Computational Fluid Dynamics

The behaviour of a fluid in any engineering problem can be described in terms of the variation of its properties (such as velocity, temperature and pressure) in space and over time. It is this information which is of importance in the modelling of a fluid flow system. For example, in the modelling of a clarifier, it is the fluid velocity variations within the vessel that are of primary interest. Applying the principles of conservation of mass, energy and momentum to a fluid produces a system of partial differential equations describing the fluid properties as functions of time and position. This system of partial differential equations forms the mathematical description of the fluid flow.

The governing equations for most fluid flows of practical interest are so complicated that they can not be solved analytically, and thus some sort of numerical technique must be used for their solution. These numerical techniques approximate the governing partial differential equations with systems of algebraic equations, so that a computer can be used to obtain the solution.

In computational fluid dynamics, the fluid flow system to be modelled is broken up into discrete elements, forming a grid-like structure. The approximations to the governing partial differential equations are then solved simultaneously at each of the intersection points on the grid, to obtain a full description of the fluid behaviour within the region of interest. A description of the numerical solution of partial differential equations, with specific reference to the issues important for the solution of fluid flow equations, can be found in Anderson (1995).

Previous Studies

Almost all of the CFD modelling work which has been carried out in the sugar industry has taken place in Australia, where many unit operations have been extensively studied using this technique. Some of these studies have included flow and heat transfer in a vertical crystalliser (Harris *et al.*, 1995), ignition instability in a bagasse-fired boiler (Woodfield *et al.*, 1998) and the optimisation of the design of the SRI short retention time clarifier (Steindl, 1995a; Steindl, 1995b; Steindl, 1996; Steindl *et al.*, 1998).

The SMRI Project

Introduction

At the beginning of 1998, the SMRI undertook a collaborative project to introduce the technique of CFD modelling in the South African sugar industry. The main aims of the project were to become familiar with the technology and to prove its viability as a tool for the detailed study of equipment in a sugar mill, with a view to process optimisation.

Due to the high costs involved in the purchase of CFD modelling software, the project was undertaken in conjunction with the Pollution Research Group (PRG) in the School of Chemical Engineering of the University of Natal. The PRG have been utilising the FLUENT CFD modelling package for consulting purposes for a number of years, for such applications as the modelling of clarifiers in the water industry (Brouckaert *et al.*, 1998).

The application chosen for study was the modelling of the multi-tray Rapidorr 444 clarifier. This clarifier type is quite widely used in the South African sugar industry. An interesting phenomenon associated with these clarifiers is their ability to cope with much higher juice throughputs than those for which they were originally designed. It was considered that further capacity improvements may be possible if the flow patterns within the vessel could be properly understood.

Objectives

Four main objectives were set for the project, namely:

- to produce a model of a Rapidorr 444 clarifier using the technique of CFD modelling,
- to perform appropriate experiments, using an industrial clarifier, for the purposes of validation of the CFD model,
- to use the model to evaluate the effects of simple structural and / or operational modifications on clarifier performance, and
- to test such modifications on an industrial scale to verify the proposed benefits.

Clarifier Modelling

In order to generate a CFD model of the Rapidorr 444 clarifier, it was necessary to select one particular industrial clarifier upon which to base the study. For this purpose, the number 2 clarifier at the Maidstone sugar mill was chosen, due to the proximity of this mill to the SMRI and the University of Natal, and the existence of a spare clarifier at Maidstone which meant that, if required, the number 2 clarifier could be taken off-line for inspection or modification.

As the Rapidorr 444 clarifier consists of four almost identical compartments, it was considered necessary to model only one compartment of the clarifier. Thus, detailed measurements were made of the internals of the second compartment of the Maidstone number 2 clarifier during the first few months of 1998. These measurements were used to generate a detailed drawing of the compartment for use in the CFD modelling.

