17th Australasian Fluid Mechanics Conference
Auckland, New Zealand
5-9 December 2010

Optimising a Numerical Model for Flow Pattern Studies of Stormwater Retention Ponds

S.Khan[1], B.W. Melville[1], C. Fischer[2] and A.Y. Shamseldin[1]

[1]Department of Civil Engineering
University of Auckland, Auckland 1142, New Zealand

[2]Department of Mechanical Engineering
University of Auckland, Auckland 1142, New Zealand

Abstract

Numerical simulations were carried out using Computational Fluid Dynamics (CFD) to optimise a CFD model for investigating the hydrodynamics of storm water retention ponds. A rectangular retention pond (4.1x1.5x0.23m) with sloping sides was modelled, with the flow pattern and velocity distribution in 3-D analysed using the commercial CFD software package ANSYS CFX 12 (CFX). A comparison between CFD results was made at different boundary conditions and grids for different flow rates to optimise the numerical model for this type of problem at moderate computational cost.

Reynolds-averaged Navier-Stokes equations were solved using the 3-D finite volume numerical code (CFX). Simulations were undertaken in two steps. First, the model was run for steady state simulation and secondly, the model was run for transient conditions. The CFD results were also compared to experimental results from a laboratory physical model.

Introduction

The hydraulics of storm water retention ponds are very important and directly affect their settling efficiency. There are two main ways to study the hydrodynamics of a pond, numerical modeling and physical modeling.

The advantage of numerical modeling is that it makes it possible to simulate the flow in the pond before the construction of a real pond and provides much more detailed information, such as flow structure and dynamics, velocity distributions and velocity components at any defined point.

CFD provides access to a broad range of data on the flow field, infinite variations of the geometry, infinite scale-up possibilities, and visually appealing results for use by engineers. In addition, numerical simulations require less investment than experimental work, which may be difficult or impossible under field conditions or in the laboratory.

The recent advances in computing power have boosted the development of user-friendly and commercially available flow simulation codes for numerical modeling, which has increased the interest in CFD modeling for the water industry. For retention ponds, CFD provides full flow field data at a low cost [1].

There are two major challenges in using CFD. The first and most important is the correct specification of the physical conditions and the second is the proper attention to numerical issues. The boundary conditions, turbulence models, rheological model, and other physical models such as reaction kinetics, heat transfer, and phase interactions (for multiphase flow) need to be correctly specified. The numerical issues include grid definition and convergence criteria [2]. If any of these issues are neglected, the results generated could be misleading. This paper addresses the optimisation of a 3-D numerical model of stormwater retention ponds using different boundary conditions.

Methodology

For the present study, 3-D numerical models were developed using CFX. To accurately model the hydrodynamics of the flow, the geometry of the numerical models was kept similar to that used in the field and in a laboratory physical model. The model ponds were trapezoidal in cross-section with side slopes of 2:1 (h:v). The circular inlet and outlet were placed at the ends of the pond. The geometry of the pond and the inlet and outlet dimensions were the same for all tests. The dimensions of the model were: top length = 4.1m; top width = 1.5m; bottom length = 3m; bottom width = 0.5m; and depth = 0.23m. The inlet and outlet pipes had diameters of 45mm and 105mm, respectively. A physical model of the same dimensions was also constructed in the laboratory (Table 1, Case 19) where particle tracking velocimetry (PTV) techniques were employed to get the full velocity vector field to validate the CFD results.

The flow region was modelled as a 3-D region. The inlet was assumed to be a mass flow boundary condition with a flow direction normal to the inlet boundary and at the outlet the static pressure was specified as zero except where mentioned in the results. This is a robust combination of boundary conditions available in CFX for this type of problem. The other boundary conditions included in defining the problem are the walls and free surface. The sides and the floor were assumed to be walls with a no-slip condition. Buoyancy has negligible effects for flows in retention ponds and therefore was not considered. No consideration was given to the wall roughness because for a large body of slow moving water, a wall roughness value has little effect on the bulk water flow [3].

The simulations were undertaken in two steps. For each case the model was first run under steady state conditions to obtain the solution for the three components of velocity, pressure, momentum, and turbulence components. The second order discretisation scheme was used. Secondly, the model was solved for transient conditions. The Second Order Backward Euler scheme was used for time-step discretisation during the transient solution. The initial time-step was very small, increasing to a relatively large time step until the end of the simulation with three loops in each time-step. A Root Mean Square (RMS) residual of 10^{-6} was used in the transient simulation to get a high level of convergence of the simulated solution. The simulations were run for a time equal to more than twice the nominal residence time.

