THESIS FOR THE DEGREE OF LICENTIATE OF ENGINEERING

Computational Fluid Dynamics for Detention Tanks

Simulation of Flow Pattern and Sedimentation

ÅSA ADAMSSON

Department of Hydraulics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 1999

Computational Fluid Dynamics for Detention Tanks
Simulation of Flow Pattern and Sedimentation
ÅSA ADAMSSON

ISSN 0348-1050
Report Series A:31

Department of Hydraulics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone + 46 (0)31 772 1000

Chalmers reproservice
Göteborg, Sweden 1999

Computational Fluid Dynamics for Detention Tanks
Simulation of Flow Pattern and Sedimentation
ÅSA ADAMSSON
Department of Hydraulics
Chalmers University of Technology

ABSTRACT

The performance of ponds, wetlands, storage tanks and sedimentation tanks is dependent on the flow conditions. The increase in more efficient and sustainable treatment processes to protect receiving waters from metals and nutrients demands better design and understanding of what happens in these structures. The tradition in sanitary engineering is to use simplified methods and rules of thumb to construct wetlands and ponds. One-dimensional simulations give sufficient accuracy for pipe flow, but in more complex structures the flow pattern becomes more three-dimensional. The rapid increase in computer capacity and more user-friendly, commercially available numerical simulation tools make it possible to simulate the flow in two and three dimensions, which was not possible 15 years ago. The aim of this thesis is to study the applicability and possibilities of using Computational Fluid Dynamics (CFD) in sanitary engineering, with special attention to three-dimensional simulations of detention tanks.

Two studies have been carried out to study the applicability of CFD simulations as a design tool in sanitary engineering. The first study deals with the flow pattern in a detention tank and the second with sedimentation in a storage tank. In the first study the flow pattern and residence time in a large-scale model (12.9 x 8.9 x 1 m) of a detention tank are studied. The flow pattern is measured using drogue tracking and a motion analysis system. The residence time is measured be instantaneous tracer injection. The measurements are compared with both two-dimensional depth-integrated and three-dimensional computer simulations. The agreement is good between measurements and simulated flow pattern and residence time distribution curves for high inflow velocities. For the lowest flow rate it was difficult both to measure and to simulate the flow pattern.

The behaviour of particles at boundaries is difficult to model. In the second study a boundary condition for particles based on the bed shear stress on the tank bottom is studied. Both the sedimentation efficiency and the spatial distribution of the sediment are studied. The simulations using particle tracking are compared with measurements from a small-scale model (2 x 0.97 x 0.196 m) of a storage tank. The results from the new modelling approach are promising especially for high flow velocities.

The thesis also includes a literature review where CFD simulations in various sanitary engineering problems are presented.

The project shows the possibilities and potential but also the limitations of using numerical simulations in two (depth-integrated) and three dimensions in sanitary engineering. However, the turbulence modelling for low flow velocities and more comparative studies between two-dimensional depth-integrated and three-dimensional simulations and measurements are areas where further research is of interest.

Keywords: CFD, numerical simulation, sanitary engineering, detention tank, flow pattern, tracer study, sedimentation, bed shear stress, storage tank.

SAMMANFATTNING

Hur väl dammar, våtmarker, utjämningsmagasin och sedimenteringstankar fungerar är bland annat beroende av flödesförhållandena. För att skydda våra vattendrag från metaller och näringsämnen ställs ökande krav på effektivare rening och hållbarare reningsmetoder för dag- och spillvatten. Detta innebär krav på bättre utformning och till följd av detta krävs bättre förståelse och kunskap om vad som händer i denna typ av anläggningar. Traditionellt inom VA-tekniken används förenklade metoder och tumregler vid utformningen av våtmarker och dammar. Endimensionella beräkningar ger tillräcklig noggrannhet för t. ex. flöden i rör, men i mer komplexa strukturer blir flödesmönstret mer och mer tredimensionellt. Den snabba utvecklingen vad gäller datorkapacitet och användarvänliga programvaror gör det numera möjligt att simulera flöde i både två och tre dimensioner, vilket inte var möjligt för 15 år sedan. Syftet med den här studien är att studera tillämpbarheten och möjligheterna med datorsimulering av strömning (computational fluid dynamics) inom VA-tekniken, med fokus på tredimensionella simuleringar i utjämningsmagasin.

Två studier har utförts för att studera tillämpbarheten av datorsimuleringar som ett verktyg för utformning och konstruktion inom VA-tekniken. I den första studien studerades flödesmönstret i ett avsättningsmagasin och i den andra studien sedimentering i ett utjämningsmagasin. I den första studien studerades uppehållstid och flödesmönster i en storskalemodell (12.9 x 8.9 x 1 m) av ett avsättningsmagasin. Flödesmönstret mättes med strömkors och analyserades med ett rörelsemätningssystem. Även spårämnesförsök utfördes. Mätningarna jämfördes med både tvådimensionella djupintegrerade och tredimensionella simuleringar. Överensstämmelsen mellan mätta och simulerade flödesmönster och uppehållstider är god för höga flöden. För det lägsta flödet var det svårt både att göra mätningar och simuleringar.

Partiklars beteende nära väggar är svårt att modellera. I den andra studien har ett randvillkor för partiklar baserat på skjuvspännningen vid magasinsbotten studerats. Både avskiljning och utbredning av sediment på botten studerades. Simuleringar som utfördes med en partikelspårningsmetod har jämförts med mätningar i en skalmodell (2 x 0.97 x 0.196 m) av ett utjämningsmagasin. Det nya randvillkoret ger lovande resultat, speciellt när det gäller höga flödeshastigheter.

Uppsatsen innehåller också en litteratursammanställning av två- och tredimensionella simuleringar av olika VA-tekniska problem.

Studien visar på möjligheterna och fördelarna men också begränsningarna med både tvådimensionella djupintegrerade och tredimensionella simuleringar inom VA-tekniken. Områden där vidare forskning bör ske är främst inom turbulensmodellering för strömning som består av regioner med både låga och höga hastigheter. Vidare behövs fler jämförande studier av tvådimensionella djupintegrerade och tredimensionella simuleringar för att studera tillämpbarheten av tvådimensionella djupintegrerade modeller.

LIST OF APPENDED PAPERS

This thesis is based on the work in the following papers:

I **Flow Pattern in a Rectangular Detention Tank**
Three-Dimensional Simulation and Measurement
To be submitted to Urban Water.

II **Numerical Simulation and Large-Scale Physical Modelling of Flow in a Detention Basin**
Åsa Adamsson, Jesper Persson and Sven Lyngfelt (1999). In: *Proc. 8th Int. Conf. Urban Storm Drainage*, Sydney, Australia, 30 August – 3 September 1999, Vol. 3, 1175-1183.

III **Modelling of Particle Deposition using a Bed Shear Stress Boundary Condition**
To be submitted to a scientific journal (Co-author: V. Stovin).

ACKNOWLEDGEMENTS

This research has been carried out at the Department of Hydraulics, Chalmers University of Technology. The project was financially supported by the Swedish Council for Building Research (BFR), which is gratefully acknowledged. I would also thank Adlerbertska forskningsfonden, Anna Ahrenbergs fond, the Swedish Institute and Åke och Greta Lissheds stiftelse for extra financial support for my time abroad.

I would like to thank my supervisors at the Department of Hydraulics, Professor Lars Bergdahl and Ass. Prof. Sven Lyngfelt. Among my colleagues at the department I would like to express my special thanks to Jens-Uwe Friemann, Ph.D. student, and Dr. Jesper Persson for valuable discussions and help with laboratory measurements. I would also like to thank all my colleagues at the Departments of Hydraulics and Sanitary Engineering, who have been involved in my work in one way or another, helping me with computers and laboratory measurements, with fruitful discussions or just encouragement. Special thanks go to my fellow Ph.D. students for their friendship and for having shared their experiences of being a Ph.D. student.

I have had the opportunity to spend three months at the Department of Civil and Structural Engineering at the University of Sheffield, United Kingdom. I would like to thank Professor Adrian Saul and Dr. Virginia Stovin for making the stay possible. I am most grateful to Dr. Virginia Stovin for having shared her knowledge of CFD simulations, for interesting discussions, her enthusiasm and also for the very warm welcome I got in Sheffield. Ass. Prof. Gustavo Perrusquia is also to be thanked for arranging the contacts with the University of Sheffield.

I would also like to thank Johan Lennblad at Caran AB for valuable discussions and for helping me getting started with the CFD simulations.

Finally, I would like to take this opportunity to thank my parents, my brother and my closest friends for their never-ending understanding and all their encouragement.

Göteborg, November 1999

Åsa Adamsson

CONTENTS

1. BACKGROUND

The interest in environmental problems has increased in recent years. People have become more concerned about the increased impact of pollutants, for instance on our lakes and seas and in the ground. The population on Earth is increasing. There are now over six billion people, and the reservoirs of clean water are limited in many areas. We have to economise on water, and there is a tendency towards short circulation of water and more efficient and sustainable treatment. Detention ponds and wetlands are examples of structures that are more and more used in the ambition to achieve a sustainable sewer system and improved treatment of sewage and stormwater to protect receiving waters from metals and nutrients.

The goal of more efficient and sustainable treatment processes demands better design and understanding of what happens, for example, in a detention pond or a settling tank at the wastewater treatment plant. In such structures the flow pattern plays a major role. The tradition in sanitary engineering is to use simplified methods and rules of thumb to construct wetlands and ponds. One-dimensional simulations are sufficiently accurate for pipe flow, but in more complex structures the flow pattern becomes more three-dimensional. Rodi (1995) comments on one-dimensional calculations: 'these formulae can describe only the simplest phenomena of interest and are not suitable for complex geometries'. The rapid increase in computer capacity and more user-friendly, commercially available numerical simulation tools make it possible to simulate the flow in two and three dimensions, which was not possible 15 years ago.