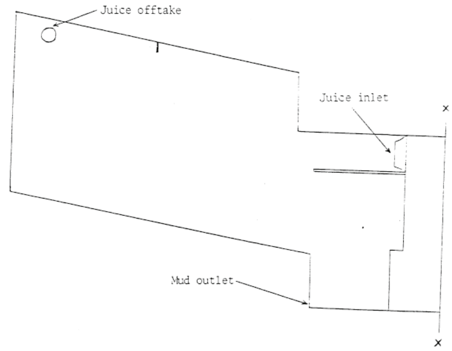

Figure 1. Axi-symmetric clarifier slice used in the CFD modelling.

In order to simplify the problem at hand, and to maintain the computational complexity within reasonable limits, it was decided to base the CFD model on an axi-symmetric two dimensional slice of the clarifier[1]. Furthermore, the mud scrapers were ignored for the purposes of the model. A diagram of the two dimensional slice used in the modelling is shown in Figure 1.

Unfortunately, the Rapidorr 444 clarifier is not symmetrical in design, and thus any axi-symmetric approach to modelling the clarifier is an approximation. The most notable deviation from symmetry in the Rapidorr 444 involves the juice inlet arrangement, as juice enters the compartment through four nozzles located on the central shaft. As the central shaft rotates at about one revolution per minute, the juice inlets may be considered to be symmetrical on a time-averaged sense. However, this is still an approximation of reality. For the purposes of the CFD modelling, it was found to be most appropriate to assume that the juice enters the compartment through a slot, of the same height as the diameter of the current feed inlets, in the central shaft. In this way, the juice was assumed to enter the compartment symmetrically. Similarly, the juice offtake was assumed to be a symmetrical slot cut into the top of the offtake pipes, and the mud withdrawal was assumed to be through a circumferential slot in the mud boot.

Once the geometry of the clarifier had been described fully, it was necessary to provide the CFD modelling software with a description of the problem boundary conditions (*i.e.* the inlet flows, temperatures, etc.) and the physical properties of the

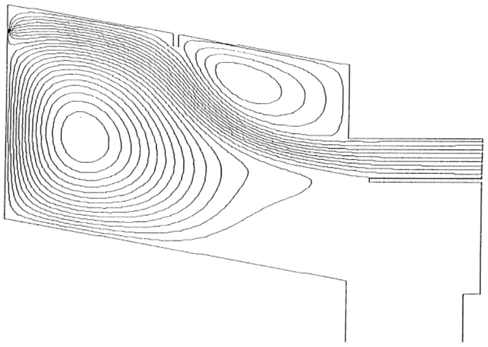

Figure 2. Stream function contour plot for the current clarifier design.

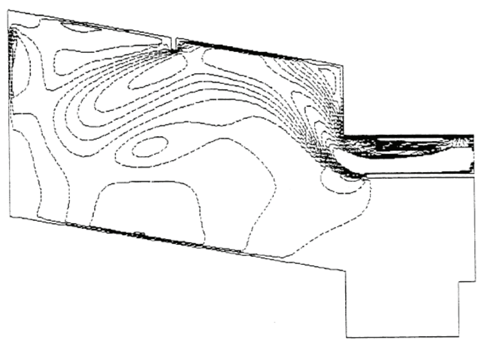

Figure 3. Velocity magnitude contour plot for the current clarifier design.

Table 1. Base-case boundary conditions for CFD modelling.

Juice feed rate	71,88 t/h
Juice viscosity	0,416 cP
Juice density	1009,0 kg/m^3
Mud underflow rate	7,19 t/h
Mud particle density	1009,2 kg/m^3

fluid within the vessel. Based on previous work on the modelling of clarifiers (Steindl, 1995a; Steindl, 1995b; Steindl, 1996; Steindl *et al.*, 1998; Brouckaert *et al.*, 1998), it was decided to ignore the mud phase within the vessel as being too complex to model. To this end, it was assumed that the entire volume of fluid within the vessel consisted of juice, with no mud present. The physical properties of the juice within the vessel were calculated using the published correlations recommended by Peacock (1995). Tests to investigate the settling of mud particles within the clarifier made use of the mud density correlation of Nix (1972). However, in these tests, it was still assumed that the bulk of the fluid within the clarifier was juice, with the presence of only a small number of mud particles assumed. The base-case boundary conditions for the CFD modelling are shown in Table 1.