The high resolution scheme (HRS) was used. The laminar model and also both the standard k-ε turbulence model and SST turbulence model were tested and compared. The laminar model was considered because of the low Reynolds Numbers at low flow rates [1].

Results and Discussion

In this study, 19 cases (listed in Table 1) of different boundary conditions were studied and compared to optimise the CFD model for the flow patterns in stormwater retention ponds. All the cases are shown in Figure 1 and are described below.

Effect of Flow Rates

Three different flow rates of 0.16, 0.5 and 5 l/s were tested in the numerical model. It was observed that the cases with low flow rate (0.16 l/s) resulted in no particular flow patterns (Figure 1, Cases 1, 3 and 5). However, when the flow rate was set to 0.5 l/s, there was some improvement in developing the flow pattern (Figure 1, Case 10) but at this flow rate the pattern consisted of two eddies in which the first eddy was developed but the second eddy was not developed fully. At high flow rates (5 l/s) the flow pattern was fully developed featuring one or two eddies (Figure 1, Cases 11-18).

Effect of Defined Turbulence Model

The effect of the defined flow model was tested for two different flow rates.. First, the flow rate of 0.16 l/s was simulated based on a laminar model. The laminar model resulted in no particular flow pattern (Figure 1, Case 1). Secondly, the SST or k-ε turbulence models were used to investigate the effect of using turbulence models rather than a laminar model. The SST turbulence model resulted in one small eddy in the pond centre in front of the inlet (Figure 1, Case 3). For the rest of the pond, low velocities were observed which were not influenced by the inlet. For the case based on the k-ε turbulence model, again no flow pattern can be distinguished (Figure 1, Case 5).

Case 4 is a repeat of Case 3 with the difference of using a smaller physical time scale. For the former case, no eddy is apparent and the flow in the pond does not seem to be influenced by the inflow. A small physical time scale did not improve the results.

The SST and k-ε models were also tested for the flow rate of 5 l/s. The results shown in Figure 1, Case 14, are based on the SST turbulence model while the results shown in Case 15 are based on the k-ε turbulence model. Comparison of these cases reveals that the turbulence model has minimal effect on the results, showing that the flow is driven by free stream turbulence and that the near wall regions have a marginal influence on the overall flow.

In summary, the use of a turbulence model, rather than a laminar model, improves the results in terms of flow pattern and both the turbulence models (k-ε and SST) gave similar results.

Effect of Velocity Ramp Function

A velocity ramp function is a function which provides relatively high velocities at the inlet for the first few iterations. This function was employed in cases where the flow pattern was not fully developed especially for low flow rates.

The results shown in Figure 1, Case 1, were simulated using a laminar model and a flow rate of 0.16 l/s and resulted in no particular flow pattern. However, when this case was re-run using a velocity ramp function (Figure 1, Case 2); there was some degree of improvement in the development of flow patterns. Similarly, the results shown in Figure 1, Case 3, were re-run using the velocity ramp function and are given in Figure 1, Case 6. Two large eddies, which dominate the whole flow domain, are apparent in this case. The resultant flow pattern is very similar to that for the laboratory physical model at this velocity. This model was then run for a further 2,000 iterations (Case 7), when the flow field changed with the two eddies splitting into three eddies. This feature was not observed in any of the laboratory physical model results.

The simulations featured in Figure 1, Case 10, did not employ a velocity ramp function. The Case 9 is a re-run of Case 10 using the velocity ramp function at 0.5 l/s. The comparison between Case 9 and Case 10 shows that use of the velocity ramp function improves the numerical results and helps in developing the flow pattern.

In summary, the use of the velocity ramp function improves the simulation results.

Effect of Inlet/Outlet Type

The flow pattern for the run shown in Figure 1, Case 12, was simulated using holes in the opposite ends of the pond as inlet and outlet. This case features two eddies. However, the laboratory physical model experiments featured only one large eddy for high flow rates (Figure 1, Case 19).

The outlet boundary condition for the case shown in Figure 1, Case 13, was set to a free surface, such that the outlet was not submerged. The first eddy is smaller than the second eddy, the pattern being similar to that shown in Figure 1, Case 12.