1.1 Aim and contents of the study

The aim of the study is to highlight the use, the benefit, the applicability and the possibilities of Computational Fluid Dynamics (CFD) in sanitary engineering with special focus on detention tanks. Shilton *et al.* (1999) write: 'We need only to look at other industries to see how CFD can be used reliably for complex design. It is a powerful tool, whose potential has been practically untapped by the water industry. As awareness of this grows there can be little doubt that we shall be seeing much more of CFD in the future.' The aim is also to compare CFD simulations in three dimensions (3-D) with measurements to find limitations in the modelling approach. The purpose is not to develop guidelines on how different sanitary structures should be designed.

This thesis consists of three appended papers and an introductory part divided into four chapters. Chapter 1 gives some background to the study. Chapter 2 includes a brief literature review of computational fluid dynamics applications in sanitary engineering. The literature review starts with an introduction (Section 2.1) and a brief presentation of the theory and some numerical codes (Section 2.2). In Section 2.3 the literature treated is presented. The literature is divided according to application into ponds, storage tanks, reservoirs, Combined Sewer Overflow (CSO) structures and miscellaneous applications. In Chapter 3 the appended papers are briefly presented. The introductory part ends with conclusions, discussion and some suggestions for further work in Chapter 4.

To investigate the possibilities, potentials and limitations of the use of CFD in sanitary engineering more studies must be performed. Two different studies have been carried out in this thesis. The contents of these studies are presented in the appended papers. The principal aim of both the studies is to study the applicability of CFD in sanitary engineering. The first study is a comparison between measurements in a large-scale model of a detention tank and 3-D numerical simulations (Paper I). The second study focuses on the modelling of sedimentation in storage tanks and involves simulation of particle deposition. The emphasis of the second study is on the boundary condition on the bottom of the storage tank (Paper III). The measurements in the large-scale model and the 3-D simulations (Paper I) have also been compared with 2.5-D (depth-integrated) simulations. The results from this comparison are presented in Paper II. This paper was not written by the author alone. The author has written the sections 'Test basin and measuring program' and '3-D simulations', as well as taking part in the writing of 'Discussion' and 'Conclusions'.

2. COMPUTATIONAL FLUID DYNAMICS APPLICATIONS IN SANITARY ENGINEERING – A LITERATURE REVIEW

2.1 Introduction

The basis of all water treatment, irrespective of whether it is sedimentation of flocs in a settling basin at a wastewater treatment plant, an oil separator or a detention pond for stormwater, is to understand the hydraulics. How is it possible to achieve a better comprehension of the hydraulics? One way involves studies of full-scale models or scale models; another is to use computers to simulate the flow. Due to the increase in powerful computers and more user-friendly commercial codes it is now possible to solve complex fluid flow problems. The space industry was one of the early users of Computational Fluid Dynamics (CFD) models. However, CFD is now more widespread and also used in other areas. One step in the development towards a more sustainable sewer system is to study how such models can be adapted as an engineering tool for hydraulic problems.

This literature review is a summary of some of the literature where CFD has been applied to sanitary engineering problems. The following questions can be seen as a basis for the literature review. Is CFD a design tool in sanitary engineering? What has been done? How does it work? What are the limitations? The review makes no claim to be complete, but is believed to give an overview of the possibilities of using CFD simulations in sanitary engineering. Different types of problems and the way CFD can be utilised to study and in some cases solve these problems are presented. The literature review can be seen as a smorgasbord of diverse problems in sanitary engineering where rules of thumb and one-dimensional (1-D) models are not enough to give satisfactory results. Only literature in English and Swedish has been covered.

2.2 A brief overview of computational fluid dynamics

2.2.1 CFD background

Computational fluid dynamics is the science of using computers to solve the fluid flow equations. Versteeg and Malalasekera (1995) define CFD as 'the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation'. Another definition by Olsen (1997) is 'CFD is the science about calculation of fluid flow and related variables using a computer'. The aerospace industry started to use CFD as a design tool in the 1960s. In the 1980s the solution of fluid flow problems was something for researchers and specialists. During the last few years there has been an enormous increase in computer capacity and availability of workstations as well as the development of more user-friendly CFD codes, which nowadays enable the use of CFD by engineers for research and design tasks. However, the user of the code still needs fluid-dynamics and numerical skill.

Almost everything around us involves fluid flow or depends in one way or another on fluid flow: the air we breathe, the blood flow in our body, the water flow in the seas, rivers or in the bathtub, the weather, aerodynamics of aircraft and cars, turbomachinery, distribution of pollutants and effluents. The list could be made very long. Consequently there are several different areas where the use of CFD is applicable. The use of CFD as a design tool in sanitary engineering is fairly new. The earliest references represented in this literature review are from the late 1980s. However, the technique was at that time mostly used as a research tool. Shilton *et al.* (1999) write: 'In many industries CFD has become an essential design tool, but to date its use in the water industry has been noticeably lacking.' They continue: 'The water industry may currently consider that CFD gives a technical overkill compared with the more traditional empirical "black box" techniques with which we are familiar. However, in many cases CFD can actually offer the more appropriate technology for problems we are trying to solve. The reason that CFD might seem too technically complex is perhaps because we perceive the water industry as too "low tech".' They conclude that: 'CFD is a powerful tool, whose potential has been practically untapped by the water industry. As awareness of this grows, there can be little doubt that we shall be seeing much more of CFD in the future.'

2.2.2 Basic theory, turbulence models and sedimentation modelling

This chapter gives an overview of the basic theory and turbulence models in CFD.

Equation of motion

The governing principles of fluid flow are that the mass of fluid is conserved (continuity equation), that the rate of change of momentum equals the sum of the forces acting on a fluid particle (Newton's second law) and that the total energy is conserved. These fundamental principles can be expressed by a set of partial differential equations, the continuity equation and the Navier-Stokes equations (Rodi, 1984). The Navier-Stokes equations were discovered independently by the French engineer Claude Navier (1785-1836) and the Irish mathematician George Stokes (1819-1903).

Turbulence modelling

Most flows of practical interest are turbulent. Turbulence in hydraulics is important because it contributes to the transport of momentum, heat and mass, and therefore influences the velocity, temperature and concentration in the water (Rodi, 1995). In lakes and reservoirs, wind shear and solar radiation are important factors. The wind shear generates turbulence, while solar radiation causes a stable stratification that inhibits the turbulent mixing between the surface layer and the lower layers. Another area in hydraulics, where turbulence is important, is in sedimentation tanks. Rodi (1995) writes: 'The sedimentation efficiency depends strongly on the flow developing in the tank, which is often far from the assumed plug flow, and on the turbulence which counteracts the settling processes.'

'Turbulence models consist of sets of differential equations, and associated algebraic equations and constants, the solutions of which, in conjunction with those of the Navier-Stokes equations, closely simulate the behaviour of the real turbulent fluids. A good turbulence model has extensive universality, and is not too complex to develop or use' (Launder and Spalding, 1974). The complexity of turbulence has, until very recently, made it impossible to solve these equations. Due to advances in computer capacity it is now possible, for low Reynolds numbers, to use Direct Numerical Simulations (DNS) to solve the Navier-Stokes equations exactly. However, the most common alternative is to consider the pattern of the mean turbulent motions. This means that all solution variables in the original Navier-Stokes equations are decomposed into a mean and a fluctuating part. This approach gives the time-averaged continuity equations and the time-averaged Navier-Stokes equations, also called Reynolds time-Averaged Navier-

Stokes (RANS) equations and Reynolds equations. By doing this, a new term in the time-averaged Navier-Stokes equations is created, namely the Reynolds stress tensor, which represents the turbulence. This term needs to be modelled in order to close the equations. Up until today, this approach has been the most common in sanitary engineering. Rodi (1995) writes about turbulence modelling in hydraulics: 'for practical calculations, the Reynolds-average-modelling approach is presently the most advanced and most suitable.'

There are several turbulence models. Versteeg and Malalasekera (1995) divide the models into two main groups: classical models and Large Eddy Simulations (LES). The classical models can further be divided into the following groups, depending on level of approximation (Versteeg and Malalasekera, 1995): zero-equation models, two-equation models, Reynolds Stress Models (RSM) and Algebraic Stress Models (ASM). Also one-equation models belong to this group. The classical models are based on the time-averaged Navier-Stokes equations. The first two models (zero-equation models and two-equation models) are also based on the assumption made by Boussinesq that the Reynolds stresses are linked to the velocity gradients via a turbulent viscosity equation. In the zero-equation models and two-equation models an expression for the turbulent viscosity is needed.

By using dimensional analysis it is possible to express the kinematic turbulent viscosity as a product of a turbulent velocity and a length scale. In zero-equation models, the velocity gradient is used as a velocity scale, and some physical length is used as the length scale. The most common zero-equation model is called the Prandtl's mixing-length model. The advantages of this model are that it is easy to implement and cheap in terms of computing resources. One of the disadvantages is its incapability of describing flow separation and flow with recirculation (Versteeg and Malalasekera, 1995).

The standard two-equation k-ε turbulence model has two model equations: one for k, kinetic energy, and one for ε, eddy dissipation. The turbulent viscosity for the k-ε model is formulated as a coefficient times the ratio of the squared kinetic energy divided by the eddy dissipation. Versteeg and Malalasekera (1995) summarise some of the advantages and disadvantages of the k-ε model. The advantages are that it is the most widely used and validated turbulence model, is simple and performs excellently for many industrial flows. The disadvantages are that the model is more expensive than the mixing-length model and that it performs poorly for e.g. unconfined flows, swirling flows and rotating flows.

