EXECUTIVE SUMMARY

MOTIVATION FOR STUDY

Sedimentation tanks are the workhorses of any water purification process. It is thus crucial for the sedimentation tank to be operated to its full potential. It is not only the chemical aspects of flocculation that cause problems, however. Hydraulics also play a prominent part. Overdesign of plant is common, leading not only to unneceassary capital expenditure, but also to water wastage in the form of excessive sludge. Inadequate design causes overloading of filters, leading to frequent backwashing which also wastes a significant percentage of treated water. Many plants are already a few decades old and do not incorporate the latest developments in technique, e.g. inlet design. Sedimentation tank performance is strongly influenced by hydrodynamic and physical effects such as density driven flow, gravity sedimentation, flocculation and thickening. In turn the velocity and density patterns in tanks influence these processes and are therefore of great interest to design engineers.

The primary performance indicator for sedimentation tanks is the fraction of the solids present in the raw water removed by the sedimentation step. However, the absolute percentage is not sufficient – an overdesigned tank will remove a large percentage without being really efficient. Therefore it is necessary to qualify it in terms of the design capacity. It must also be qualified in terms of the smallest size of floc particles that will still be removed by it.

Computational Fluid Dynamics (CFD) is the analysis of systems involving fluid flow (gases or liquids) by means of computer-based simulation. It is a research tool and a design tool and it is complementary to theory and experiments. CFD can also be described as a method to investigate and simulate fluid flow by means of iterative calculations on computers. It was developed originally to study aerodynamics, but has since been applied to many different types of flow under a great variety of conditions. A recent application (since ca 1990) is to use it for the simulation of unit processes in water treatment, e.g. chlorine and ozone contactors, sedimentation tanks and sludge thickeners.

The basic technique utilized in CFD is to divide the region or component in or through which the flow is to be investigated (the flow domain) into a grid of thousands of small blocks. Boundary types and conditions for the domain (inflow, outflow, pressure, symmetry), values for inflow (flow rate, concentrations, temperature) and physical values for the fluid (density, viscosity) are specified. The computer then calculates values for velocity components, pressure, momentum, energy and concentrations for each cell iteratively until the values converge for a steady state solution, or for a required number of iterations that yield sufficient accuracy for nonsteady flow cases. The results can then be visualized by displaying it graphically by looking at views (slices) through the flow domain, for example as velocity vectors, pressure contours or particle tracks. It can also be imported into applications like spreadsheets for further mathematical analysis.

The application of CFD to the simulation of sedimentation tanks is fairly recent because it involves modelling of the movement of solid particles in water, i.e. twophase flow. This increases computer memory and processor speed needs. With the advent of PC's with processor speeds in the Giga-Herz range and Gigabytes of RAM it has become feasible and has put CFD as a tool within the scope of standard consulting practice in the design and operation of water purification and treatment works.

RESEARCH OBJECTIVES

- Evaluate the suitability of CFD as a technique for design and research of rectangular sedimentation tanks.
- Design CFD models for simulation of sedimentation tanks, i.e. grids and numerical descriptions.
- Validate the models with experimental data.
- Use CFD to investigate the effects of design parameters and operational parameters.
- Make recommendations for improved design and operation of sedimentation tanks.

The study was conducted in three stages. First a model was developed to compare with results published for a clarifier at a sewage treatment plant at Jönköping, Lund, Sweden. This clarifier is well studied and often used for benchmarking. It also enabled the research workers to become proficient with the Flo++ CFD package that was used for this project, as well as with the technique of developing a suitable grid for the model, which is the most time-consuming aspect of CFD modelling.

This was followed by experiments with a physical, small-scale (100 l) perspex model of a rectangular sedimentation tank. (Hereafter it will be referred to as the labtank.) It was used to provide information for validation of CFD models of it. In order to obtain realistic experimental results, it was necessary to test a number of substances for the simulation of the floc particles. Finally ground and sieved crystalline polystyrene was used.

Finally, a CFD model was developed to simulate the full scale rectangular sedimentation tanks at the Midvaal purification works near Orkney.

