

**APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS
MODELLING IN WATER TREATMENT**

Report to the Water Research Commission

by

C.J. Brouckaert, T. Huang and C.A. Buckley

Pollution Research Group
University of Natal
Durban

on the Project

"Computational Fluid Dynamics Support to Water Research Projects"

WRC Report No : 1075/1/05

ISBN No : 1-77005 –271-2

March 2005

Disclaimer

This report emanates from a project financed by the Water Research Commission (WRC) and is approved for publication. Approval does not signify that the contents necessarily reflect the views and policies of the WRC or the members of the project steering committee, nor does mention of trade names or commercial products constitute endorsement or recommendation for use.

EXECUTIVE SUMMARY

Almost all water and wastewater treatment equipment rely on continuous through flow of water. Some equipment requires this flow to be well-mixed, whereas other equipment requires plug-flow. Examples of well-mixed systems are activated sludge plants, chemical dosing zones and anaerobic digesters while sand filters (in both filtration and back-wash modes), clarifiers, adsorption columns (ozone, activated carbon and ion exchange) and dissolved air flotation cells are examples of the plug-flow systems. Some processes such as nutrient removal activated sludge plants require the combination of both plug flow and completely mixed reaction zones.

The laboratory-scale experiments that are used to obtain design data for a plant are usually operated under ideal flow conditions; unfortunately it is usually not feasible to carry this through to full-scale plants, due to the greater difficulties and expense of achieving similar ideal conditions on a large scale. The complexity of the flow patterns, and the uncertainties about how they affect the relevant performance indicators for the process involved have led designers of equipment to use *safety factors* based on experience to ensure that the process achieves its required objectives. This means that equipment that is installed is often larger and more expensive than it needs to be.

Computational fluid dynamics (CFD) is a numerical procedure to calculate the properties of moving fluid. Most water treatment processes involve the movement of water. This motion is often complex and difficult or very expensive to observe. The prediction of the flow patterns and other properties of flowing fluids would provide insight into processes which otherwise would not have been possible. A previous WRC project (No 648) indicated the value of CFD modelling of clarifiers and an anaerobic compartment. It was able to both logically explain the unexpected behaviour of the clarifier and in designing features to modify the undesirable flow pattern.

Apart from its use in design of water treatment equipment, CFD modelling can also assist in research into water treatment processes. The project on which this report was based was unusual in that it was initiated to provide a CFD modelling service to assist water researchers who felt that it could enhance their investigations. As a result the project did not have a specific research focus of its own, but adapted to the objectives of each research project that it became involved with. Furthermore, not all the collaborations that were started were fruitful. The main content of the report is a series of case studies, each corresponding to a different investigation. To give the report some thematic consistency, it has been compiled with a view to illustrating the kinds of situations where CFD modelling is useful. To this end, the case studies that were selected for inclusion in the report are those which best fitted this purpose, i.e. they each involved an appropriate application of CFD, and yielded some useful conclusions. Various investigations which for one reason or another did not fit these criteria have been left out. The investigations which are presented have also been *cleaned up* to reflect a logical development which did not always take place in reality; i.e. the various misapplications, misconceptions and dead ends which occurred along the way have been removed from the narratives.

1 Project Objectives

- To provide a service to water researchers by undertaking modelling exercises on proposed and existing equipment so that they may design more efficient experimental equipment and to better understand their experimental results.
- To promote the use of computational fluid mechanics by water authorities, consultants and water researchers.
- To model wastewater treatment secondary settling tanks.
- To assist in the training of academics and students in the practical use of computational fluid mechanics.

2 Overall Course of the Project

At the inaugural meeting in 1999, a list of water-related projects with potential for CFD input was put forward. Seven activities were originally placed in the work programme for 1999, which would have involved collaborations with researchers from around South Africa. These were:

- | | |
|-----------------------------------|------------------------------|
| • Ozone Contactor | • Filter back washing |
| • Anaerobic baffled reactor (ABR) | • Capillary membrane modules |
| • Secondary settling tanks | • Ultrafiltration manifolds |
| • Passive mine water treatment | |

Not all of these evolved into fruitful collaborations; the direction and extent of progress depended on unpredictable factors which emerged. During the course of the project, reasons for making progress, or failing to do so, largely revolved around personnel capacity to undertake the work, though the logistics of collaborating with remote groups also played a part in some cases. No postgraduate students were directly associated with the project during 2000, although some work was done by undergraduate students taking vocational employment.

Between December 2000 and January 2001 modelling of the secondary settling tanks at Durban's Northern Wastewater Treatment Works was carried out to see if it was possible to achieve a significant increase in capacity without extensive redesign. Unfortunately this exercise was inconclusive. Promising results which were initially obtained later proved to be a result of incorrect flow rate data supplied by the Metro. When these were corrected, the improvements that could be achieved using simple baffles were found to be inadequate for the requirements of the treatment works. In view of the lack of a conclusive outcome, this investigation has not been included in the report.

At the beginning of 2001 Tzu-Hua Huang, a chemical engineering graduate of the University of Natal, enrolled for a MScEng, taking the study of the ozone contactor as her thesis topic. Chapter 3 of this report is based on her work. Additionally, Emilie Pastre, a chemical engineering graduate from ENSIGC in Toulouse, undertook her MScEng studies in Durban, in terms of an exchange agreement between ENSIGC and the University of Natal. The topic that she chose involved modelling the final product water reservoirs at the Wiggins Water Treatment Plant, in order to develop an effective control strategy for chlorine dosing. The primary focus of her work was the control, with the CFD model contributing only to the conceptual development of a control model. This work will be reported in another WRC report, so has not been included here. During 2001 the pilot plant Anaerobic Baffled Reactor (ABR) was installed at the Umbilo Wastewater Treatment Works and was operated to gather data for the design of a system for treating wastewater from informal settlements. The CFD support that was provided to this investigation was never a central issue; rather the

CFD modelling was used to help deciding on structural details such as the shape and placement of baffles, by allowing visualisation of the consequences of different options. Operating experience led to the conclusion that the concerns which motivated the CFD modelling were not of crucial importance, and that the CFD work had not made a significant contribution to the investigation. This work has also been omitted from this report as it will be part of the ABR report.

In July of 2001 Ms. Huang visited Institut National des Sciences et Appliquée (INSA) de Toulouse, for an month. There, with the help of Prof. A. Liné, she undertook some two phase modelling of the ozone contactor. The investigation formed the basis of a paper presented at the *IWA Conference on Water and Wastewater Management for Developing Countries*, Kuala Lumpur, Malaysia, 29-31 October 2001, which was subsequently accepted for publication in *Water Science and Technology*. Ms. Huang subsequently applied successfully to have her MScEng registration changed to PhD.

Also during 2001, an investigation into the backwashing of sand filters at the Faure Water Treatment Plant operated by the Cape Metropolitan Council. The motivation for the investigation was that the efficiency of back washing the sand filters was not uniform over their whole area, and that excessive quantities of backwash water were required to get some areas of the filter clean. A number of other treatment works under the control of Cape Metro have filters of similar design, with similar problems, so a solution to the problem would be widely applicable. This study is the basis of chapter 4.

in 2002 an extension to the project was granted, in order to complete the study on the ozone contactor at the Wiggins Water Treatment Plant. A series of studies was carried out to monitor the contactor at the Wiggins Waterworks, to obtain data to be used in partial validation of the computational fluid dynamic model.

The studies reported in chapters 5 and 6 resulted from suggestions made by members of the steering committee. During 2002 CFD modelling was undertaken to support the design of modifications to a potable water clarifier at the Hazelmere Water Treatment Works north of Durban, operated by Umgeni Water. This was the same clarifier that had been reported in WRC Report No 648/1/02 *The Application of Computation Fluid Dynamics to Water and Wastewater Treatment Plants*, and the upgrade was a direct sequel to the previous investigation. The final investigation was prompted by a suggestion that a CFD analysis of batch settling tests might be useful for developing a strategy to advance the CFD modelling of secondary settling tanks.

3 Literature Survey: the use of CFD in water treatment

A survey was carried out on the literature relating to the use of CFD modelling in water and wastewater treatment. This covered recent papers on CFD in water research, CFD in the water and wastewater industry, and CFD in environmental studies. It was found that, although CFD has a very extensive literature, very little of this is related to water treatment. In a search conducted through the *ISI Web of Knowledge* site, the keyword *CFD* found 2196 references in the 12 months preceding September 2003, but *CFD water treatment* found only 8, and of these only 3 referred to water treatment as understood in this report.

4 Case Studies

The case studies carried involved a range of aspects that covered many of the broad issues found in the literature.

4.1 The Ozone Contactor at the Wiggins water treatment works

Ozonation is used in drinking water treatment primarily to oxidise iron and manganese, to remove odour- or taste-causing compounds, and to destroy micro-organisms. The peripheral benefits include possible reduction in coagulant demand, enhancement of algae removal and the colour removal. Ozonation of water is typically carried out by dispersing gas containing ozone into the liquid phase. The contact between the two phases accompanied by an ozone mass transfer takes place in ozone contactors.

The pre-ozonation system at Wiggins Waterworks, operated by Umgeni Water in Durban, consists of four contactors. Each of the contactors is preceded by a static mixer such that every chamber can operate individually or in parallel with another contactor. An ozone-oxygen gas mixture is injected as a side-stream through the static mixer which is employed to achieve high mass transfer of ozone to water. The Wiggins pre-ozonation system has an unusual configuration, as it was adapted from an existing structure, which had originally been designed for a different purpose. Water enters from the static mixer at the bottom, and passes through three horizontal compartments before it exits over the weir at the top.

The objectives of the investigation were:

- To determine the actual liquid residence time distribution as a function of flow conditions through the contactors
- To optimise the disinfection efficiency of the contactors by
 - ♦ selecting the most appropriate position for monitoring residual ozone concentration.
 - ♦ achieving the most efficient use of the ozone dosed to the system.

In outline, the phases of the investigation were approximately as follows (various mistaken or misguided excursions have been excised from the sequence):

- 1) A single phase (water only) CFD model was set up to provide an initial understanding of the flow patterns in the contactor.
- 2) Tracer tests using lithium chloride were carried out to compare with the model. These were conducted with and without gas injection into the static mixer. Although the gas injection did cause some noticeable difference in the measured outlet concentrations, the effect on the overall residence time distribution was very small. After some adjustment to the model, it was concluded that a single phase (i.e. water only) model would give an adequate representation of the RTD for modelling the ozone reactions.
- 3) An ozone reaction scheme was added to the model, using kinetic data obtained from the literature. From this it was evident that the ozone consumption is very dependent on the local characteristics of the water, which need to be determined experimentally.
- 4) Sampling lines were installed on the contactor which allowed ozone concentrations to be measured at various points. The positions of these were chosen with reference to the CFD model results. Consideration of both the model results and the measurements suggested that the best point for monitoring the ozone concentration for control purposes was located between the 2nd and 3rd compartments, rather than at the outlet of the 3rd compartment as at present.
- 5) A laboratory study was initiated to obtain rate constants for the reaction scheme. At the time of writing, this was in progress.
- 6) To check the validity of neglecting the effect of gas injection, some two phase modelling (gas bubbles in liquid) was carried out. First a two dimensional model was tried, and when this proved successful, a full three dimensional model was implemented. However satisfactory results were not achieved, due to grid resolution and convergence

difficulties. These might well have been resolved with more powerful computers than those available.

The main conclusions related to the objectives of the case study were:

- The location of the residual ozone monitor used in the control of the ozone dose should be moved from its current position close to the outlet weir to a point at the end of the middle compartment, level with the position of the experimental sampling point 4. However, although sampling point 4 was located on the right side of the contactor, the model suggested that a more consistent and reliable signal would be obtained if the sensor were located on the left.
- The disinfection efficiency of the system is very sensitive to the level of ozone consuming substances in the raw water feed. Since this factor can be expected to vary seasonally and with weather conditions, operational procedures should be developed to take it into account in determining the ozonation control strategy.

The investigation was aimed at improving the operating rules for the contactor rather than changing any aspects of its design, however the mass of detailed information provided by the models did indicate aspects of the design which could be improved.

- The Residence Time Distribution exhibited by the contactor (both simulated and actual) is somewhat disappointing given its structure. The US EPA *Disinfection Benchmarking and Profiling Guidance Manual* presents a broad classification of contactors in terms of their general configuration, the Wiggins contactor might be placed under the category of being provided with “serpentine intra-basin baffles”. According to this classification the “baffling condition” should fall somewhere between “average” and “superior” The measured residence time distribution fell between “average” and “poor” The model results showed that this degradation in performance was mainly due to left-right asymmetry in the flow distribution, and recirculating vortices in the bottom compartment. The latter occur because the feed is concentrated at one point rather than being distributed evenly across the width of the compartment. CFD modelling could be used to design modifications (such as strategically placed baffles) to reduce these non-idealities and therefore improve the contactor performance.

Since the study was by far the most comprehensive undertaken during the project, it provided the broadest illustration of the use of CFD in research into water treatment processes, and some of its strengths and weaknesses.

- **The large size and geometrical complexity of the contactor:** on the one hand this meant that the CFD model provided insight into details of the processes within the reactor that would be almost impossible to obtain experimentally, on the other hand it led to a very large model with convergence difficulties and long solution times.
- **The hydraulic sub-model:** the water-only hydraulic model gave very good agreement with experimental data, with only minimal calibration (i.e. the adjustment of the turbulent intensity of the feed from the static mixer). Together with similar experiences in other investigations, this indicates that CFD predictions of straightforward flow patterns and RTDs can be usually be used with a high degree of confidence.
- **The two phase (gas-liquid) sub-model:** two phase modelling was undertaken, but the conclusion that could be drawn were limited. Difficulties associated with grid resolution and convergence are very much greater than with the water-only model. These were overcome successfully in the 2-D case by making the computational grid fine enough, but the 2-D model proved to be an inadequate representation of the system. In the 3-D model, making the grid fine enough for stability resulted in a model that was too large for the

computers available to handle. Experimental verification would have also presented much greater difficulties than in the single phase case.

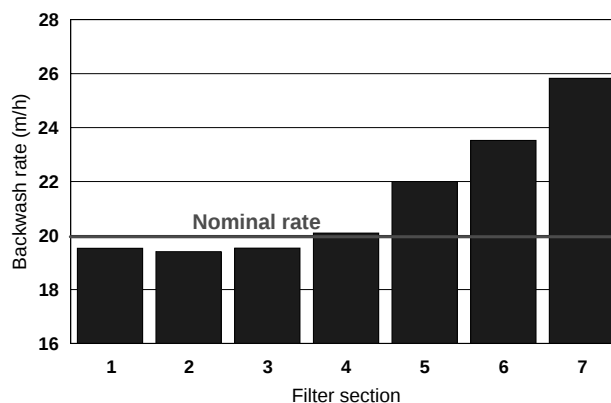
- **The reaction sub-model :** in this study, adding the reaction sub-model did not involve much extra difficulty as far as the modelling was concerned. This fortunate result was due to the relatively slow reaction rates, with time constants of the same order as the hydraulic residence time of the system. The main difficulty associated with the reaction modelling is due to the lack of detailed knowledge of the reaction mechanisms and the kinetic parameters involved. This meant that model predictions could not be trusted without experimental verification and calibration. It also meant that extrapolation of the model to operating conditions different from the calibration set was unreliable. In an attempt to improve the situation, a laboratory study of the reaction kinetics has been initiated, but is incomplete at the time of writing.

These conclusions can be generalised to an extent by noting that CFD modelling is very successful where the underlying physics of the process are very well understood, but becomes less useful and reliable when sub-models are added which involve approximations and uncertainties.

4.2 The sand filter backwash system at the Faure Water Treatment Works

A CFD model was set up to model the water-only phase of the backwash cycle one of the sand filters operating at the Faure treatment works. The objective of the investigation was to determine why parts of the filter take much longer to clean than others, and to propose modifications that would lead to improved operation. The modelling was accordingly divided into two phases: modelling of the existing configuration and modelling of the proposed improvement.

The model of the existing configuration showed that the pressure in the underdrain tends to increase towards the far end from the feed, due the general deceleration of the flow. Because of this increase in pressure, the flow through the nozzle slabs also tends to increase towards the far end, where the model predicted that the flow would be about 30% higher than the average for the filter. This means that the parts of the filter close to the feed end get less than their fair share of the flow, which explains why they take longer to clean.



Simulated backwash rates in the seven filter sections

Having satisfactorily explained the reason for the operational problem, a CFD model of a proposed solution to the problem was set up. This was to install a flow distributor down the centre of the under drain, which ensures an even supply of water to each section. This would take the form of a pipe laid along the length of the under drain, with holes on each side. The

diameters and spacing of these holes would need to be carefully gradated down the length of the pipe to deliver a uniform volumetric flow per unit length in spite of the pressure rise.

The model was constructed in two parts, one for the flow inside the distributor, and one for outside the distributor. The model predicted that the variation over the filter surface should be reduced to less than 1%.

Although the distributor appeared to be relatively inexpensive to install, at the time of writing it had not been installed so its efficacy had not been verified.

4.3 The clarifier upgrade at the Hazelmere Water Treatment Works

In this case study, a series of CFD models were generated to support the design work for modifications to a clarifier which needed to have its performance upgraded. The peripheral feed arrangement for this clarifier was particularly unusual, and caused it to be plagued by poor feed distribution resulting in severe short-circuiting. An investigation into the maldistribution of flow occurring in this clarifier using tracer testing and a CFD model was reported in WRC Report No 648/1/02 *The Application of Computation Fluid Dynamics to Water and Wastewater Treatment Plants*. The conclusion of that investigation had been that converting the clarifier to a central feed arrangement was the only way to obtain a significant improvement in its performance.

In June 2000 Umgeni Water reviewed the existing design and made recommendations on proposed improvements to Clarifier 1 and Clarifier 2. The working group tabled the following design proposal:

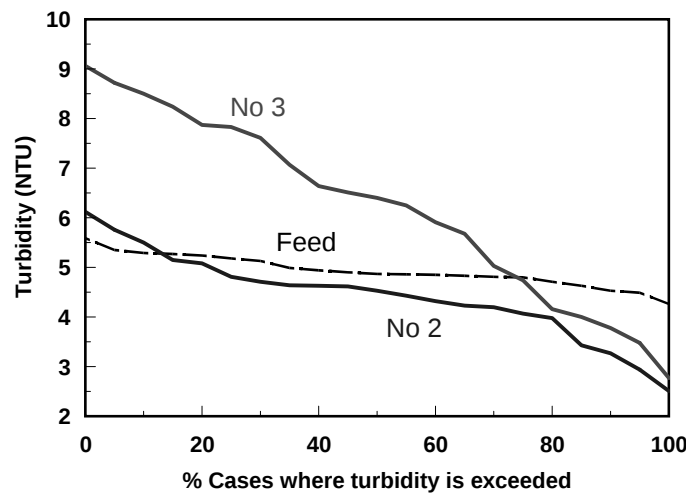
- to convert the existing peripheral inlet system to a central inlet, which required construction of the inlet pipe below the existing floor;
- to install baffles on the central inlet port imparted a rotational component to the flow in the flocculation zone to enhance passive flocculation;
- To provide a 10 m diameter flocculator zone in the centre of the clarifier providing, a flocculation time of 30 minutes;
- To install two paddle flocculators within the new flocculator chamber.

It was expected that these modifications would increase the clarifier capacity from about 9 ML/d to 15 ML/day at an up flow velocity of 1.2 m/h (within the design guideline value of 1.5 m/h overflow rate for this type of clarifier). While the design work was being carried out, CFD modelling was undertaken to help evaluate various design options. This interaction led to a number of changes to the design:

- a single central sludge discharge hopper was designed in place of the hopper originally located within the sedimentation zone.
- the floor within the flocculation zone was sloped at 1:12 towards the centre, to aid transport of concentrated sludge into the central hopper against the outward flow of water.
- at the point where the water flow passed under the skirt between the flocculation zone and the settling zone a step was made for the sludge to flow over, to prevent it being re-entrained.
- the dimensions of the flow area immediately beneath the flocculation skirt were decided after evaluating several CFD models.

The modelling was based on data supplied by the Umgeni Water design team as the design work was proceeding, and the configuration was continually being changed while the modelling exercise was in progress.

The design modifications were implemented on the No. 2 clarifier at Hazelmere, which was re-commissioned in August 2002. A comparative performance test between clarifiers No. 2 and No. 3, which also has a central feed configuration but none of the other CFD-designed features, was carried out during August 2003.



Turbidities measured at each clarifier during the comparative test

The graph shows the inlet and outlet turbidities measured during the test, plotted first against the elapsed time of the test, and also on a percentile basis. During the test period the feed water turbidity was extremely low, so that flocculant dosage increased the turbidity significantly, which is how the turbidity from No. 3 comes to be higher than the water feeding it at times. The superior performance of the No. 2 is clearly evident, which vindicates the use of CFD in its design.

4.4 Batch settling of secondary sewage sludge

The modelling of solids settleability is essential for modelling settling tanks in water and wastewater treatment. Until the advent of hydrodynamic models, the focus of modelling solids settleability was on describing the behaviour of the solids in the water while the water itself was considered a stationary or ideally moving medium in which the solids settled. Hydrodynamic models now allow the behaviour of the water in the settling tank to be modelled. While the modelling of the water flow has made extraordinary advances in the past 20 years, modelling the settleability of the solids has not improved much over this time. In fact, the weakest part of hydrodynamic models of settling tanks may be the modelling of settleability of the solids. This investigation explored methods for measuring and modelling solids settleability with the view of improving these for hydrodynamic models of settling tanks.

The design and operation of secondary clarifiers is commonly based on the solid flux theory. The basic data required for the application of this theory can be obtained from multiple batch tests by which the stirred zone settling velocities over a range of sludge concentrations are measured (dilution experiments).

Many CFD modellers of settling tanks have used the *Takács equation* to describe the settling velocity of the solids, however the equation is not well formulated for experimental calibration. It contains 4 constants that require measurement to calibrate it. Only 2 of these constants are readily measurable from laboratory scale tests, the remaining 2 usually have to be inferred from measured values of the suspended solids in the effluent from the full-scale

clarifier. This is unsatisfactory, in that the clarifier cannot be properly modelled without using its own operating data.

The strategy of this investigation was to incorporate the Takács settling model into the simulation of batch settling tests, in an attempt to identify characteristics which might be amenable to experimental measurement, and which might allow the complete set of Takács parameters to be estimated.

From the batch settling simulations, two characteristic settling behaviours were identified, dependent on the initial concentration of sludge in the settling test. The features of *Type I* settling, which occurred at higher sludge concentrations were:

The notable features of this result are:

- The sharp interface between “clear” liquid and sludge settling at the initial concentration.
- The residual concentration of non-settling sludge in the “clear” liquid.
- The smooth build-up of a concentrated layer of settled sludge on the floor of the column.

Type I settling occurs for initial sludge concentrations higher than a critical value C_m for which the sludge settling velocity is a maximum. For tests starting from concentrations below C_m the a qualitatively different *Type II* settling behaviour takes over. Under these conditions, the interface with the “clear” liquid is diffuse, whereas the interface with the settled sludge at the bottom of the column is sharp. These qualitative features correspond well to experimental observations.

Although the simulation results indicated there was no direct way to determine all the Takács equation parameters from a single batch settling test (which confirms practical experience), a series of experiments with different starting concentrations could be conducted to determine the value of the critical concentration C_m at which the settling behaviour switches between Type I and Type II. This value, together with the settling velocity v_m of the sludge at this concentration, could then be used to infer the remaining Takács equation parameters.

The character of this case-study was somewhat different to the others undertaken during the project, in that the CFD model was used to suggest a direction for further research, rather than to interpret or extrapolate research results. An experimental investigation needs to be undertaken to verify the suggested protocol.

4.5 Conclusions and recommendations

This section presents the more general conclusions which arise from considering the project as a whole.

4.5.1 The scope for application of CFD modelling in water treatment

It is interesting that most of the broad issues identified the literature were touched on in one form or another during the investigations undertaken during this project. The Wiggins ozone contactor started with simple hydraulic modelling and prediction of the residence time distribution, and progressed to more complex physical modelling of reaction kinetics, disinfection performance and 2-phase flow. The Faure filter backwashing investigation looked at simple hydraulic modelling of flow distribution in the context of an equipment re-design exercise, concentrating entirely on the one specific issue for the design, and ignoring or approximating all other aspects of the system. The Hazelmere clarifier investigation was similarly a re-design exercise, but this time it involved two-phase modelling. It also provided experience of working interactively in the design team, with the concomitant time and budget constraints, requiring strict focus on the specific design objectives, at the expense of realism and unnecessary detail. Finally the batch settling investigation again involved two phase modelling, but this time addressed a purely

theoretical question. Thus the experience gained allows a reasonably comprehensive assessment of the role that CFD can play in water and wastewater treatment.

The very fundamental nature of the CFD approach has the advantage of being able to represent appropriate systems (see below) in great detail with minimal requirements for empirical data, but the disadvantages of complexity and difficulty in solving the resulting systems of equations. These practical difficulties prevent CFD from being a universally appropriate approach to all problems involving fluid flow, in spite of its fundamental basis. Generally, CFD is most useful for systems which are *well-connected*, that is, where **all** the boundary conditions have relatively strong influences on **all** parts of the flow field. This applies to many systems found in water treatment, such as reservoirs, contact chambers, sedimentation basins, ponds and even lakes and lagoons. However there are also many systems in water treatment where CFD does *not* provide an effective approach, for example a set of equipment connected by a pipe network, or a long reach of a river. In such cases the CFD model would expend enormous computational effort on calculating the practically negligible effects of remote boundary conditions.

The simplest CFD models consider only the hydraulic aspects of a system. Frequently these models are used to predict the residence time distribution (RTD), which often provides a link to more direct performance indicators through empirical rules based on experience (e.g. the disinfection CT rule). As CFD modelling has become more established, more detailed models are appearing which attempt direct representations the physical and chemical processes taking place in the treatment processes, such as sedimentation, flocculation, inter-phase mass transfer and chemical reaction. In all cases these more complex models need to be supported by experimental studies to establish the parameters for the physical and chemical parts of the models. The CFD modelling thus has a role in both the interpretation of results from experimental apparatus, and in extrapolating research results to the design of full-scale processes.