Almost all practical calculations in hydraulics use eddy-viscosity models. The mixing-length models and the k-ε model, described above, are the most widely used turbulence models in sanitary engineering. However, some other models have also been used, for example, constant eddy viscosity models, the RSM model and the Renormalization Group (RNG) k-ε turbulence model. The RNG k-ε model accounts for low-Reynolds-number effects. Examples of flows where the RNG model is applicable are separated flows, recirculating flows and low-Reynolds-number flows, e.g. where the flow is turbulent in regions with limited extent and otherwise laminar (Fluent, 1997). The Reynolds Stress Model (RSM) is the most complex classical turbulence model. In RSM the Boussinesq assumption is not used. Instead a partial differential equation for the stress tensor is derived from the Navier-Stokes equation. Davidson (1997) discusses eddy viscosity models versus Reynolds Stress Models, writing: 'Whenever non-isotropic effects are important we should consider using RSMs. Examples where non-isotropic effects often are important are flows with strong curvature, swirling flows, flows with strong acceleration/retardation.' The main disadvantages of the RSM model are its large computing cost, complexity and numerical instability.

In Section 2.3.6 some opinions from the authors of the different papers, as regards turbulence modelling, are presented. The other turbulence models named above, the Algebraic Stress Models (ASM) and Large Eddy Simulations (LES), have not yet been used to a wide extent in sanitary engineering and are therefore not discussed any further here.

Sedimentation modelling

Sedimentation is important in sanitary engineering. Almost all flow involves solid transport in one way or another. For storage tanks it is important to prevent settling of particles; for sedimentation tanks at wastewater treatment plants and CSO structures it is the opposite. Here the main goal is to achieve a flow that enables settling. Modelling of multiphase flow is therefore an important task in sanitary engineering problems.

In CFD there are mainly two different approaches to calculating multiphase flow: the Eulerian and the Lagrangian approaches. In Eulerian models all phases are calculated as continua using Navier-Stokes equations. The Lagrangian models compute momentum, heat and mass transfer between particles and the continuous phase, and stochastic models account for turbulent particle dispersion effects, while the carrier phase is calculated as continua by the Eulerian approach. There are many other numerical approaches used for sediment transport, settling and erosion. However, such models are not discussed here. Some of the papers using different approaches for sedimentation modelling are discussed in Section 2.3.6.

2.2.3 CFD codes

There are mainly three numerical solution techniques to solve the governing equations: the Finite Difference Method (FDM), the Finite Element Method (FEM) and the spectral method (Versteeg and Malalasekera, 1995). The main difference between the models is in which way the different flow variables are approximated. Most CFD codes are based on the Finite Volume Method (FVM). This method was originally developed from the finite difference method.

There are several different commercially available codes to simulate flow, ranging from 1-D models for closed conduits or open channel flow to 3-D models with advanced turbulence modelling. Mouse and HydroWorks for sewage water and Licwater for potable water are examples of models especially used in hydraulics for 1-D modelling of pipe flow. Most of these models have been refined and can now offer simulations in both 2-D, 2.5-D (depth-integrated) and 3-D. SSIIM is a freeware code used for hydraulic/river/sedimentation engineering. TELEMAC, Mike3D, AquaDyn and Delft3D are other codes mainly used for rivers, lakes, reservoirs and coastal and ocean waters. Examples of CFD codes applicable in a wide range of fluid flow problems, often with several options concerning turbulence models, are: CFX, FLUENT, PHOENICS, Star-CD and FLOW-3D. These codes are all finite volume programs. An example of a finite element program is FIDAP. More information about the codes can be found on the Internet.

2.3 Applications in sanitary engineering

To simulate flow with computers is not new. Computer models in 1-D have been used by engineers since the late 1970s. The 1-D models are sufficiently accurate as long as the flow is one-dimensional like that in pipes but as soon as the flow pattern is dominated by flow in more than one direction, such models are too simple. Examples of structures where the flow pattern is no longer one-dimensional are ponds, sedimentation tanks, reservoirs and oil separators. These are also the types of structures that are presented in most papers discussed in this section.

The literature review is primarily focused on simulations performed by commercially available software. The emphasis is put on the way the author of the paper has approached the hydraulic problem, for instance in setting up the problem, selecting boundary conditions and turbulence model, and validating the simulations. A compilation of the literature studied is given in the Appendix.

2.3.1 Ponds

Ponds (sometimes called surface water wetlands) in this study are characterised by a large water body with a complex geometry. The ponds may be constructed, real ponds or small lakes. They are mainly used to remove suspended solids in stormwater or as the last step in the treatment of sewage water, especially to reduce nitrogen, at wastewater treatment plants. It is therefore important to achieve a flow that improves the possibilities for settling as well as minimises the risk of erosion of already settled material. The simulation of flow in ponds is complicated since the geometry is often very complex and not well known and the boundary conditions are difficult to estimate. In most ponds the bottom consists of clay or stone and in some cases concrete. The sidewalls are often covered with grass and reed. The surface is a free boundary.

Persson (1999a) used a 2.5-D model to simulate a vegetated pond in a wetland system consisting of three different ponds. The volume of the studied pond was 24 300 m^3. The depth varied between 2 and 0.5 m. Time-dependent simulations were performed with a varying flow rate. The simulations were compared with tracer studies, comparing simulated and measured Residence Time Distribution (RTD) curves. Persson highlighted some difficulties with simulations and measurements in vegetated ponds. The major difficulty with the simulations was the roughness of the boundaries. Persson used the Mannings number to calibrate the numerical model from the measurements. The roughness in the region with dense vegetation was set to n=2. In the other regions, n was set to 0.1 and, where the water depth was over 1 m, to 0.03. The main conclusion drawn is that it is possible to simulate the flow in a pond with a 2.5-D model, but extensive measurements and experience are needed to verify and calibrate the model.

Benelmouffok and Yu (1989) developed a vertically averaged 2-D model (2.5-D model) to simulate water movements and pollutant transport. With the developed model it was also possible to simulate the level variations of the free surface. They used the model to study pollutant removal performance of an urban stormwater detention pond. The surface area of the pond was 6 800 m^2 and the average depth 2.3 m. The measurements in the pond included both the dissolved and particulate part of the pollutants. The simulations were divided into two steps. The first step was to simulate the transport and dispersion without deposition. The pollutants were assumed to follow the water pattern. The agreement was good for NO_3 and NH_3 and ortho-phosphate, which are known to be present mainly in the dissolved phase. Large deviations between simulated and observed values were noted for Total Suspended Solids (TSS), Total Phosphorus (TP) and Total Kjeldahl Nitrogen (TKN). The second step is to simulate the particulate pollutants. In this simulation the deposition term must be included in the transport equation. No measurements of particle distribution were performed and, therefore, a 75-15-10 particle size distribution was assumed. This corresponds to a material where 75% of the particles are less than one micron, 15% have a size between 1 and 15 microns and 10% have a size over 75 microns. The settling velocities were used to calibrate the model. This method gave better agreement for the suspended particles.

Pettersson *et al.* (1998) performed simulations and measurements of flow pattern in a detention pond for stormwater. The detention pond has a surface area of 6 200 m^3. The pond depth varies between 0.5 to 1.6 m. The pond has a longish shape with a narrow and shallow section in the middle. The free surface was modelled as a "rigid lid". The equivalent sand roughness on the bottom was set to 91 mm for macadam, 3.3 mm for concrete and 2.5 mm for clay. The simulated flow patterns were compared with drogue tracking measurements of the flow pattern. Simulations were performed in both 2-D and 3-D. A 2-D model implies that the water depth is infinite. The standard k-ε turbulence model was used with standard wall functions. Conclusions drawn from the study are that the flow pattern in the studied pond is similar for all inflow intensities. Pettersson *et al.* (1998) also emphasise that it is important to pay attention to the topography of the bottom. They showed that the 2-D model was not able to simulate the correct flow pattern in the pond.

In all studies presented above the influence of wind on the surface has been neglected. Shaw *et al.* (1997) presented a study where the effect of wind on the flow pattern was examined. The pond is elliptic and is approximately 90 m long and 60 m wide. The average depth is 1.2 m. Velocity measurements were performed and the average wind speed and direction were recorded. The simulations were performed in three dimensions and the turbulence was described by the k-ε model. However, the simulations were performed without including the wind stress. Shaw *et al.* (1997) concluded that the flow pattern in the pond was very complex, mainly due to the wind stress on the surface. They found that 'With relatively minimal wind stress, the velocity field in the pond is determined by inflow momentum, shaped by pond bathymetry, and results in circulation in the horizontal plane.' This was also the result obtained from the computer simulations. The flow pattern described above also occurs during high flow events when the effect of the wind is secondary.

Pettersson (1997) studied sedimentation in a stormwater pond with a surface area of 530 m^3. He used particle tracking techniques to simulate the particle deposition. For the simulations four different particle sizes were used: 1.5, 10, 20, 40 μm. The density of the particles was 1300 kg/m^3. The simulated results were compared with measured values of the sedimentation efficiency in the pond. The agreement was good.

2.3.2 Storage tanks

Storage tanks mostly have a more simple geometry than ponds, but this is not always an advantage. A geometry where the flow is governed by the geometry is often easier to simulate once the mesh has been built. The examples described below have, in many respects, a simpler geometry and more well-defined boundary conditions.