The CFD simulations of the laboratory model and the Midvaal tanks were done by setting up standard cases for each, *i.e.* a configuration and operating conditions that represented the physical tanks as they were built, and then varying different aspects of the configuration or operating conditions one or two at a time to determine the effect. Only type 1-settling (discrete particles in dilute suspension) was simulated, as it is the applicable type for the operating conditions in rectangular sedimentation tanks foir potable water treatment.

CONCLUSIONS

From the results of the various investigations and simulations the following conclusions can be made:

CFD has reached a level of development where it has become a useful tool for research, design and operation in water technology. It is now possible to use standard software packages and desktop computers to analyse and investigate fairly complex situations in terms of configurations and conditions within reasonable time periods and at lower cost than doing experimental work. Very realistic models that mimic nature in such a way that the behaviour of the models cannot be distinguished from that of the physical systems they are modelling can be developed. Such models can

therefore be used with confidence to investigate the characteristics and behaviour of the physical systems, including the effects of changes in or to it.

The realism of the CFD models developed in this project did lead to valuable insight into the real dynamics of the tanks. For example, it was observed that the systems studied were behaving non-linearly (i.e. chaotically) at higher temperatures in both the numerical CFD and the physical laboratory models. In another case removing some sumps in a model of the Midvaal tanks showed that a configuration modification that was intuitively expected to simplify the dynamics of the tank, would instead cause it to become significantly less stable and robust and thus susceptible to changes in operational conditions.

The results of the physical experiments and CFD simulations done showed that the presence of the solid phase, *i.e.* floc particles, in the water affected the dynamics of sedimentation tanks. Therefore two-phase simulation will normally be required to obtain realistic information for design and operational purposes. To do two-phase modelling the solid phase can be simulated as either another fluid (*i.e.* sludge) completely mixed with the water entering the tank, known as the scalar approach, or as discrete solid particles. Both approaches have their own advantages and disadvantages. Discrete particle tracking yields more information about the interaction of the water and the particles and insight into the development of the flow patterns than the scalar approach. It is computationally much more intensive and time consuming and is unnecessary if the modeller is not particularly interested in the particle trajectories. It is also almost impossible to use it to calculate solids concentration in the outflow, since one needs a very exact particle size distribution to do so. The scalar approach can do this much easier.

The complexity of the computational grid of cells used for the CFD simulations always requires a that a balance be struck between the resolution required and the computational requirements. In order to obtain mesh-independent solutions that are realistic simulations of the physical reality the grid must be as detailed and as fine as possible, but to optimise time frames and hardware requirements as few cells as possible must be used. Detailed grids must be developed for inlets and outlets, i.e. the

grid must be refined in these areas. The same should be done for any area with large variations in velocity, e.g. vortices.

It was found that the physical characteristics of the flocs (size, shape, density, concentration and composition) are not such significant parameters in the operation and performance of sedimentation tanks used for potable water treatment as originally assumed, due to the much lower solids concentrations and greater particle size distributions than those encountered in sewage water treatment.

In order to obtain information on floc particle composition, solids present in the Mooi River in Potchefstroom was flocculated by means of conventional jar tests with different metal hydroxides as coagulants. The flocs formed were observed under a microscope to determine their composition in terms of the different types of solid material suspended in the water. All the flocs observed consisted of a random mixture of the mineral particles, bacteria, algae and diatoms present in the water. The extent of this work is too little to make any definite or final conclusions, but it does show that floc composition will tend to be very variable, depending on seasonal and environmental conditions. This implies that when designing actual tanks, a large variation in floc particle composition (and therefore size, density and shape) must be considered - at least the possible extremes and typical values of mineral and microbial content. More research into this aspect is recommended.

A set of criteria were compiled for suitable floc simulants in physical experiments and a number of substances were tested viz. synthetically prepared flocs, sago grains, maize meal and, finally, ground polystyrene granules. The latter was found to be ideal in all respects and is recommended to any future researchers.

