NUMERICAL INVESTIGATION OF THE REYNOLDS NUMBER DEPENDENCE OF THE FLOW WITHIN MODEL PONDS AT DIFFERENT GEOMETRIC SCALE RATIOS

Sher Khan[1], Bruce W. Melville[1], Roger Nokes[2], Asaad Y. Shamseldin[1], and Heide Friedrich[1]

[1]Department of Civil and Environmental Engineering, Faculty of Engineering, The University of Auckland, Private Bag 92019, Auckland 1142, New Zealand, Tel: (+ 64 9) 3737599 ext: 88503 Fax: (+ 64 9) 3737462; email: skha121@aucklanduni.ac.nz
[2]Department of Civil and Natural Resources Engineering, College of Engineering, University of Canterbury, New Zealand, Tel: (+64 3) 364 2987 ext: 7319 Fax: +64 3 364 2758 Email: roger.nokes@canterbury.ac.nz

Abstract

This paper addresses the numerical investigation of the Reynolds Number dependence of the flow in scale models of stormwater retention ponds. Scale modeling is a method to understand the hydrodynamics of field ponds. However, the basis of the scaling criteria for modeling the ponds hydraulics used in most of the previous research is questionable and hence its reliability is limited.

This paper for the first time elucidates the scaling issues focusing on the Reynolds Number dependence and geometric scale ratios. The overall conclusion of this paper is that Froude Number similarity should not be taken as the only key criterion for scaling the hydraulics of retention ponds, but careful consideration should be given also to the geometric scale ratios and/or the Reynolds Number independence.
Key Words: Retention ponds, Hydrodynamics, Numerical models, Reynolds Number

1. INTRODUCTION

The performance of stormwater retention ponds is directly linked to their hydrodynamics. To obtain hydraulic data from field ponds is difficult and the results may only be applicable for the conditions which were present during the field data collection. Further it is difficult to obtain repeatable data from field ponds. Due to these difficulties the use of scale models is a common practice to improve the understanding of pond hydraulics. However for meaningful results the model should be based on sound principles of hydraulic modeling and should have flow Reynolds Number above a certain value to represent the fully turbulent field flows. For accurate simulation of flow in a scale model, the model should display similarity of length ratio, velocity ratio and force ratio with those of the prototype. These ratios are called geometric, kinematic and dynamic similarities, respectively.

Geometric similarity is not enough to ensure the comparison of flow conditions between model and prototype. It is kinematic similarity which makes the flow conditions in a model scalable to those of the prototype. The combination of these two similarities leads to scale ratios of time, discharge and other variables between the model and the prototype.

To maintain kinematic similarity some dimensionless parameters (Froude Number, Reynolds Number) must have the same value in the model and the prototype. However it is not possible to maintain Froude Number and Reynolds Number similarity at the same time in the prototype and the model, if the fluid used is the same in both (i.e. water). Froude Number similarity implies $V_r = \sqrt{L_r}$ while Reynolds Number similarity implies $V_r = 1/L_r$, where V_r

and L_r are velocity ratio and length ratio between model and prototype respectively. Clearly these two equations are incompatible.

In most of the previous research regarding scale modeling of pond hydraulics, the compromise was made on the Reynolds Number similarity and the Froude Number was considered as the key criterion for scaling the kinematic similarities between model and prototype (Adamsson et al. 2005; Adamsson et al. 2002; Adamsson et al. 2003; Quarini et al. 1996; Shilton 2001).

However for small flow rates and/or laboratory models of very large field ponds, with Froude Number similarity the flow may be laminar with a very small Reynolds Number (Adamsson 2004; Shilton 2001). Hence the reliability of the model pond becomes questionable when applied to the full scale pond.