Rectangular tanks
Waste stabilisation ponds are a popular way to treat wastewater from small towns and many industries, especially the meat industry, in Australia. A survey showed that 74% of the 39 municipal wastewater treatment pond systems had hydrodynamic problems. Wood *et al.* (1995) used CFD to study the performance of such a pond. They simulated different ways, for example baffles, to improve the function of the pond. The pond was rectangular with a length of 100 m and a width of 50 m. The simulations were performed for steady-state conditions in 2-D with a laminar flow regime. No-slip was assumed on the walls. No validation was performed at this stage. However, a study to compare simulations and experimental Residence Time Distribution (RTD) curves from the literature was performed (Wood *et al.*, 1998). The studied pond was a scale model 12.2 m long and 6.1 m wide. The simulations were performed in 2-D with the k-ε turbulence model. The inlet had the biggest impact on the flow pattern and CFD was considered a possible tool to predict *a priori* the flow pattern with different inlet arrangements. However, there were discrepancies between simulated and measured RTD curves. The main reason for this was the difficulty of representing a 3-D inlet in a 2-D model. Wood *et al.* (1998) suggested that a 3-D model would correspond better to the real situation.

Another type of rectangular tanks is on-line storage tanks for combined sewage water. These tanks store the sewage water during heavy storms and are then more or less emptied. It is important to avoid sedimentation in these structures since it decreases the available storage volume and the sediment might also cause problems with odour. Another problem is the cleaning of the storage tanks. A better-designed storage tank, where sedimentation is minimised, decreases the maintenance cost for the tank. CFD simulations of the 3-D flow pattern for steady flow in a model of a storage chamber have been performed by Stovin and Saul (1996). The scale model was 2 m long and 0.972 m wide. The inlet and outlet were positioned opposite each other in the middle of the short sides. Both velocity measurements and measurements of sediment deposition and spatial distribution of sediment were performed in the model and compared with CFD simulations. A stochastic particle tracking routine was used to simulate the sediment deposition. A sensitivity test for different parameters in the particle tracking routine was also performed (Stovin and Saul, 1998).

Ta (1999) simulated sedimentation in a storm tank at a wastewater treatment plant. The tank was studied during filling, spilling and emptying. The time-dependent simulations were performed in 3-D with the k-ε turbulence model. The surface was simulated as a moving boundary using the deforming mesh technique. Simulations of sediment transport were also done. No validation with measurements is presented in the paper.

The flow in final settling tanks is complicated due to density stratification. The sludge content in the incoming jet makes it dive to the bottom due to density effects. Krebs (1991) performed a study where the aim was to reduce the velocities in the bottom jet.

Both computer simulations and hydraulic model tests were performed. The simulations were performed in 2-D and the turbulence was approximated by a constant viscosity. The velocity and sludge concentration were uniform over the whole inlet. A flight scraper was simulated as a given return velocity in the bottom cells. The conclusions were that the simulations gave flow fields close to those described in the literature and in the model test. Simulations were also performed with a cross-sectional wall. The general effect of the wall is to stop the bottom current. The result with a cross-sectional wall gave positive effects on the velocity and the sludge distribution.

Lyn *et al.* (1992) present a similar study of sedimentation tanks. The numerical simulations are performed with a vertical 2-D model using the k-ε turbulence model. The code is extended to include suspended solids with a settling velocity distribution, the treatment of sediment-induced density currents and a simple flocculation model. The simulations were compared with field measurements reported by Larsen (1997). The simulated flow pattern was similar to the one described by Krebs (1991) with a bottom current due to density effects.

Circular tanks
Circular sedimentation tanks are common at sewage treatment plants in the United Kingdom. Quarini *et al.* (1996) studied a 1:20 model of a circular sedimentation tank. Traditionally, sediment tanks are designed with a sufficient residence time to allow separation. In this study 3-D CFD simulations were performed. The solution was found not to be sensitive to mesh density. A standard k-ε turbulence model was used. There were difficulties in obtaining converged solutions. The simulations were compared with flow pattern measurements and dye studies. The flow pattern measurements were performed with small glass balls and aluminium flakes. The trajectories of the particles were recorded by photos and video. The dye studies indicated that the flow in the tank was very strongly affected by short-circuiting. The measured and simulated flow velocities agree reasonably well. However, some differences occurred in the surface layer and in the central regions of the measured profiles. The measured flow velocities in the surface layer were lower than the simulated ones. Quarini *et al.* (1996) explain these discrepancies by two effects. The first is the difficulty of measuring the velocities near the surface. The second concerns the zero shear boundary condition imposed at the free surface. Quarini writes: 'This boundary condition will not underpredict and is likely to be a good representation of the true physics, provided there are no surface effects present, such as scum accumulation or oil contamination.' The measured flow profile in the mid-region shows a richer structure than the simulated profile. Quarini *et al.* (1996) explain this problem: 'It is well known that the k-ε turbulence model is poor in the region of low flow and inadequate where the mean velocity gradients go to zero, as occurs in this region of the tank.'

Matako *et al.* (1996) present a similar study of the flow in secondary circular sedimentation tanks. However, they performed measurements and simulations in both a pilot-scale model (d=3.7 m) and a full-scale model (d=25.5 m). The tank was modelled in 2-D with respect to water depth and tank radius. Simulations were performed with a steady flow rate for the pilot-scale model and with variable and mean flow rates for the full-scale model. Simulations were performed for a vertical and a horizontal inlet to compare the obtained flow pattern. Also in this study the standard k-ε turbulence model was used. The free surface was modelled as a symmetry plane "rigid lid". The simulations were compared with tracer tests. The most important finding in the paper is the importance of modelling a variable flow rate if relevant.

2.3.3 Water reservoirs

Hult (1994) discusses problems that can occur in water reservoirs for potable water. One of the points in the paper is the deficient hydraulic performance of reservoirs. This may lead to zones with more or less stagnant water. It is important that the circulation of water is good to avoid these stagnant or dead zones.

Glynn *et al.* (1997) performed CFD simulations for three different types of potable water reservoirs, two rectangular and one circular. The RTD simulations were compared with field data. The agreement for two of the models was good. The authors could only explain the mismatch for the third model (rectangular) by the determination of field data. Glynn *et al.* (1997) concluded that 'The model predictions were not in general greatly sensitive to modelling refinements or to fine geometrical detail. This observation lends confidence to the use of CFD for reservoir studies in the future, since highly refined models are unlikely to be necessary for obtaining useful analyses of reservoir flows.'

Some studies have also been performed on raw water reservoirs of various shapes. Ta and Brignal (1998) used CFD to study the residence time distribution in a raw water reservoir. The reservoir is used for both storage and preliminary treatment of raw water. The main problem was short-circuiting, which caused retention times much shorter than the nominal detention time. Both flow pattern and tracer distribution were simulated. A standard k-ε turbulence model was used as well as simple boundary conditions. All walls had a no-slip boundary condition except for the free surface where a slip wall condition was applied. No consideration was given to the effect of density stratification or impact of wind. To improve the flow conditions in the reservoir, different inlet and outlet configurations were studied. No comparison with field data or laboratory data was performed.

Ali and Othman (1997) also performed 2.5-D simulations of different types of scale models of water-supply reservoirs and with different inlet and outlet arrangements. The simulations were compared with velocity measurements and tracer studies in scale models of reservoirs. Another way to improve the water quality and reduce the effects of temperature stratification and dead zones is to use artificial mixing. Cox *et al.* (1998) used CFD to compare two different mixing devices in order to improve the performance of a water reservoir. The objectives were to reduce stratification, improve oxygenation of the water, and hence minimise the solution of manganese from the sediment. The bubble mixer provided best top-to-bottom mixing and was installed in the reservoir.

2.3.4 CSO structures

Combined Sewer Overflow (CSO) structures have two main purposes. One is to control the through flow to the downstream sewer. The other purpose is to retain the pollutants in the sewer system. The excess storm flow is spilled to a receiving watercourse. Several CFD simulations have been performed to study these structures. Saul and Svejkovsky (1994) studied a vortex chamber with a spherical spill weir. Harwood and Saul (1996) studied a geometry similar to a Stormking separator. Tyack and Fenner

(1998) simulated the flow in a hydrodynamic separator. The flow in these types of structures is in general more swirling than in ponds and tanks. The initial simulations were performed with the standard k-ε model to get a converged solution. After that, the RNG k-ε model was used since this model gives better accuracy for swirling flows. The simulated flow velocities were compared with measured velocities.

Another type of CSO structure is rectangular overflow chambers. Three-dimensional simulations and particle tracking have been performed by Harwood and Saul (1999) and Stovin *et al.* (1999). Stovin *et al.* (1999) simulated the gross solids separation and flow field. The simulations were compared with field measurements in a full-scale model. Reuber *et al.* (1998) studied similar constructions, namely combined sewer detention tanks. In their study a scale model consisting of a pipe with an inner diameter of 0.441 m and a length of 9.3 m was used. The inlet pipe had a diameter of 0.15 m and there were two outlets: one circular overflow with a diameter of 0.15 m and a throttle pipe of 0.06 m diameter. Flow simulations in 3-D were performed and compared with velocity measurements. The flow simulations were performed with a finite element code, RISMO (River Simulation Model). The turbulence was modelled with a k-ε model modified for flows with low Reynolds numbers. Kluck (1996) simulated filling and particle distribution in a stormwater settling tank for CSOs in 2-D.

2.3.5 Miscellaneous applications

Other examples of structures where CFD has been used to model the flow to understand the hydraulics better, and in some cases to improve the design, are horizontal primary oil/water separators (Wilkinson and Waldie, 1994), sandtraps (Olsen and Skoglund, 1994) and manholes (Asztély and Lyngfelt, 1994). Hamille and Rowe (1996) present a study where CFD has been used to simulate the flow in a pump sump, a Francis turbine and a turbo-pump.