The relationship between tracer testing and CFD modelling to determine the RTD of a system is worth mentioning. The ozone contactor study demonstrated the use of tracer testing to verify the CFD model, and concluded that CFD modelling is often able to predict the RTD very accurately. However, if the RTD is all that is required, the tracer test may be quicker and less expensive to perform than to develop a CFD model. However this depends on the size of the system: for many water treatment systems the size is such that a very large dose of tracer is required, together with an elaborate and expensive sampling and chemical analysis programme, and the time required to complete the test is so long that it is not feasible to maintain conditions steady for long enough.. Nevertheless, tracer testing should always be considered as a possible alternative to a CFD study, as long as the RTD is adequate to address the required purpose. A less tangible factor that should be borne in mind is the extra insight that the CFD model is able to bring to the investigator.

4.5.2 The costs involved in CFD modelling

The literature does not reflect a widespread acceptance of CFD modelling in water and wastewater treatment, and this is mirrored in the South African water industry. The cost involved in undertaking such modelling is undoubtedly one of the factors contributing to this situation. To some extent this is a matter of perception, but the reality is that the overall cost of a CFD investigation is likely to be fairly high.

To start with, the skills required are relatively rare, and take some time to develop. CFD has not yet found a place in undergraduate curricula, so postgraduate training is involved. The underlying mathematics is complex and not easy for practising engineers to master on their own. It is true that the CFD software now available takes care of almost all the mathematical complexities, but paradoxically this may make the problem worse rather than better, because

it makes it so easy to obtain plausible results which one does not really understand, increasing the potential for making serious errors. The large international water treatment firms such as Veolia and Thames Water have a small number of CFD specialists who act as internal consultants for equipment design. During this project, the example of the Hazelmere clarifier illustrated how such an arrangement might work..

CFD modelling does require more than usually powerful computing hardware, and the requirements escalate rapidly when modelling the more complex physical processes, as illustrated by the difficulties encountered with the gas phase in the ozone contactor. However the hardware cost is less and less significant as a cost factor, as it continues to decline steadily, and since the skilled personnel costs involved in such advanced modelling are likely to be very significant.

The cost of CFD software has come down steadily during the duration of the project, but is still high, even with a discounted academic licence. A non-academic licence would have been much more expensive. The development of the software seems to be driven by much higher cost applications than water treatment - aerospace, chemical manufacture, power generation, automotive design etc., and the pricing appears to reflect this kind of market. Many of the models, for instance combustion or solidification, that are available in the software are not relevant to water treatment - it could be that more limited package could be marketed to the water and wastewater industry.

The wide range of problems which might be tackled with CFD makes it very difficult to make any general statement about the cost of undertaking CFD modelling. However, some idea can be obtained by considering a relatively straightforward investigation, such as the Hazelmere clarifier (chapter 5). The time involved was about 40 h (excluding report-writing), so the personnel cost would be of the order of R6 000 (2003 rand values). The software licence cost for commercial use of the Fluent software was about R16 000 per month. Since the minimum period made available by Fluent was 1 month, it depended whether other jobs were available to share the cost, so the software cost would be between R4 000 and R16 000, and the overall cost would be between R10 000 and R22 000, to which would be added the costs for computer time: if done with a suitable personal computer (1GHz pentium with 500Mb RAM) this would amount to about R100.

This cost might be considered reasonable for a large clarifier, but appear excessive for a small unit. The problem is that the cost would probably be about the same irrespective of the size of the installation. For small units the costs could be effectively reduced by developing standard designs which could be reused a number of times. The extra design cost should be recoverable through lower operating cost, however this will probably be difficult to quantify beforehand.

4.6 Recommendations

The recommendations are divided into those that relate to the individual case studies, and the more general ones that relate to the application of CFD to water and wastewater treatment.

4.6.1 The Wiggins ozone contactor

- 1) The position of the ozone sensor monitoring the residual ozone should be moved to the position identified in the investigation.
- 2) A new strategy for the control of the ozonation should be developed that takes into account the ozone demand of the raw water and the disinfection efficiency of the contactor.
- 3) The cost-benefit balance of the ozonation in the overall water purification process is not easy to quantify; it could even be the case that the benefits do not justify the cost. The

model that has been developed for the contactor could be very useful as a component in a wider investigation of the role of ozonation in the overall treatment process.

4.6.2 The Faure filter backwash system

A backwash distributor should be installed on a trial basis on one of the filters at the Faure Water Treatment Works. If this proves as successful as predicted, the system could then be installed on the other filters at the works, and on other filters of similar design.

4.6.3 The Hazelmere clarifier

The design modifications which proved so successful should be implemented on the remaining peripherally fed clarifier at the Hazelmere Water Treatment Works when appropriate. The design should also be considered for new clarifiers of a similar size.

4.6.4 The batch settling test for sewage sludge

An experimental project should be undertaken to test the protocol suggested by the modelling results. This would involve carrying out laboratory settling tests to determine the sludge settling parameters, using these parameters in a CFD model of a full-scale clarifier, and testing the model predictions experimentally on the full-scale unit.

4.7 General recommendations

The South African water industry still needs to develop an adequate pool of CFD expertise that can be called upon when appropriate. This might involve:

- Introducing CFD in a limited way in undergraduate university courses, to promote awareness of the technique.
- Postgraduate university training to build up the CFD skills base in the country.
- Courses, symposia and workshops designed to promote awareness of the benefits of CFD among water industry managers and policy-makers.
- Centres of expertise which can undertake research or consultancy work to provide CFD support to water researchers and operators, and promote CFD by providing the academic and industrial training mentioned above.
- CFD specialists employed in the design teams of the larger water authorities, municipalities and engineering consultants.
- A way should be sought to engage the developers of CFD software in order to make appropriate CFD packages available at costs which are appropriate to the water and wastewater treatment industry.

ACKNOWLEDGEMENTS

The research in this report emanated from a project funded by the Water Research Commission and entitled :

COMPUTATIONAL FLUID DYNAMICS SUPPORT TO WATER RESEARCH PROJECTS

The authors thank the of the Steering Committee responsible for this project for its guidance. Its membership varied during its course; the following persons were members at various times:

Dr G. Offringa	Water Research Commission (Chairman)
Mr JN Bhagwan	Water Research Commission
Dr IM Msibi	Water Research Commission
Dr LJ du Preez	Water Research Commission (Secretary)
Ms SG Matthews	Water Research Commission (Secretary)
Mr JF Taljaard	Water Research Commission (Secretary)
Prof GA Ekama	University of Cape Town
Mr SA Pieterse	City of Cape Town
Mr P Thompson	Umgeni Water
Prof Haarhoff	Rand Afrikaans University
Prof CG du Toit	University of Potchefstroom
Mr MJ Pryor	Umgeni Water
Prof EP Jacobs	University of Stellenbosch
Prof M Starzak	University of Natal
Prof O Onyejekwe	University of Durban-Westville
Mr CF Pretorius	Wates, Meiring and Barnard
Mr DJ Nozaic	Umgeni Water
Mr SA van der Merwe	Tshwane Metropolitan Municipality

The financing of the project by the Water Research Commission and the contribution of the members of the Steering Committee is acknowledged gratefully.

This project was only possible with the co-operation of several individuals and institutions. The authors therefore wish to record their sincere thanks to the following :

Mr R Rajagopaul	Umgeni Water
Mr D Naidoo	Umgeni Water
Mr T Naidu	Umgeni Water
Mr M Docrat	Umgeni Water

TABLE OF CONTENTS

1	INTRODUCTION	1
1.1	Project Objectives	2
1.2	Overall course of the project	
2	THE USE OF CFD IN WATER TREATMENT	4
2.1	Some recent references involving CFD in water treatment	4
2.1.1	CFD in research	
2.1.2	CFD in the water and wastewater industry	7
2.1.3	CFD in environmental studies	8
2.2	Conclusions	8
3	THE OZONE CONTACTOR AT THE WIGGINS WATER TREATMENT WORKS	10
3.1	The pre-ozonation system at the Wiggins Water Treatment Work	10
3.2	The overall course of the investigation	12
3.3	The initial hydraulic model	13
3.3.1	Geometry and meshing	13
3.3.2	Physical Models, Boundary Conditions and Input Parameters	13
3.3.3	Experimental tracer tests	13
3.3.4	Simulated tracer tests	15
3.3.5	Conclusions from the initial model and tracer tests	17
3.4	Reaction kinetics modelling	18
3.4.1	Ozone reactions	18
3.4.2	Reaction simulations	19
3.5	Monitoring the ozone contactor	20
3.5.1	Equipment	20
3.6	Comparison of experimental results with CFD model	21
3.7	Conclusions from the monitoring program	23
3.8	Disinfection modelling	23
3.9	Gas phase modelling	25
3.9.1	2-dimensional 2-phase modelling	26
3.9.1.1	Boundary conditions and other model parameters	26
3.9.1.2	Model results	27

3.9.1.3	Conclusions from the 2-D model	28
3.9.2	3-dimensional 2-phase modelling	28
3.10	Conclusions	28
3.10.1	Conclusions regarding the ozone contactor	29
3.10.2	General conclusions regarding CFD modelling	29
3.11	Acknowledgements	30
4	THE SAND FILTER BACKWASH SYSTEM AT THE FAURE WATER TREATMENT WORKS	31
4.1	The existing filter backwash system	31
4.1.1	Model geometry	31
4.1.2	Model results	31
4.2	The model of a proposed modification	34
4.2.1	Model results	34
4.2.2	Hydraulic design of the proposed distributor	34
4.3	Acknowledgement	36
5	THE CLARIFIER UPGRADE AT THE HAZELMERE WATER TREATMENT WORKS	37
5.1	Background	37
5.2	Design of the clarifier modifications	38
5.3	The CFD model	39
5.3.1	Model basis and assumptions	39
5.3.2	Modelling results	40
5.4	Evaluation tests	42
5.5	Conclusion	72
6	BATCH SETTLING OF SECONDARY SEWAGE SLUDGE	44
6.1	Modelling procedure	46
6.2	Results	46
6.3	Discussion	47
6.4	Conclusion	48

7	CONCLUSIONS AND RECOMMENDATIONS	49
7.1	The scope for application of CFD modelling in water treatment	49
7.2	The costs involved in CFD modelling	50
7.3	Recommendations	51
9	REFERENCES	52

LIST OF FIGURES

3	THE OZONE CONTACTOR AT THE WIGGINS WATER TREATMENT WORKS	10
3.1	View of the ozone contactor, showing the static mixer at the inlet.	10
3.2	Geometry of the ozone contactor.	11
3.3	Experimental tracer response curves at various points along the outlet weir without gas injection	14
3.4	Experimental tracer response curves at various points along the outlet weir without gas injection.	14
3.5	Simulated tracer response for 10% turbulence intensity ratio	15
3.6	Simulated tracer response for 20% turbulence intensity ratio	16
3.7	Simulated tracer response for 50% turbulence intensity ratio	16
3.8	Comparison of predicted RTD density function with experimental data. The mean residence time and T10 are calculated from the experimental data with gas	17
3.9	Schematic ozone reactions	18
3.10	Sample valves and ozone sensors installed on the contactor	20
3.11	Position of sample points in the contactor	20
3.12	Comparison of measured and simulated ozone concentrations for a feed flow rate of 106 ML/d and an ozone dosage of 2.5 mg/L	21
3.13	Comparison of measured and simulated ozone concentrations for a feed flow rate of 120 ML/d and an ozone dosage of 2.5 mg/L	21
3.14	Simulated ozone concentrations on a plane in the middle compartment for a feed flow rate of 120 ML/d and an ozone dosage of 2.5 mg/L.	22
3.15	Comparison of measured and simulated ozone concentrations at sample point 4 for a flow rate of 106 ML/d and an ozone dosage stepped from 2.5 mg/L to 3.5mg/L	22
3.16	The effect of increasing the TOC on simulated ozone concentrations on a vertical plane situated 2m from the right hand side of the contactor. Flow: 106 ML/d., ozone dose: 2.5 mg/L	23
3.17	The effect of increasing the TOC on simulated ozone concentrations on a horizontal plane through the top compartment. Flow: 106 ML/d., ozone dose: 2.5 mg/L	24
3.18	The effect of increasing the TOC on simulated cryptosporidium survival on a vertical plane situated 2m from the right hand side of the contactor. Flow: 106 ML/d, ozone dose: 2.5mg/L	25

3.19	The effect of increasing the TOC on the overall simulated residual ozone and cryptosporidium survival in the contactor effluent. Flow: 106 ML/d, ozone dose: 2.5mg/L.	25
3.20	Geometry of the 2-D ozone contactor model	26
3.21	Contours of gas volume fraction for the 2-D model	27
3.22	Simulated tracer responses for the 2-D model	28
4	THE SAND FILTER BACKWASH SYSTEM AT THE FAURE WATER TREATMENT WORKS	31
4.1	Schematic diagram of porous floor model.	31
4.2	Geometry of the CFD model of the filter underdrain	32
4.3	Simulated backwash rates in the seven filter sections	32
4.4	Contours of pressure at the mid plane surface (side view)	33
4.5	Contours of upflow velocity at the nozzle slab surfaces (plan view)	33
4.6	Flow pattern in filter under drain (plan view)	33
4.7	Geometry of model simulating effect of a flow distribution pipe.	34
4.8	Geometry of model of the flow distributor.	35
4.9	Simulated distributor pressure profile.	35
5	THE CLARIFIER UPGRADE AT THE HAZELMERE WATER TREATMENT WORKS	37
5.1	View of the central part of the clarifier before the upgrade.	37
5.2	View of the peripheral feed inlet to the clarifier.	38
5.3	(a) Streamlines and (b) contours of sludge concentration (initial model).	40
5.4	(a) Streamlines and (b) contours of sludge concentration (final model)	41
5.5	Flow rate through each clarifier during the comparative test	42
5.6	Turbidities measured at each clarifier during the comparative test	43
6	BATCH SETTLING OF SECONDARY SEWAGE SLUDGE ..	44
6.1	Sludge settling rates according to the Takács model.	45
6.2	Simulated concentration profiles in a batch settling test for C_I higher than C_M .	47
6.3	Simulated concentration profiles in a batch settling test for C_I lower than C_M .	47

1 INTRODUCTION

Almost all water and wastewater treatment equipment rely on continuous through flow of water. Some equipment requires this flow to be well-mixed, whereas other equipment requires plug-flow. Examples of well-mixed systems are activated sludge plants, chemical dosing zones and anaerobic digesters while sand filters (in both filtration and back-wash modes), clarifiers, adsorption columns (ozone, activated carbon and ion exchange) and dissolved air flotation cells are examples of the plug-flow systems. Some processes such as nutrient removal activated sludge plants require the combination of both plug flow and completely mixed reaction zones.

The laboratory-scale experiments that are used to obtain design data for a plant are usually operated under ideal flow conditions; unfortunately it is usually not feasible to carry this through to full-scale plants, due to the greater difficulties and expense of achieving similar ideal conditions on a large scale. The complexity of the flow patterns, and the uncertainties about how they affect the relevant performance indicators for the process involved have led designers of equipment to use *safety factors* based on experience to ensure that the process achieves its required objectives. This means that equipment that is installed is often larger and more expensive than it needs to be.

Computational fluid dynamics (CFD) is a numerical procedure to calculate the properties of moving fluid. Most water treatment processes involve the movement of water. This motion is often complex and difficult or very expensive to observe. The prediction of the flow patterns and other properties of flowing fluids would provide insight into processes which otherwise would not have been possible. A previous WRC project (No 648) indicated the value of CFD modelling of clarifiers and an anaerobic compartment. It was able to both logically explain the unexpected behaviour of the clarifier and in designing features to modify the undesirable flow pattern.

Apart from its use in design of water treatment equipment, CFD modelling can also assist in research into water treatment processes. The project on which this report was based was unusual in that it was initiated to provide a CFD modelling service to assist water researchers who felt that it could enhance their investigations. As a result the project did not have a specific research focus of its own, but adapted to the objectives of each research project that it became involved with. Furthermore, not all the collaborations that were started were fruitful. The main content of the report is a series of case studies, each corresponding to a different investigation. To give the report some thematic consistency, it has been compiled with a view to illustrating the kinds of situations where CFD modelling is useful. To this end, the case studies that were selected for inclusion in the report are those which best fitted this purpose, i.e. they each involved an appropriate application of CFD, and yielded some useful conclusions. Various investigations which for one reason or another did not fit these criteria have been left out. The investigations which are presented have also been *cleaned up* to reflect a logical development which did not always take place in reality; i.e. the various misapplications, misconceptions and dead ends which occurred along the way have been removed from the narratives.

1.1 Project Objectives

- To provide a service to water researchers by undertaking modelling exercises on proposed and existing equipment so that they may design more efficient experimental equipment and to better understand their experimental results.

- To promote the use of computational fluid mechanics by water authorities, consultants and water researchers.
- To model wastewater treatment secondary settling tanks.
- To assist in the training of academics and students in the practical use of computational fluid mechanics.

1.2 Overall course of the project

At the inaugural meeting in 1999, a list of water-related projects with potential for CFD input was put forward. Seven activities were originally placed in the work programme for 1999, which would have involved collaborations with researchers from around South Africa. These were:

- | | |
|-----------------------------------|------------------------------|
| • Ozone Contactor | • Filter back washing |
| • Anaerobic baffled reactor (ABR) | • Capillary membrane modules |
| • Secondary settling tanks | • Ultrafiltration manifolds |
| • Passive mine water treatment | |

As was anticipated, not all evolved into fruitful collaborations; the direction and extent of progress depended on unpredictable factors which emerged. During the course of the project, reasons for making progress, or failing to do so, largely revolved around personnel capacity to undertake the work, though the logistics of collaborating with remote groups also played a part in some cases. No postgraduate students were directly associated with the project during 2000, although some work was done by undergraduate students taking vocational employment.

Between December 2000 and January 2001 modelling of the secondary settling tanks at Durban's Northern Wastewater Treatment Works was carried out to see if it was possible to achieve a significant increase in capacity without extensive redesign. Unfortunately this exercise was inconclusive. Promising results which were initially obtained later proved to be a result of incorrect feed flow rates. When these were corrected, the improvements that could be achieved using simple baffles were found to be inadequate for the requirements of the treatment works. In view of the lack of a conclusive outcome, this investigation has not been included in the report.

At the beginning of 2001 Tzu-Hua (Jenny) Huang, a chemical engineering graduate of the University of Natal, enrolled for a MScEng, taking the study of the ozone contactor as her thesis topic. Chapter 3 of this report is based on her work. Additionally, Emilie Pastre, a chemical engineering graduate from ENSIGC in Toulouse, undertook her MScEng studies in Durban, in terms of an exchange agreement between ENSIGC and the University of Natal. The topic that she chose involved modelling the final product water reservoirs at the Wiggins Water Treatment Plant, in order to develop an effective control strategy for chlorine dosing. The primary focus of her work was the control, with the CFD model contributing only to the conceptual development of a control model. This work will be reported in another WRC report, so has not been included here. During 2001 the pilot plant Anaerobic Baffled Reactor (ABR) was installed at the Umbilo Wastewater Treatment Works and was operated to gather data for the design of a system for treating wastewater from informal settlements. The CFD support that was provided to this investigation was never a central issue; rather the CFD modelling was used to help deciding on structural details such as the shape and placement of baffles, by allowing visualisation of the consequences of different options. Operating experience led to the conclusion that the concerns which motivated the CFD modelling were not of crucial importance, and that the CFD work had not made a significant

contribution to the investigation. This work has also been omitted from this report as it will be part of the ABR report.

In July of 2001 Ms. Huang visited Institut National des Sciences et Appliquée (INSA) de Toulouse, for an month. There, with the help of Prof. A. Liné, she undertook some two phase modelling of the ozone contactor. The investigation formed the basis of a paper presented at the *IWA Conference on Water and Wastewater Management for Developing Countries*, Kuala Lumpur, Malaysia, 29-31 October 2001, which was subsequently accepted for publication in *Water Science and Technology*. Ms. Huang subsequently applied successfully to have her MScEng registration changed to PhD.

Also during 2001, an investigation into the backwashing of sand filters at the Faure Water Treatment Plant operated by the Cape Metropolitan Council. The motivation for the investigation was that the efficiency of back washing the sand filters was not uniform over their whole area, and that excessive quantities of backwash water were required to get some areas of the filter clean. A number of other treatment works under the control of Cape Metro have filters of similar design, with similar problems, so a solution to the problem would be widely applicable. This study is the basis of chapter 4.

At the time of the steering committee meeting in 2002, it was considered to be the penultimate meeting, however later in 2002 an extension to the project was requested and granted, primarily in order to complete the study on the ozone contactor at the Wiggins Water Treatment Plant. A series of studies was carried out to monitor the contactor at the Wiggins Waterworks, to obtain data to be used in partial validation of the computational fluid dynamic model.

The studies reported in chapters 5 and 6 resulted from suggestions made by members of the steering committee. During 2002 CFD modelling was undertaken to support the design of modifications to a potable water clarifier at the Hazelmere Water Treatment Works north of Durban, operated by Umgeni Water. This was the same clarifier that had been reported in WRC Report No 648/1/02 *The Application of Computation Fluid Dynamics to Water and Wastewater Treatment Plants*, and the upgrade was a direct sequel to the previous investigation. The final investigation was prompted by a suggestion that a CFD analysis of batch settling tests might be useful for developing a strategy to advance the CFD modelling of secondary settling tanks.

2 THE USE OF CFD IN WATER TREATMENT

This review does not consider the basis and development of CFD modelling, which is can be found in a number of textbooks, for instance Anderson (1995) and Shaw (1992).

The term *computational fluid dynamics* is, strictly speaking, wider than what is commonly understood as *CFD*. Various specialised models, such as those simulating piping networks, could properly be described as computational fluid dynamics, but fall outside the scope of CFD as considered in the context of this report. Here we will consider CFD to apply to models which represent fluid systems by sets of coupled partial differential equations representing the fundamental mass, momentum and energy balances in the space-time continuum.

In an article promoting the use of CFD in the water industry Shilton, Glynn and Phelps (1999) remarked that “the reason that CFD might seem too technically complex is perhaps that we perceive the water industry as too *low tech*.” The remark was offered as a possible explanation of why CFD had found so little use in the water industry, whereas it had become an essential design tool in several other industries. The relatively high cost of CFD software and the computing power required to run it also contributed to the negative perceptions. However it was pointed out that these costs had fallen dramatically, (a trend which has subsequently continued) and that the detailed insight provided by CFD simulation could greatly improve the design and performance of apparently *low tech* systems, such as waste stabilisation ponds for example.

To an extent, the subsequent literature bears out their prediction that “we shall be seeing much more of CFD in the future”. The current situation is that the use of CFD in water treatment is still relatively limited. To illustrate this, in a search conducted through the *ISI Web of Knowledge* internet site (<http://access.isiprducts.com>), the keyword *CFD* found 2196 references in the 12 months preceding September 2003, but *CFD water treatment* found only 8, and of these only 3 referred to water treatment in the sense understood here. If the use of CFD in water treatment is growing at all, it is doing so very slowly.

2.1 Some recent references involving CFD in water treatment

An exhaustive review of the literature is not attempted here; what follows is a selection of cases which illustrate the range of applications that have been undertaken.

2.1.1 CFD in research

(Do-Quang *et al.*, 1999) Several CFD applications in the field of water or wastewater treatment was presented to highlight the feasibility and provisional limitations of CFD. A single-phase reactor example of a chlorine contactor for drinking water treatment was motivated by bad hydrodynamic performance such as short-circuiting or dead-zones formation arising often in such large storage facilities. In the example, baffles designed with the CFD model allowed a 50% reduction in chlorine consumption with improved disinfection.

Three multi-phase contactors were then discussed: bubble ozone contactor, for drinking water treatment, a circulating bed floating bio-reactor and activated sludge aeration tank for wastewater treatment. Different arrangements of the internal baffles and the bubble diffusers in a multichamber ozonation contactor were studied in order to find the optimum configuration for achieving the highest water disinfection level. Numerical tracer studies were performed in order to obtain information about the residence time distribution and

quantify the degree of dispersion in the system. Experimental tracer study performed with LiCl on the full-scale reactor was used to validate the CFD prediction.

A new type of mobile bed reactor was presented, namely the circulating bed biological reactor (air-lift). Since biological efficiency depends to a great extent on the hydrodynamic parameters, the study focused on the hydrodynamics modelling of a two-phase air-lift internal loop reactor. In the numerical simulations, different kinds of modelling of local physical phenomena were tested. The numerical results have been compared to experimental results.

In oxidation ditches mainly used for waste water treatment, the number, the type and the spatial distribution of aeration systems is a complex problem. The interaction between the liquid circulation created by mechanical agitation and the hydrodynamics induced by air injection at the bottom of the tank was analysed. From these simulations it was possible to estimate the global interfacial volumetric area for the different cases. The results explain the fact that the oxygen transfer was 30 to 40% more efficient in the aeration tanks equipped with horizontal flux agitators.

(Haarhoff and Van der Walt, 2001) Hydraulic flocculators are principally characterised by their volume (which determines the time of flocculation) and the water level difference between inlet and outlet (which determines the average energy dissipation). Within these constraints, however, the designer has many degrees of freedom, such as the average water depth, the number and spacing of baffles, the length of the gap at the baffle ends, and the degree to which adjoining baffles overlap. In this paper, these ratios are further investigated by CFD to find their optimal values. The validity of CFD modelling is first verified by comparing experimentally measured velocities in two flocculators against modelled velocities for similar hydraulic conditions. CFD is then used to systematically optimise the three critical design ratios for flocculator design.

(Jones, Sotiropoulos and Amirtharajah, 2002) To improve the understanding of how static mixers work and how to better use them in drinking water treatment, a numerical model for simulating turbulent flows in helical static mixers was developed. Numerical simulations were carried out for a two-element helical static mixer, and the computed results were analysed to elucidate the complex, three-dimensional features of the flow. As an example of the kind of practical insights that can be gained from such detailed three-dimensional computations, the simulated flow field was used to investigate two quantities that are often used to characterise mixing within a static mixer, the G-value and the turbulent energy dissipation, and to discuss the merits of these quantities for coagulant mixing in drinking water treatment.