To the best of our knowledge, this paper for the first time focuses on the Reynolds Number dependence of the flow in laboratory model ponds in connection with different geometric scale ratios and Froude Number and Reynolds Number Similitude criteria. In this study the hydrodynamics of a field retention pond in the Auckland Region, New Zealand were studied using scale models at four different geometric scale ratios between model and prototype. For kinematic similarity between model and prototype all the models were operated first according to Froude Number similarity and then according to Reynolds Number similarity. Also, the Reynolds Number dependence was investigated under conditions where both Froude Number and Reynolds Number similitude was considered.

2. METHODOLOGY

2.1 Case studies

In this study the Reynolds Number dependence of the flow in laboratory models of a field retention pond was studied. Fourteen different cases were investigated having similar pond shape but with different sizes. The sizes were selected in such a way that the numerical models cover a large range of geometric scale ratios of model ponds from a very small size to a relatively large size of the models. The cases studied are as follows:

- Case A (Full scale pond with geometric scale ratio of 1:1)
- Case F1 (The full scale pond was scaled down at a geometric scale ratio of 1:40)
- Case F2 (The full scale pond was scaled down at a geometric scale ratio of 1:10)
- Case F3 (The full scale pond was scaled down at a geometric scale ratio of 1:7)
- Case F4 (The full scale pond was scaled down at a geometric scale ratio of 1:5)

The other cases were the same in their size and shape to the above cases. Cases R1, R2, R3 and R4 are similar to F1, F2, F3 and F4 respectively in their geometric scale ratios except that the 'R' cases were operated according to Reynolds Number similarity between model and prototype and the 'F' cases were operated according to the Froude Number similarity. A different case, Case R5, was also tested in which a compromise between Froude Number and Reynolds Number similarity was considered. The Case R5 was operated at 1.25 l/s discharge which was 4 times less discharge compared to its corresponding Case R2.
All the above cases were tested for a symmetrical inlet. However, the Cases F1, F2, R1 and R4 were also tested for asymmetrical inlet location and are referred to as $F1_{S,}$ $F2_{S,}$ $R1_S$ and $R4_S$.

2.2 Model set-up

For the present study, fourteen 3D numerical models were developed using Ansys-CFX (Baawain et al. 2006; Khan 2009a; Rasmussen and McLean 2004). To accurately model the hydrodynamics of the flow, the geometries of all the model ponds were kept similar to that used in the field. 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 ponds. The geometry of the full scale pond and the inlet and outlet dimensions were scaled down at four different geometric scale ratios. The geometric scale ratios tested were 1:40, 1:10, 1:7, 1:5 and 1:1 respectively. The flow rate and water level in the model ponds were measured first according to their geometric scale ratios and Froude Number similarity and then also according to Reynolds Number similarity. The dimensions of the full scale pond were: top length = 41m; top width = 15m; bottom length = 30m; bottom width = 5m; depth = 2.3m. The inlet and outlet pipe diameters were 450mm and 1050mm, respectively, while the inlet discharge was 50 l/s based on a storm event discharge in the field, around the Auckland Region, New Zealand.

For this study, the most robust boundary conditions were applied. The flow region to be modeled was identified as a 3D region. A constant mass flow rate was applied at the inlet while the outlet was modeled as a pressure outlet. The flow direction was set normal to the inlet boundary conditions and the turbulence intensity at inlet was set at a high level, a value of 30-40%. All other default 2D regions, like the sides and bottom of the pond, were modeled as walls with no slip boundary conditions. The top of the ponds is a free surface and was modeled as a symmetry surface for simplification, which provides free slip conditions for tangential fluid motion with no fluid motion across the surface (Adamsson 2004; Khan 2009b).

The well known k-ε turbulence model was used in this study. The simulation was undertaken for steady state conditions to obtain the solution for the three components of velocity, pressure, momentum, and turbulence components. The second order discretization scheme was used with a physical time step of 2 to 5 seconds.