Results

CFD Clarifier Model

Figures 2, 3 and 4 show the results of CFD modelling of the clarifier. Figure 2 shows the stream function contour plot for the clarifier, where the contour lines in the diagram represent streamlines[2]. The figure shows an extensive region of juice recirculation in the lower portion of the vessel, as well as a small circulation pattern in the upper portion of the clarifier. Qualitatively, it may be assumed that the circulation cell in the lower portion of the compartment negatively impacts on clarification by impeding settling and disturbing the layer of mud on the clarifier floor.

Figure 3 shows a contour plot of the magnitude of the velocity of flow in the clarifier. The contours represent lines of constant velocity magnitude. A region of upward velocities is evident near the inlet of the clarifier, which may impede effective settling of the mud.

The particle path plot for a 0,22 mm diameter particle of mud is shown in Figure 4. This is the largest diameter mud particle which would be carried out of the clarifier in the clear juice stream. All particles of greater diameter will settle. This characteristic particle diameter (*i.e.* the largest diameter particle to be carried over) can be used as a measure of clarification efficiency, in that more favourable settling conditions would result in a smaller size for the largest particle carried over, and *vice versa*.

Model Validation

Validation of the CFD modelling results was found to be an extremely difficult task. As a result of the design of the Rapidorr 444 clarifier, it was found to be impossible to make use of any sort of probe to determine the velocity profiles within the vessel. Furthermore, due to the dark colour and high turbidity of cane sugar juices, dye injection methods would have been futile. A number of tracer tests have been carried out on the clarifier. However, none of these tests has, of yet, provided results suitable for the purposes of model validation. Consequently, no reliable model validation has thus far been carried out. Work on this aspect of the project is continuing.

Model Testing of Modifications

Ideally, in order to maximise the efficiency of clarification, the incoming juice should move uniformly through the body of the clarifier, with no recirculation taking place, from the juice inlets near the centre of the vessel to the juice offtakes at the periphery of the vessel. Similarly, the settling mud should fall evenly towards the bottom of the compartment, where it can be moved by the mud scrapers towards the mud boot. A conceptual diagram of the ideal flow pattern within a clarifier was given by Sabi (1956), as shown in Figure 5.

By contrast, the CFD model of the Rapidorr 444 clarifier shows the occurrence of significant recirculation within the vessel. Enhanced clarification performance should thus be possible by breaking the momentum of these recirculating cells of juice. This can most easily be achieved by the appropriate placement of baffles within the clarifier.

In an effort to improve the performance of the modelled clarifier, five different baffle placement alternatives, as shown in Figure 6, were investigated using CFD modelling. In the figure, baffle A represents the circular ceiling stiffener which currently exists on the roof of the clarifier compartment. Baffles B through F represent those baffle placement alternatives investigated using CFD modelling.

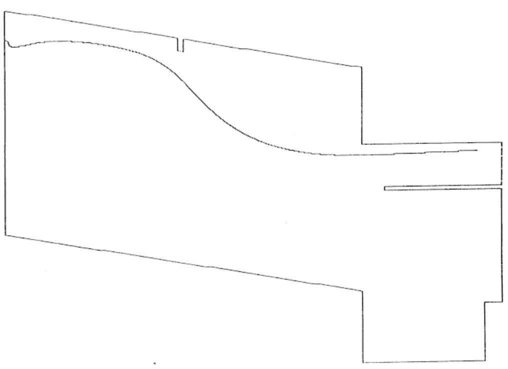

Figure 4. Particle path plot (for a 0,22 mm diameter particle) for the current clarifier design.