(Sherwin and Ta, 2002) Flow characteristics of the anaerobic zone of an activated sludge plant were investigated using CFD. The tank contained an baffle to reduce short-circuiting between the inlet and outlet, and two impellers to prevent solid settlement. From a reaction point of view, the ideal residence time distribution (RTD) for the system would correspond to plug flow, however the actual tank arrangement corresponded to mixed flow with short circuiting. This was confirmed by both the CFD model and experimental tracer testing. The CFD model was used to explore various options with respect to the location of the inlet and outlet and the placement of additional baffles in the tank to improve the RTD by reducing the degree of short-circuiting.

(Park, Park and Kim, 2003) In order to investigate the effect of mixer shape and mixing intensity on hydraulic turbulence and velocity field in a rapid mixer and relate the results of the investigation to the performance of the, rapid mixer with respect to coagulant dispersion and turbidity removal, this study conducted wet tests, CFD simulation and Particle Image Velocimetry (PIV) analysis, using three different shapes of jar: a circular jar with squared

baffles, a circular jar without baffles and a Hudson jar. From the results of the wet tests, it was observed that the performance of rapid mixing in the circular jar without baffles was better than in the other shapes of jar. Also, the shape of jar is found to be a factor affecting the performance of the rapid mixer and ultimately the efficiency of coagulation. The results of CFD simulation and PIV analysis confirmed this by showing that, since it forms moderate turbulence throughout the jar and minimises localised dead zones, the circular jar without baffles produced the best mixing conditions among the jars. From all these results, this study concludes that turbulent fluid conditions in a rapid mixer, including distribution of turbulence and formation of dead zones, are important factors in determining performance of the rapid mixer. Furthermore, it is suggested that mixing intensity and mixer shape are determined considering those fluid conditions.

(Ta, Beckley and Eades, 2001) A Eulerian-Eulerian multiphase CFD model of a Dissolved Air Flotation (DAF) process is presented. A 3D structure grid is used to incorporate the air nozzle and tank geometry. The floc particle is then introduced and is tracked in the air/water fluid using a dispersed Lagrangean model. The fate of these flocs depends on their sizes and density. Flocs therefore can either escape through the top water surface, settles in the main tank or breakthrough under the outlet weir. The CFD model is developed for a full scale DAF tank to predict the flow dynamics, particle removal and settled solid profile. The general flow pattern was compared with Row visualisation using a underwater camera. Comparison of average fluid velocities was carried out using Acoustic Doppler Velocimetry (ADV) measurements.

(Baléo, Humeau and Le Cloirec 2001) This study used CFD to characterise a laboratory scale apparatus intended for obtaining design data for a wastewater treatment pond. The study focussed on the RTD and related it to details of the flow distribution. Tracer tests with lithium chloride were used to validate the model.

(Greene, Haas and Farouk, 2002) A CFD model was used to predict flow structure, mass transport and chlorine decay in a continuous flow pilot scale reactor. These predictions were compared with experimental measurements for model validation. The study demonstrated that inlet configurations can significantly impact reactor hydrodynamics.

(Salter, Ta, Williams, 2000) A series of facultative lagoons operated by Thames Water treating industrial wastewater in Thailand were found to be performing poorly, particularly with respect to the removal of biological oxygen demand (BOD). A review of the design parameters for the site found that all the lagoons are of a sufficient area for the flow and BOD load. However, observations of the lagoons suggested that there may be significant hydraulic short-circuiting. Computational fluid dynamics (CFD) modelling was therefore carried out on one of the lagoons to establish the hydraulic regime. Two consecutive simulations were carried out, both with and without baffles; the first to establish steady flow conditions, and the second using a chemical species transport model to obtain the residence time distribution (RTD). The results of the modelling indicate that the lagoons do currently suffer from significant short-circuiting, and large dead-zones are present. The installation of baffles in the CFD model improved the plug-flow characteristics of the lagoons, substantially reducing the short-circuiting and the size of the dead-zones. It has therefore been concluded that the installation of baffles in the lagoons will lead to an improvement in their performance, by increasing the retention time of the system.

(de Kretser, Matthews and Williams, 2003) The behaviour of water flow in an Australian aerated lagoon was assessed with CFD. The researchers calculated the detailed water velocity distributions to determine if mixers could replace aerators as a way to increase the degree of mixing and reduce the settling of solids. The natural flow was found to short-circuit mainly along the dividing wall, in keeping with the results of physical experiments carried out

previously with surface flotation devices. The mixing characteristics of individual aerators and mixers were examined, and the flow patterns under the influence of both aerators and mixers were determined for various parts of the lagoon. The CFD results suggest that the mixing characteristics of the aerators and mixers are complementary and that an alternate placement of each would produce adequate vertical mixing between the surface and floor of the lagoon.

(Kamimura, Furukawa and Hirotsuji, 2002) A CFD simulator for an ozone/UV reactor where ozone dissolved water flows under the irradiation of UV, has been developed by combining a fluid dynamics model with a complex radical reaction model. The radical reaction model used in this simulator with the simulation of a completely stirred tank reactor (CSTR) system was in good agreement with the experimental results, in terms of the concentrations of total organic carbon (TOC), hydrogen peroxide and dissolved ozone obtained from a lab-scale CSTR. Using this CFD simulator, the distributions of substances such as hydroxyl radical (OH.) and hydrogen peroxide in the ozone/UV reactor were investigated.

Wilderer *et al.*, (2002), in a review of new developments in wastewater science and technology, highlighted the numerical simulation of fluid flow, nutrient transport and biomass growth as a significant new tool to augment traditional experimental techniques for the investigation of microscopic biofilms.

Van der Walt (2002) undertook CFD modelling of process tanks for potable water treatment, in order to gain new insight into their hydraulics and to develop design guidelines and performance indicators that can assist in developing more effective units.

CFD modelling of secondary settling tanks has a fairly extensive literature, some of which was reviewed in a previous WRC report (Brouckaert and Buckley, 2002), e.g. Dahl and Larsen (1994); Imam and McCorquodale (1983a, 1983b); Krebs (1995); Lyn, Stamou and Rodi, (1992); Matko *et al.*, (1996); McCorquodale and Zhou, (1993); Szalai, Krebs and Rodi, (1994); Zhou and McCorquodale, (1992a, 1992b); Zhou, McCorquodale and Vitasovic (1992). In those papers the modelling dealt with flow patterns and particle settling in the settling tanks. Recently researchers have started to consider additional phenomena taking place in such units:

(Brannock *et al.*, 2002) A model of the anoxic denitrification zone of a wastewater treatment reactor was developed which incorporated both biological reactions and particulate settling coupled to the hydrodynamic model. Particle settling was represented by the exponential Vesilind model, and the biological reactions by a simplified version of the Activated Sludge Model No 1 (ASM1). Experimental tracer tests were used to verify the hydrodynamic part of the model. Brannock (2003) presented a more complete account of this study.

(de Clercq, 2003) As part of the CFD modelling of a settling tank, sub models were implemented for solids sedimentation and sludge rheology, and for a scraper mechanism. A new rheological model was proposed which is especially applicable to low shear rates that prevail in sludge blankets.

2.1.2 CFD in the water and wastewater industry

(Lyonnaise des Eaux, 2000) The *Virtual Plant* is a computer modelling program developed by Lyonnais des Eaux that is used in the design of new water treatment processes. With Virtual Plant technology, a full scale plant operation can be reproduced on a computer so that plant designers can see how changes in water flow or water quality will affect the water treatment process. While design projects carried out using traditional methods usually require a three to six month study at very high costs, this evaluation can now be accomplished in

only a matter of days at a fraction of the cost. Overall, it is claimed that Virtual Plant in the water treatment cycle:

- replaces long and expensive studies;
- provides faster, more accurate results than existing physical models;
- guarantees maximum efficiency of the applied treatment process;
- allows for more innovative and adaptable designs.

(Craig *et al.*, 2002) The use of Computational Fluid Dynamics (CFD) software enables the dimensioning of water treatment processes by taking into account the real hydraulic behaviour of processes. That has been done for the Coliban Water Aqua 2000 project, which consists of the construction of three water treatment plants. The disinfection performance of three ozone contactors were compared using CFD. Moreover, the CFD application has been extended to a large range of water treatment processes in recent years. The paper presents several of these: flocculation tanks, UV reactors and secondary settling tanks.

(Chataigner *et al.*, 1999) FLUENT was used to evaluate the hydraulic efficiency of chlorination tanks. The aim was to provide a homogeneous flow within the tank, a measure needed to ensure good water treatment quality. The simulations demonstrated that the use of a CFD code is a practical engineering tool and led to practical recommendations for the construction of chlorination tanks.

2.1.3 CFD in environmental studies

(Cook, Orlob and Huston, 2002) The Salton Sea in California, was formed in 1904 as the result of a levee failure along the Colorado River. At present the surface elevation of the Sea stands near -69.5 m and salinity is approaching 45 g/L, about 30% above ocean salinity. Circulation in the Sea is driven primarily by wind stresses imposed on the water surface, and circulation changes are likely to affect the Sea's quality and ecology. A mathematical model for simulation of the hydrodynamic behaviour of the Sea was developed, calibrated to data gathered by a field investigation conducted in 1997, and applied to alternative schemes that will isolate sections of the southern basin, thus changing the natural wind induced circulation in areas that are ecologically sensitive. Using data derived from 1997 field measurements of velocities using acoustic Doppler current profilers and temperature sensors, the model was calibrated to reproduce mathematically the historic experience of field observation. The model successfully replicated principal features of the Sea's behaviour. Once calibrated, the model was applied to evaluate the possible effects of changing water surface elevations in the Sea and altering its configuration to isolate sections for evaporative concentration of salts.

2.2 Conclusions

The very fundamental nature of the CFD approach has the advantage of being able to represent appropriate systems in great detail with minimal requirements for empirical data, but the disadvantages of complexity and difficulty in solving the resulting systems of equations. These practical difficulties prevent CFD from being a universally appropriate approach to all problems involving fluid flow, in spite of its fundamental basis. Generally, CFD is most useful for systems which are *well-connected*, that is, where all the boundary conditions have relatively strong influences on all parts of the flow field. This applies to many systems found in water treatment, such as reservoirs, contact chambers, sedimentation basins, ponds and even lakes and lagoons. However there are also many systems in water treatment where CFD does *not* provide an effective approach, for example a set of equipment connected by a pipe network, or a long reach of a river. In such cases the CFD model would expend enormous computational effort on calculating the practically negligible effects of remote boundary conditions.

The simplest CFD models consider only the hydraulic aspects of a system. Frequently these models are used to predict the residence time distribution (RTD), which often provides a link to more direct performance indicators through empirical rules based on experience. As CFD modelling has become more established, more detailed models are appearing which attempt direct representations the physical and chemical processes taking place in the treatment processes, such as sedimentation, flocculation, inter-phase mass transfer and chemical reaction. In all cases these more complex models need to be supported by experimental studies to establish the parameters for the physical and chemical parts of the models. The CFD modelling thus has a role in both the interpretation of results from experimental apparatus, and in extrapolating research results to the design of full-scale processes.

3 THE OZONE CONTACTOR

AT THE WIGGINS WATER TREATMENT WORKS

Ozonation is used in drinking water treatment primarily to oxidise iron and manganese, to remove odour- or taste-causing compounds, and to destroy micro-organisms. The peripheral benefits include possible reduction in coagulant demand, enhancement of algae removal and the colour removal. Ozonation of water is typically carried out by dispersing gas containing ozone into the liquid phase. The contact between the two phases accompanied by an ozone mass transfer takes place in ozone contactors.

The most important hydraulic characteristic of an ozone contactor is its residence time distribution (RTD). For example, under the US EPA Interim Enhanced Surface Water Treatment Rule (IESWTR) the physical removal or the inactivation of waterborne pathogens during disinfection of drinking water is specified in terms of CT , which is the product of the residual ozone outlet concentration (C) and a characteristic contact time (T) (USEPA, 1999). The characteristic contact time is taken to be T_{10} , rather than the mean hydraulic retention time, $\bar{T}$. T_{10} is the time required for 10% of a pulse of a tracer introduced at the disinfectant dosing point to have reached the residual sampling point. Thus, unlike the mean residence time which depends only the volume of the contactor and the flow rate, T_{10} is dependent on the RTD of a contactor, which, in turn, is affected by its geometry and operating conditions. The use of T_{10} in CT consequently is an advance over the older practice of using $\bar{T}$, which took no account of the configuration of the contactor apart from its volume, however it clearly will not give a characterisation of the process that is as accurate as one which uses the whole RTD.

In contrast to a mixed-compartment modelling represented by the CT approach, CFD modelling offers the advantage of a fundamental physical basis for representing the complex interaction between flow and chemical reaction phenomena. CFD modelling is based on the solution of sets of partial differential equations, which express the local and temporal phenomena, i.e.: local momentum, energy, mass and transport balances. Because of the increasing power of computers, CFD modelling is an effective tool to analyse a complex system, such as an ozone contactor.

3.1 The pre-ozonation system at the Wiggins Water Treatment Works



Figure 3.1: View of the ozone contactor, showing the static mixer at the inlet.

The pre-ozonation system at Wiggins Waterworks, operated by Umgeni Water in Durban, consists of four contactors. Each of the contactors is preceded by a static mixer such that every contactor can operate individually or in parallel with another contactor. An ozone-oxygen gas mixture is injected as a side-stream through the static mixer which is employed to achieve high mass transfer of ozone to water. The Wiggins pre-ozonation system has an unusual configuration, as it was adapted from an existing structure which had originally been designed for a different purpose. Water enters from the static mixer at the bottom, and passes through three horizontal compartments before it exits over the weir at the top. (Figure 3.2)

The operating rule for selecting the number of contactors in use aims to achieve an acceptable efficiency of ozone transfer in the static mixers, before the water enters the contactor. If the flow is too high, the injected ozone is unable to dissolve in the available contact time in the static mixer. Therefore as the flow through the plant changes, the number of contactors in operation is changed. However, the effectiveness of disinfection or destruction of taste and odour compounds also depends on the *CT* in the contactor, which implies the need for more sophisticated criteria for deciding when a contactor should be brought into or out of operation.

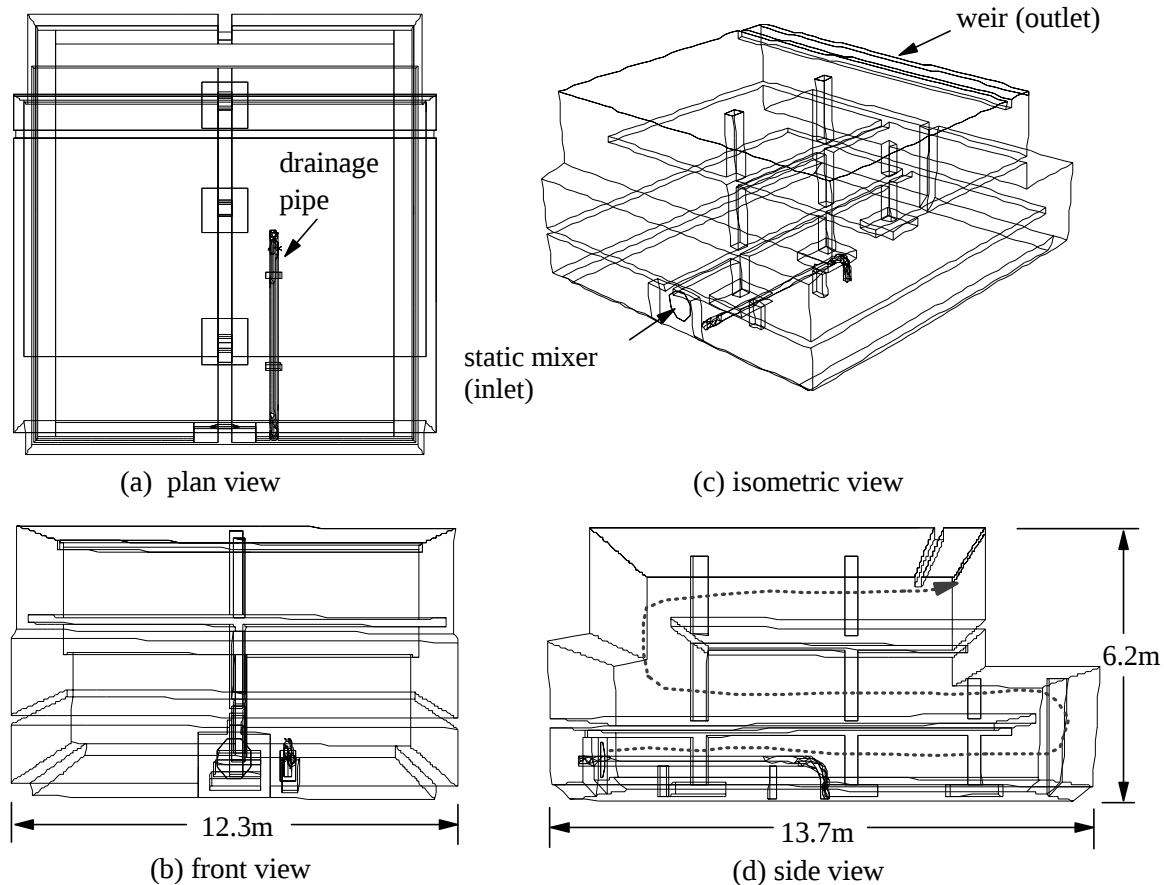


Figure 3.2: Geometry of the ozone contactor. The diagrams were generated from the CFD model, and consequently represent only surfaces in contact with water.

In order to ensure that sufficient ozone is added to the water, provision was made to measure the residual ozone concentration at the outlet from the contact chambers. Water leaves the ozone contactors over a broad-crested weir, and turbulence and de-gassing effect causes the release of residual ozone. Although this is not enough to exceed the safety exposure limit, it is a health concern as well as an operating inefficiency. A previous investigation showed that

the measurement of ozone residual at the outlet is probably not the most appropriate position. One of the operational objectives for the ozone contactors is to achieve the desired residual ozone concentration in water. It is obvious that the operating cost increases with higher dosage of ozone. Moreover, overdosing with ozone could also increase the disinfection by-products to an unacceptable level (Rakness et al., 2000).

3.2 The overall course of the investigation

The objectives of the investigation were:

- To determine the actual liquid residence time distribution as a function of flow conditions through the contactors
- To optimise the disinfection efficiency of the contactors by
 - ♦ selecting the most appropriate position for monitoring residual ozone concentration.
 - ♦ achieving the most efficient use of the ozone dosed to the system.

Of the four contactors at the works, No 3 was used for all the investigations. The contactors are not completely identical, but the differences are relatively minor.

In outline, the phases of the investigation were approximately as follows (various mistaken or misguided excursions have been excised from the sequence):

- 1) A single phase (water only) CFD model was set up to provide an initial understanding of the flow patterns in the contactor.
- 2) Tracer tests using lithium chloride were carried out to compare with the model. These were conducted with and without gas injection into the static mixer. Although the gas injection did cause some noticeable difference in the measured outlet concentrations, the effect on the overall residence time distribution was very small. After some adjustment to the model, it was concluded that a single phase (i.e. water only) model would give an adequate representation of the RTD for modelling the ozone reactions.
- 3) An ozone reaction scheme was added to the model, using kinetic data obtained from the literature. From this it was evident that the ozone consumption is very dependent on the local characteristics of the water, which need to be determined experimentally.
- 4) Sampling lines were installed on the contactor which allowed ozone concentrations to be measured at various points. The positions of these were chosen with reference to the CFD model results. Consideration of both the model results and the measurements suggested that the best point for monitoring the ozone concentration for control purposes was located between the 2nd and 3rd compartments, rather than at the outlet of the 3rd compartment as at present.
- 5) A laboratory study was initiated to obtain rate constants for the reaction scheme. At the time of writing, this was in progress.
- 6) To check the validity of neglecting the effect of gas injection, some two phase modelling (gas bubbles in liquid) was carried out. First a two dimensional model was tried, and when this proved successful, a full three dimensional model was implemented. However satisfactory results were not achieved, due to grid resolution and convergence difficulties. These might well have been resolved with more powerful computers than those available.

3.3 The initial hydraulic model

The initial single phase (i.e. water only) model was used to determine the residence time distribution of the contactor, and to help with the interpretation of tracer tests carried out on the contactor. The software used for the CFD modelling was FLUENT 5.5.14.

3.3.1 Geometry and meshing

The initial geometry of the contactor was taken directly from design drawing. The wire frame diagrams (see Figure 3.2) were created using GAMBIT 1.3, the grid generation pre-processor. They represent only the surfaces in contact with water, instead of the entire concrete structure. The upper surface represents the free surface of water, which was modelled as a rigid frictionless surface, to avoid the extra complexity of a more rigorous free surface model.

The meshing was also carried out using GAMBIT 1.3. The contactor volume was divided into 103105 tetrahedral volume elements, with an average volume of 0.0082 m³ (8.2 L) and an average side length of about 20 cm.

3.3.2 Physical Models, Boundary Conditions and Input Parameters

The initial model considered only the flow of water, neglecting the effect of inert gas injected with the ozone. The FLUENT software provides a number of established models for turbulence and boundary layer flow in the vicinity of walls. The widely used k - ϵ model was chosen for the turbulence modelling.

Before the inlet to the contactor, the water passes through a static mixer, where the gas is injected. This consists of about 10 % ozone and 90 % oxygen. The gas flow rate is about 1 % v/v of the water flow rate. By the time the flow emerges from the static mixer into the contactor, almost all the ozone has dissolved, but the oxygen remains as gas bubbles. In the model, the presence of the bubbles was not taken into account explicitly, however it was assumed that its effect on the flow could be indirectly represented by an increase in the turbulent mixing in relation to the case with water only. Because of the difficulty of predicting the intensity of turbulence at the inlet, the turbulent intensity was adjusted to give the best fit to the experimental data.

A wall roughness of 0.5 mm was used to represent the concrete surfaces. The free water surfaces were represented as rigid frictionless planes. More rigorous modelling of free surface flow would have led to a much more complex formulation, without significant benefits in terms of the objectives of the investigation. For similar reasons, the outlet weir was represented as a rectangular slot, the height of which was set to the measured height of liquid flowing over the weir. To simulate the head loss over the weir, a frictional loss coefficient was assigned to the slot and set to a value, which was high enough to ensure that the flow would be evenly distributed along the length of the weir.

Apart from the turbulent intensity at the inlet, the only other physical parameter required from the user was the inlet flow rate. The inlet flow rates during the period when the tracer tests were carried out were 107.3 ML/d and 108.3 ML/d, with and without gas respectively. These values were used in the simulations.

3.3.3 Experimental tracer tests

Two tracer tests were carried out on the ozone contactor under different operating conditions, i.e.: with and without ozone injection. The tests were carried out using lithium chloride (LiCl), which was dissolved in water and then dosed into the feed chamber of the contactor.

Approximately 7.8 kg of LiCl was used in each of the tracer tests. During both tracer tests, the water and gas were maintained at steady flow rates and were recorded at 5 min interval.

Samples were taken simultaneously at three points along the outlet weir. Since the CFD model indicated a variation in the flow between the right and the left side of the contactor, three sampling points were positioned along the weir, one on each side measured 2 m away from the edge, and the third one in the centre. Samples were subsequently analysed using an atomic absorption spectrophotometer.

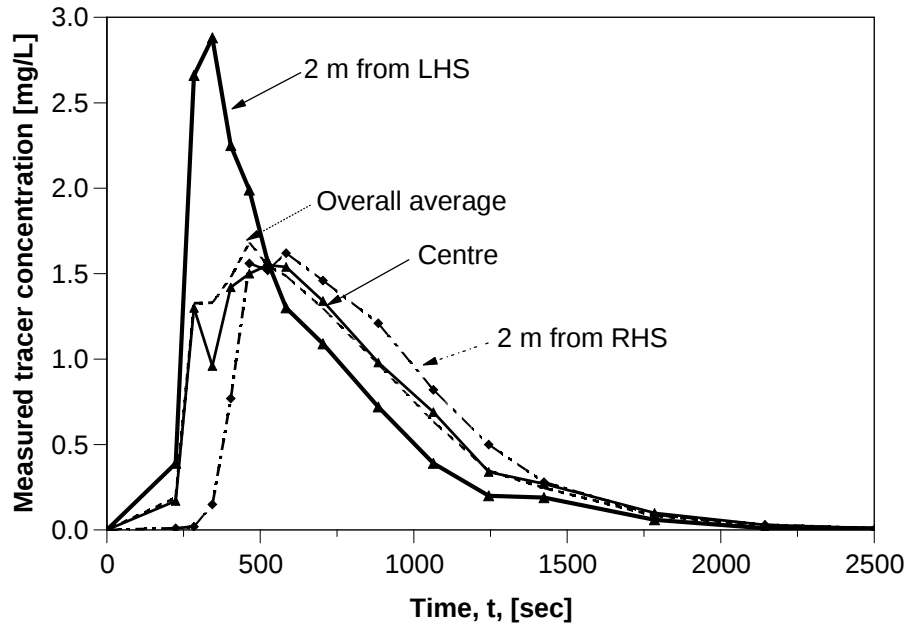


Figure 3.3: Experimental tracer response curves at various points along the outlet weir without gas injection.