The ability of sedimentation the tanks to clarify water by letting suspended solids settle out as flocculated particles depends on two aspects: (a) The water flow pattern through the tank, which in turn is determined by the configuration of the tank and by operational parameters (solids concentration, water flow rate and temperature). (b) The settling characteristics of the particles as determined by their shape, size and interaction with the water through drag and buoyancy forces. Water flow patterns dominated particle settling in determining the dynamics and efficiency of rectangular

sedimentation tanks experiments and imulations done in the project. It was also observed that the water flow patterns were remarkably stable and robust for any particular configuration. With regard to the water flow pattern through tanks it was found that the most important aspect was the design of the inlets, especially their placement.

The experiments and CFD simulations showed that baffles were not necressarily effective to dissipate kinetic energy and prevent short-circuiting between inlets and outlets. Putting baffles at the inlets did not affect flow patterns or particle trajectories through a tank as a whole or at the outlets specifically. Neither did it affect particle settling patterns on the bottom of the tank. The baffles did have a significant effect on the vortices and particle trajectories directly at the inlets. The vortices were split up into more vortices and the density waterfalls became steeper between the baffles and the inlets. This can have an effect on the flocculation and settling of real floc particles in practice because more vortices and higher velocity gradients at the inlets were observed to capture more of the floc particles and recirculate them. This effectively forms a sludge bed or blanket in inlet zone. Further research into this aspect is recommended.

The hydrodynamic parameters were much less important than expected. (*I.e.* the water flow rate and velocity components in the different axial directions.) The position and relative velocities of the inlet vortices changed to an extent, but the basic flow patterns remained the same. Temperature, which affects water viscosity and density, did not affect the flow patterns, but did affect the stability of the flow and the settling behaviour of the particles.

Outlet configuration is generally assumed to influence the flow significantly due to short-circuiting, but was found to have minor effects on flow patterns and particle trajectories. Only minor differences were observed in flow patterns and particle trajectories in the inlet and settling zones, whether the outlets were in the form of systems of distributed outlet channels, or overflow weirs at the back of the tank, or even merely a holes in the back walls of the tanks.

The presence of the settling particles themselves also affected the flow dynamics. Due to their interaction with the water through drag and buoyancy, they cause the development of density waterfalls in inlet zones. The extent to which this happened depended on the concentration of the particles. Without particles in the inflow streams the water flow patterns in the simulations and labtank model experiments invariably developed direct shortcuts between the inlets and the outlets. With particles, the flow patterns were more complex because the particle-induced density waterfalls interacted with the inlet vortices.

As far as floc particle settling behaviour is concerned, the water flow patterns in the tank had the most significant effect. This mostly stems from the development of density waterfalls.

Water temperature did affect tank efficiency in the physical labtank to an extent which is usually not considered in the design of the tanks. The increased viscosity and density of water at the lower end in the normal operational temperature range of 10 to 25 °C caused significantly larger fractions and sizes of the particles to be washed out of the tank via the outlets. The Flo++ models of the labtank and Midvaal tank also showed an effect on the particles, but very much less so than in the case of the physical tank. However, the water temperature did have a significant effect on the numerical behaviour of the models. At the lower temperatures $(4 - 15 \, ^{\circ}\text{C})$ the models converged much more quickly than at the higher ones $(15 - 25 \, ^{\circ}\text{C})$. In the case of the labtank, the CFD models ususally did not converge at all at the higher temperatures, but could only be solved as unsteady cases. Again, it is an open question for further research whether this instability is due to numerical or physical reasons, or both.

The conventional approach for the design of sedimentation tanks is still valuable as a tool for the first stage of tank design to obtain general conceptual information, but such a design should then be subjected to a rigorous CFD analysis of the its configuration and of the effect of changes in operational parameters. Apart from the obvious benefits of proper understanding of tank dynamics and optimization of design, overdesign is wasteful and can be detrimental, because clarified water in the vicinity of the outlets is an ideal medium for algal regrowth.

Another aspect for further research to come out of this project is what contribution the high-velocity vortices in the inlet zones make to effective clarification in practice. They trap particles, effectively forming sludge blankets where the particles can flocculate further, or simply recirculate until they move into a lower velocity region from which they can settle out. Could this effect be enhanced by increasing the intensity of the density waterfall?