3. RESULTS AND DISCUSSION

3.1 Flow dynamics

The flow regime within the model pond was measured for four different geometric scale ratios. The reference pond for this study was the Alpurt B2 Motorway flocculation pond in Auckland, New Zealand, which is the area under consideration for this study. Two dynamic similarities (Froude Number and Reynolds Number) were tested to investigate the Reynolds Number dependence of the flow in the model ponds. The inlet jet Reynolds Number (Rej) was calculated using inlet diameter and jet velocity and the Reynolds Number within the main circulation cell (Rec) was calculated using average velocity (averaged over the horizontal area of the pond) at 60% depth of the flow within the pond and flow depth.

The results show that the models based on Froude Number scaling criterion have Reynolds Numbers many orders of magnitude lower than the Reynolds Number of the prototype in both regions, i.e. the main circulation zone and the region near and around the inlet (Tables 1 and 3). The results also show that the Rec values are approximately 30% of the Rej values for both scaling criteria (Froude Number and Reynolds Number) for all the studied cases.

However, the Reynolds Number within the dead zones and the regions near walls are typically lower than 30% of Rej due to smaller velocities in these regions. The velocities in these regions are less accurate which make the estimation of Reynolds Number more difficult in these regions. The Reynolds Numbers (Rec) in the main circulation cell for Froude Number similarity based models were very low at larger geometric scale ratios and were found to be in the laminar flow range (Table 1, Case F1). The Reynolds Number in Cases F1 and $F1_S$ was only 146-147 indicating a completely laminar flow which is not representative of fully turbulent flow in the field with Reynolds Number above 10^5. Similarly the Reynolds Number for Cases F2 and $F2_s$ was relatively low and there is a chance that in regions of low velocities (dead zones) the flow may fall in the laminar regime for this value (Tables 1 and 3). Case F4 resulted in high enough Reynolds Number but this case is physically large and may be difficult to build in the laboratory due to limited space.

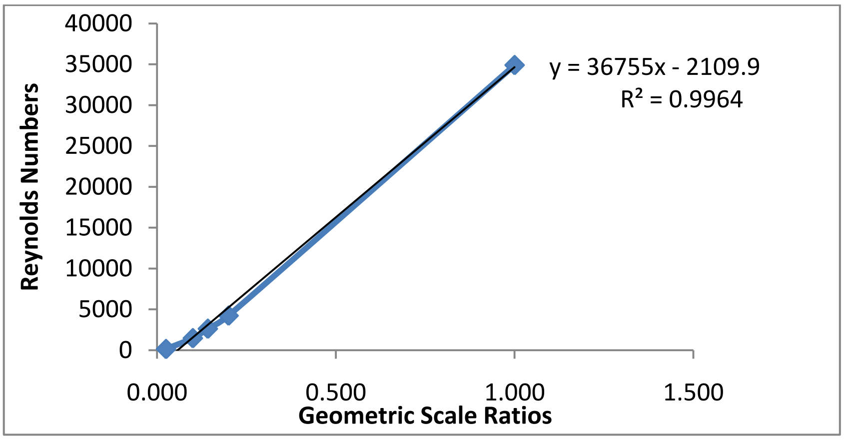


Fig 1. Reynolds Numbers in main circulation cell (Rec) at different geometric scale ratios for Froude Number similitude

The Reynolds Numbers (Rec) in model ponds with Froude Number similitude were found to increase with the increase in the size of the model pond. Figure 1 shows that there is a linear relationship between the geometric scale ratios and the Reynolds Numbers of the model ponds. The relationship can be described as $y = 36755x - 2109.9$ with $R^2 = 0.9964$, where y = Reynolds Number (Rec) within the main circulation cell and x = geometric scale ratio and R^2 = coefficient of determination with a range of 0-1. To avoid such situations where longer model ponds lead to higher Reynolds Number, all the above cases were again tested according to Reynolds Number similarity between model and prototype. The cases based on Reynolds Number similarity resulted in turbulent flow. In all cases, the Reynolds Number was above 1×10^4.