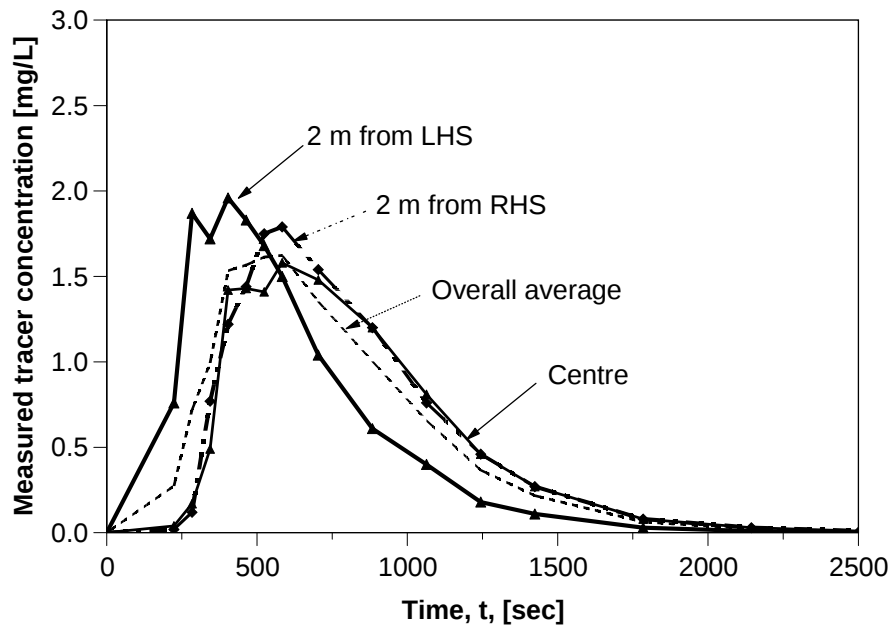


Figure 3.4: Experimental tracer response curves at various points along the outlet weir with gas injection.

The tracer response curves for the cases with and without gas injection showed a generally similar pattern, with some differences in detail. Without gas injection, the measured response appeared slightly higher on the left side and the residence time residence distribution was narrower (Fig. 3.3) than when gas was present (Figure 3.4). However the shape of the overall average response was little changed. The uneven distribution of flow, with more going to the left side, was observed with and without gas injection, but was more pronounced when the gas was absent. This observation seems consistent with the idea that gas bubbles increased the turbulent mixing in the contactor.

3.3.4 Simulated tracer tests

Since the effect of gas injection on the overall RTD was slight, and the gas injection could not be represented directly in the single phase model, it was tried to represent the effect by adjusting the turbulent intensity of the flow entering the contactor through the static mixer. The turbulent intensity of the incoming flow is a required boundary condition for the turbulence model. Even without gas injection, its value was not known, because of the effect of the static mixer blades, so it made sense to treat it as an unknown parameter that could be adjusted to match the experimental tracer response as closely as possible. FLUENT provides a number of different options for specifying the turbulent intensity boundary condition at an inlet; the one that was chosen was the *Intensity and length scale* option. The length scale was set to 0.1 m (the blade spacing) and the intensities of 10%, 20% and 50% were tried.

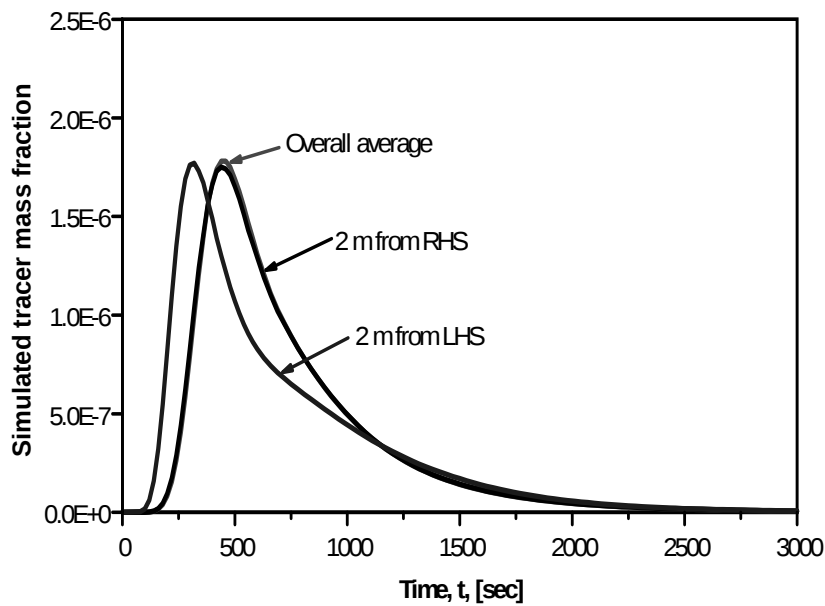


Figure 3.5: Simulated tracer response for 10% turbulence intensity ratio.

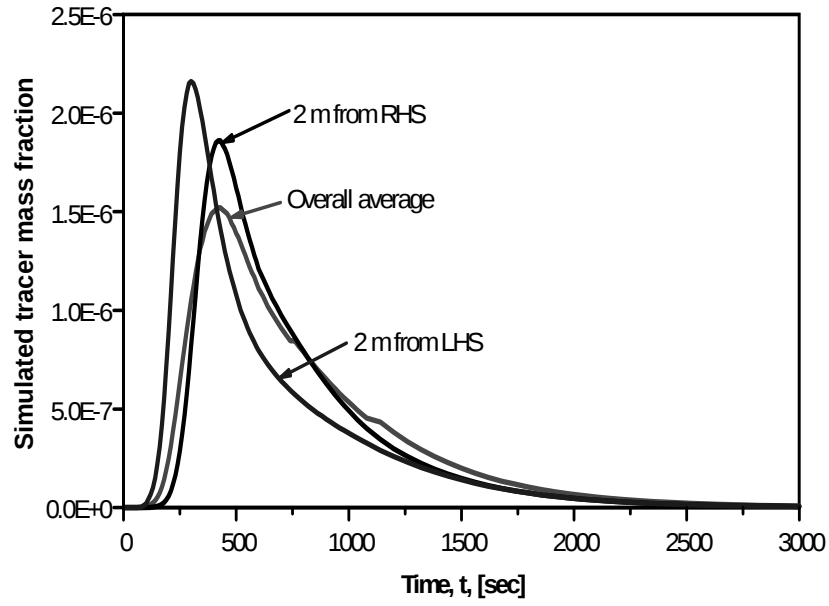


Figure 3.6: Simulated tracer response for 20% turbulence intensity ratio.

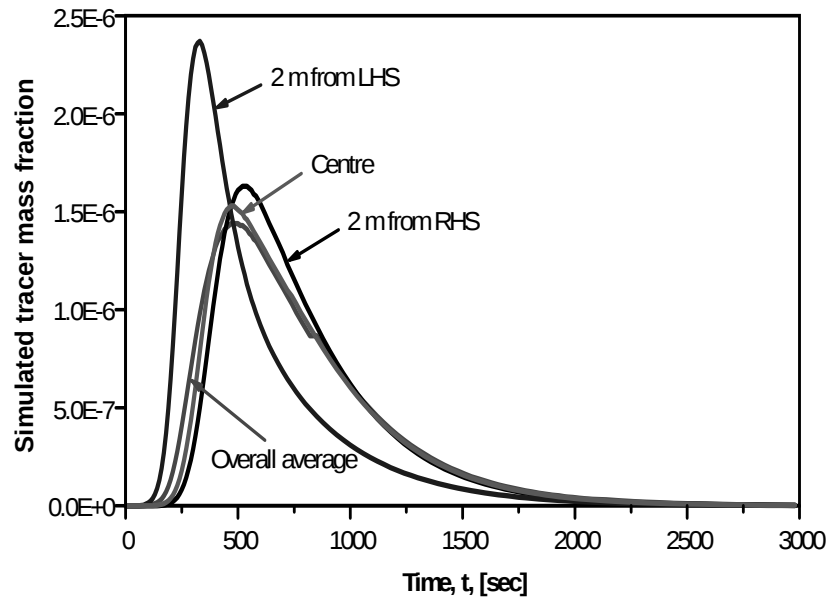


Figure 3.7: Simulated tracer response for 50% turbulence intensity ratio.

Comparing Figures 3.5 to 3.7 with Figures 3.3 and 3.4, it appears that the gas injection is better represented by a low turbulent intensity, because the difference between the simulated responses of the left hand side *increases* with turbulent intensity. However when one considered the overall average RTD curve a contrary trend was observed, as illustrated in Figure 3.8

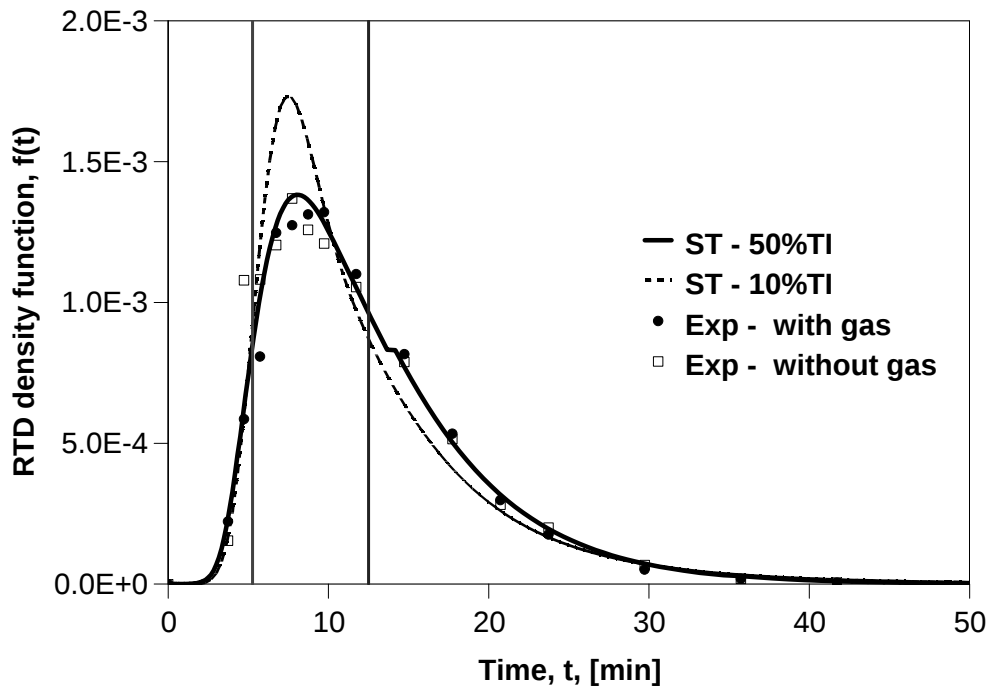


Figure 3.8: Comparison of predicted RTD density function with experimental data. The mean residence time and T_{10} are calculated from the experimental data with gas.

Figure 3.8 is plotted in terms of the RTD density function in order to achieve a comparable basis between the experimental data (tracer concentration measured in mg/L) and the simulation results (obtained as the average mass fraction of the tracer) the RTD density. The RTD density function is obtained from the concentration data (both experimental and simulated) by dividing by the mass of tracer involved, so that the area under the response curve is made the same. This makes differences or similarities in the shapes of the curves easier to judge. As can be clearly seen, the higher turbulent intensity model gives a much better fit to the experimental data than the lower intensity model. Furthermore, in terms of the overall RTD curve, very little difference is discernible between the experiments with and without gas injection.

The best match between experiment and simulation is obtained between Figures 3.3 and 3.7, i.e. between the high turbulent intensity model and the tracer response test *without* gas injection.

3.3.5 Conclusions from the initial model and tracer tests

The initial model with adjusted inlet turbulent intensity was able to represent the overall RTD of the contactor, as determined from experimental tracer tests, very accurately. It correctly predicted an asymmetry in the distribution of flow, with more flow going to the left hand side of the contactor, however the predicted effect was greater than indicated by the measurements. In the model, the overall RTD and the left-right asymmetry responded in opposite ways to the turbulent intensity of the feed, so a choice had to be made as to which to accept to represent the goodness-of-fit to the measured data. It was decided, that it would more important to match the overall RTD in the next stage of the investigation where ozone reaction kinetics were brought into the model. Furthermore, it appeared that gas injection affected the overall RTD very little, although it did affect the left-right distribution of flow.

Although the adjustment of the inlet turbulent intensity was originally an attempt to provide an approximation to the effect of gas injection, the results suggest that it was rather related to

the effect of the static mixer than the gas, because there was no discernible difference in overall RTD with and without gas. The effect of the gas was manifested in the shift in the left-right distribution of flow in the contactor. Since this was considered of lesser importance to the reaction modelling, it was decided to continue with the single phase, water only model.

3.4 Reaction kinetics modelling

CFD modelling can be used simply to determine the RTD of a system, for example, as in Sherwin and Ta (2002). In terms of the *CT* concept mentioned in the introduction to this chapter, this amounts to using the CFD purely to determine just *T*. This, however does not exploit the full potential of CFD modelling. Moreover, in the cases of the Wiggins ozone contactor, the standard *CT* would not have been appropriate, as the *C* that would standardly be used is the residual ozone concentration at the contactor outlet. While this would be suitable for a mixed reactor, the Wiggins contactors are closer to plug-flow reactors, and are operated to have the residual outlet concentration as close to zero as possible. Consequently it was necessary to undertake more detailed reaction modelling in order to have an appropriate representation of the contactor's performance.

3.4.1 Ozone reactions

Only a fraction of the ozone introduced into the contactor reacts with the target waterborne substances or pathogens. The distribution of the ozone introduced can be globally categorised into off-gas losses, consumption, and loss by self-decomposition (Bredtmann, 1982).

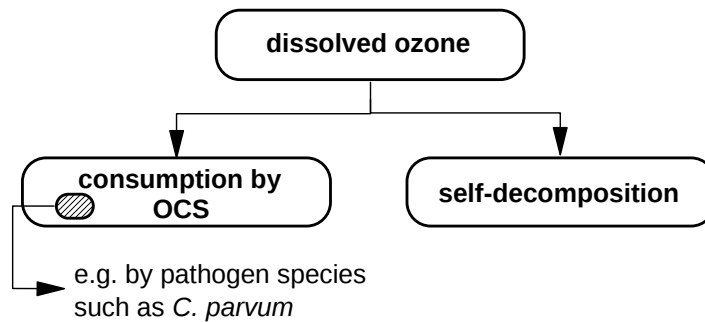


Figure 3.9: Schematic ozone reactions

Ozone reactions in water are generally rather complex. The depletion of the dissolved ozone was simplified for the model as shown in Figure 3.9. The ozone-consuming substances (OCS) in raw water consist of oxidisable species such as dissolved organics, or reduced inorganic species such as Fe (II) and Mn (II). Although pathogens form part of the OCS, they consume a tiny amount of ozone due to their extremely low concentrations. The disinfection reaction is therefore modelled as a separate reaction.

The overall decrease in dissolved ozone concentration was modelled as:

$$\frac{\partial[O_3]}{\partial t} = -k_s[O_3] - k_r[O_3][OCS] \quad [3.1]$$

The first term on the right hand side of the equation describes the ozone self-decomposition reaction, the second the reaction with OCS. Values for the kinetic constants k_s and k_r were obtained from the literature.

Self decomposition

The self-decomposition of ozone in water has been studied by several authors as a first-order reaction (Beltrán et al. 1995, Muroyama et al. 1999). The reaction rate was observed to increase with increasing pH and dissolved ozone concentration. The self-decomposition rate

constant, k_s was examined by Beltrán et al. (1995) at a wide range of pH values. For the present model, the value at pH 7 was used.

$$k_s = 4.8 \times 10^{-4} \text{ s}^{-1}$$

Reaction with ozone consuming substances

The exact nature of OCS in natural waters is variable and cannot be determined. The dissolved OCS may react with ozone or be oxidised by $\text{OH}\cdot$ radicals which are formed from ozone decomposition (Hoigné and Bader, 1983a,b). However, no general correlation is currently available to relate the ozone consumption with the decomposition of OCS. Muroyama (1999) assumed the reaction between ozone and the OCS is second order with respect to ozone and OCS. The ozone consumption reaction rate constant, k_r was inferred from experimental results to be:

$$k_r = 21.33 \text{ m}^3/(\text{kmol.s})$$

The amount of OCS present in the water was estimated from the total organic carbon (TOC) content in water, using glucose as a representative molecule. A typical value of TOC in the Wiggins raw water is 3.3 mg/L, which yields 8.26 mg/L OCS.

Disinfection

The reaction kinetics between *Cryptosporidium parvum* and ozone were based on the integrated form of equation 3.3 (the classical Chick-Watson inactivation model) :

$$\ln \frac{N}{N_0} = -k_\mu [\text{O}_3] t \quad [3.3]$$

Here $\frac{N}{N_0}$ is the micro-organism survival ratio, $[\text{O}_3]$ is the ozone concentration maintained constant for the contact time t and k_μ is the inactivation rate constant.

The value of k_μ appears to be influenced by the quality of water, such as pH and temperature. This is reflected by the wide range of k_μ reported in the literature (Joret et al. 1997, Gyürék et al. 1999, Do-Quang et al. 2000, Rennecker et al. 2000, Li et al. 2001). For this study, the disinfection rate constant, k_μ was calculated from the experimental results reported by Li et al. (2001) using an average residual ozone concentration 0.85 mg/L and a contact time of 4 min, with an observed kill of 1.5 log-units. From equation 3.3 this gave:

$$k_\mu = 352.9 \text{ m}^3/(\text{kmol.s})$$

3.4.2 Reaction simulations

The Fluent reaction modelling framework requires the reactant concentrations to be specified as mass fractions. In the case of pathogens, the mass fraction is neither known nor relevant. The measure of interest is the survival ratio $\frac{N}{N_0}$, representing the probability that a viable organism will survive the process. To represent this in the model, the inlet mass fraction was set to an arbitrary value of 10^{-8} , and survival ratios were calculated by ratioing simulated mass fractions to this value. The value 10^{-8} was simply chosen to be low enough for its effect on the ozone consumption to be negligible.

The initial simulation corresponded to the typical Wiggins raw water value 3.3 mg/L TOC (8.26 mg/L OCS). The main input parameters are summarised in Table 3.1.

Table 3.1: Input values for kinetic modelling of ozone reactions

Species	Mass Fraction
Ozone	2.5×10^{-6}
OCS	8.26×10^{-6}
<i>Cryptosporidium parvum</i>	1.0×10^{-8}

3.5 Monitoring the ozone contactor

A programme of measurements was undertaken to obtain ozone concentration data for verification and calibration of the model.

3.5.1 Equipment

Three residual ozone sensors (Orbisphere) were installed on the contactor. Figure 3.10 shows the six sampling lines with sample point selection valves, and the three sensors connected in series. The positions in the contactor from which the samples were drawn are indicated in Figure 3.11. Water from a sample line was pumped through all three sensors to increase the confidence in the measurements, and to help detect instrument malfunctions. Only one sample point could be analysed at a time.

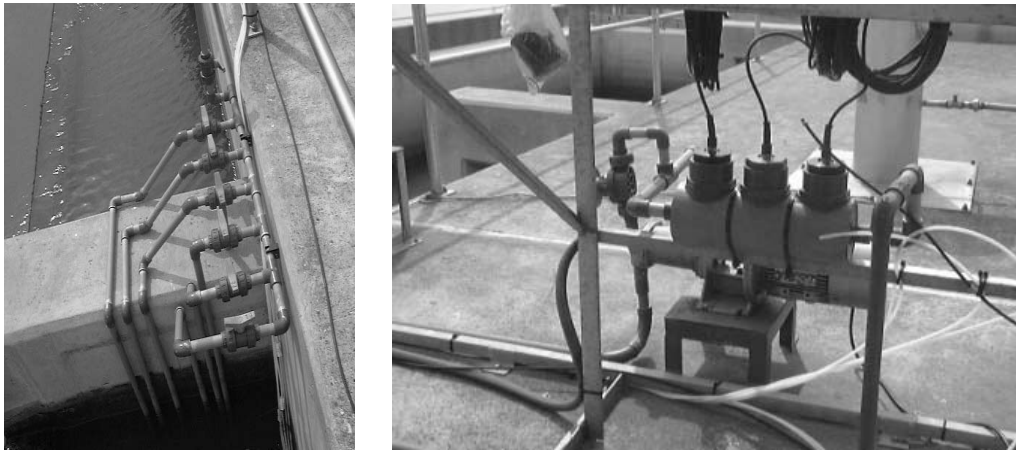


Figure 3.10: Sample valves and ozone sensors installed on the contactor

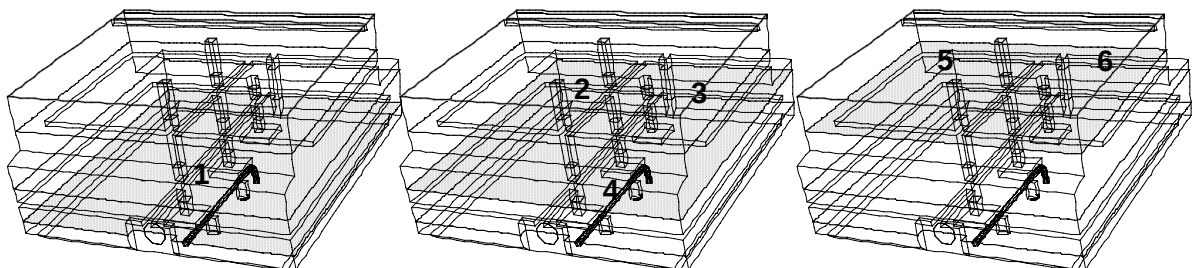


Figure 3.11: Position of sample points in the contactor. Point 1 was located in the bottom compartment; 2,3 and 4 in the middle compartment; 5 and 6 in the top compartment.

3.6 Comparison of experimental results with CFD model

Figure 3.12 compares the measured ozone concentrations with concentrations predicted by the model using all the conditions and parameter values presented sections 3.5 and 3.7, apart from k_r , for which results for 2 values are shown. The simulated ozone concentrations using the literature value for k_r of $21.33 \text{ m}^3/(\text{kmol.s})$ are much too high. Since it was clear from the literature that the value k_r of is very dependent on the local water, it was adjusted to give a better fit to the measurements. Figure 3.13 shows the comparison for a test at a higher flow rate, using the adjusted value of $58 \text{ m}^3/(\text{kmol.s})$ for k_r .

A curious feature of figures 3.12 and 3.13 is the relative positioning of sample points 4, 5 and 6. At the lower flow rate, the measured concentration for position 4 is lower than for 5 and 6, however this relationship is not reflected by the simulation. At the higher flow rate the relationship between the measured and the simulation is reversed. A possible explanation for this can be seen in figure 3.14, which shows the simulated ozone concentration field on a plane in the middle compartment of the contactor, where sample point 4 is located. It can be seen that the sample point is located in an area of sharp ozone concentration gradient, brought about by local peculiarities in the flow distribution. If this reflects reality correctly, then one could expect both the measured and simulated concentrations to appear somewhat erratic.

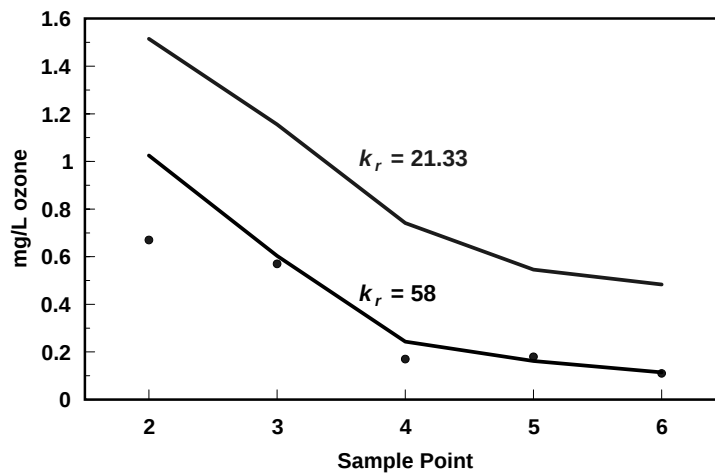


Figure 3.12: Comparison of measured and simulated ozone concentrations for a feed flow rate of 106 ML/d and an ozone dosage of 2.5 mg/L.

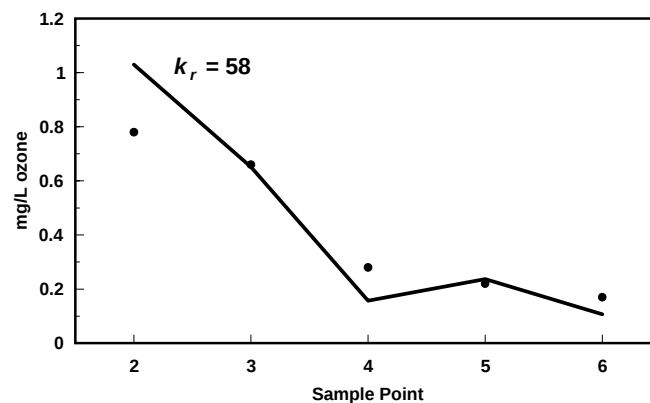


Figure 3.13: Comparison of measured and simulated ozone concentrations for a feed flow rate of 120 ML/d and an ozone dosage of 2.5 mg/L.

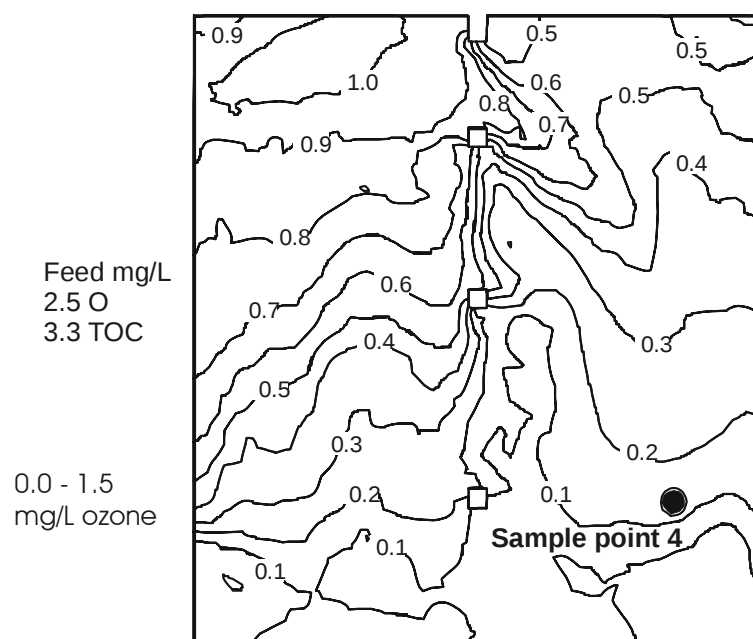


Figure 3.14: Simulated ozone concentrations on a plane in the middle compartment for a feed flow rate of 120 ML/d and an ozone dosage of 2.5 mg/L.