In the simulation shown in Figure 1, Case 14, the inlet and outlet were modelled as small pipes (0.5m length) to investigate the influence of the different boundary conditions. The flow pattern features a larger first eddy compared to that shown in Figure 1, Case 13. The resultant larger eddy indicates that this boundary condition is more suitable as the simulated flow pattern is closer to the laboratory physical model results.

The run shown in Figure 1, Case 16 used the resultant flow velocity at the pipe inlet, while for the Cases 14 and 15, the inlet velocity was defined in terms of the three Cartesian velocity components at the inlet. That Case 16 results were the same as those for cases 14 and 15, confirms that using the resultant flow velocity (i.e. the pipe exist velocity) is acceptable.

Effect of Defined Mesh

Cases 7 and 8 have similar boundary conditions except that Case 7 is based on a structured mesh while Case 8 is based on an unstructured mesh. The comparison of these cases shows that the flow pattern for this problem is relatively insensitive to the mesh defined for the simulations.

Again, Case 12 used the same boundary conditions as used for Case 11, but employed an unstructured mesh rather than a structured mesh. The flow pattern from this run showed good agreement with the results for the structured mesh indicating that structured and unstructured meshes can result in similar flow patterns for such simulations.

Symmetry Boundary Condition

For Case 17, a symmetry boundary condition was used on top of the pond. This run resulted in a similar flow pattern to that for the laboratory physical model results (Case 19). However, the use of the symmetry condition may not represent the actual physical situation.

Transient Simulation

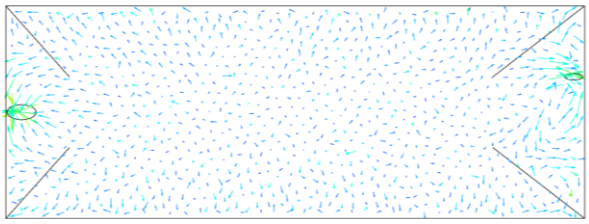
Case 14 was re-run in a transient simulation and the results are shown in Case 18, for which the results showed good agreement with the results from the laboratory physical model (Case 19). This means that at higher velocities, transient effects influence the flow which may not be captured by the steady state runs.

Conclusions

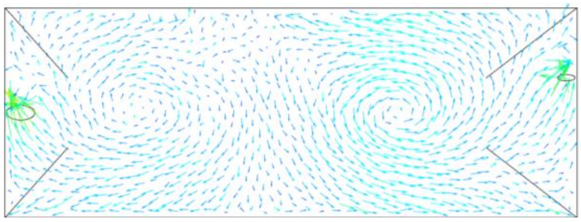
To get reliable results in terms of the flow pattern, the boundary conditions of the problem should be carefully selected. Wrong selection of the boundary conditions can produce misleading results. The velocity ramp function helps in developing the flow pattern for the simulations where the inlet velocity is slow. The turbulence models, k-ε or SST, produce similar flow patterns for this particular problem. Also, the structured and unstructured meshes result in similar flow patterns for steady state simulations. Adding a pipe inlet or outlet improves the overall flow pattern. Also, using a symmetry boundary condition, rather than a free surface, can result in a similar flow pattern to that from the laboratory physical model.

	Discharge (Q) in l/s	Mesh	Flow Model	Velocity Ramp Function	Comments
Case 1	0.16	Unstructured	Laminar	No	
Case 2	0.16	Unstructured	Laminar	Yes	
Case 3	0.16	Structured	SST	No	
Case 4	0.16	Structured	SST	No	Smaller physical time scale
Case 5	0.16	Unstructured	k-ε	No	
Case 6	0.16	Structured	SST	Yes	Same as Case 3
Case 7	0.16	Structured	SST	yes	Re-run of Case 6 for a further 2000 iterations
Case 8	0.16	Unstructured	SST	Yes	Same as simulation in Case 7
Case 9	0.5	Unstructured	SST	Yes	
Case 10	0.5	Unstructured	SST	No	
Case 11	5.0	Structured	SST	No	
Case 12	5.0	Unstructured	SST	No	
Case 13	5.0	Unstructured	SST	No	Free surface outlet
Case 14	5.0	Unstructured	SST	No	Inlet and outlet as pipes
Case 15	5.0	Unstructured	k-ε	No	Inlet and outlet as pipes
Case 16	5.0	Unstructured	SST	No	Inlet and outlet as pipes and using normal speed boundary conditions at the inlet
Case 17	5.0	Unstructured	k-ε	No	Inlet and outlet as pipes using symmetry boundary conditions at the top
Case 18	5.0	Unstructured	SST	No	Re-run of Case 14 in transient conditions
Case 19	2.0	Laboratory physical model			