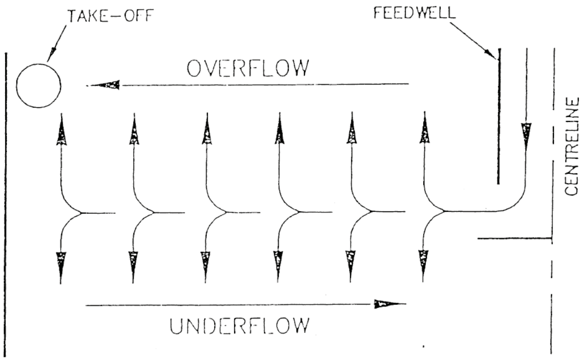

Figure 5. Ideal juice flow pattern within a clarifier (Sabi, 1956).

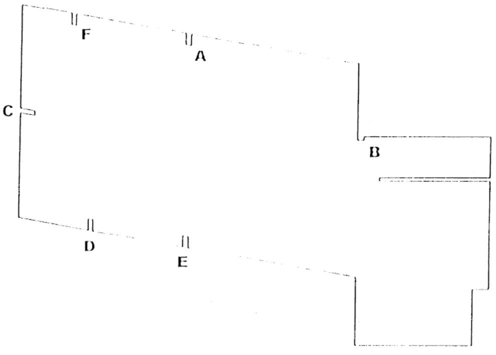

Figure 6. Baffle position alternatives tested using CFD modelling.

The first baffle placement tested was position B, near the juice inlet. As the fluid flow velocities near the centre of a cylindrical clarifier are higher than the velocities near the outer wall, a small baffle near the central juice inlet has a significant effect on the flow patterns within the vessel. This effect is shown in the stream function contour plot in Figure 7. The incoming juice is deflected downwards by the baffle, onto the floor of the compartment. It is obvious that this flow pattern would severely impact on the settling of mud on the floor of the compartment, as is confirmed by the particle path plot, for 0,5 mm and 1,0 mm particles, shown in Figure 8. While a particle of diameter 0,5 mm would have settled in the original design, the figure shows that this size particle would be carried over in the clear juice following the introduction of baffle B. Thus, it would appear that baffle placement B would degrade the performance of the clarifier.

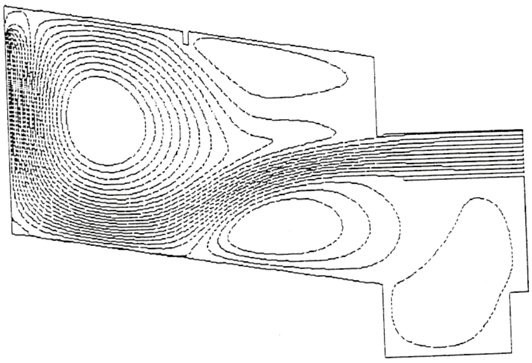

Figure 7. Stream function contour plot for baffle B.

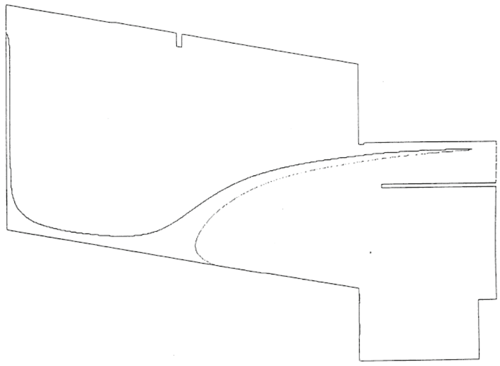

Figure 8. Particle path plot (for 0,5 mm and 1,0 mm particle diameters) for baffle B.

Figure 9 shows the stream function contour plot for a baffle placed at position C, on the outer wall of the clarifier compartment. This baffle position appears, at least qualitatively, to be reasonably effective at breaking the momentum of the recirculating cells of juice. The circulation zone in the lower portion of the compartment has been significantly reduced, which should enhance settling in the clarifier.

However, the particle path plot displayed in Figure 10 shows that the settling behaviour is not substantially better than that for the original design. A particle of diameter 0,22 mm would still be carried over in the clear juice stream.

Similarly, baffle positions D and F were found to be ineffective at enhancing the clarification performance of the modelled clarifier.