2.3.6 Summary

Turbulence modelling
The most widely used turbulence model is, as described before, the standard k-ε model. However, there are some limitations in the applicability of this model. It is known that the k-ε model is not good for low flow velocities (Quarini *et al.*, 1996). The k-ε model assumes that the turbulent viscosity is equal in all directions. Measurements and simulations in a sandtrap showed deviations in the length of the recirculation zone. The problems were assumed to be caused by the turbulence model (Olsen and Skoglund, 1994). For swirling flows with high curvature the RNG k-ε model is a better choice (Tyack and Fenner, 1998). Tyack and Fenner used the RNG k-ε model to simulate the flow in a hydrodynamic separator. Harwood and Saul (1996) used the Reynolds Stress Model (RSM) since it was considered to be more applicable to the swirling flow regime found in dynamic separators. Another reason is its possibility of modelling the anisotropy of the turbulent viscosity, which occurs in swirling flow.

Boundary conditions
The choice of boundary conditions is difficult especially for ponds where the boundaries are not well defined. Persson (1999a) addressed the problem of estimating the boundary conditions for real ponds, especially vegetated ponds. Most studies simulate the free water surface as a "rigid lid", which means that links with all velocity components

normal to the surface are cut and the surface is shear-free. This implies that waves and other unsteady surface motions are not permitted. However, some attempts have been made to use a free boundary (Ta, 1999; Benelmouffok and Yu, 1989). Rodi (1995) suggests an alternative to using a symmetry plane at the free surface by using a boundary condition on the dissipation rate ε. Wind stress on the free surface in ponds and reservoirs is another factor that may affect the flow pattern and mixing. Both Shaw *et al.* (1997) and Matako *et al.* (1996) mention this problem. However, none of the studies account for the effect of wind in the simulations.

1-D, 2-D, 2.5-D and 3-D modelling

One of the objectives in this thesis is to present problems in sanitary engineering where 1-D models are not sufficiently accurate to resolve the flow pattern. However, are 2-D models enough or are 2.5-D or even 3-D models required? Several authors bring up this question. Pettersson *et al.* (1998) emphasise that it is important to consider the topography of the bottom. They showed that the 2-D model was not able to simulate the correct flow pattern in the pond. Wood *et al.* (1998) draw the same conclusion. A 2.5-D (depth-integrated) model implies that a logarithmic velocity profile is assumed. This means that the fluid is affected by the bottom, which is not the case with a 2-D simulation. Several studies are presented where reasonable agreement between field measurements and simulations in 2.5-D is obtained (Persson, 1999a; Ali and Othman, 1997; Benelmouffok and Yu, 1989). Wilkinson and Waldie (1994) studied horizontal primary oil/water separators. Measurements in a 2-D model were compared with 2-D simulations, and simulations in 3-D were compared with measurements in a 3-D model. They found that the CFD results were in reasonable agreement with the velocity measurement in a small 2-D rectangular model separator, but that there were substantial differences between measured flow patterns and CFD prediction in a larger three-dimensional cylindrical separator. They assumed that the main reason for the discrepancies was caused by limitations in the CFD model. An improved CFD model with more and smaller computational cells would probably give a better result.

Ekarna *et al.* (1997) write about the theory, modelling, design and operation of secondary settling tanks. They conclude that the flow pattern in full-scale settling tanks is very complex. They write: 'Although some success has been achieved through 1-D or 2-D models, several field conditions cannot be adequately represented by 1-D or 2-D models. Circular tanks generally exhibit axisymmetric flow and can therefore be adequately modelled by 2-D models with the exception of the local field around the removal system. Rectangular and square tanks are prone to more complex flow and might require 3-D models.' Rodi (1995) writes: 'As three-dimensional calculations are still too expensive for many practical situations, such depth average calculations are often used, in practice, whenever the water body is relatively shallow, i.e. its horizontal dimensions are much larger than its vertical extent, and when buoyancy effects are weak.

Krebs (1995) presents another study discussing different modelling approaches for final or secondary clarifiers. One of the conclusions was that design improvements, for example inlet and outlet design, require 2-D or even 3-D modelling. Krebs (1995) also writes: 'For investigation of structure details, supporting experiments are still essential.'

Sedimentation modelling

The sediment deposit in storage tanks and ponds is complex and depends on many factors. As described previously, the simulation of sediment transport in CFD is mainly done by two approaches; the Eulerian and the Lagrangian. Sediment transport is generally divided into bedload and suspended load. Olsen and Skoglund (1994) simulated bedload and suspended load to study the sediment flow in a sandtrap. Ta (1999) used the chemical species transport model to simulate the dissolved solids; the suspended particles were simulated with a dispersed phase model. To simulate the sedimentation, the particles were injected at the inlet, and to simulate the resuspension the particles were injected at the bottom using a saltate boundary condition. A similar approach using two different models for suspended and dissolved pollutants was used by Benelmouffok and Yu (1989).

Most simulations of particle sedimentation are concentrated on the suspended load and use the Lagrangian approach. Most simulations also separate the two phases and neglect the interaction between the second phase (the dispersed phase) and the fluid phase. Another simplification that is often assumed is that the particles are spherical and do not interact with each other (Pettersson, 1997; Stovin and Saul, 1996; Saul and Svejkovsky, 1994; German and Kant, 1998). De Cock *et al.* (1998) implemented a model for floc growth and breakup in the numerical code to study how coagulants improved the settling efficiency in storage sedimentation tanks. The particle trajectories may be calculated for the mean flow, neglecting that the turbulence affects the particle (Pettersson, 1997; German and Kant, 1998). In stochastic particle tracking the influence of turbulence on the particle is included (Stovin and Saul, 1996; Saul and Svejkovsky, 1994). At the inlet boundary the particle size, distribution and density must be set. Another difficulty is to model the boundary condition for the particle at the bottom. The simplest boundary condition is that the particle trajectories are terminated once they reach the bottom (Pettersson, 1997; German and Kant, 1998). It is also possible to allow the particles to be reflected back into the flow (Stovin and Saul, 1996).

Design

One of the main advantages with CFD is the possibility to investigate the influence of different inlet and outlet arrangements, different shapes and baffles on the flow pattern. Persson (1999b) performed simulations with a 2.5-D model to investigate how different inlet and outlet positions affected the flow in a tank. The tank models all had the same surface area. Tracer simulations were performed and the parameters studied were short-circuiting, effective volume and hydraulic efficiency. No comparison was performed with real tanks. Van Buren *et al.* (1996) briefly discussed how baffles can be used to extend the mean residence time in a stormwater pond. The design and location of the baffle were based on computer simulations. Another study, where simulations with different baffle arrangements, have been performed is the study by Wood *et al.* (1995). The baffle was used to better utilise the pond volume and decrease short-circuiting. German and Kant (1998) simulated how the flow pattern in a stormwater pond changed when small islands were placed in front of the inlet pipe. The purpose of the islands was to reduce the energy and the turbulence in the water to obtain a better flow regime for particle settling. Another example is the study by Ali and Othman (1997). They studied different inlet and outlet arrangements and various shapes of water-supply reservoirs.

3. SUMMARY OF PAPERS

3.1 Paper I: Flow Pattern in a Rectangular Detention Tank — Three-Dimensional Simulation and Measurement

The scope of paper I was to study the applicability of 3-D simulations as an engineering tool in detention tank design. This was done by simulating the flow pattern and tracer concentration in a large-scale model of a detention tank. The simulations were compared with measurements in a large-scale model.

The test tank is rectangular, 12.9 x 8.9 m. The inlet and outlet are positioned opposite each other in the middle of the short sides. The water depth in the tank is 1.0 m. Three different flow rates, 2, 8 and 20 l/s, were selected to study the performance of the tank during different flow conditions. This corresponds to an inflow velocity of 0.48, 0.19 and 0.048 m/s. The flow pattern was measured by drogue tracking. The drogues consisted of a floater, a thin string and a stream cross. The flow pattern was measured at three different depth levels, 0.85, 0.5 and 0.15 m, below the surface. The movements of the drogues were recorded by a motion analysis system using two video cameras. Tracer studies were also performed. Rhodamine was used as the tracer and added instantaneously upstream of the inlet. The Rhodamine concentration was measured continuously at the outlet by a fluorometer

The simulations were performed with the finite volume code FLUENT. The turbulence was modelled with the standard k-ε model. Simple boundary conditions were used. The free surface was modelled as a symmetry plane and the walls and bottom had imposed no-slip conditions. The computational mesh was a rectangular box with 144 000 cells. The computer simulations were divided into two steps. First a stationary flow simulation to obtain the flow pattern was performed, and secondly a transient simulation solving the advection-diffusion equations for the tracer transport. The tracer was injected as a pulse at the inlet.

The comparison between measurements and simulations showed similar flow patterns for the two higher discharges. The flow pattern consisted of a jet with two big whirls, one on each side of the jet. However, for the low flow rate 2 l/s, the measured and simulated flow patterns were different. The simulated flow pattern was similar to the simulated flow pattern for the two higher discharges. The measured flow pattern had no clear jet and no recirculation flow on the sides.

The simulated and measured Residence Time Distribution (RTD) curves showed the same thing. The agreement for the two higher discharges was good, but for 2 l/s there were discrepancies. For 20 and 8 l/s the measured and simulated RTD curves consisted of a fast and high peak and a long tail with low concentrations. This indicates a tank with strong short-circuiting and dead regions having little exchange with the rest of the water volume. The RTD curves also showed a second peak. This is explained by the recirculation in the tank. For the measured RTD for 20 l/s, the second peak was smeared out due to turbulence and strong mixing. The measured RTD curve for 2 l/s had a more widely spread shape.