In addition to the steady state ozone concentration profiles, a step test was undertaken in which the ozone dose to the contactor was changed while the flow rate was held constant. As can be seen from Figure 3.15, the simulation using the fitted value of k_r does not predict the measured response well. The concentration response was recorded only for sample point 4, because of the time required to do a scan of all the sample points. At the time point 4 was considered to be the most likely candidate to selected to monitor the contactor for control purposes, based on the preliminary CFD modelling that had been carried out to help decide on the experimental strategy.

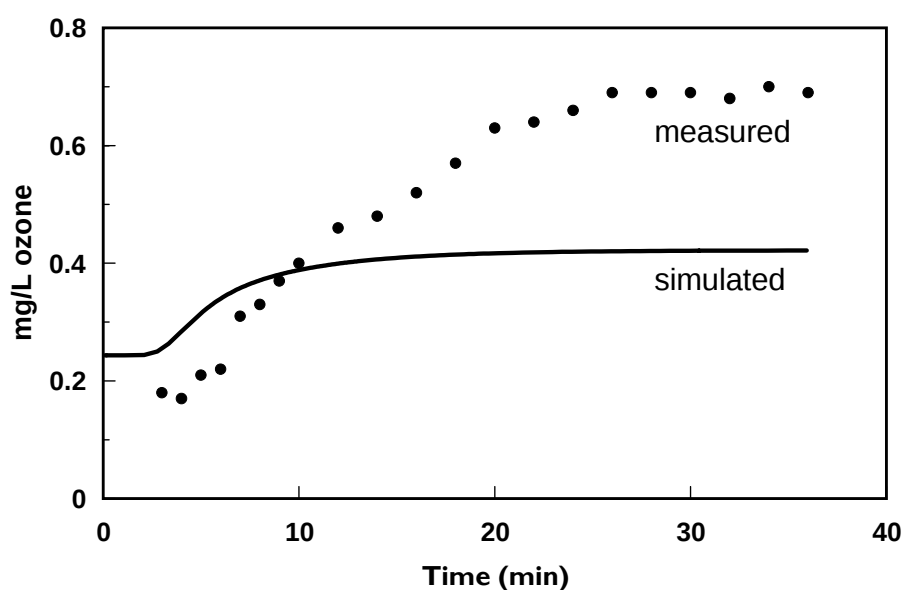


Figure 3.15: Comparison of measured and simulated ozone concentrations at sample point 4 for a flow rate of 106 ML/d and an ozone dosage stepped from 2.5 mg/L to 3.5 mg/L.

3.7 Conclusions from the monitoring program

The model calibrated at 106 ML/d (Figure 3.11) gives quite a good prediction of the profile when the flow changes to 120 ML/d (Figure 3.12), but not when the ozone dose changes. This suggests that the hydraulic part of the model is sufficiently accurate, but that the reaction model needs improvement. To this end a laboratory study has been undertaken to attempt to determine reaction rates for the water being treated at the Wiggins Works. At the time of writing this study is incomplete.

The issue of the location of sample point 4 in a particularly sensitive and uncertain position in the contactor (Figure 3.14) does raise some questions about the above conclusion. It also suggests that point 4 may not be the ideal monitoring point for control purposes, because it may be subject to erratic influences. Figure 3.14 suggests that it would be better to locate the sample point in the left half of the contactor rather than the right. It is noteworthy that this conclusion would not have been reached from consideration of the experimental data alone.

3.8 Disinfection modelling

An aim of this case study has been to demonstrate the use of CFD to make direct predictions of the performance of a water treatment unit in terms of the function that it is supposed to perform, instead of surrogate measures such as the residence time distribution. Once the CFD model has been validated it should be possible to make reliable predictions of the effects of changes in various operating conditions. Although the reaction model has not been fully validated at the time of writing, it will be used to illustrate the effect of a change in OCS on the disinfection performance of the system.

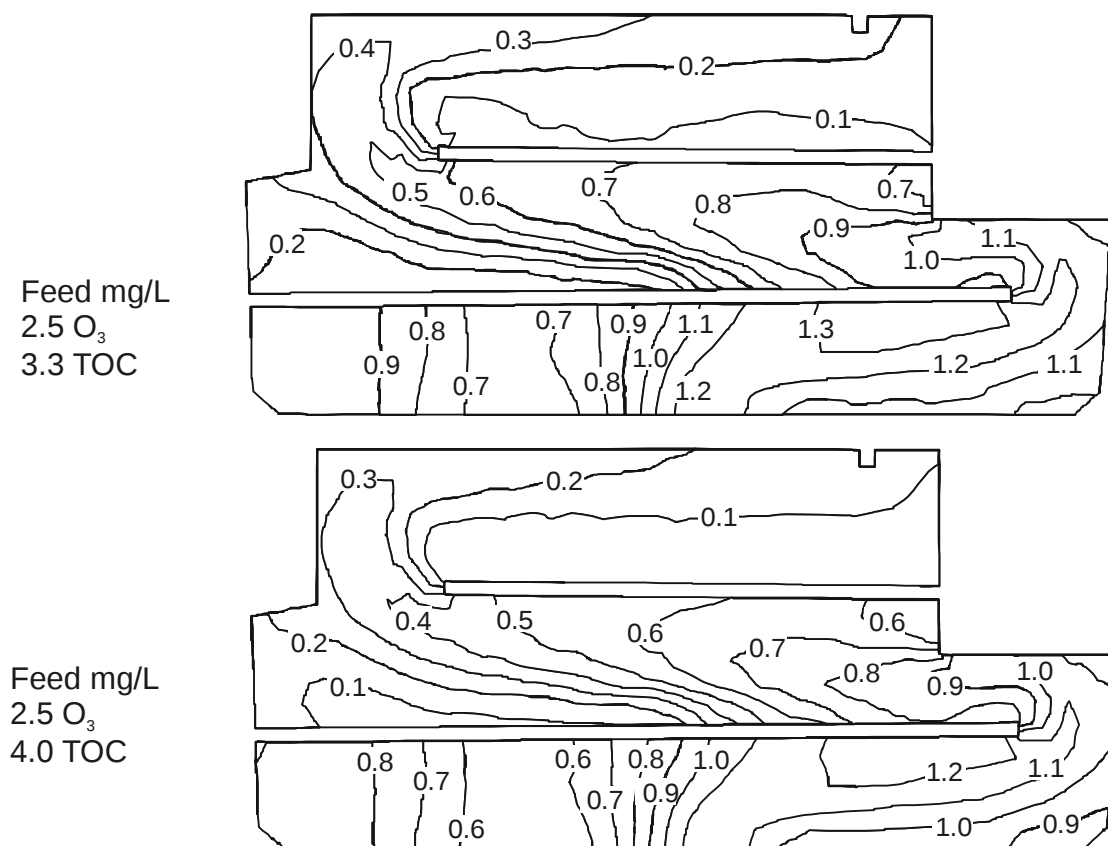


Figure 3.16: The effect of increasing the TOC on simulated ozone concentrations on a vertical plane situated 2m from the right hand side of the contactor. Flow: 106 ML/d., ozone dose: 2.5 mg/L.

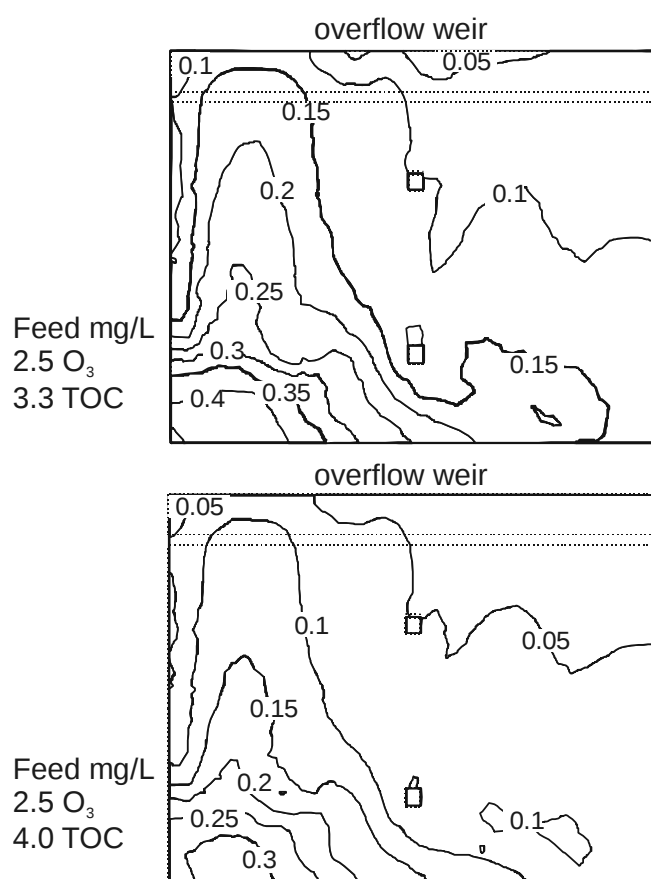


Figure 3.17: The effect of increasing the TOC on simulated ozone concentrations on a horizontal plane through the top compartment. Flow: 106 ML/d., ozone dose: 2.5 mg/L

The level of OCS in the feed water can be expected to vary seasonally and with weather conditions. By reducing the concentration of ozone in the contactor, the OCS will reduce its disinfection efficiency. If the effect is significant, this means that the OCS level should be taken into account in the control strategy for the contactor. To investigate the effect of an increase in the OCS level on deactivation of *cryptosporidium* oocysts, a simulation was performed for a TOC increase of 20 % over the base case, i.e. from 3.3 mg/L to 4.0 mg/L. (Recall that for the base case the OCS was related entirely to the TOC in the water. At certain times of the year dissolved Fe II and Mn II also contribute to the OCS). Figure 3.16 illustrated the effect of increasing the OCS on the ozone concentrations in the contactor. The general reduction in ozone concentrations at the higher OCS level is evident.

These concentrations are on the left side of the reactor; because of the left-right asymmetry in the flow distribution there are gradients from left to right as illustrated by Figure 3.17.

Figure 3.18 shows the corresponding contours of predicted survival of the *cryptosporidium* oocysts on a plane 2 m from the left hand side. It is clear that there is a significant loss of disinfection performance.

To assess the effect properly, the overall survival ratio must be obtained by integrating over the entire length of the outlet weir. Figure 3.19 plots integrated ozone concentrations and *cryptosporidium* survival ratios for a range of TOC values. The contactor control strategy will need to strike a balance between these two quantities, neither residual ozone nor viable oocysts are desirable in the effluent. However it should be recognised that the oocysts will be subjected to further attack in the subsequent chlorination stages, and it is known that having been exposed to ozone makes them much more vulnerable to chlorine.

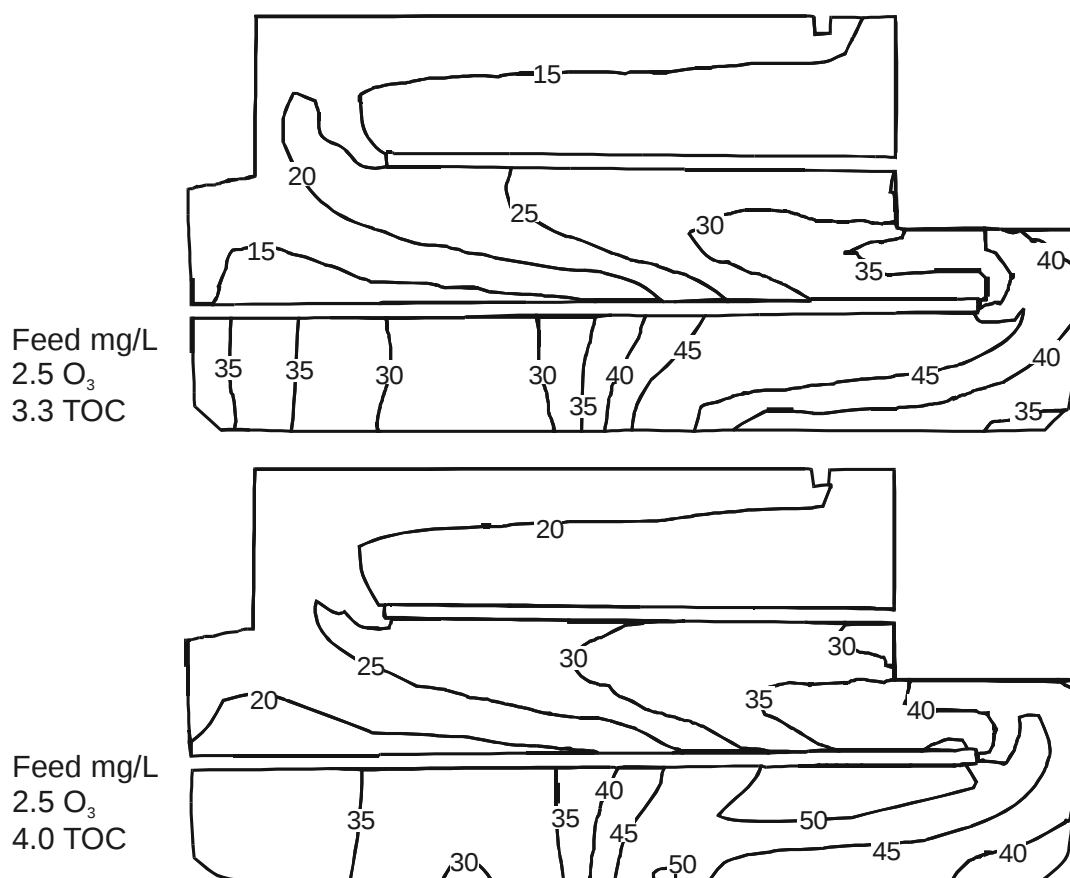


Figure 3.18: The effect of increasing the TOC on simulated % cryptosporidium survival on a vertical plane situated 2m from the right hand side of the contactor. Flow: 106 ML/d, ozone dose: 2.5 mg/L

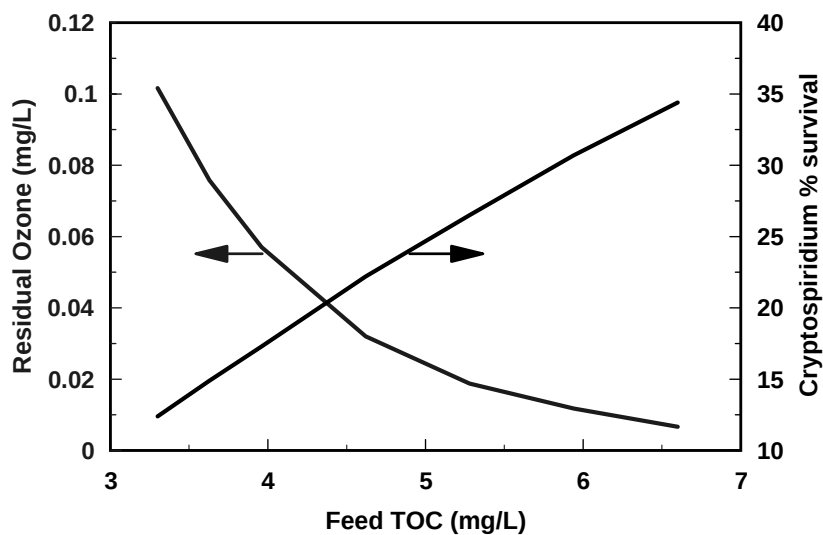


Figure 3.19: The effect of increasing the TOC on the overall simulated residual ozone and cryptosporidium survival in the contactor effluent. Flow: 106 ML/d, ozone dose: 2.5 mg/L

3.9 Gas phase modelling

All the modelling of the contactor described so far did not directly include the presence of gas bubbles in the contactor, but its effect on the overall hydrodynamics was modelled by increased turbulence intensity. The injected gas contains about 10% ozone which dissolves almost completely, and the remainder (oxygen) passes through the contactor as bubbles. The tracer tests described in sections 3.3.3 to 3.3.5 showed that the presence of these bubbles had an appreciable, though apparently small effect on the flow of water through the reactor. At the static mixer the gas flow constitutes only about 1% of the total flow by volume, however as it passes through the reactor it tends to separate from the water and follows a preferential path along which its concentration is much higher. It therefore seemed worthwhile to attempt to model the effect of gas to estimate its effect on the water flow distribution, and to check whether neglecting this was indeed justified. Unfortunately these attempts were only partially successful. An initial two-dimensional model yielded some interesting results, but a more realistic three-dimensional model ran into numerical stability problems that were not solved. It is probable that these problems would have been overcome if a faster computer with more memory had been available, as the problem appeared to become less severe as the grid resolution was increased; however the size of the resulting model exceeded the capability of the machine before a stable solution was reached.

3.9.1 2-dimensional 2-phase modelling

To reduce the 3-D model to a 2-D model requires some rather unrealistic approximations. The model considers a longitudinal slice of a reactor which is infinitely wide, with every longitudinal section identical. The main problem with this assumption is the water inlet, which in reality exists only at one lateral location, whereas in the 2-D model it has to be represented as an infinitely wide slot. Other 3-D features such as the internal pillars had to be left out of the model.

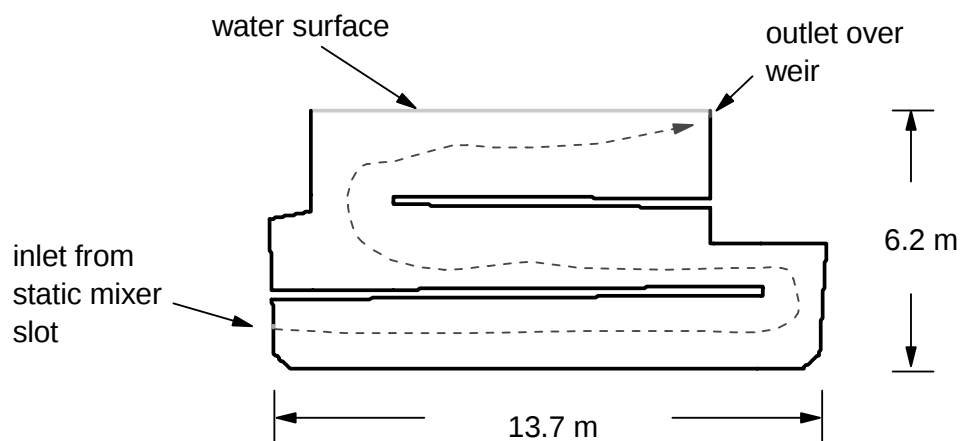


Figure 3.20: Geometry of the 2-D ozone contactor model

The *Algebraic Slip Mixture* (ASM) multiphase model was chosen to represent the gas in the reactor. The ASM model represents the phases as interpenetrating fluids which can move at different velocities. Partial differential equations describe the mass and momentum balances for the mixture and a volume fraction equation for the secondary (gas) phase. An algebraic equation describes the relative velocity between the phases (the *slip velocity*). To reduce this from a partial differential equation to an algebraic equation, acceleration terms for the slip velocity are neglected. The model also involves an assumption that gas bubbles maintain a constant and uniform size, which means that it does not account for phenomena such as bubble coalescence.

3.9.1.1 Boundary conditions and other model parameters

The *inlet boundary condition* presented a particular problem because of the distortion involved in its 2-D representation. Since it was impossible to match all the conditions which apply to the physical 3-D inlet, some ultimately arbitrary decisions had to be taken as to what aspects would be represented. The 2-D model can be considered as a 1m wide slice of an infinitely wide reactor. It was decided to set the mass and momentum injection rates *per metre of width* to match the corresponding values averaged over the width of the 3-D reactor. Thus the model would represent some kind of average of the range of conditions prevailing at different section across the width of the reactor. The gas at the inlet was set to a typical gas/water ratio for the contactor (1 % by volume) distributed uniformly across the inlet.

The other boundary that required special consideration was the upper liquid surface, corresponding to the free liquid surface in the actual reactor. As mentioned in section 3.3.2, it was desirable to represent the free surface as a rigid frictionless wall, rather than using a more rigorous 2-phase free surface model, because of the much reduced computational requirement and greater numerical stability. The problem with this representation was the presence of gas bubbles: the wall representation meant that they were trapped below the surface instead of passing through to the atmosphere above - an issue which had not come up with the water-only model. This was solved relatively simply and satisfactorily by adding a user written subroutine to set the gas-phase volume fraction to zero at the surface, which meant that any gas that reached the surface simply disappeared from the model.

The remaining boundary surfaces, i.e. walls and outlet were handled using the standard Fluent features as described previously. The standard $k-\varepsilon$ model provided by Fluent was used to model turbulence. The only remaining parameter was the bubble size for the gas phase. This was set to 3 mm as suggested by Cockx *et al.* (1999).

3.9.1.2 Model results

Figure 3.21 shows the predicted gas distribution through the contactor. Most of the gas separates rapidly from the bulk of the liquid, a flows along very narrow path, which follows the compartment ceilings. This might explain why, in the experimental tracer tests, the gas injection showed very little effect on the overall residence time distribution. However, this suggested conclusion is apparently contradicted by the simulated tracer test that was carried out with the 2-D model.

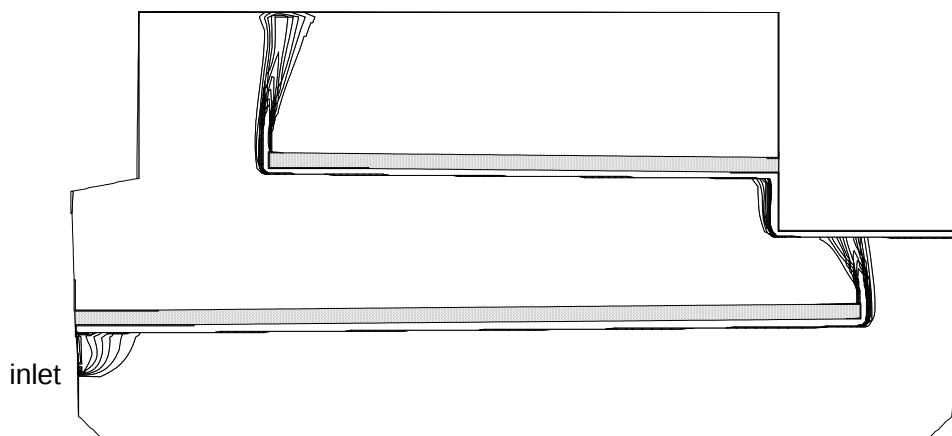


Figure 3.21: Contours of gas volume fraction for the 2-D model

As can be seen in Figure 3.22, gas injection has a marked effect in broadening the tracer response curve. This is because some water is dragged through the contactor by the bubbles a little faster than the rest .

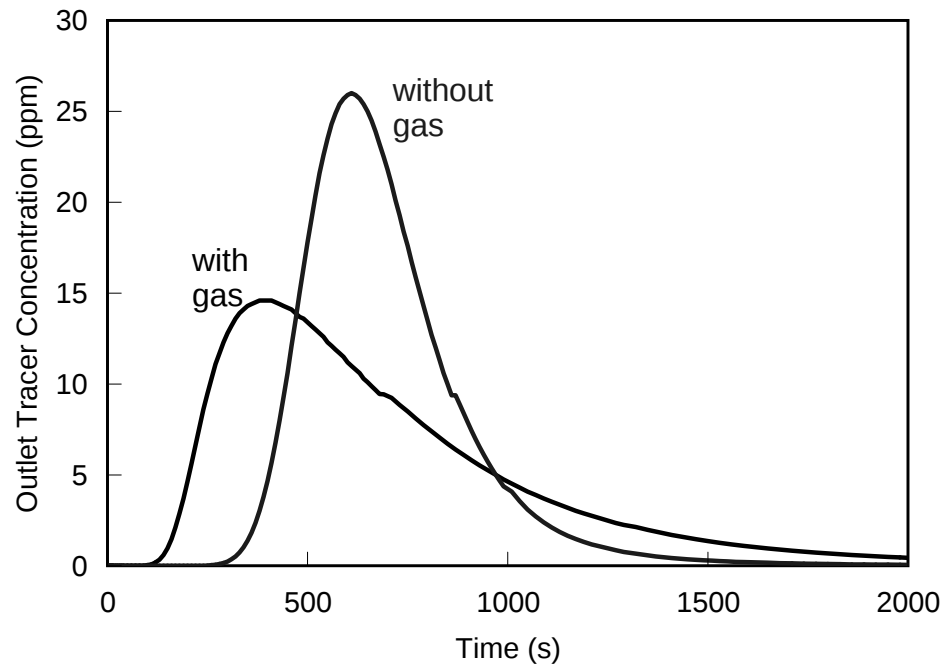


Figure 3.22: Simulated tracer responses for the 2-D model .

The experimental tracer responses (Figures 3.3 and 3.4) exhibited a similar qualitative tendency, but to a considerably lesser extent. It is interesting that the measurements taken on the left hand side of the contactor correspond better to the model than the rest, which would suggest that the gas tends to be channelled to the left.

3.9.1.3 Conclusions from the 2-D model

Because of these contradictory indications, few firm conclusions could be drawn from the 2-D model results. The most optimistic interpretation was that the gas flows in a relatively narrow path through the reactor, within which it has quite a marked effect on the flow, but outside of which its effect is small: the 2-D model best represents what occurs in the section of the contactor which contains the gas path. However, in view of the uncertainties involved, this interpretation is clearly speculative, and a 3-D model offered the only hope of settling the questions involved.

3.9.2 3-dimensional 2-phase modelling

Several attempts were made to add gas-phase modelling to the 3-D model described in section 3.3, however all failed to overcome the problem of numerical stability. The instabilities tended to occur where the gas volume fraction became high immediately under the compartment ceilings. The degree of instability appeared to be reduced by making the computational grid finer in these areas, however this came at the expense of a much greater number of computational elements. It seems probably that, if the memory capacity and processor speed of the available computer had been greater, the problem would have eventually been overcome by refining the grid even further.