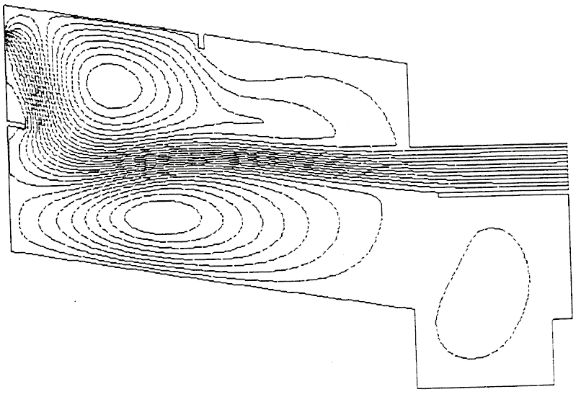

Figure 9. Stream function contour plot for baffle C.

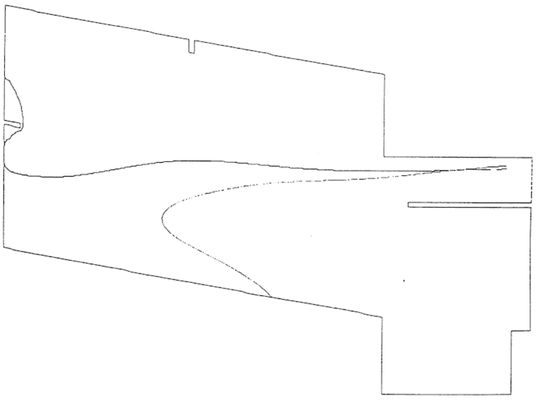

Figure 10. Particle path plot (for 0,22 mm and 0,5 mm particle diameters) for baffle C.

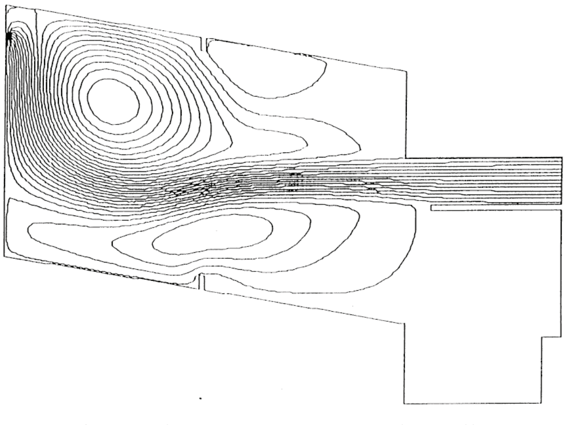

Figure 11. Stream function contour plot for baffle position E.

Figure 11 shows the stream function contour plot for the most effective baffle placement tested, namely a 90 mm high baffle placed in position E. The figure shows the circulation cell in the lower portion of the compartment being almost entirely eliminated, leading to improved settling. Circulation in the upper portion of the compartment was found to remain strong. However, this is presumed not to impact too heavily on the settling characteristics of the mud, as the circulation cell is located near the clear juice offtake.

Figure 12 shows a contour plot of the magnitude of the velocity for the flow of juice within the compartment, for baffle position E. The figure shows an improvement over the original design, as shown in Figure 3. No strong upward velocities exist within the compartment, with the highest upward velocity components existing near the clear juice offtake, where the juice should be reasonably clear. There is, however, still room for improvement, in that the incoming juice crosses the compartment in a "stream", not making efficient use of the entire volume of the clarifier compartment.

Figure 13 shows particle path plots for 0,5 mm and 1,0 mm particles for baffle placement E. The rapid settling of large particles is thus clearly demonstrated. Figure 14 shows a similar particle path plot for a particle of diameter 0,22 mm. It can be seen that the 0,22 mm particle settles under these conditions, which did not occur with the original design. It is thus apparent that a baffle placed in position E within the clarifier (directly beneath the ceiling stiffener A), of optimum height 90 mm, would produce enhanced settling behaviour in the Rapidorr 444 clarifier.