The variation of the flow pattern in the measurements is interesting and some suggestions of what might cause the variation are presented in the paper. Assuming that the measured flow pattern is right, this implies that the CFD simulation did not reproduce the measured flow pattern. One reason for this might be that the *k*-ε turbulence model is not suitable for this problem. The *k*-ε turbulence model is a high-Reynolds-number turbulence model. A better choice might have been the RNG *k*-ε turbulence model. This model accounts for regions with low flow velocities, as described in Section 2.2.2. On the other hand, there were some problems during the measurements indicating that the measurements are not reliable. The incoming jet had a lower temperature than the water in the tank and was diving towards the bottom. This might have caused the change in flow pattern. The flow pattern is the reason why the measured RTD curve for 2 l/s, is more widely spread and has no clear peak compared to the simulated RTD curve. Another reason might be that the flow in the physical model was not fully developed when the measurements started. Before the measurements started, the flow was kept running for a couple of hours to allow the flow pattern to develop. Perhaps this time was too short in the 2 l/s case.

The main conclusions from Paper I are that the 3-D simulations reasonably well predict the measured flow pattern and the tracer transport for higher discharges. However, for the lowest discharge 2 l/s there were difficulties in both the measurements and the simulations. The study also shows the possibilities of using 3-D numerical simulations as a tool in detention tank design.

3.2 Paper II: Numerical Simulation and Large-Scale Physical Modelling of Flow in a Detention Basin

The aim of Paper II was to compare 3-D simulations with 2.5-D (depth-integrated) simulations. The simulations are compared with measurements in a large-scale physical model of a detention tank. The 3-D simulations and measurements are presented in the previous section in Paper I. To study the hydraulic performance of a tank and evaluate the RTD curve, different measures can be used. In this study the short-circuiting, the effective volume, the number of equivalent tanks in a tank-in-series model (which is a measure of mixing) and the hydraulic efficiency were used.

The results showed that the 2.5-D and the 3-D simulations both simulated the main flow pattern and tracer transport well for high discharges. For the lowest flow rate, it was possible with the 2.5-D model, after some calibrations and change of turbulence model, to achieve a flow pattern close to the simulated one. No calibrations were performed for the 3-D model. However, as indicated in the previous paper, there are some uncertainties in the measurement for the lowest flow rate. The parameters of hydraulic performance showed that approximately 50% of the tank volume was active. There was a small indication that the lowest flow rate has less short-circuiting and less effective volume compared to the two higher flow rates. However, too few measurements have been performed to draw general conclusions from the data.

The main conclusion is that for this rectangular geometry with constant water depth a 2.5-D model is enough to obtain good results for the main flow pattern, which implies that the vertical velocities had little influence on the dominating flow pattern. The advantage of the 2.5-D model is that it is easier to build the mesh and that the computational time is shorter. However, the 2.5-D model demands more modelling skill and more calibrations. The 3-D approach is more general and simulates the flow pattern and velocities in more detail. Rodi (1995) discusses the limitations of the 2.5-D approach. He writes: 'it is not suitable whenever strong three-dimensional effects involving vertical motions prevail, caused, for example, by the boundaries or by buoyancy forces; and as computer power increases so does the application of three-dimensional models.'

3.3 Paper III: Modelling of Particle Deposition using a Bed Shear Stress Boundary Condition

In the third paper an on-line storage tank is studied. Storage tanks are used in the sewer system to store sewage water during heavy storms when the capacity downstream is limited. The main purpose is consequently to store water. However, it is also important to avoid settling in storage tanks since the capacity of the tanks is then decreased. Sediment may also cause odour and the maintenance frequency for cleaning of the tank will thereby increase. Stovin and Saul (1996, 1998) have performed several studies on storage tanks. The objective of this study is the boundary condition for the particles on the tank bottom.

The computer simulations were compared with measurements in a small-scale model of a storage tank. The measurements have been extensively reported by Stovin (1996). The dimensions of the scale model are 2 x 0.972 m. The water depth is 0.196 m. Velocity measurements were performed as well as measurements of sedimentation efficiency and the spatial distribution of the sediment on the tank bed. Crushed olive stone was used to model the sediment. The density of the crushed olive stone was 1500 kg/m^3 and the median particle diameter was d_{50}=47 μm.

The computer simulations could be divided into two parts. First the flow field and then the sediment transport was simulated. The flow simulations were performed in 3-D using the same finite volume code as in the two previously discussed studies, namely FLUENT. The standard k-ε turbulence model and standard wall functions were used.

To model the sedimentation the Lagrangian stochastic particle tracking approach was used. Uncoupled simulations were performed, which means that the particles do not affect the flow field. The particle tracks are simulated one by one, which means that no interactions between the particles are taken into account. However, the stochastic particle tracking used implies that the turbulence affected the particle trajectory. Fifty simulations using 48 particles in each simulation were performed for each set-up. The particles were evenly distributed over the inlet. Another important aspect when simulating particle transport is the behaviour of the particles at the boundary. Previous studies have been performed. Pettersson (1997) used the trap condition. The trap condition means that the particle trajectories are terminated once the particles reach the boundary. Stovin and Saul (1996) used the reflect condition. The reflect condition enables the particles to bounce at the bottom. The change of momentum due to the bounce, the reflection angle and the maximum number of reflections in each cell are parameters that can be varied. Since these parameters are difficult to estimate, Stovin and Saul (1996) suggested and performed initial studies with another approach based on the shear stress distribution on the bed. This new modelling approach was used in the present study.

The new boundary condition works as follows. When a particle reaches a boundary cell with the defined Bed Shear Stress (BSS) boundary condition, the bed shear stress in the cell is calculated. The calculated BSS for the cell is compared with a critical bed shear stress for deposition, set by the user. If the bed shear stress is below the critical value for deposition, the particle is reported as aborted; otherwise the particle will be reflected. The critical bed shear stress for deposition was set in this study to 0.04 N/m^2. The value was obtained from the laboratory measurements. With the BSS boundary condition it is possible to get a report of the x-, y-, z-positions of the settled particle, the particle number and the bed shear stress at the cell where the particle settled. This report facilitates the estimation of the spatial distribution of sediment.

Many simulations were done to investigate the adequacy of the BSS boundary condition. The sensitivity of the sediment pattern and efficiency to cell size, choice of particle dispersion model and selected value of critical bed shear stress was tested. Simulations were performed for different inlet velocities ranging from 0.005 to 0.8 m/s and for different particle diameters (28, 47, 88 and 110 μm). Four different meshes were used. The number of cells varied between 36 700 and 142 700. The simulations using the BSS boundary condition were compared with simulations using the trap condition and with measurements in the scale model. Comparisons were performed for both the sedimentation efficiency and the spatial distribution. The efficiency was defined as the number of particles that deposit in the tank divided by the number of input particles.

Some simulations were also performed for a narrower tank geometry with a length-to-width ratio of 2.5. The observed flow pattern was not reproduced in the simulations, and therefore also the spatial distribution of sediment differed.

The conclusion from the study was that the bed shear stress boundary condition predicted the sedimentation efficiency well for high inflow rates. The trap condition overestimated the sedimentation efficiency for high velocities, which was also found by Stovin (1996). For low inflow velocities and small particles, the discrepancies between measurements and simulations are larger. This is the case for both the BSS and trap boundary conditions. By studying the particle tracks it is possible to see that most of the particles do not come into contact with the bottom. Possible explanations for this are that the particles are too light to settle by gravity and that the turbulence keeps the particles in the flow. The sedimentation efficiency was found to be sensitive to particle size, particle density and flow rate.

The obtained spatial particle distributions from the simulations are promising. The distribution with the bed shear stress boundary condition is very similar to the photos and sketches of the sediment distribution obtained from the measurements. There were some small regions where sedimentation occurred in the simulations but not in reality. The explanation for this could be that in reality the flow conditions are rarely steady. Small oscillations in the inlet jet could explain why no sediment was observed in these regions. The trap condition is not able to predict the observed sediment pattern. The spatial distribution of particles is sensitive to inlet velocity. The influence of cell size is small and does not affect the main features of the sediment distribution and coverage.

The study showed the possibilities of predicting the sediment efficiency and sediment distribution using a bed shear stress boundary condition. Both the spatial distribution and the simulated sedimentation efficiency gave reasonable agreement with measured values.

4. CONCLUSIONS, DISCUSSION AND SUGGESTIONS FOR FURTHER WORK

The literature review shows that there are many different hydraulic problems in sanitary engineering where CFD simulations are applicable. The main area seems to be modelling of sedimentation and residence time in tanks and ponds. Almost all authors of the papers treated describe the possibilities with CFD simulations as important and useful. There is an almost uniform belief in CFD simulations as an engineering tool in sanitary engineering. Glynn *et al.* (1997) write: 'it is very difficult to study the flows in existing reservoirs by direct observation, CFD is likely to prove a highly useful tool in this field, and offers a means whereby potential design improvement can be evaluated.' Wood *et al.* (1995) have a more cautious approach. They think modern CFD techniques require significant development and validation. They write 'CFD modelling is not an easy universal answer. It involves a considerable investment of time, money and expertise.' However, the literature study presented shows the possibilities with CFD simulations and the development in CFD codes and computer capacity has accelerated in just the last few years. Many authors consider it necessary to compare the simulations with measurements at least at the initial stage. More measurements in full-scale and scale models and simulations in two and three dimensions are therefore required to build a basis of knowledge and skill for how to use CFD in sanitary engineering.

This was one of the objectives in the study where comparisons were performed between 3-D simulations and measurements in a large-scale model of a detention tank. The main conclusion from this study was that the agreement between simulations and measurements was good for high inlet velocities. However, for low flow velocities there were differences between the measured and simulated flow patterns. This difference in flow pattern is interesting and further studies need to be performed to determine its causes. Examples of suggested studies are improved measurements where there are no temperature differences between the incoming jet and the water in the tank. It is also possible to simulate the flow pattern with a lower temperature of the incoming jet. Probably another type of turbulence model should be used for the low flow rate; see discussion below.