The main limitation with the use of the Reynolds Number similarity criterion is that it implies unnecessarily high velocities in the model pond which may cause erosion of the already settled particles and hence may cause difficulties while studying particle settling behaviour in a model pond. Reynolds Number similitude also leads to very short residence times due to high velocities in the model pond. The flow within the model pond will become Reynolds Number independent at a particular value of Reynolds Number. Operating the model in the Re-independent range would be advantageous. However, the value of Rec for completely turbulent flow is unknown. Case R5, which was operated under conditions where Froude Number and Reynolds Number similitude were both considered, resulted in 73-77% lower velocities compared to the corresponding Case R2 (Table 2). The Reynolds Number (Rec) for Case R5 remains above 1×10^4 within the main flow cell of the pond (Table 2), which is adequate to ensure fully turbulent flow.

Changing the symmetrical inlet to an asymmetrical inlet has a lesser effect on Rec. The Cases $R1_S$ and $R4_S$ have Rec very near to the Cases R1 and R4 and similarly the Cases $F1_S$ and $F2_S$ result in a Rec which is comparable to that for Cases F1 and F2.

3.2 Flow patterns

The simulated flow patterns for all the cases of symmetrical inlet are shown at three different depths in Figure 2. The flow patterns are depicted at 15%, 50% and 85% depth from the water surface. The flow paths are represented by stream lines. All the flow patterns for symmetrical inlet feature a strong jet, which attaches quickly to one of the walls. The jet induces either one large eddy near one of the walls or several eddies along the length of the pond. The swirling regions are unstable which make the flow very complex. The flow patterns are very similar at all depths for a given model pond size. However, the eddies and swirling regions were found to be stretched near the top compared to the bottom of the pond. In all the cases the eddies are relatively small near the bottom. Although the flow patterns are similar at all depths of the pond, the velocities were higher at the top compared to those near the bottom of the pond.

However, the flow patterns for a symmetrical inlet are inconsistent, which inherently results in an unstable jet. For the asymmetrical inlet, the flows result in a stable flow pattern for a wide range of geometric scale ratios. All cases result in one rotating cell (Figure 3). The comparison of Cases $F1_S$ and $R1_S$ in Figure 3 shows that, as the Reynolds Number of flow within the pond increases, the velocities at the circumference of the flow cell increase and the flow structure stretches towards walls. Once the Reynolds Number of the flow is above a certain value, the flow becomes stable irrespective of the size of the model pond (Figure 3). The results in Figures 4 and 5 show that there is no further change in velocity distribution for the cases of high Reynolds Number. The Cases $R1_S$ and $R4_S$ show the same velocity distribution, which also shows that the numerical model could be employed with a wide range of model pond sizes without geometric scale effects. Among other parameters, it is the flow rate or the Reynolds Number of the flow which establishes the flow pattern of the model pond. The flow pattern is consistent for Rec above a certain value (Figure 3). Cases $F1_S$ and $R1_S$ have the same dimensions but result in different velocity distributions due to the Reynolds Number of the flow within the model ponds. Case $F2_S$, which has a Reynolds Number = 1702, indicates the flow to be on the borderline for turbulent flow. The velocity distribution of Case $F2_S$ is different from that for Cases $R1_S$ to $R4_S$ and shows a velocity distribution which is more similar to Case $F1_S$.

Table 1. Reynolds Number within model ponds for Froude Number similitude

Case	Inlet velocity (m/s)	Outlet Velocity (m/s)	Average Velocity at 60% Depth (m/s)	Velocity at Bottom (m/s)	Inlet Jet Reynolds No. (Rej)	Reynolds No. at 60% Depth (Rec)
F1	0.055	0.0096	0.00255	0.0023	618.75	146
F2	0.1015	0.019	0.0064	0.0052	4567.5	1472
F3	0.13	0.025	0.008	0.007	8359	2628.8
F4	0.152	0.02611	0.0092	0.0086	13680	4232
A	0.342	0.06868	0.01517	0.012	153900	34891