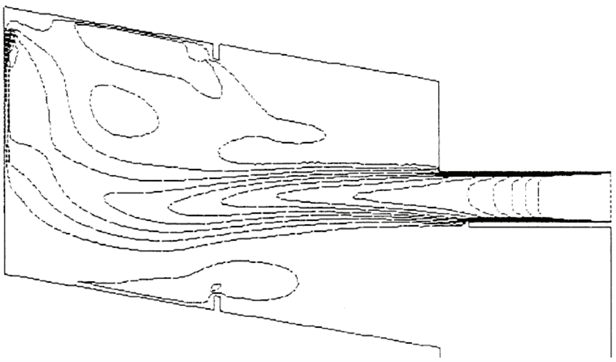

Figure 12. Velocity magnitude contour plot for baffle E.

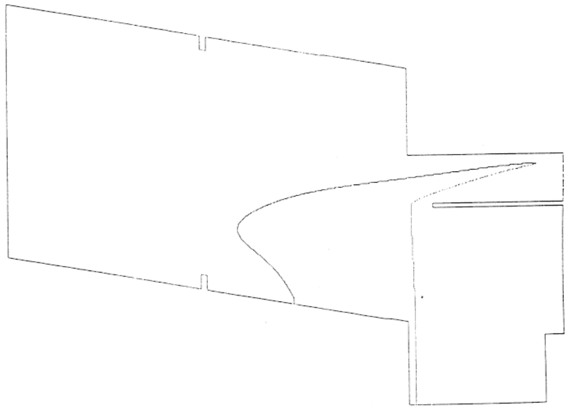

Figure 13. Particle path plots (for 0,5 mm and 1,0 mm diameter particles) for baffle E.

Discussion

While the results of the CFD modelling show an advantage to be gained from placing a baffle on the floor of the clarifier compartment, this is not a particularly useful result in practice. The positioning of the baffle is such that it would obstruct the flow of mud along the floor of the compartment. This would result in a build-up of mud until the baffle is covered, at which point it would cease to impact upon the juice flow in the vessel. This practical problem was, of course, not predicted by the CFD model due to the simplifying assumptions used, namely that the entire volume of the clarifier was filled with juice alone.

It may be possible to construct a suitable "baffle" within the clarifier. Angled mud scrapers, such as those on the mud rake arms, could be hung from a suitable support in a circumferential fashion to construct the required obstruction to juice flow. This would simulate the effect of a baffle to a certain degree, without hindering the free flow of mud across the clarifier floor. However, this arrangement would be costly, and is not guaranteed to be successful. Such alternatives to the originally proposed baffle should ideally be separately modelled as such, as the effect on the flow will be different. Careful design is required to properly interfere with the recirculating juice flow without hindering the flow of mud.

As a result of the difficulties inherent in applying the recommendations of the two-dimensional axi-symmetric clarifier model, and due to the inherent non-symmetrical nature of the Rapidorr 444 clarifier, it has been decided to develop an appropriate three-dimensional model of the clarifier in the coming year. The three-dimensional model will be more capable of accurately predicting the behaviour of the clarifier, leading to a greater degree of confidence in the model results. Furthermore, the three-dimensional model should provide for more flexibility in the testing of modifications to the clarifier.

Conclusions

CFD modelling has found application as a useful design tool in many industries. A collaborative project was undertaken in 1998 to investigate the use of CFD modelling in the South African sugar industry. The project undertook to model one compartment of a Rapidorr 444 clarifier, and to recommend modifications to enhance clarifier performance, based on the CFD modelling results.

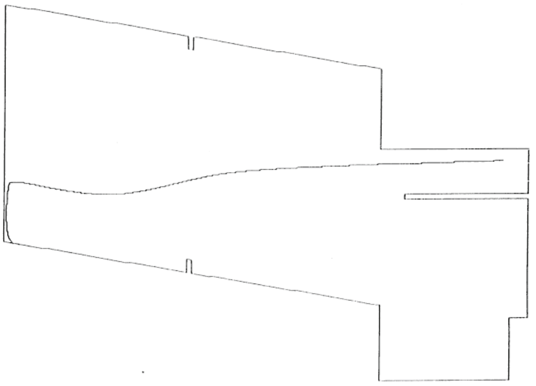

Figure 14. Particle path plot (for a 0,22 mm diameter particle) for baffle E.