The comparison between a 2.5-D and a 3-D simulation and measurements showed the possibilities of using 2.5-D simulations to predict the large-scale flow pattern. The advantages of a 2.5-D model are that it is less expensive and demands less time with respect to data input, mesh generation and computer capacity. However, it needs more calibration and modelling skill. The 3-D approach is more general and evaluates the flow properties in more detail. The simulated geometry was a rectangular box with constant depth. A more general use of the 2.5-D approach requires better knowledge of its limitations with respect to tank geometry, for example varying bottom shape, inlet and outlet arrangements and inlet velocities. It is believed that this knowledge is preferably obtained by comparative studies of 2.5-D and 3-D simulations. The 3-D approach is believed to have a potential to develop towards a more practically useful tool through the development of both hardware and software, such as mesh generators, and also increased practical experience from new studies.

In the last study the possibilities of using CFD to simulate sediment transport and settling in storage tanks are examined. Many researchers have studied these problems. The focus in the present study is on the boundary condition on the tank bed. The boundary condition used is based on the bed shear stress on the tank bottom. The modelling approach is promising and predicts the sediment distribution and sedimentation efficiency well for high inflow velocities. For low inflow velocities the agreement is not very good. More studies are therefore needed for small particles and low inflow velocities. The Lagrangian or dispersed phase modelling approach is a very simplified way to model particle transport. Interaction between particles is not considered, nor do the particles have any effect on the flow pattern, and it would be interesting to study under which circumstances these assumptions are valid. It is not possible to simulate the build-up of sediment at the bottom with this modelling approach. In hydraulic problems where this is of considerable significance, the Eulerian two-phase model is perhaps a better choice.

In both of the performed studies, the agreement between measurements and simulations for low flow velocities is not satisfactory. In the detention tank a different flow pattern occurred, and in the storage tank the simulated sedimentation efficiency was too low for low flow velocities. The flow pattern simulations for the narrower tank also indicated difficulties in obtaining the measured flow pattern. As suggested before, these problems could be explained by the turbulence modelling. The standard two-equation k-ε turbulence model is the most commonly used turbulence model in hydraulic problems, and it has also been used in these studies. The k-ε turbulence model is a high-Reynolds-number model. One of its limitations is that it cannot account for regions with stagnant flow and low flow velocities. Most flows in environmental fluid dynamics, especially flow problems in ponds and tanks, are flows that have low flow velocities and consist of regions with more or less stagnant flow. On the other hand, there are swirling flows in vortex separators that demand turbulence models suited for swirling flow, which can account for the anisotropy of the turbulent viscosity. Further studies are needed to determine how the turbulence should be modelled in different cases, as well as in what cases or under what circumstances the most commonly used k-ε turbulence model is not applicable.

Is CFD a design tool in sanitary engineering? What has been done? How does it work? What are the limitations? These were the questions asked before the literature review and the different studies were performed. The discussion presented shows that there are several studies demonstrating the possibilities of using CFD in sanitary engineering. The results are promising in most cases. However, it seems that until today CFD has mainly been a tool for researchers and not for engineers working with sanitary engineering problems. With the development of both hardware and software such as mesh generators, 3-D simulations are expected to develop towards a more practically useful tool in the future. However, more studies and simulations need to be performed to obtain more knowledge of how to use the tools in best practice. Examples of further work have been suggested in the different papers and above. I would like to emphasise the necessity of studying turbulence modelling in sanitary engineering problems, especially for low flow velocities, and the need of more comparative studies between 2.5-D and 3-D simulations and measurements in full-scale models.

REFERENCES

Ali, K. H. M. and Othman, K. (1997). Investigation of jet-forced water circulation in reservoir. *Proc. Inst. Civ. Engrs. Wat., Marit. & Energy,* Vol. 124, 44-62.

Asztély, M. and Lyngfelt, S. (1996). Three-dimensional numerical modelling of energy losses in a manhole. In: *Proc. 7th Int. Conf. on Urban Storm Drainage*, Hannover, Germany, Vol. 2, 647-652.

Benelmouffok, D. E. and Yu, S. L. (1989). Two dimensional numerical modeling of hydrodynamics and pollutant transport in a wet detention pond. *Wat. Sci. Tech.,* Vol. 21, 727-738.

Bergdahl, L., Lyngfelt S. and Adamsson, Å. (1999). Tredimensionell analys av strömningen i va-anläggningar. (Three-dimensional analysis of the flow in sanitary engineering structures). *Bygg & Teknik,* Nr 2, 57-60 (in Swedish).

Cox, J. D., Padley, M. B. and Hannon, J. (1998). Use of computational fluid dynamics to model reservoir mixing and destratification. *Wat. Sci. Tech.,* Vol. 37, No. 2, 227-234.

Davidson, L. (1997). *An introduction to turbulence models.* Department of Thermo and Fluid Dynamics, Chalmers University of Technology, Sweden.

De Cock, W., Blom, P. and Berlamont, J. (1998). The feasibility of flocculation in a storage sedimentation tank. In: *Proc. Novatech'1998,* 379-385.

Ekarna, G. A., Banard, J. L., Günthert, F. W., Krebs, P., McCorquodale, J. A., Parker, D. S. and Wahlberg E. J. (1997). *Secondary settling tanks: Theory, modelling design and operation.* Scientific and Technical Report, No. 6, International Association on Water Quality.

Fluent (1997). *Fluent User's Guide* – Version 4.4. Fluent Inc., USA.

German, J. and Kant, H. (1998). FEM-analys av strömningsförhållanden i en dagvatten-Damm (FEM-analysis of the hydraulic conditions in a stormwater detention pond). *Vatten,* Nr 54, 183-190.

Glynn, D. R., Creasey, J. and Oakes, A. (1997). Simulation of flow patterns in potable water reservoirs. Presented at: *7th International PHOENICS User Conference*, Seville, Spain.

Hamill, N. and Rowe, G. (1996). Computational fluid dynamics for pump related designs. *World Pumps,* No. 358, 50-53.

Harwood, R. and Saul, A. J. (1996). CFD and novel technology in combined sewer overflow. In: *Proc. 7th Int. Conf. on Urban Storm Drainage,* Hannover, Germany, Vol. 2, 1025-1030.

Harwood, R. and Saul, A. J. (1999). The influence of CSO chamber size on particle retention efficiency performance. In: *Proc. 8th Int. Conf. on Urban Storm Drainage,* Sydney, Australia, Vol. 1, 1-9.

Hult, A. (1994). Dålig tillsyn av dricksvattenreservoarer (Maintenance of potable water reservoirs). *VAV-Nytt,* 20-22 (in Swedish).

Kluck, J. (1996). Design of storm water settling tanks for CSOs. In: *Proc. 7th Int. Conf. on Urban Storm Drainage,* Hannover, Germany, Vol. 1, 181-186.

Krebs, P. (1991). The hydraulics of final settling tanks. *Wat. Sci. Tech.,* Vol. 23, 1037-1046.

Krebs, P. (1995). Success and shortcomings of clarifier modelling. *Wat. Sci. Tech.,* Vol. 31, No. 2, 181-191.

Larsen, P. (1997). *On the hydraulics of rectangular settling tanks.* Report 1001, Department of Water Resour. Engrg., Lund Institute of Technology, University of Lund, Sweden.

Launder, B. E. and Spalding, D. B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, 269-289.

Lyn, D. A., Stamou, A. I. and Rodi, W. (1992). Density currents and shear-induced flocculation in sedimentation tanks. *J. Hydr. Eng.*, Vol. 118,No. 6, 849-865.

Matako, T., Fawcett, N., Sharp, A. and Stephenson, T. (1996). A numerical model of flow in circular sedimentation tanks. *Trans IChemE,* Vol. 74, Part B, 197-204.

Olsen, N. R. B. and Skoglund, M. (1994). Three-dimensional numerical modeling of water and sediment flow in a sand trap. *J. Hydr. Res.*, Vol. 32, No. 6, 833-843.

Olsen, N. R. B. (1997). Class notes: *Computational fluid dynamics in hydraulic and sedimentation engineering*, Division of Hydraulic and Environmental Engineering, The Norwegian University of Science and Technology.

Persson, J. (1999a). Hydraulic performance of a constructed pond in Oxelösund. Submitted to: *J. of Ecological Engineering.*

Persson, J. (1999b). The hydraulic performance of ponds of various layouts. Submitted to: *J. Urban Water.*

Pettersson, T. J. R. (1997). FEM-modelling of open stormwater detention ponds. *Nordic Hydrology*, Vol. 28(4/5), 339-350.

Pettersson, T. J. R., German, J. and Svensson, G. (1998). Modelling of flow pattern and particle removal in an open stormwater detention pond. In: *Proc. HydraStorm'98*, Adelaide, Australia, 63-68.

Quarini, G., Innes, H., Smith, M. and Wise, D. (1996). Hydrodynamic modelling of sedimentation tanks. *Proc. Inst. Mech. Engrs.,* Vol. 210, 83-91.

Reuber, J., Herbertz, F., Kowalski, R. and Köngeter, J. (1998). Performance – optimisation of combined sewer detention tanks. In: *Proc. UDM,* London, UK, 425-432.

Rodi, W. (1984). *Turbulence models and their application in hydraulics.* A state of the art review, IAHR, Delft, The Netherlands.

Rodi, W. (1995). Impact of Reynolds-average modelling in hydraulics. *Proc. R. Soc.* Lond. A, Vol. 451, 141-164.

Saul, A. J. and Svejkovsky, K. (1994). Computational modelling of a vortex CSO structure. *Wat. Sci. Tech.*, Vol. 30, No. 1, 97-106.

Shaw, J. K. E., Watt, W. E., Marsalek, J., Anderson, B. C. and Crowder, A. A. (1997). Flow pattern characterization in an urban stormwater detention pond and implications for water quality. *Water Qual. Res. J. Canada*, Vol. 32, No. 1, 53-71.