Table 2. Reynolds Number within model ponds for Reynolds Number similitude

Case	Inlet Velocity (m/s)	Outlet Velocity (m/s)	Average Velocity at 60% Depth (m/s)	Velocity at Bottom (m/s)	Inlet Jet Reynolds No. (Rej)	Reynolds No. at 60% Depth (Rec)
R1	13.65	2.436	0.879	0.837	153562	50542
R2	3.2	0.607	0.234	0.2	144000	53820
R3	2.4	0.46	0.1822	0.1625	154320	59870
R4	1.699	0.285	0.1115	0.1089	152910	51290
R5	0.852	0.161	0.0531	0.052	38340	12213
A	0.342	0.06868	0.01517	0.012	153900	34891

Table 3. Reynolds Number within model ponds having asymmetrical inlet

Case	Inlet Velocity (m/s)	Outlet Velocity (m/s)	Average Velocity at 60% Depth (m/s)	Velocity at Bottom (m/s)	Inlet Jet Reynolds No. (Rej)	Reynolds No. at 60% Depth (Rec)
$R1_S$	13.57	3.02	0.974	0.6365	152662	56005
$R4_S$	1.69	0.39	0.12	0.076	152910	55200
$F1_S$	0.0535	0.0098	0.0026	0.00173	602	149
$F2_S$	0.1064	0.023	0.0074	0.0042	4788	1702

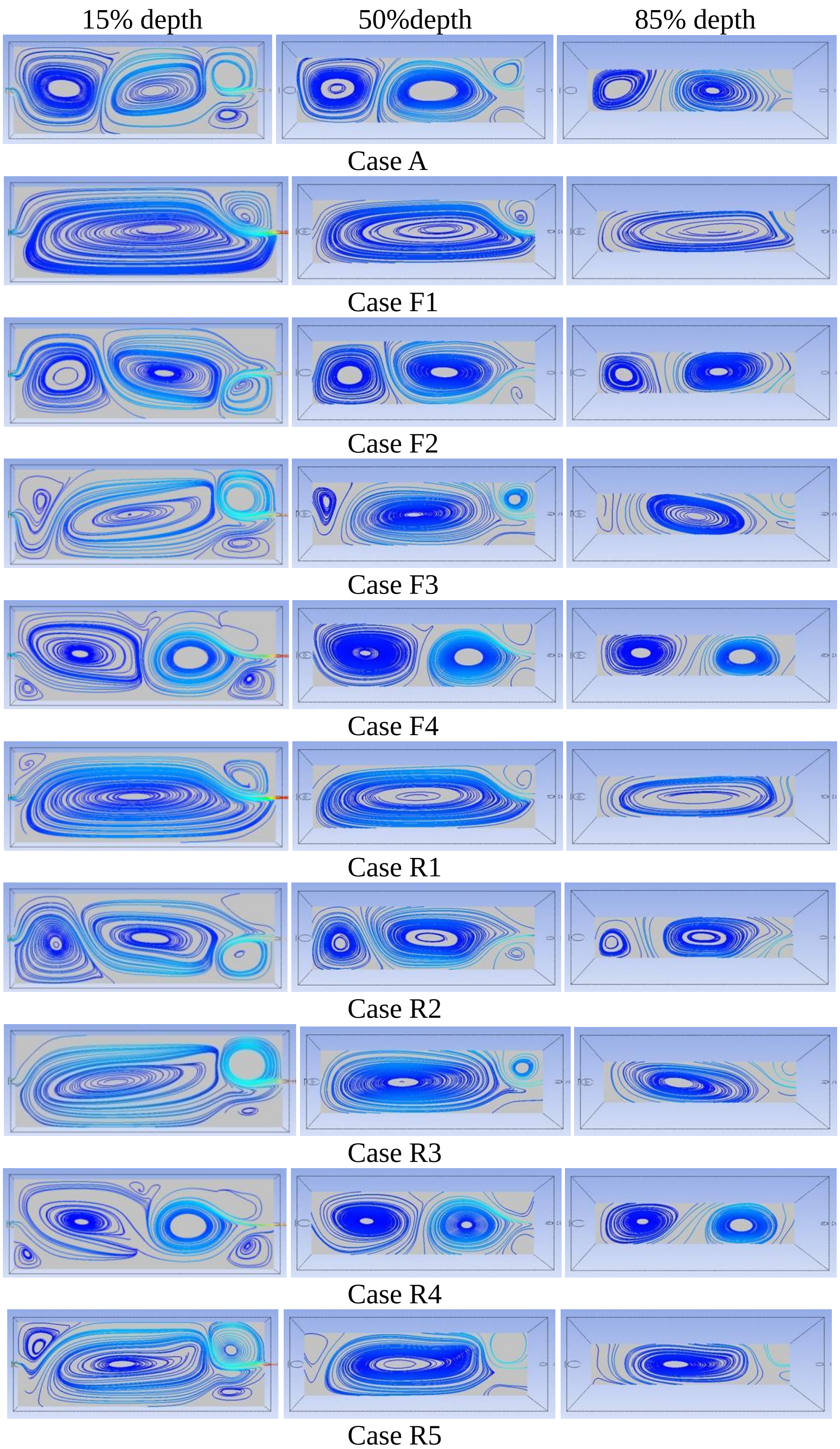
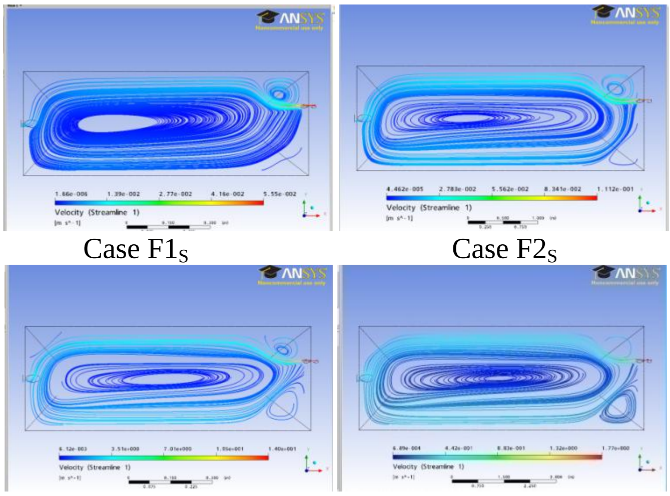