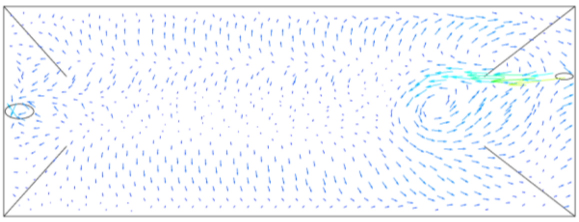
Table 1: List of all the simulated cases



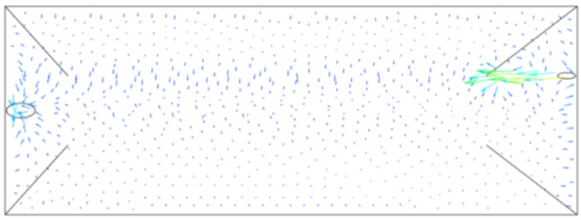
Case 1



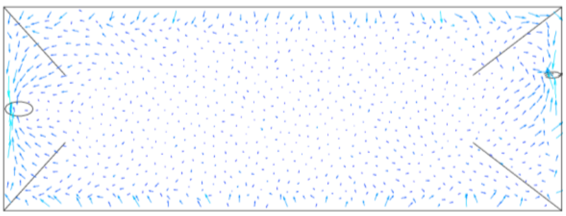
Case 2



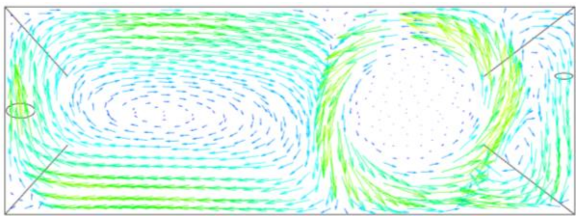
Case 3



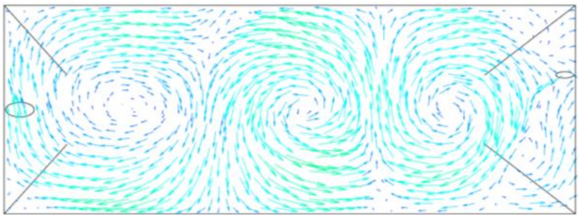
Case 4

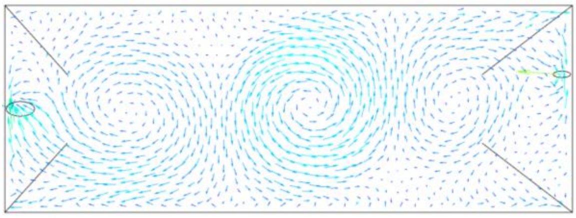


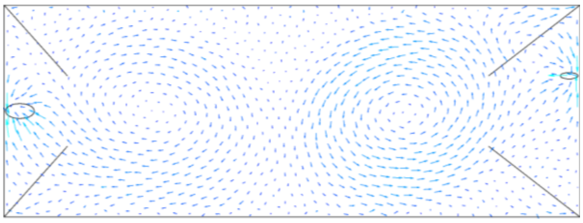
Case 5

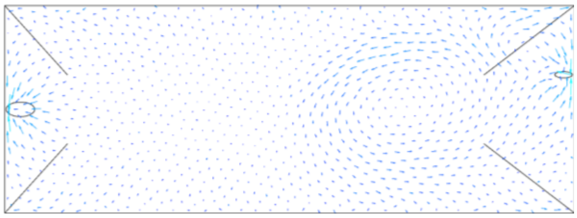


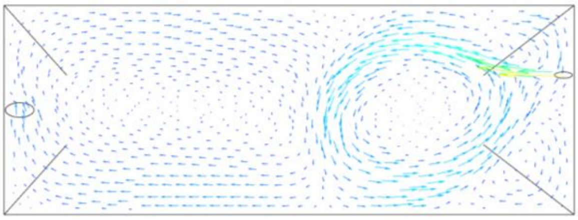
Case 6

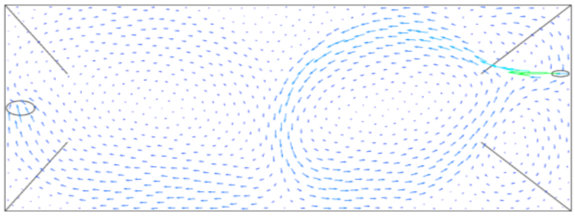

Case 7

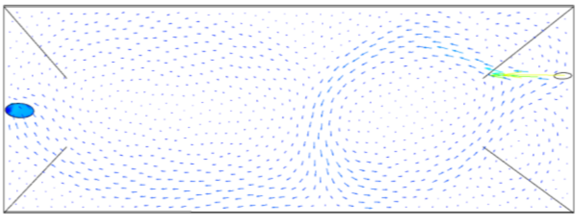

Case 8

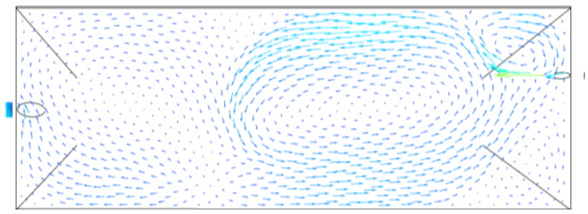

Case 9

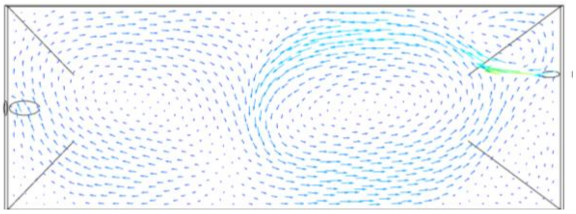

Case 10

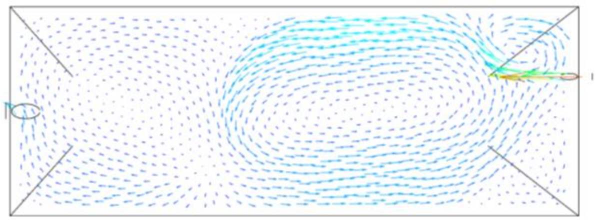

Case 11

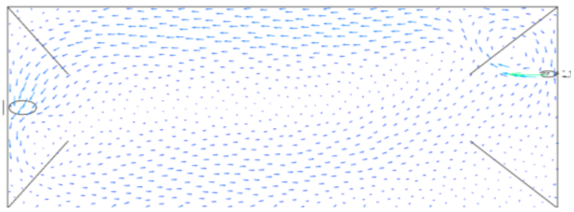

Case 12

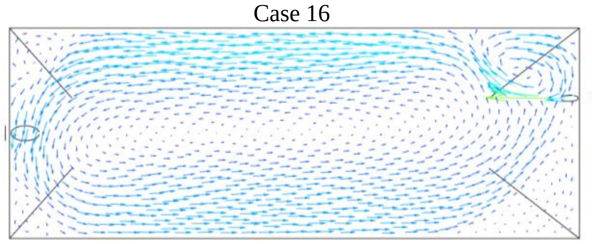