3.10 Conclusions

Conclusions from this case-study can be divided into specific conclusions concerning the ozone contactor, and conclusions regarding the application of CFD to such water-treatment systems. The study was by far the most comprehensive undertaken during the project, and provides the broadest illustration of the use of CFD in research into water treatment processes, and some of its strengths and weaknesses.

3.10.1 Conclusions regarding the ozone contactor

- The location of the residual ozone monitor used in the control of the ozone dose should be moved from its current position close to the outlet weir to a point at the end of the middle compartment, level with the position of the experimental sampling point 4. However, although sampling point 4 was located on the right side of the contactor, the model suggested that a more consistent and reliable signal would be obtained if the sensor were located on the left.
- The disinfection efficiency of the system is very sensitive to the level of ozone consuming substances in the raw water feed. Since this factor can be expected to vary seasonally and with weather conditions, operational procedures should be developed to take it into account in determining the ozonation control strategy.

The investigation was aimed at improving the operating rules for the contactor rather than changing any aspects of its design, however the mass of detailed information provided by the models did indicate aspects of the design which could be improved.

- The Residence Time Distribution exhibited by the contactor (both simulated and actual) is somewhat disappointing given its structure. The US EPA *Disinfection Benchmarking and Profiling Guidance Manual* (USEPA, 1999) presents a broad classification of contactors in terms of their general configuration, the Wiggins contactor might be placed under the category of being provided with “serpentine intra-basin baffles”. According to this classification the “baffling condition” should fall somewhere between “average” ($T_{10}/\bar{T} = 0.5$) and “superior” ($T_{10}/\bar{T} = 0.7$). The measured value for $T_{10}/\bar{T}$ was 0.44, which in fact falls between “average” and “poor” ($T_{10}/\bar{T} = 0.3$). The model results show that this degradation in performance is mainly due to the left-right asymmetry in the flow distribution, and recirculation vortices in the bottom compartment. The latter occurs because the feed is concentrated at one point rather than being distributed evenly across the width of the compartment. CFD modelling could be used to design modifications (such as strategically placed baffles) to reduce these hydraulic non-idealities and therefore improve the contactor performance.

3.10.2 General conclusions regarding CFD modelling

- **The large size and geometrical complexity of the contactor:** on the one hand, this meant that the CFD model provided insight into details of the processes within the reactor that would be almost impossible to obtain experimentally. On the other hand, it led to a very large model with convergence difficulties and long solution times.
- **The hydraulic sub-model:** the water-only hydraulic model gave very good agreement with experimental data, with only minimal calibration (i.e. the adjustment of the turbulent intensity of the feed from the static mixer). Together with similar experiences in other investigations, this indicates that CFD predictions of straightforward flow patterns and RTDs can be usually be used with a high degree of confidence.
- **The two phase (gas-liquid) sub-model:** two-phase modelling was undertaken, but the conclusion that could be drawn were limited. Difficulties associated with grid resolution and convergence are very much greater than with the water-only model. These were

overcome successfully in the 2-D case by making the computational grid fine enough, but the 2-D model proved to be an inadequate representation of the system. In the 3-D model, making the grid fine enough for stability resulted in a model that was too large for the computers available to handle. Experimental verification would have also presented much greater difficulties than in the single phase case.

- **The reaction sub-model :** in this study, adding the reaction sub-model did not involve much extra difficulty as far as the modelling was concerned. This fortunate result was due to the relatively slow reaction rates, with time constants of the same order as the hydraulic residence time of the system. The main difficulty associated with the reaction modelling is due to the lack of detailed knowledge of the reaction mechanisms and the kinetic parameters involved. This meant that model predictions could not be trusted without experimental verification and calibration. It also meant that extrapolation of the model to operating conditions different from the calibration set was unreliable. In an attempt to improve the situation, a laboratory study of the reaction kinetics has been initiated, but is incomplete at the time of writing.

The above conclusions can be generalised to an extent by noting that CFD modelling is very successful where the underlying physics of the process are very well understood, but becomes less useful and reliable when sub-models are added which involve approximations and uncertainties.

3.11 Acknowledgements

The support and assistance of Umgeni Water staff at the Wiggins Waterworks and Process Evaluation Facility, particularly Martin Pryor, Rachi Rajagopaul, Dan Naidoo and Mahomed Docrat, are gratefully acknowledged.

4 THE SAND FILTER BACKWASH SYSTEM AT THE FAURE WATER TREATMENT WORKS

A CFD model was set up to model the water-only phase of the backwash cycle one of the sand filters operating at the Faure treatment works. The objective of the investigation was to determine why parts of the filter take much longer to clean than others, and to propose modifications that would lead to improved operation. The modelling was accordingly divided into two phases: modelling of the existing configuration and modelling of the proposed improvement.

4.1 The existing filter backwash system

The sand in the filter is supported on a concrete floor which has flow distribution nozzles regular spaced over its area. Below the floor is a space which serves as the under drain during filtration, and as a flow distributor during backwash. The CFD model was concerned only with the backwash phase of operation. It also considered only the flow in the under drain space and through the flow nozzles, on the basis that during backwash the sand above the floor is fluidised. During fluidisation, the flow has to support the weight of the sand, which means that the pressure drop across the sand bed is virtually independent of the flow rate through it. Thus the effect of the sand bed could be represented as a constant pressure boundary condition at the flow nozzle exits. The 5712 nozzles in the floor were not modelled individually, instead the floor area was modelled as a porous medium, with pressure drop/flow rate characteristics set to match those of the nozzles.

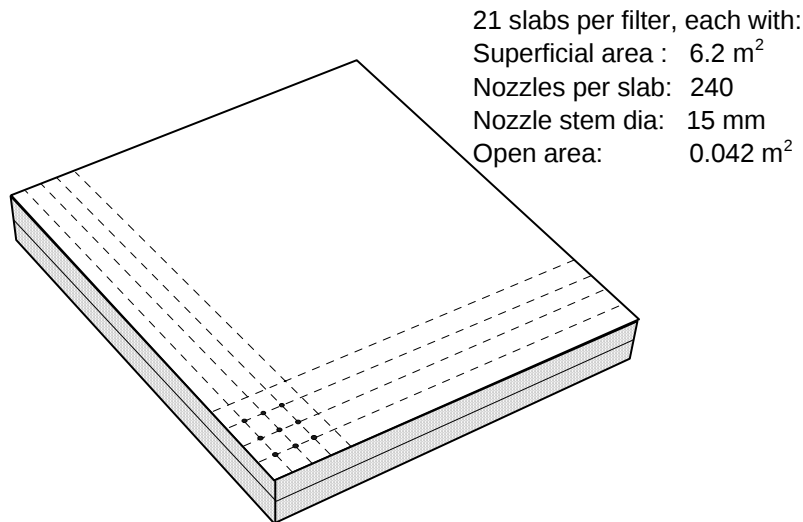


Figure 4.1: Schematic diagram of porous floor model

4.1.1 Model geometry

The model considers only the left hand half of the under drain of the filter, on the assumption that the right half is symmetrically similar. This assumption is not entirely accurate, as there is a air distribution pipe which runs the length of the filter, offset from the centre. This complication was neglected. The feed inlet is a circular opening, 610 mm in diameter. To simplify the meshing, it was modelled as a square opening of the same area. Figure 4.2 illustrates the geometry of the model.

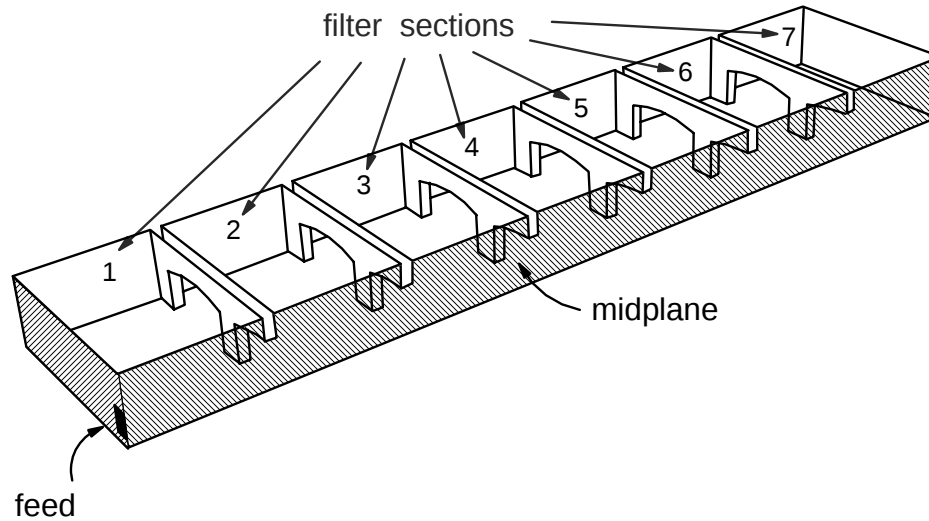


Figure 4.2: Geometry of the CFD model of the filter underdrain.

The mid plane is the plane of symmetry. The upper surfaces of the model represents the nozzle slabs which form the filter floor, and are the outlets for the model. They were modelled as porous surfaces with pressure-drop characteristics calculated to match those of the nozzles. The arches supporting the slabs provided convenient boundaries to divide the outlet into 7 sections.

4.1.2 Boundary conditions

The model was run with a feed rate of 363 kg/s for the half model (i.e. 726 kg/s or 2620 m³/h for the whole filter), which corresponds to a nominal backwash rate of 20 m/h. The pressure at the upper surface of the porous zone was set to a uniform 40 kPa gauge, which corresponds approximately to the pressure drop across the fluidised bed of sand, with the surface at atmospheric pressure (in fact the figure is arbitrary, since only the relative pressure distribution is important in the model, all that matters is that the pressure is uniform over the surface representing the floor).

4.1.3 Model results

Figure 2 shows the resulting backwash rates averaged over each of the seven sections. There is a marked tendency for the rate to increase towards the end of the filter far from the feed, which corresponds to the reported behaviour of the filters during back washing.

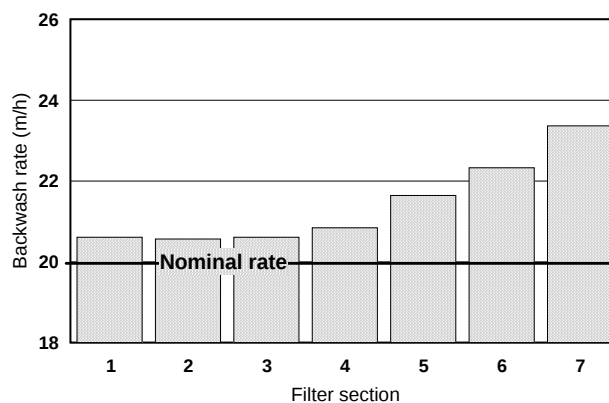


Figure 4.3: Simulated backwash rates in the seven filter sections (see Figure 4.2)

The reason for this behaviour can be seen from the simulated pressure distribution along the length of the filter, shown in Figure 4.4 on the mid plane surface.

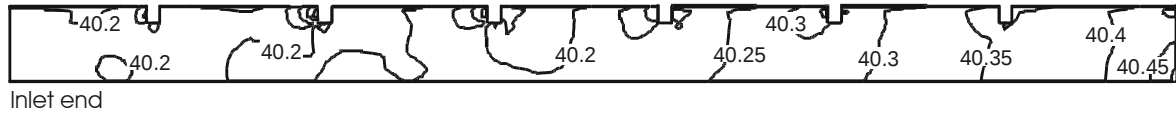


Figure 4.4: *Contours of pressure at the mid plane surface (side view)*

It can be clearly seen how the pressure tends to increase towards the far end from the feed, due the general deceleration of the flow. Because of this increase in pressure, the flow through the nozzle slabs also tends to increase towards the far end, and shown in Figure 4.5. The general tendency is complicated by local variations caused by the presence of the arch supports. The bars in Figure 4.3 show the same upflow rates as Figure 4.5 averaged over each section of the filter.

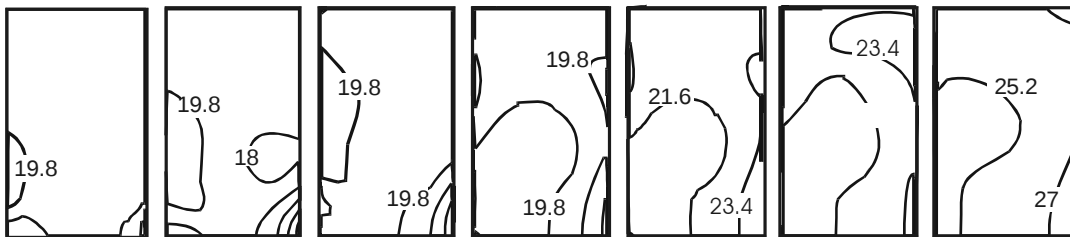


Figure 4.5: *Contours of upflow velocity (m/h) at the nozzle slab surfaces (plan view)*

Further insight into the phenomenon is obtained by considering the flow pattern in the under drain. Figure 4.6 shows *streak lines* (paths that would be followed by particles suspended in the flow) together with contours of velocity magnitude, plotted on a horizontal plane located halfway between the floor and top of the under drain.

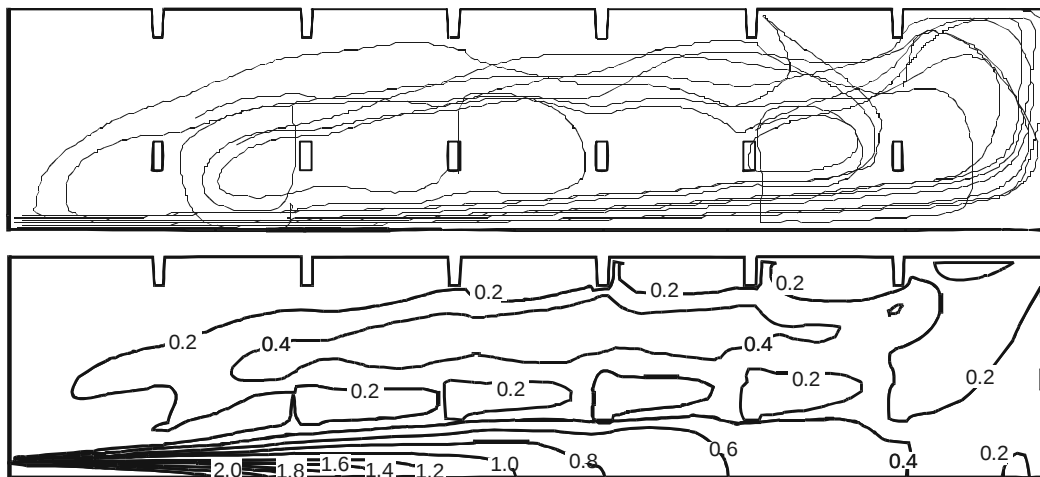


Figure 4.6: *Flow pattern in filter under drain (plan view)*

It can be seen that the incoming flow forms a jet which runs through the central set of arches, hits the far wall, with return flows through the side arches. Thus, the pressure rise can be seen to be intensified as a consequence of the excess momentum in the feed stream, although there would be some pressure rise even if the flow were perfectly distributed across the width of the filter.

4.2 The model of a proposed modification

The model offers an apparently satisfactory explanation of the problems encountered with the back washing of the Faure filters, and suggests a solution in terms of distributing the feed over a wider area to reduce the excess momentum of the feed. There are a number of ways that this could be achieved. Probably the simplest and cheapest to implement would be to install a number of judiciously placed baffles which redirect the flow so as to even out the pressure distribution. However, although the effect of such baffles could be simulated, it would be a difficult task to find appropriate locations, because it has to be done by trial-and-error with a sequence of models, each of which takes significant time and effort to set up.

An alternative idea would be to install a flow distributor down the centre of the under drain, which ensures an even supply of water to each section. This would take the form of a pipe laid along the length of the under drain, with holes on each side. The diameters and spacing of these holes would need to be carefully gradated down the length of the pipe to deliver a uniform volumetric flow per unit length in spite of the pressure rise. Although this will probably be more expensive to install than baffles, its design is a relatively straightforward CFD problem, and its performance in delivering a uniform distribution of flow is virtually guaranteed.

To check the effect of such an arrangement on the flow through the nozzles, an approximate model was set up with the flow introduced via a long narrow slot located on the mid plane, to simulate the effect of the distributor (Figure 4.7).

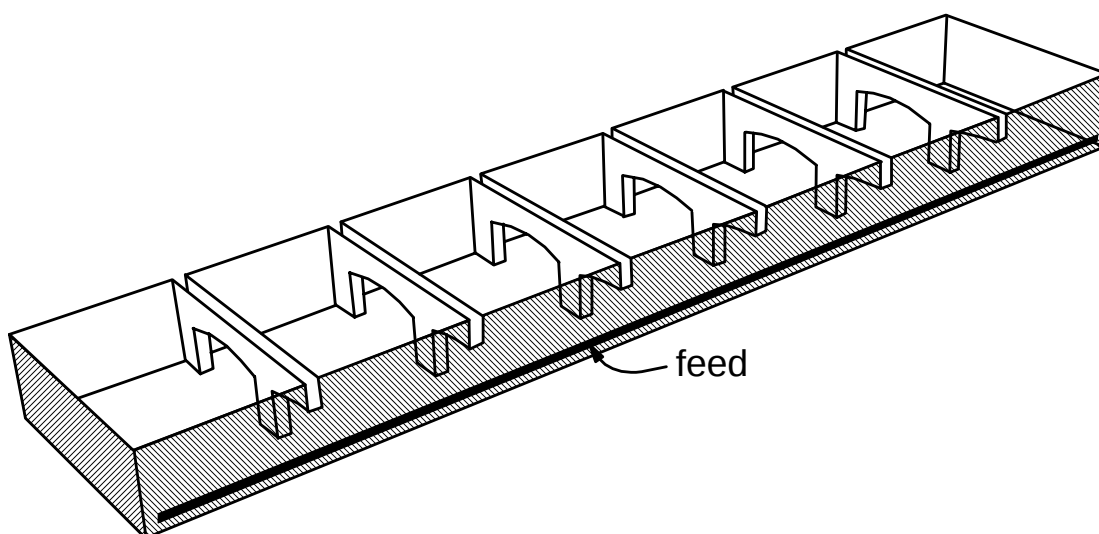


Figure 4.7: Geometry of model simulating effect of a flow distribution pipe

4.2.1 Model results

In this model the variation in upflow rates between filter sections is almost entirely eliminated. The vertical velocities through the porous zone representing the nozzle slabs vary over a range of only 5.9 to 6.1 mm/s (21.2 to 22 m/h : note that these are slightly higher than the nominal 20 m/h, because the area is reduced from the nominal 131 m² by the presence of the arched supports, which block off part of the area.).

4.2.2 Hydraulic design of the proposed distributor

The basic concept used for the distributor design was a large pipe laid down the centre of the filter, with holes drilled into the sides, with diameters and spacing chosen to give a uniform

distribution of outflow along the length. The velocity at the inlet is quite high (2.5 m/s), so there should be no reduction in diameter that would cause the pumping head to increase. This means that the pipe needs to be 600 to 610 mm diameter, at least at the feed end. As the flow decreases along the pipe, the need for such a large diameter falls away. However, the cost of having a large diameter pipe for the full length must be offset against the cost of manufacturing a reducer fitting, such standard fittings are not available in these large sizes.

As a first iteration, a uniform 610 mm ID distributor has been modelled. Figure 4.8 shows the geometrical basis of the model. Because of symmetry, only a quarter of the pipe needed to be represented. The outlet was modelled as a long slot. (The holes shown in the diagram illustrate how the distributor would be set up in practice, they were not set up explicitly in the CFD model). As the distributor was to be designed to give a uniform flow along its length, the outflow boundary condition was set as such. The CFD model then calculated the resulting pressure distribution along the length. (Figure 4.9).

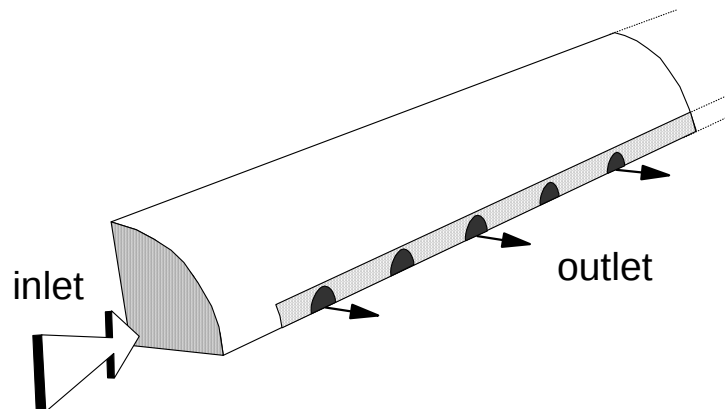


Figure 4.8: Geometry of model of the flow distributor

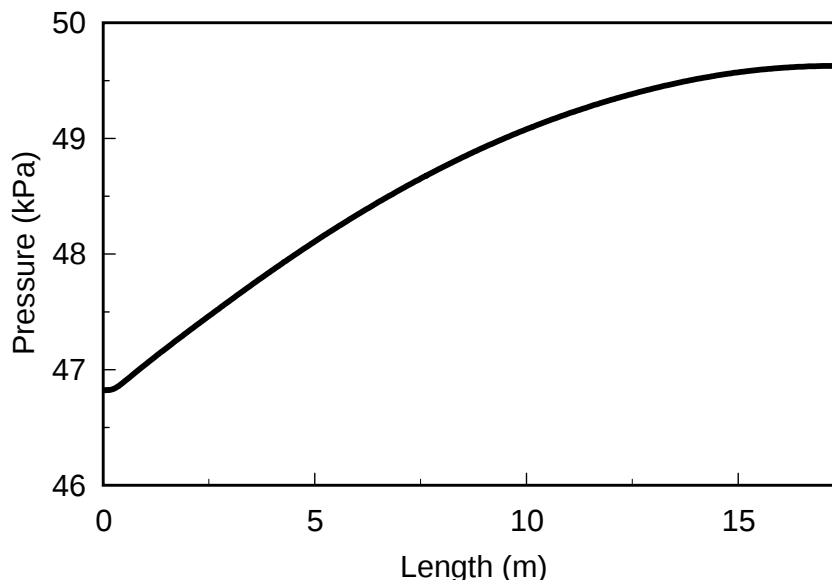


Figure 4.9: Simulated distributor pressure profile

In this solution only the relative values of pressure are relevant. The arrangement of outlet holes has some degrees of freedom. The total area of holes roughly determines the overall pressure drop of the distributor, after which the diameter of holes determines the number that have to be drilled. A spreadsheet was set up with the hole diameter and the outside pressure

(assumed constant, since the distributor is designed to eliminate pressure gradients) as user inputs. The distributor was divided into 0.5 m lengths, over which the interior pressure was taken as the average value from the profile, and the number of holes calculated to give the required flow. The flow through each hole was calculated according to the formula (Perry, *et al.*, 1999) :

$$Q = C_D \cdot A \sqrt{\frac{2 \cdot \Delta P}{\rho}}$$

Where the symbols stand for:

Q	-	volumetric flow rate, m ³ /s
C_D	-	discharge coefficient, dimensionless, taken as 0.68.
A	-	area of hole, m ² .
ΔP	-	pressure drop across hole, Pa.
ρ	-	liquid density, kg/m ³ .

As an example, for an overall pressure drop across the manifold of 8 kPa and a hole diameter of 25 mm, the distributor requires 300 holes on each side, with 10 per side in the first half metre, and 8 per side in the last.

4.3 Conclusions

The proposed distributor should be highly effective and inexpensive to construct. A problem which was identified during a visit to Faure was that limited access space made it difficult to get a long pipe into the underdrain. This could possibly be overcome by constructing the distributor from short sections, or considering some other physical way to implement the principle which would be easier to install. This might require a new CFD model for the distributor, but this would probably be quite a simple exercise.

4.4 Acknowledgement

Thanks to Sarel Pieterse of the Cape Metropolitan Council for providing the motivation and background information for this investigation.

5 THE CLARIFIER UPGRADE AT THE HAZELMERE WATER TREATMENT WORKS

In this case study, a series of CFD models were generated to support the design work for modifications to a clarifier which needed to have its performance upgraded.

5.1 Background

Clarifiers No. 1 and No. 2 at Hazelmere Waterworks north of Durban were identical units with a peripheral inlet unit with clarified water draw-off through slotted pipes positioned radially (Figure 5.1). Flocculation was aided by water-jets from nozzles positioned on a rotating arm. A de grit hopper was positioned on the outer flocculation zone to remove heavier solids. The level of the sludge blanket was controlled by the height of a wall at the 10-metre diameter. Sludge flowing over this wall settled in the central settling zone and was transported via wooden scrapers into the desludge hopper.



Figure 5.1: View of the central part of the clarifier before the upgrade.

The peripheral feed arrangement for these clarifiers (Figure 5.2) was particularly unusual, and caused them to be plagued by poor feed distribution resulting in severe short-circuiting. An investigation into the maldistribution of flow occurring in this clarifier using tracer testing and a CFD model was reported in WRC Report No 648/1/02 *The Application of Computation Fluid Dynamics to Water and Wastewater Treatment Plants*. The conclusion of that investigation had been that converting the clarifier to a central feed arrangement was the only way to obtain a significant improvement in its performance.

The existing hydraulically operated scraper system was close to mechanical failure and had required maintenance and repair at least every three months. The system had failed on a number of occasions resulting in lengthy maintenance shutdowns.

The under performance of the clarifiers posed the following operating risks and problems:

- The poor flocculation and hydraulic short-circuiting on the clarifiers result in continuous carryover of suspended solids onto the filters resulting in shortened filter run times of 12 to 24 hours where filter run times are normally in the region of 24 to 36 hours. The higher backwash frequency also increases operating costs and reduces production. There would be a reduced life of the filter sand from higher solids loading and increased potential for filter mud-balling.
- The increased pathogen loading (attached to suspended solids) onto the filters increases the risk of pathogen breakthrough into treated water and place the consumer's health at risk.
- Reliability of production was not guaranteed as the hydraulic scraper system is close to mechanical failure and had failed on a number of occasions. Production at the waterworks peaks during holiday periods, notably December. The peaks also falls within the high rainfall season when raw water quality is at its poorest.