Shilton, A., Glynn, D. and Phelps, P. (1999) An inside look – the potential of CFD technology. *Water21, Magazine of the International Water Association,* July-August, 37-38.

Stovin V. R. (1996). *The prediction of sediment deposition in storage chambers based on laboratory observations and numerical simulation.* Ph.D thesis, University of Sheffield, UK.

Stovin, V. R. and Saul, A. J. (1996). Efficiency prediction for storage chambers using computational fluid dynamics. *Wat. Sci. Tech.*, Vol. 33, No. 9, 163-170.

Stovin, V. R. and Saul, A. J. (1998). A Computational Fluid Dynamics (CFD) particle tracking approach to efficiency prediction. *Wat. Sci. Tech.*, Vol. 37, No. 1, 285-293.

Stovin, V. R., Saul, A. J., Drinkwater, A. and Clifford, I. (1999). Field testing CFD-based predictions of storage chamber gross solids separation efficiency. *Wat. Sci. Tech.*, Vol. 39, No. 9, 161-168.

Ta, C. T. and Brignal, W. J. (1998). Application of computational fluid dynamics technique to storage reservoir studies. *Wat. Sci. Tech.*, Vol. 37, No. 2, 219-226.

Ta, C. T. (1999). Computational fluid dynamic model of storm tank. In: *Proc. 8th Int. Conf. Urban Storm Drainage*, Sydney, Australia, Vol. 3, 1279-1286.

Tyack, J. N. and Fenner, R. A. (1998). Computational fluid dynamics modelling of velocity profiles within a hydrodynamic separator. In: *Proc. UDM,* London, UK, 441-449.

Van Buren, M. A., Watt, W. E. and Marsalek, J. (1996). Enhancing the removal of pollutants by an on-stream pond. *Wat. Sci. Tech.*, Vol. 33, No. 4-5, 325-332.

Versteeg, H. K. and Malalasekera, W. (1995). *An introduction to Computational Fluid Dynamics, The finite volume method.* Longman Group Ltd.

Wilkinson, D. and Waldie, B. (1994). CFD and experimental studies of fluid and particle flow in horizontal primary separators. *Trans IChemE*, Vol. 72, Part A, 189-196.

Wood, M. G., Greenfield, P. F., Howes T., Johns, M. R. and Keller, J. (1995). Computational fluid dynamic modelling of wastewater ponds to improve design. *Wat. Sci. Tech.*, Vol. 31, No. 12, 111-118.

Wood, M. G., Howes, T., Keller, J. and Johns, M. R. (1998). Two dimensional computational fluid dynamic models for waste stabilisation ponds. *Wat. Res*, Vol. 32, No. 3, 958-963.

APPENDIX: A compilation of literature

Author	Structure	Dimensions / Mesh	CFD-code	Dim	Re-number Inlet/Flow	Turbulence model	Performed studies	BC	Validation
Wood *et al.*, 1995	Wastewater pond	100 x 50 m	FIDAP	2-D	30	Laminar	Tried baffles (2 different) and aerators (modelled as an accelerator)		
Wood *et al.*, 1998	Waste stabilisation pond	12.2 x 6.1 m (scale model)	FIDAP	2-D	9 200	k-ε	Different inlet and outlet position		Tracer studies from literature
De Cock *et al.*, 1998	Storage sedimentation tank	32 x 8 x 2.75 m	PHOENICS		145 000		Baffles A floc growth and break-up model is implemented in the numerical model		
Matako *et al.*, 1996	Circular sedimentation tank	Pilot-scale d=3.7 m Full-scale d=25.5 m	CFDS-Flow-3D	2-D	Pilot-scale: 32 000-36 000 Full-scale: 39 000	k-ε	Vertical and horizontal inlet		Tracer studies (sodium chloride and lithium chloride)
Quarini *et al.*, 1996	Circular sedimentation tank	1/20 scale model d=1.4 m 9 000 cells	Flow-3D	3-D	3 200	k-ε			Experimental tracer study and flow pattern visualization
Reuber *et al.*, 1998	Combined sewer detention tank (CSDT)	Pipe l=9.3 m d=0.441 m 4532 elements	RISMO FEM code	3-D	8 000	k-ε modified for flow with low Reynolds numbers		Free surface	Flow field measurements using Digital Particle Velocimetry and sedimentation measurements
Ta, 1999	Storm tank	166 x 39 m Sloping bottom average depth 3.3 m 110 x 32 x 10		3-D		k-ε time-dependent	Simulated emptying and filling, simulated both dispersed (particles) and dissolved (chemical species transport)	Free surface: deforming mesh technique	
Lyn *et al.*, 1992	Sedimentation tank	Length 30 m depth 3.4 m	Own code	2-D	9 100	k-ε			Compared with measurements from literature
Krebs, 1991	Final settling tanks		PHOENICS	2-D		Constant turbulent viscosity	Simulated a sludge hopper by a return velocity in bottom cells, baffles		Model tests, velocity measurements

Stovin *et al.*, 1999	Storage chamber	2 different (long and short) Long 15 x 8 m	FLUENT	3-D		k-ε	Particle tracking, BC reflect		Field measurements, Visual flow pattern observations and sedimentation measurement
Harwood and Saul, 1999	CSO chamber	C_1=2.0 x 0.7 m C_2=4.1 x 1.4 m C_3=15 x 5 m	FLUENT	3-D	Q_1=0.019 Q_2=0.116 Q_3=0.294 m^3/s		Particle tracking		
Stovin and Saul, 1996	Storage chamber	2 x 0.972 x 0.196 m	FLUENT	3-D	9 500-150 000	k-ε	Particle tracking, BC reflect		Measurements of velocity and sedi-mentation efficiency
Benelmouffok *et al.*, 1989	Detention pond	Surface area 6 800 m^2 depth 2.3 m	Own code	2.5-D				Free surface	Measurements of pollutants removal at 5 rain events
Shaw *et al.*, 1997	Stormwater detention pond	Elliptic shape Appr. 90 x 60 x 1.2 m	PHOENICS		Q=0.056 0.08, 0.165 m^3/s	k-ε	Emphasised the importance of wind		Measured velocities ADV
Van Buren *et al.*, 1996	Pond	Surface area 5 200 m^2	PHOENICS	3-D			Baffles		
Pettersson, 1997	Stormwater detention pond	Surface area: 530 m^2, average depth 1.5 m	FIDAP	3-D	180 000	k-ε	Particle tracking, BC trap. Density 1300 kg/m^3 d= 1.5, 10, 20, 40 μm		Measured settling efficiencies
Pettersson *et al.*, 1998	Stormwater detention pond	Surface area 6 200 m^2, depth 0.5-1.6 m	FIDAP	3-D 2-D	Q=0.02, 0.8 m^3/s	k-ε		Bottom roughness	Flow pattern measurements
Persson, 1999a	Pond	Surface area 24 300 m^2 depth 0.5-0.7 m	Mike21	2.5-D	Q=0.012 m^3/s	Smagorinsky		Vegetated	Tracer measurement in field. Tracer Bromide
Persson, 1999b	Tank	V=2 700 m^3 depth=1.5 m	Mike21	2.5-D	Q=0.04 m^3/s	Smagorinsky	Tried different inlet and outlet arrangements and l/b ratio		
German and Kant, 1998	Pond	V=6 000 m^3 depth 1.6 m 2-D 38 000 cells 3-D 87 000 cells	FIDAP	2-D 3-D	Q=0.02, 0.8 m^3/s	k-ε	Particle tracking 1.5,10,20,40 μm, density 1300 kg/m^3		Particle settling efficiency and flow pattern

Wilkinson and Waldie,1994	Oil water separator	Small scale: length=0.875 m depth=0.25 m Large scale: length=3.77 m, diameter=1.0 m depth 0.5 m	FLUENT	2-D 3-D	v=0.05, 0.01 m/s For small scale also v=0.012 m/s	k-ε	Particle tracking 50-300 μm		Measurements of velocity and oil drop size distribution
Kluck 1996	Stormwater settling tanks for CSO	30 m long weir 1.6 m 75 x 25 cells	PHOENICS	2-D	Q=0.1 m^3/s	k-ε		Free surface. Simulated filling and emptying	
Saul and Svejkovsky, 1994	CSO vortex chamber	Chamber diameter=1.1 m	FLUENT	3-D	Q=0.03, 0.045, 0.06 m^3/s		Particle tracking. Sinkers d= 40 mm, density 1005 kg/m^3 Floaters d= 20 mm, density 997 kg/m^3		Observations in full scale laboratory model
Harwood and Saul, 1996	Stormking hydrodynamic separator	Chamber diameter=1.45 m	FLUENT	3-D	Q=0.06 m^3/s	RSM	Particle tracking. d=10, 15, 20, 25 mm 940-1010 kg $/m^3$		Laboratory measure-ments, video recording of posi-tion and movements of 3 mm diameter polystyrene beads
Tyack and Fenner, 1998	Hydrodynamic separator	Chamber diameter=1.6 m 56 824 cells	FIDAP	3-D	Q=0.03, 0.052, 0.0636 m^3/s	k-ε RNG			Velocity measurements with ADV
Olsen and Skoglund, 1994	Sandtrap	Length 18 m width 1 m depth 1.54 m Grid A: 41 x 11 x 11 B: 96 x 11 x 21	SSIIM	3-D		k-ε	Tried different geometries and roughness and moving-water surface, sedimentation simulations d=0.1, 0.2, 0.3, 0.45 mm		Velocity measurements with current meter and measurement of sediment concentration
Glynn *et al.*, 1997	Reservoir	Rectangular 78 x 114 x 4 m 30 x 50 m Circular d=73.2 m	PHOENICS	3-D		k-ε and LVEL	Simulated chlorine concentration, Free surface for the time dependent simulations		Comparison with tracer studies in the field