CFD modelling of the current clarifier design showed the existence of extensive recirculation cells within the vessel. It should be possible to enhance the clarification efficiency by eliminating these recirculation zones. This can be achieved by the appropriate placement of baffles within the clarifier.

Five baffle positions within the vessel were tested using CFD modelling. One of the baffle configurations tested, namely a 90 mm high baffle on the floor of the compartment, directly beneath the ceiling stiffener, was found to be effective at enhancing the settling behaviour of the clarifier. However, it was found to be impractical to implement the model recommendations on a factory scale.

Future work on the project will focus on the development of a three-dimensional clarifier model, which should allow for more flexibility in the modelling of various modifications to the clarifier, and lead to a greater degree of confidence in the modelling results.

REFERENCES

Anderson, JD (1995). *Computational fluid dynamics: the basics with applications*. McGraw Hill Publishing Company, New York.

Brouckaert, CJ, Fortmann, ME and Buckley, CA (1998). A computational fluid dynamics investigation of a secondary clarifier. *Poster Presentation: 1998 Biennial Conference and Exhibition of the Water Institute of Southern Africa, Cape Town, 4 to 7 May*.

Coulson, JM, Richardson, JF, Backhurst, JR and Harker, JH (1990). *Chemical engineering. Volume 1: Fluid flow, heat transfer and mass transfer*. Pergamon Press, Oxford. 27-28.

Davis, SB (1999). The results of tracer tests on Maidstone clarifier 2. *SMRI Technical Note* 25/99. 29 September 1999. 6pp.

Harris, JA, Robinson, JA and Vigh, S (1995). Mathematical modelling of flow and heat transfer in a vertical crystalliser. *Proc Aust Soc Sug Cane Technol* 17: 216-221.

Massey, BS (1989). *Mechanics of fluids*. Sixth edition. Chapman and Hall, London. 70-71.

Nix, KJ (1972). The density of primary mud. *Proc Aust Soc Sug Cane Technol* 39: 281-287.

Peacock, SD (1995). Predicting physical properties of factory juices and syrups. *Int Sug J* 97: 571-577.

Sabi, N (1956). Principles of clarifier design. *Proc Int Soc Sugar Cane Technol* 9: 619-635.

Steindl, RJ (1995a). Modelling of flows in factory vessels. *SRI Technical Report* 10/95. 35pp.

Steindl, RJ (1995b). Optimum performance through CFD modelling of clarifier designs. *Proc Aust Soc Sug Cane Technol* 17: 207-215.

Steindl, RJ (1996). Optimum performance through CFD modelling of the SRI clarifier design. *Proc S Afr Sug Technol Ass* 70: 145-148.

Steindl, RJ, Fitzmaurice, AL and Alman, CW (1998). Recent developments in clarifier design. *Proc Aust Soc Sug Cane Technol* 20: 477-483.

Woodfield, PL, Kent, JH and Dixon, TF (1998). Computational modelling of a bagasse-fired furnace – effects of fuel moisture. *Proc Aust Soc Sug Cane Technol* 20:458-464.

---

1 The slice is axi-symmetric in that, if rotated through 360° around its axis (x-x in Figure 1), the entire clarifier shape would be generated. The slice is thus not actually parallel-sided, but wedge-shaped.

2 A streamline is an imaginary line across which no fluid flows. In other words, therefore, the direction of the velocity of every particle on the line is along the line (Massey, 1989). Constant velocity over a cross-section of a vessel is shown by equidistant streamlines, and increasing velocity by closer spacing of the streamlines. Similarly, a decrease in velocity would be represented by more widely spaced streamlines (Coulson *et al.*, 1990). Mathematically, this phenomenon results from the definition of the stream function, which is such that its partial derivative in any direction yields the component of fluid velocity at a direction 90° anti-clockwise to that direction.