Figure 5.2: View of the peripheral feed inlet to the clarifier

5.2 Design of the clarifier modifications

In June 2000 Umgeni Water reviewed the existing design and made recommendations on proposed improvements to Clarifier 1 and Clarifier 2. The working group tabled the following design proposal:

- to convert the existing peripheral inlet system to a central inlet, which required construction of the inlet pipe below the existing floor;
- to install baffles on the central inlet port imparted a rotational component to the flow in the flocculation zone to enhance passive flocculation;
- To provide a 10 m diameter flocculator zone in the centre of the clarifier providing, a flocculation time of 30 minutes;
- To install two paddle flocculators within the new flocculator chamber.

It was expected that these modifications would increase the clarifier capacity from about 9 ML/d to 15 ML/day at an up flow velocity of 1.2 m/h (within the design guideline value of

1.5 m/h overflow rate for this type of clarifier). While the design work was being carried out, CFD modelling was undertaken to help evaluate various design options. This interaction led to a number of changes to the design:

- a single central sludge discharge hopper was designed in place of the hopper originally located within the sedimentation zone.
- the floor within the flocculation zone was sloped at 1:12 towards the centre, to aid transport of concentrated sludge into the central hopper against the outward flow of water.
- at the point where the water flow passed under the skirt between the flocculation zone and the settling zone a step was made for the sludge to flow over, to prevent it being re-entrained.
- the dimensions of the flow area immediately beneath the flocculation skirt were decided after evaluating several CFD models.

5.3 The CFD model

The modelling was based on data supplied by the Umgeni Water design team as the design work was proceeding, and the configuration was continually being changed while the modelling exercise was in progress. A number of different models were tried, of which two basic variants are presented here, with different arrangements for sludge withdrawal. The second model configuration presented corresponds to the design that was eventually implemented.

5.3.1 Model basis and assumptions

The model used a two-dimensional, axi-symmetric representation of the geometry, which assumed that the clarifier had exactly the same cross-sectional profile in all radial directions. Certain aspects of the physical clarifier did not match this assumption: the most important of these was the rotating sludge scraper, which moved sludge that had settled on the floor towards the central withdrawal hopper. There is no way of representing this realistically without going to a much more complex 3-dimensional model, so the main effect of moving the sludge towards the centre was modelled by setting a moving surface boundary condition on the floor, with an inward velocity set to match the average distance that solids would be moved by the scraper during a revolution.

The above description applies to the final model configuration with the central sludge hopper. For the earlier design variant with a radial hopper, the sludge withdrawal was modelled as a uniform withdrawal through the clarifier floor, at a rate set to match the overall volumetric underflow averaged over the entire area of the clarifier floor. Again, a more realistic treatment would have required a 3-dimensional model.

The sludge was modelled using the *Algebraic Slip Model* (ASM) provided by Fluent. This model assumes that the relative velocity between solids and liquid (the *slip velocity*) has reached a pseudo-steady-state, so that it can be described by an algebraic equation instead of a differential equation. This assumption should be very good under the low velocity conditions found in clarifiers. The model represents the solids as mono-sized spherical flocs, and the parameters required are floc diameter and density. The diameter arbitrarily set to 1 mm, and the floc density adjusted to give a settling velocity which corresponded to 6 cm/min, which was the average value observed in tests at Hazelmere (4-8 cm/min). Because the flocs are assumed to have a constant density, the model is unable to represent sludge compression, where the weight of overlying sludge squeezes water out of the sludge lying on the clarifier floor. However, this phenomenon was not expected to be limiting on the clarifier performance.

The feed nozzle design had three baffles which imparted a swirling component to the flow entering the flocculation compartment. This was modelled by adding swirl to the flow inlet boundary condition. The magnitude of this component were estimated from the angle of the baffle blades and the inlet flow velocity.

The conditions used in the modelling are summarised in Table 5.1.

Parameter		First Model	Final Model
Feed flow rate	m ³ /h	531.0	625.0
Feed sludge concentration	g/L	75.0	150.0
Feed swirl velocity	m/s	0.0	0.1
Moving floor velocity	m/h	-	1.0

5.3.2 Modelling results

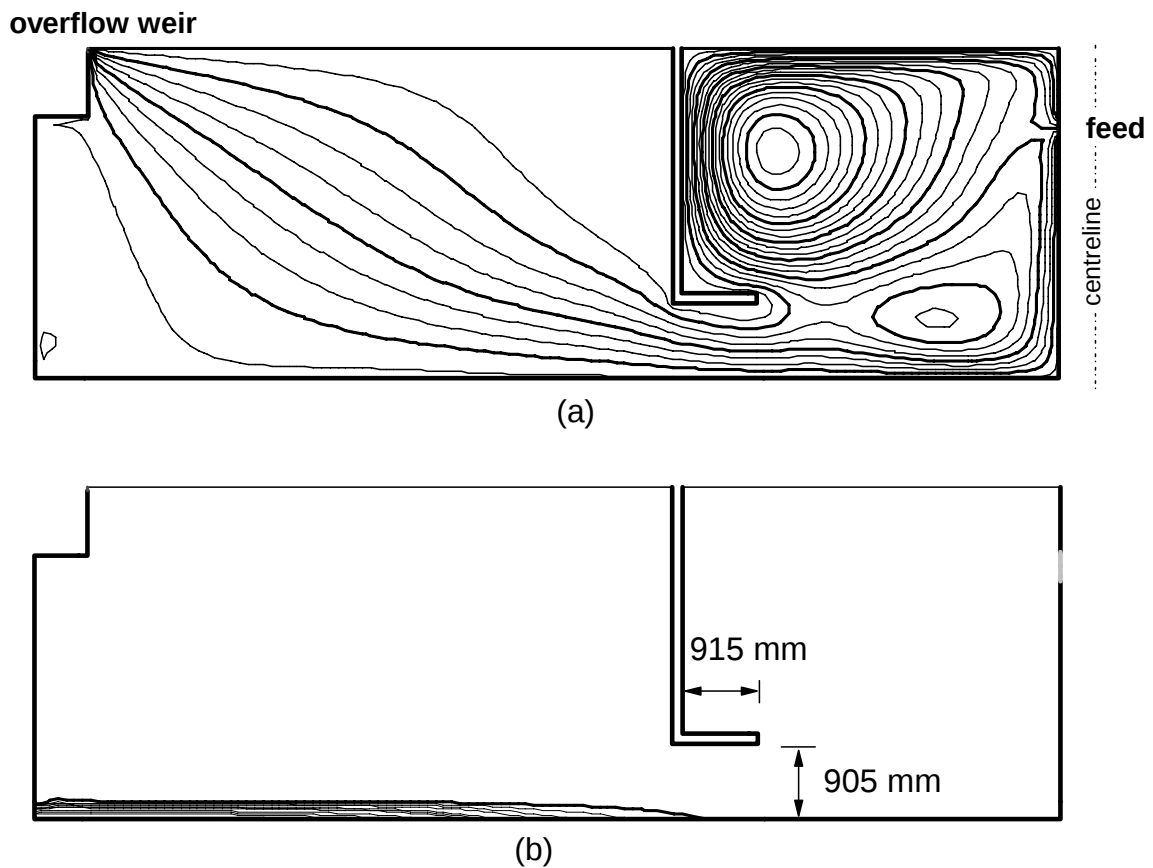


Figure 5.3: (a) Streamlines and (b) contours of sludge concentration (initial model)

Figure 5.3 illustrates one example of the several variants of the first model that were tried with different dimensions of the flow gap and horizontal baffle length as shown in Figure 5.3(a). Note that it shows a half cross-section of the clarifier, since the model considered the clarifier to be radially symmetrical. This model was considered satisfactory as far as it went, however it was felt by the Umgeni design team that the sludge collection system could be improved by moving the discharge hopper from the periphery of the clarifier to the centre. This was based on reasons related to the movement of sludge across the clarifier floor to the

hopper by the scraper system, which was not represented in the radially symmetrical CFD model.

When it was decided to move the sludge hopper to the centre, it was of concerning that the model showed scouring of the sludge by the relatively high water velocities in the gap below the flocculation skirt (see Figure 5.3(a)). The new design required the sludge to be moved to the centre by the scraper, and it was feared that the scouring might interfere with the efficient transfer of solids through the gap. To assist this process, the floor in the flocculation zone was sloped, and a step-down was provided just below the gap (see Figure 5.4, which is derived from the final model in the sequence, and corresponds to what was eventually installed).

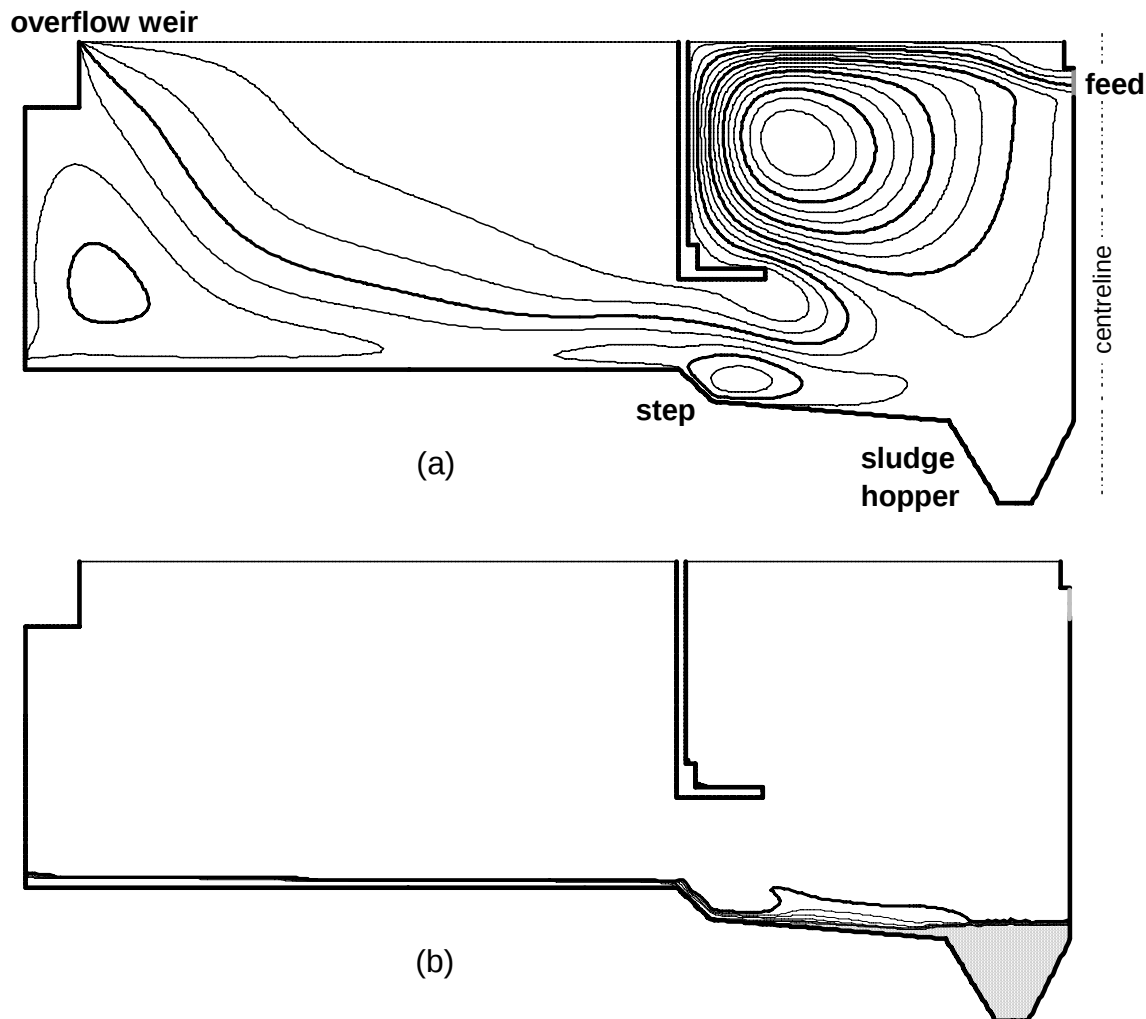


Figure 5.4: (a) Streamlines and (b) contours of sludge concentration (final model)

It can be seen clearly in Figure 5.4(b) how the presence of the step assists the transfer of the settled sludge through the gap below the flocculation zone skirt and prevents re-entrainment of the sludge by the relatively high water velocities in this area. This is because the step induces a vortex which results in a reversed secondary flow (visible in Figure 5.4(a)), which actually helps the sludge through the flocculation skirt gap. It can also be seen that the region immediately above the sludge hopper has particularly low flow velocities (indicated by widely spaced streamlines), which is conducive to producing well consolidated sludge in the hopper, resulting in efficient sludge discharge with minimal entrainment of the overlying water.

5.4 Evaluation tests

The design modifications were implemented on the No. 2 clarifier at Hazelmere, which was re-commissioned in August 2002. There were then three variations on the clarifier design present at the works, No. 1 with the original peripheral feed configuration, No. 2 with the newly modified central feed, clari-flocculator configuration, and No. 3 which had originally been designed with a central feed sludge blanket configuration. A comparative performance test between clarifiers No. 2 and No. 3 was carried out during August 2003. A comparison between No. 1 and No. 2 would not have proved much, since the peripheral feed arrangement was known to give inferior performance to the central feed.

In the test, No. 2 and No. 3 were the only clarifiers online. The feed to the works is distributed to the clarifiers by a splitter box which is designed to split the flow equally, however there is no separate measurement of flows to the individual clarifiers. Figure 5.5 shows the flow rate through each clarifier (i.e. half the measured total flow) during the test. For most of the time the flow rate stood at about 80 % of the design capacity of 15 ML/d.

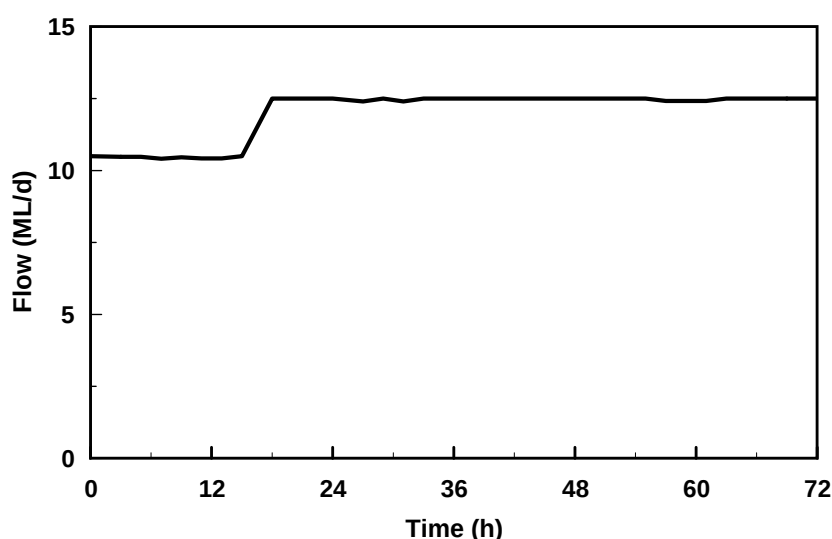


Figure 5.5: Flow rate through each clarifier during the comparative test.

Part of the upgrade of clarifier No. 2 had been to install mechanical mixers in the flocculation zone to enhance flocculation. These has been shown in earlier tests to improve the product water turbidities significantly, however, since they had not been considered in the model, this improvement could not be attributed to the CFD modelling. Consequently the mixers were switched off during the test.

Figure 5.6 shows the inlet and outlet turbidities measured during the test, plotted first against the elapsed time of the test, and also on a percentile basis. During the test period the feed water turbidity was extremely low, so that flocculant dosage increased the turbidity significantly, which is how the turbidity from No. 3 comes to be higher than the water feeding it at times. The superior performance of the No. 2 is clearly evident, which vindicates the use of CFD in its design.

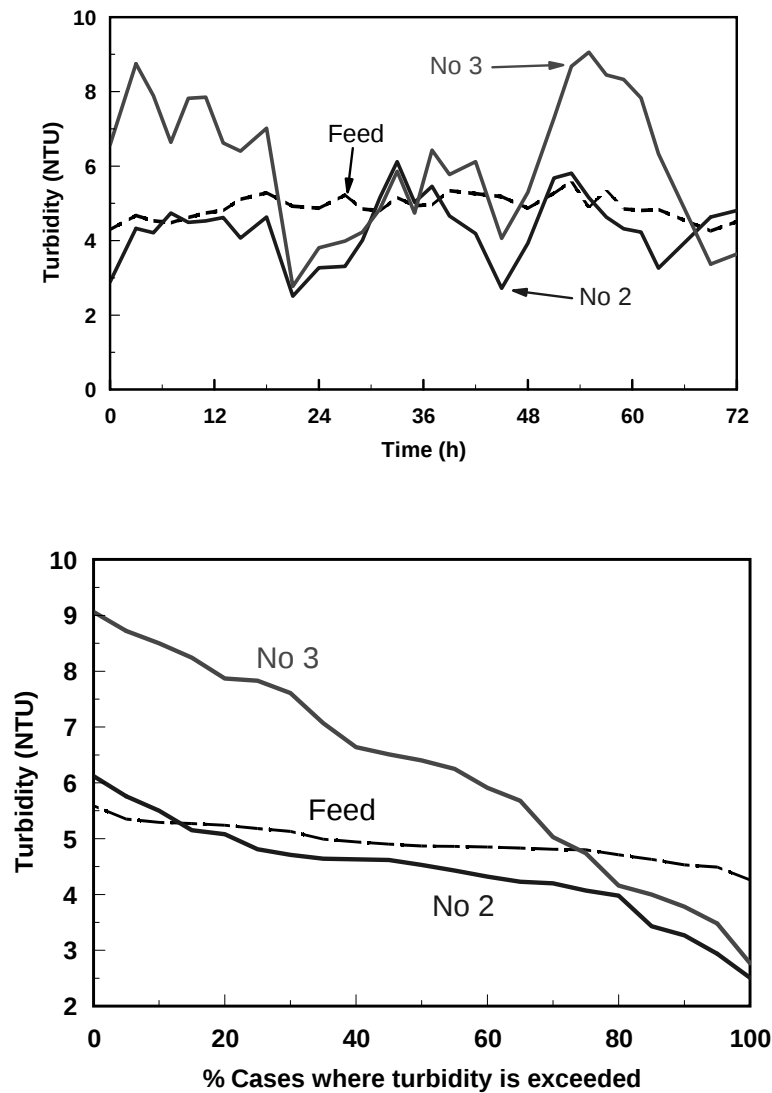


Figure 5.6: Turbidities measured at each clarifier during the comparative test.

5.5 Conclusion

This case study provides a clear and relatively straightforward example of the successful use of CFD modelling to assist in water treatment equipment design.

5.6 Acknowledgement

This case study was undertaken in co-operation with Thubendran Naidu of Umgeni Water, who was the engineer in charge of the redesign and re-commissioning of the clarifier. His contributions to the study and to the writing of the report are gratefully acknowledged.

6 BATCH SETTLING OF SECONDARY SEWAGE SLUDGE

The modelling of solids settleability is essential for modelling settling tanks in water and wastewater treatment. Until the advent of hydrodynamic models, the focus of modelling solids settleability was on describing the behaviour of the solids in the water while the water itself was considered a stationary or ideally moving medium in which the solids settled. Hydrodynamic models now allow the behaviour of the water in the settling tank to be modelled. While the modelling of the water flow has made extraordinary advances in the past 20 years, modelling the settleability of the solids has not improved much over the this time. In fact, the weakest part of hydrodynamic models of settling tanks may be the modelling of settleability of the solids. This investigation explored methods for measuring and modelling solids settleability with the view of improving these for hydrodynamic models of settling tanks.

The design and operation of secondary clarifiers is commonly based on the solid flux theory (Vanderhasselt and Vanrolleghem, 2000). The basic data required for the application of this theory can be obtained from multiple batch tests by which the stirred zone settling velocities (V_{zs}) over a range of sludge concentrations (C) are measured (dilution experiments).

The relationship between C and V_{zs} is often characterised by the *Vesilind equation*

$$V_{zs} = V_0 e^{-nC} \quad [6.1]$$

which contains two parameters: V_0 and n . The key underlying assumption is that the settling velocity is only dependent on the local sludge concentration. The Vesilind parameters can then be used to construct the flux curve, which is the key input of the solid flux theory.

The flux theory is an approach to describe *zone settling*. Zone settling can be assumed for concentrations above about 1 g/L. Even though Equation 6.1 gives settling velocities down to zero concentration, this clearly is incorrect because the settling velocity is predicted to increase for decreasing concentrations and does not reach a terminal velocity value for particles. Measurement of V_{zs} for concentrations below about 1 g/L is difficult due to the lack of a well-defined solid/liquid interface during settling at the low concentrations. Moreover particles left behind after most of the sludge has settled, settle very differently from those in the main bulk of sludge.

Analysis of samples taken from the clear water region above the zone settling region in the column or near the surface of a secondary settling tank (SST) will show low concentrations of small particles with low settling velocity. This observation does not correspond to what the flux theory predicts, and thus shows that the flux theory does not hold for low concentration zones. Consequently, should the flux theory be applied not only to model the zone settling in the lower region of the SST but also the discrete settling conditions in the upper effluent region, the approach must be modified for low concentrations.

The approaches to extend the flux model (Equation 6.1) to low concentrations may be divided in two categories:

- i) Two groups of particles are defined. Most of the particles are treated as sludge and are described according to the flux theory. A small group of single particles is additionally defined, whose settling velocity is set either to a very low value or even to zero (Dupont and Henze, 1992; Otterpohl and Freund, 1992).

- ii) The basic approach of the flux theory is kept, that is to estimate the settling velocity as a function of the concentration, but modified such that the function starts at zero settling velocity for very low concentrations rather than at some maximum value (Takács *et al.*, 1991; Dupont and Dahl, 1995).

Both approaches have been shown to significantly improve the accuracy of the dynamic model prediction of the effluent suspended solids (ESS) concentration in 1-D and 2-D CFD models.

Takács *et al.* (1991) presented a double exponential settling function to describe the change in settling velocity V_s with concentration C over the entire concentration range from zero (i.e. not only the zone settling part):

$$V_s = V_0 [e^{-n_1(C-C_0)} - e^{-n_2(C-C_0)}] \quad [6.2]$$

where V_0 , n_1 and n_2 are constant parameters, and C_0 is concentration of the non-settleable particle fraction. At high sludge concentrations, this tends to the Vesilind form.

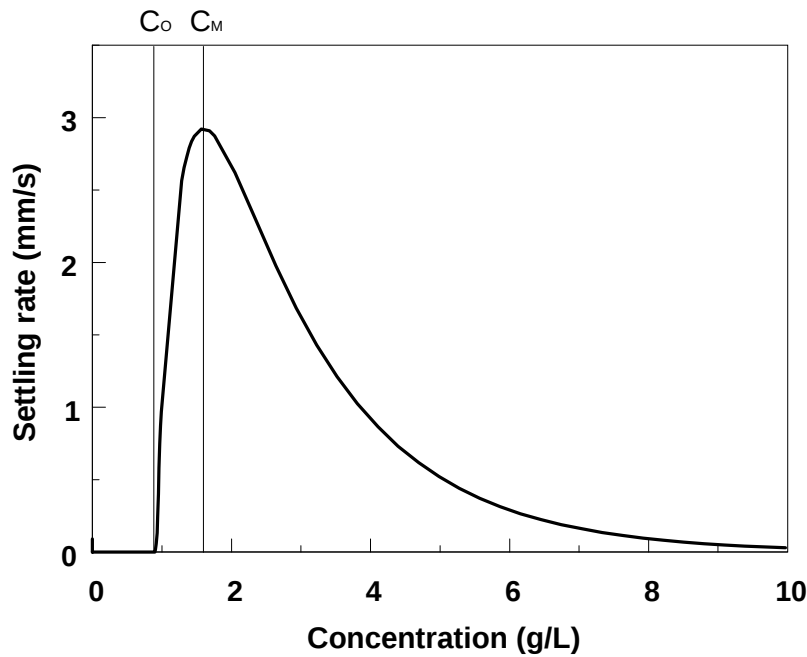
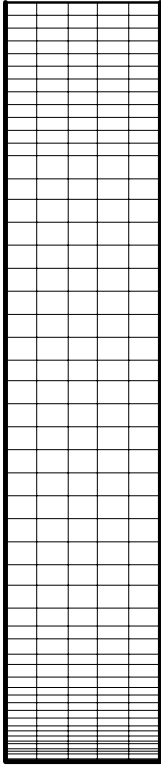


Figure 6.1: Sludge settling rates according to the Takács model with parameters: $V_0 = 5.5$ mm/s; $n_1 = 0.576$ L/g; $n_2 = 2.86$ L/g; $C_0 = 0.9$ g/L.

Many CFD modellers of settling tanks have used the Takács equation to describe the settling velocity of the solids, however the equation is not well formulated for experimental calibration. It contains 4 constants that require measurement to calibrate it, i.e. V_0 , n_1 , n_2 and C_0 . Although the determination is tedious, V_0 and n_1 are measurable on sludges provided C is large enough for zone settling (~ 1 g/L). However, once $C < 1$ g/L, n_2 has a major effect on the settling velocity (V_s). However, n_2 cannot be measured, since the settling no longer follows the zone settling pattern in reality. With Equation 6.2, n_2 and C_0 values have a profound influence on the predicted ESS. C_0 can be defined by measuring the residual suspended solids concentration in the supernatant some time after sedimentation, but experimental protocol for this has not been standardised yet. Generally, because n_2 and C_0 cannot be measured, their values have been determined by calibration (force-fitting model simulations to measured ESS data from full-scale SSTs). This is unsatisfactory, in that the clarifier cannot be properly modelled without using its own operating data.