Case 13


Case 14


Case 15


Case 16


Case 17


Case 18

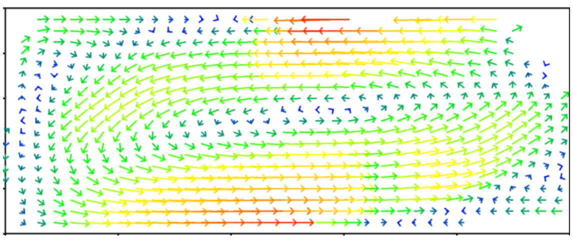

Case 19

Figure 1: Flow pattern of all cases

References

[1] Khan, S., Melville, B.W., Shamseldin, A.Y., Numerical investigation of the Reynolds Number dependence of the flow within model ponds at different geometric scale ratios in IAHR-APD 2010, Auckland, New Zealand.

[2] Kresta, S.M., Boundary conditions required for the CFD simulation of flows in stirred tanks, in Advances in Engineering Fluid Mechanics: Multiphase Reactor and Polymerization System Hydrodynamics, P.C. Nicholas, Editor. 1996, Gulf Professional Publishing: Burlington. p. 297-316.

[3] Shilton, A., Studies into the hydraulics of waste stabilization ponds, in Environmental Engineering. 2001, Massey University.