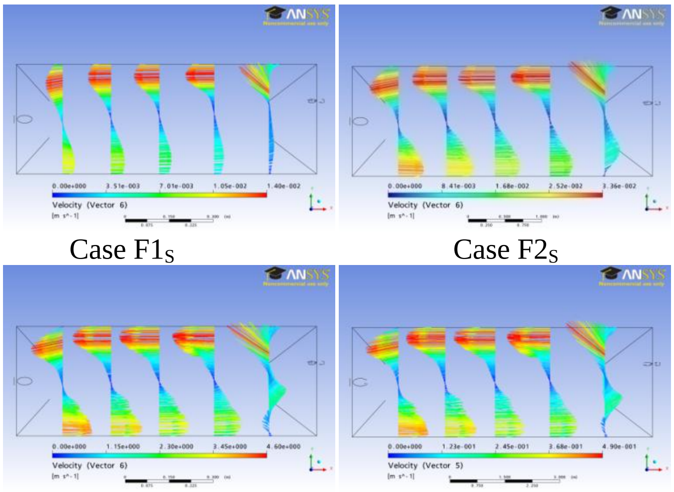
Fig 2. Simulated flow patterns for all the studied cases of symmetrical inlet



Case $F1_S$ Case $F2_S$

Case $R1_S$ Case $R4_S$

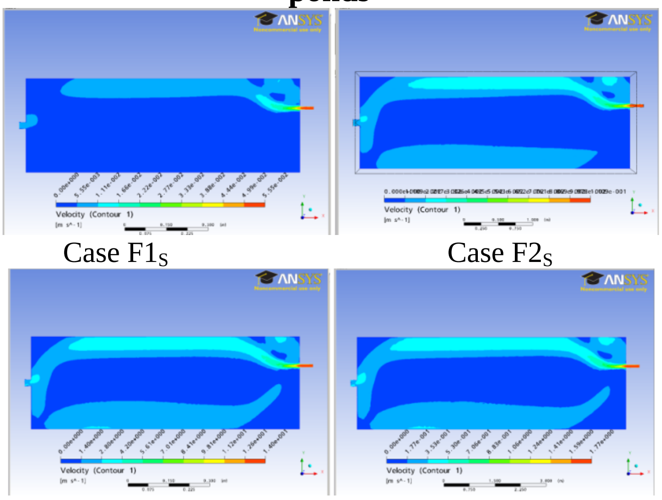
Fig 3. Simulated flow patterns for all the studied cases of asymmetrical inlet



Case $F1_S$ Case $F2_S$

Case $R1_S$ Case $R4_S$

Fig 4. X-sectional velocity distribution at five different locations of the model ponds



Case $F1_S$ Case $F2_S$

Case $R1_S$ Case $R4_S$

Fig 5. Velocity contour of the model ponds with asymmetrical inlet

4. CONCLUSIONS

The following conclusions can be drawn from this study:

1- For very large ponds, a model pond based on Froude Number similarity leads to flows in the laminar flow regime, which is not appropriate.

2- Froude Number should not be taken as the only key criterion for scale modeling of retention ponds.

3- Careful consideration should be given to Reynolds Number independence.

4- A numerical model could be employed with a wide range of model pond sizes without scale effects, provided the flow in the model pond is in the Reynolds Number independent regime.

REFERENCES

[1]. Adamsson, A. (2004). "Three-dimensional simulation and physical modelling of flows in detention tanks - Studies of flow pattern, residence time and sedimentation." *Doktorsavhandlingar vid Chalmers Tekniska Hogskola*(2116).

[2]. Adamsson, A., Bergdahl, L., and Lyngfelt, S. (2005). "Measurement and three-dimensional simulation of flow in a rectangular detention tank." *Urban Water Journal*, 2(4), 277-287.

[3]. Adamsson, A., Bergdahl, L., and Vikstrom, M. "A laboratory study of the effect of an island to extend residence time in a rectangular tank." *Global Solutions for Urban Drainage*, 1-10.

[4]. Adamsson, A., Stovin, V., and Bergdahl, L. (2003). "Bed shear stress boundary condition for storage tank sedimentation." *Journal of Environmental Engineering*, 129(7), 651-658.

[5]. Baawain, M. S., Gamal El-Din, M., and Smith, D. W. (2006). "Computational fluid dynamics application in modeling and improving the performance of a storage reservoir used as a contact chamber for microorganism inactivation." *Journal of Environmental Engineering and Science*, 5(2), 151-162.

[6]. Khan, S., Melville, B.W., Shamseldin, A.Y. (2009a). "Modeling the Layouts of Stormwater Retention Ponds using Residence Time." 4th IASME / WSEAS International Conference on Water Resources, Hydraulics & Hydrology (WHH 2009), Cambridge, U.K., 77-83.

[7]. Khan, S., Melville, B.W., Shamseldin, A.Y.,Sharma, R.N.,Nokes, R.I. (2009b). "Effect of Vertical Locations of Inlet and Outlet on Hydraulics of Stormwater Retention Ponds using Numerical Modeling." 33rd IAHR Congress: Water Engineering for a Sustainable Environment, Vancouver, British Columbia, Canada, 2627-2634.

[8]. Quarini, G., Innes, H., Smith, M., and Wise, D. (1996). "Hydrodynamic modelling of sedimentation tanks." *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, 210(2), 83-91.

[9]. Rasmussen, M. R., and McLean, E. (2004). "Comparison of two different methods for evaluating the hydrodynamic performance of an industrial-scale fish-rearing unit." *Aquaculture*, 242(1-4), 397-416.

[10]. Shilton, A. (2001). "Studies into the hudraulics of waste stabilization ponds," Massey University.