The strategy of this investigation was to incorporate the Takács settling model into the simulation of batch settling tests, in an attempt to identify characteristics which might be amenable to experimental measurement, and which might allow the Takács parameters to be estimated.



6.1 Modelling procedure

A two-dimensional model of a simple settling column was set up, 20 cm high by 4 cm wide, as shown in the diagram. The Fluent *Algebraic Slip* two phase model was used to represent the sludge. This model represents the water and sludge as interpenetrating fluids with volume fractions for each phase. Accelerational terms are neglected in the calculation of the relative velocity between the phases (the *slip velocity*) so that it is represented by an algebraic formulation rather than a differential equation. These approximations are particularly appropriate for the sludge settling problem.

The key sludge properties to be represented were the density, the conversion from g/L (which is how sludges are commonly characterised,) to volume fraction (which is how Fluent represents multi-phase systems); and the settling velocity. These are somewhat interrelated quantities, and it is desirable that they should be chosen in a consistent way. The method chosen was based on the idea that the sludge concentration at which the settling rate becomes negligible (110 g/L) corresponds a volume fraction of 0.62 (which is the maximum packing density for rigid spheres). This gives a bulk density of 1008.2 kg/m³ and a volume fraction to g/L conversion factor of 14.66 g/L.

The Takács model was introduced as a user-written subroutine which replaced the standard Fluent calculation of the slip velocity. In this routine, the local slip velocity at any point was calculated from the local volume fraction of sludge according to the double exponential formula (Equation 6.2). The parameters used in the equation were those given with Figure 6.1.

The settling test was simulated by initialising the computational domain with zero velocities and a uniform sludge volume fraction everywhere.

6.2 Results

The simulated sludge settling showed two contrasting characteristic behaviours, depending on the initial sludge concentration set for the test. Figure 6.2 shows the pattern for initial concentration C_I higher than C_M , the concentration that exhibits the maximum settling velocity (refer to Figure 6.1).

The notable features of this result are:

- The sharp interface between “clear” liquid and sludge settling at the initial concentration.
- The residual concentration of non-settling sludge in the “clear” liquid (C_0)
- The smooth build-up of a concentrated layer of settled sludge on the floor of the column.

These qualitative features of the solution are retained for all initial concentrations down to C_M . However for C_I below C_M the behaviour changes to that shown in Figure 6.3.

Under these conditions, the interface with the “clear” liquid is diffuse, whereas the interface with the settled sludge at the bottom of the column is sharp.

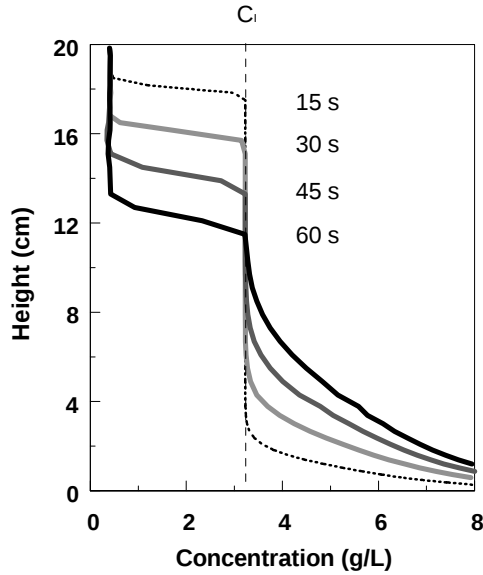


Figure 6.2: Simulated concentration profiles in a batch settling test for C_I (3.2 g/L) higher than C_M (2.0 g/L) at 4 elapsed times.

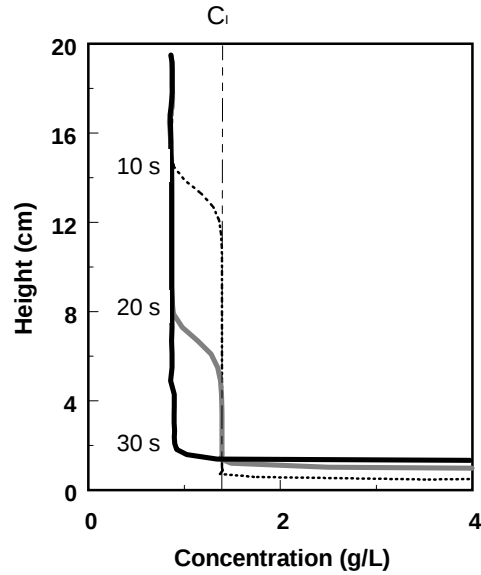


Figure 6.3: Simulated concentration profiles in a batch settling test for C_I (1.4 g/L) lower than C_M (2.0 g/L) at 3 elapsed times.

6.3 Discussion

For brevity, a settling test with C_I higher than C_M will be referred to as a *Type I* test, and one with C_I lower than C_M as a *Type II* test.

The first point to note is that the qualitative characteristics of the simulated Type I and Type II settling tests correspond reasonably well with what is observed in reality.

It is clear that, in a Type I test, no information can be obtained about settling velocities for concentrations lower than C_I , apart from the value of C_0 , since this is the concentration of sludge which remains in the liquid above the interface, and does not settle at all. In principle, it would be possible to obtain information for concentrations higher than C_I by making measurements of the concentration profiles near the floor, however this would probably be quite difficult in practice. What can be measured very easily is the rate of descent of the sharp interface, which gives the settling velocity for C_I . By performing a series of such tests with progressive dilution, the part of Figure 6.1 for concentrations above C_M could be mapped out. This corresponds to the well known result, noted in the introduction to this chapter, that V_0 and n_1 are measurable provided C is large enough for zone settling.

Figure 6.3 shows that it should be possible, in principle, to obtain information about settling velocities for concentrations between C_0 and C_M by analysing the diffuse concentration profiles of Type II tests. However, apart from the experimental difficulties involved, the Takács model is not physically realistic in this region, so that any detailed analysis of the concentration profile would be of dubious value.

However, it seems that identifying the point at the peak of the settling curve (C_M , V_M) and C_0 should give an adequate characterisation of the settling behaviour for concentrations between C_0 and C_M . This could be done by performing a series of Type I tests at progressively lower initial concentrations, until the sharp concentration interface disappears.

It is accepted that Type I tests with high initial concentrations C_I give good estimates of V_0 and n_1 : indeed a commercial device is available which automates such tests (Vanderhasselt

and Vanrolleghem, 2000). If the proposed procedure is able to provide estimates of (C_M, V_M) and C_0 , then one could possibly use the following characteristics of the Takács equation to estimate n_2 :

$$C_M = C_0 + \frac{\ln(n_1/n_2)}{(n_1 - n_2)} \quad \text{and} \quad V_M = V_0 \left[\left(\frac{n_2}{n_1} \right) \frac{n_1}{n_1 - n_2} - \left(\frac{n_2}{n_1} \right) \frac{n_2}{n_1 - n_2} \right]$$

This then theoretically completes the characterisation of the sludge in terms of the Takács model.

6.4 Conclusion

The character of this case-study was somewhat different to the others undertaken during the project, in that the CFD model was used to suggest a direction for further research, rather than to interpret or extrapolate research results. An experimental investigation needs to be undertaken to verify the suggested protocol.

6.5 Acknowledgement

Thanks to Professor George Ekama of the University of Cape Town for providing the motivation and background information for this investigation.

7 CONCLUSIONS AND RECOMMENDATIONS

Some conclusions and recommendations specific to each on the investigations undertaken during the project may be found in the relevant chapters; this chapter presents the more general conclusions which arise from considering the project as a whole.

7.1 The scope for application of CFD modelling in water treatment

It is interesting that most of the broad issues identified the literature survey (chapter 2) were touched on in one form or another during the investigations undertaken during this project. The Wiggins ozone contactor started with simple hydraulic modelling and prediction of the residence time distribution, and progressed to more complex physical modelling of reaction kinetics, disinfection performance and 2-phase flow. The Faure filter backwashing investigation looked at simple hydraulic modelling of flow distribution in the context of an equipment re-design exercise, concentrating entirely on the one specific issue for the design, and ignoring or approximating all other aspects of the system. The Hazelmere clarifier investigation was similarly a re-design exercise, but this time it involved two-phase modelling. It also provided experience of working interactively in the design team, with the concomitant time and budget constraints, requiring strict focus on the specific design objectives, at the expense of realism and unnecessary detail. Finally the batch settling investigation again involved two phase modelling, but this time addressed a purely theoretical question. Thus the experience gained allows a reasonably comprehensive assessment of the role that CFD can play in water and wastewater treatment.

The very fundamental nature of the CFD approach has the advantage of being able to represent appropriate systems (see below) in great detail with minimal requirements for empirical data, but the disadvantages of complexity and difficulty in solving the resulting systems of equations. These practical difficulties prevent CFD from being a universally appropriate approach to all problems involving fluid flow, in spite of its fundamental basis. Generally, CFD is most useful for systems which are *well-connected*, that is, where **all** the boundary conditions have relatively strong influences on **all** parts of the flow field. This applies to many systems found in water treatment, such as reservoirs, contact chambers, sedimentation basins, ponds and even lakes and lagoons. However there are also many systems in water treatment where CFD does *not* provide an effective approach, for example a set of equipment connected by a pipe network, or a long reach of a river. In such cases the CFD model would expend enormous computational effort on calculating the practically negligible effects of remote boundary conditions.

The simplest CFD models consider only the hydraulic aspects of a system. Frequently these models are used to predict the residence time distribution (RTD), which often provides a link to more direct performance indicators through empirical rules based on experience (e.g. the disinfection CT rule). As CFD modelling has become more established, more detailed models are appearing which attempt direct representations the physical and chemical processes taking place in the treatment processes, such as sedimentation, flocculation, inter-phase mass transfer and chemical reaction. In all cases these more complex models need to be supported by experimental studies to establish the parameters for the physical and chemical parts of the models. The CFD modelling thus has a role in both the interpretation of results from experimental apparatus, and in extrapolating research results to the design of full-scale processes.

The relationship between tracer testing and CFD modelling to determine the RTD of a system is worth mentioning. The ozone contactor study demonstrated the use of tracer testing to verify the CFD model, and concluded that CFD modelling is often able to predict the RTD very accurately. However, if the RTD is all that is required, the tracer test may be quicker and less expensive to perform than to develop a CFD model. However this depends on the size of the system: for many water treatment systems the size is such that a very large dose of tracer is required, together with an elaborate and expensive sampling and chemical analysis programme, and the time required to complete the test is so long that it is not feasible to maintain conditions steady for long enough.. Nevertheless, tracer testing should always be considered as a possible alternative to a CFD study, as long as the RTD is adequate to address the required purpose. A less tangible factor that should be borne in mind is the extra insight that the CFD model is able to bring to the investigator.

7.2 The costs involved in CFD modelling

The literature does not reflect a widespread acceptance of CFD modelling in water and wastewater treatment, and this is mirrored in the South African water industry. The cost involved in undertaking such modelling is undoubtedly one of the factors contributing to this situation. To some extent this is a matter of perception, as argued by Shilton, Glynn and Phelps (1999), but the reality is that the overall cost of a CFD investigation is likely to be fairly high.

To start with, the skills required are relatively rare, and take some time to develop. CFD has not yet found a place in undergraduate curricula, so postgraduate training is involved. The underlying mathematics is complex and not easy for practising engineers to master on their own. It is true that the CFD software now available takes care of almost all the mathematical complexities, but paradoxically this may make the problem worse rather than better, because it makes it so easy to obtain plausible results which one does not really understand, increasing the potential for making serious errors. The large international water treatment firms such as Veolia and Thames Water have a small number of CFD specialists who act as internal consultants for equipment design. During this project, the example of the Hazelmere clarifier (chapter 5) illustrated how such an arrangement might work..

CFD modelling does require more than usually powerful computing hardware, and the requirements escalate rapidly when modelling the more complex physical processes, as illustrated by the difficulties encountered with the gas phase in the ozone contactor (chapter 3). However the hardware cost is less and less significant as a cost factor, as it continues to decline steadily, and since the skilled personnel costs involved in such advanced modelling are likely to be very significant.

The cost of CFD software has come down steadily during the duration of the project, but is still high, even with a discounted academic licence. A non-academic licence would have been much more expensive. The development of the software seems to be driven by much higher cost applications that water treatment - aerospace, chemical manufacture, power generation, automotive design etc., and the pricing appears to reflect this kind of market. Many of the models, for instance combustion or solidification, that are available in the software are not relevant to water treatment - it could be that more limited package could be marketed to the water and wastewater industry.

The wide range of problems which might be tackled with CFD makes it very difficult to make any general statement about the cost of undertaking CFD modelling. However, some idea can be obtained by considering a relatively straightforward investigation, such as the Hazelmere clarifier (chapter 5). The time involved was about 40 h (excluding report-writing), so the personnel cost would be of the order of R6 000 (2003 rand values). The software licence cost for commercial use of the Fluent software was about R16 000 per

month. Since the minimum period made available by Fluent was 1 month, it depended whether other jobs were available to share the cost, so the software cost would be between R4 000 and R16 000, and the overall cost would be between R10 000 and R22 000, to which would be added the costs for computer time: if done with a suitable personal computer (1GHz pentium with 500Mb RAM) this would amount to about R100.

This cost might be considered reasonable for a large clarifier, but appear excessive for a small unit. The problem is that the cost would probably be about the same irrespective of the size of the installation. For small units the costs could be effectively reduced by developing standard designs which could be reused a number of times. The extra design cost should be recoverable through lower operating cost, however this will probably be difficult to quantify beforehand.

7.3 Recommendations

The recommendations are divided into those that relate to the individual case studies, and the more general ones that relate to the application of CFD to water and wastewater treatment.

7.3.1 The Wiggins ozone contactor

- 1) The position of the ozone sensor monitoring the residual ozone should be moved to the position identified in the investigation.
- 2) A new strategy for the control of the ozonation should be developed that takes into account the ozone demand of the raw water and the disinfection efficiency of the contactor.
- 3) The cost-benefit balance of the ozonation in the overall water purification process is not easy to quantify; it could even be the case that the benefits do not justify the cost. The model that has been developed for the contactor could be very useful as a component in a wider investigation of the role of ozonation in the overall treatment process.

7.3.2 The Faure filter backwash system

A backwash distributor should be installed on a trial basis on one of the filters at the Faure Water Treatment Works. If this proves as successful as predicted, the system could then be installed on the other filters at the works, and on other filters of similar design.

7.3.3 The Hazelmere clarifier

The design modifications which proved so successful should be implemented on the remaining peripherally fed clarifier at the Hazelmere Water Treatment Works when appropriate. The design should also be considered for new clarifiers of a similar size.

7.3.4 The batch settling test for sewage sludge

An experimental project should be undertaken to test the protocol suggested by the modelling results. This would involve carrying out laboratory settling tests to determine the sludge settling parameters, using these parameters in a CFD model of a full-scale clarifier, and testing the model predictions experimentally on the full-scale unit.

7.3.5 General recommendations

The South African water industry still needs to develop an adequate pool of CFD expertise that can be called upon when appropriate. This might involve:

- Introducing CFD in a limited way in undergraduate university courses, to promote awareness of the technique.
- Postgraduate university training to build up the CFD skills base in the country.

- Courses, symposia and workshops designed to promote awareness of the benefits of CFD among water industry managers and policy-makers.
- Centres of expertise which can undertake research or consultancy work to provide CFD support to water researchers and operators, and promote CFD by providing the academic and industrial training mentioned above.
- CFD specialists employed in the design teams of the larger water authorities, municipalities and engineering consultants.
- A way should be sought to engage the developers of CFD software in order to make appropriate CFD packages available at costs which are appropriate to the water and wastewater treatment industry.

8 REFERENCES

- Anderson JD, (1995). *Computational Fluid Dynamics. The Basics with Applications*. McGraw-Hill Book Co., Singapore.
- Baléo JN, Humeau P and Le Cloirec P (2001) Numerical and experimental hydrodynamic studies of a lagoon pilot. *Water Research* **36**(8) 2029-2039
- Beltrán FJ (1995). Theoretical aspects of the kinetics of competitive ozone reactions in water. *Ozone Science and Engineering* **17** (2): 163-181
- Bin, AK, and Roustan, M. (2000). Mass Transfer in Ozone Reactors. International Specialised Symposium IOA 2000, *Fundamental and Engineering Concepts for Ozone Reactor Design*, Toulouse, France.
- Bredtmann M (1982). Significance of the ozone to water contacting system. *Ozonation Manual for Water and Wastewater Treatment*. Masschelein. W.J. New York, John Wiley and Sons, 129-132
- Brannock MWD, Howes A, Johns M, de Clercq B and Keller J (2002) CFD modelling of particle transport and biological reactions in a mixed wastewater treatment vessel. Information accessed on 20th August 2003 from:
<http://www.leapaust.com.au/services/casestudy/casestudy.htm>
- Brannock MWD, (2003) Computational Fluid Dynamics Tools for the Design of Mixed Anoxic Wastewater Treatment Vessels, *PhD Thesis, University of Queensland*.
- Brouckaert CJ and Buckley CA (2002). The application of computational fluid dynamics to water and wastewater treatment plants. *WRC Report No. 648/1/02*. Water Research Commission, Pretoria South Africa.
- Chataigner O, Gagnon JL, Chagneau G and Chopard P (1999) FLUENT: modelling for optimising the design of post-chlorination tanks, *Houille Blanche-Revue Internationale de l'Eau*, **54** (3-4): 15-20
- Chau KW and Jiang YW (2002) Modelling of flocculation and transport of cohesive sediments from an on-stream storm water detention pond. *Water Research* **36**(15) 3849-3859.
- Cockx A., Do-Quang Z., Liné A., Roustan M. (1999). Use of computational fluid dynamics for simulating hydrodynamics and mass transfer in industrial ozonation towers, *Chemical Engineering Science*, **54**, 5085-5090.
- Cook CB, Orlob GT and Huston DW, (2002) Simulation of wind-driven circulation in the Salton Sea: implications for indigenous ecosystems. *Hydrobiologia* **473**(1-3): 59-75.
- Craig K, De Traversay C, Bowen B, Essemiani K, Levecq C, Naylor R (2002) Hydraulic study and optimisation of water treatment processes using numerical simulation. *3rd World Water Congress of the International-Water-Association*, Melbourne, Australia, 7-12 April 2002.
- Dahl C, Larsen T and Peterson O (1994). Numerical modelling and measurement in a test secondary settling tank. *Water Science and Technology*, **30**(2), 219-228.
- De Clercq B (2003), Computational fluid dynamics of settling tanks: development of experiments and rheological, settling and scraper sub models. *PhD Thesis, Universiteit Gent*.
- de Kretser D, Matthews B and Williams M (2003) Use of computational fluid dynamics to investigate natural and mechanically induced flows in an aerated lagoon, *TAPPI Journal*, **2**(4): 12-14

- Do-Quang Z, Cockx A, Liné A and Roustan M (1999) Computational fluid dynamics applied to water and wastewater treatment facility modelling. *Environmental Engineering and Policy*, **1**: 137-147
- Do-Quang Z, Roustan M, and Duguet J-P (2000). Mathematical Modelling of Theoretical Cryptosporidium Inactivation in Full-Scale Ozonation Reactors. *Ozone Science & Engineering* **22**, 99-111.
- Dupont R and Dahl C (1995) A one-dimensional model for a secondary settling tank including density current and short-circuiting. *Water Science and Technology*, **31(2)**, 215-224.
- Dupont R and Henze M (1992) Modelling of the secondary clarifier combined with the activated sludge model No.1. *Water Science and Technology*, **25(6)**, 285-300.
- Greene DJ, Haas CN and Farouk B, (2002) Numerical simulation of chlorine disinfection processes. *Water Science and Technology: Water Supply*, **2(3)**, 167-173.
- Gyürék LL, Li H, Belosevic M, and Finch GR, (1999). Ozone inactivation kinetics of Cryptosporidium in phosphate buffer. *Journal of Environmental Engineering* **125(10)**: 913-924.
- Haarhoff J and van der Walt JJ (2001). Optimal design parameters for hydraulic flocculators *Journal of Water Supply: Research and Technology - AQUA*, **50(3)**: 149-159.
- Hoigné J, and Bader H (1983a). Rate constants of reactions of ozone with organic and inorganic compounds in water - I Non-dissociating organic compounds. *Water Research* **17**, 173-183.
- Hoigné J, and Bader H (1983b). Rate constants of reactions of ozone with organic and inorganic compounds in water - II Dissociating organic compounds. *Water Research* **17**, 185-194.
- Imam E and McCorquodale JA (1983a). Simulation of flow in rectangular clarifiers. *Journal of Hydraulic Engineering, ASCE*, **109(3)**, 713 -73 .
- Imam E, McCorquodale JA and Bewtra JK (1983b). Numerical modelling of sedimentation tanks. *Journal of Hydraulic Engineering, ASCE*, **109(12)**, 1740-1754.
- Jones SC, Sotiropoulos F and Amiratharajah A (2002). Numerical modelling of helical static mixers for water treatment. *Journal of Environmental Engineering*, **128(5)**: 431-440.
- Joret J-C, Mennecart V, Robert C, Compagnon B, and Cervantes P (1997). Inactivation of indigenous bacteria in water by ozone and chlorine. *Water Science and Technology* **35** (11-12): 81-86.
- Kamimura M, Furukawa S and Hirotsuji J (2002). Development of a simulator for ozone/UV reactor based on CFD analysis. *Water Science and Technology*, **46(11-12)**, 13-19.
- Krebs P (1995) Success and shortcomings of clarifier modelling. *Water Science and Technology*, **31(2)**, 181-191.
- Krishnappan BG and Marsalek J (2002) Modelling of flocculation and transport of cohesive sediments from an on-stream storm water detention pond. *Water Research* **36(15)** 3849-3859.
- Li H, Finch GR, Smith DW, and Belosevic M (2001). Sequential inactivation of Cryptosporidium parvum using ozone and chlorine. *Water Research* **35** (18), 4339-4348.
- Lyonnaise des Eaux (2000) *Virtual Water Plant*. Information accessed on 14th August 2003 from: http://www.unitedwater.com/tour/bios_facts/virtualPlant_pressRelease.pdf.

- Lyn DA, Stamou AI and Rodi W (1992). Density currents and shear-induced flocculation in sedimentation tanks. *Journal of Hydraulic Engineering, ASCE*, **118**(6), 849-867.
- Matko T, Fawcett N, Sharp A and Stephenson T (1996). A numerical model of flow in circular sedimentation tanks. *Trans IChemE.*, **74**(B), 197-204.
- McCorquodale JA and Zhou S (1993). Effects of hydraulic and solids loading on clarifier performance. *Journal of Hydraulic Research, ASCE*, **31**(4), 461-478
- Muroyama K, Norieda T, Morioka A, and Tsuji T (1999). Hydrodynamics and computer simulation of an ozone oxidation reactor for treating drinking water. *Chemical Engineering Science* **54**: 5285-5292.
- Otterpohl R and Freund M (1992) Dynamic models for clarifiers of activated sludge plants with dry and wet weather flows. *Water Science and Technology*, **26**(5-6), 1391-1400
- Park NS, Park H, Kim JS, (2003) Examining the effect of hydraulic turbulence in a rapid mixer on turbidity removal with CFD simulation and PIV analysis, *Journal of Water Supply Research and Technology-Aqua*, **52**(2): 95-108
- Salter HE, Ta CT, Williams SC, (2000) Three-dimensional computational fluid dynamic modelling of a facultative lagoon, *Water Science and Technology*, **42**(10-11): 335-342
- Shaw CT (1992). *Using Computational Fluid Dynamics*. Prentice-Hall International, London.
- Rakness KL, Hunter GF, and DeMers LD (2000). Drinking Water Ozone Process Control and Optimisation. *International Specialised Symposium IOA2000: Fundamental and Engineering Concepts for Ozone Reactor Design*, Toulouse, France.
- Rennecker JL, Driedger AM, Rubin SA, and Mariñas BJ (2000). Synergy in sequential inactivation of *Cryptosporidium parvum* with ozone/free chlorine and ozone/monochloramine. *Water Research* **34** (17), 4121-4130.
- Sherwin C and Ta CT (2002) Investigation of anaerobic zone short-circuiting using computational fluid dynamics. *Water Science and Technology*, **46**(4-5), 333-338.
- Shilton A, Glynn D and Phelps P (1999) An inside look - the potential of CFD technology. *Water21*, July-August 1999, 37-38.
- Szalai L, Krebs P and Rodi W (1994). Simulation of flow in circular clarifiers with and without swirl. *Journal of Hydraulic Engineering, ASCE*, **120**(1), 4-21
- Ta CT, Beckley J and Eades A (2001). A multiphase CFD model of DAF process. *Water Science and Technology*, **43**(8): 153-157
- Takacs I, Patry GG and Nolasco D (1991). A dynamic model of the clarification-thickening process. *Water Research*, **25**, 1263-1271.
- USEPA (1999). *Disinfection Profiling and Benchmarking Guidance Manual*, United States Environmental Protection Agency, Office of Water, EPA 815-R-99-013.
- Vanderhasselt A, and Vanrolleghem P, (2000) Estimation of Sludge Settling Parameters from Single Batch Settling Curves, *Water Research*, **34**(2), pp. 395-406.
- Van Der Walt JJ, (2002) The modelling of water treatment process tanks, *D.Ing. Thesis, Rand Afrikaans University*.
- Wilderer PA, Bugartz HJ, Lemmer H, Wagner M, Keller J and Wuertz S (2002) Modern scientific methods and their potential in wastewater science and technology. *Water Research* **36**(2) 370-393.
- Zhou S and McCorquodale JA (1992a). Mathematical modelling of a circular clarifier. *Canadian Journal of Civil Engineering*, **19**(3), 365-374.

Zhou S and McCorquodale JA (1992b). Modelling of rectangular settling tanks. *Journal of Hydraulic Engineering, ASCE*, **118**(10), 1391-1405.

Zhou S and McCorquodale JA and Vitasovic Z (1992). Influences of density on circular clarifiers with baffles. *Journal of Hydraulic Engineering, ASCE*, **118**(6), 829-847.