Reducing Sediment Deposition in a Clarification Tank Using Numerical Modeling

C. Yilmazer*[1], A. O. Celik[2], V. Kiricci[3]

[1] Department of Civil Engineering, Anadolu University, 26555, Eskisehir, Turkey.
(Corresponding Author's E-mail: cemyilmazer@anadolu.edu.tr)

[2] Department of Civil Engineering, Anadolu University, 26555, Eskisehir, Turkey.

[3] Department of Civil Engineering, Anadolu University, 26555, Eskisehir, Turkey.

ABSTRACT

This study aims to resolve a hydraulic engineering problem using Computational Fluid Dynamics (CFD) method. The problem is related to a sedimentation basin in a water treatment plant of ESKI (Eskisehir Water and Sewerage Administration) which is one of the municipal facilities in Turkey. Accumulation of sediment in distribution channels for the clarification tanks were reported to be generating operation-wise problems. The plant is entirely gravity driven and the flow conditions at various channels are difficult to control. The manual cleaning process of deposited sediment is required periodically during operation due to accumulated sediment in the distribution channels. This work puts an effort for detecting the problem and stopping the sediment deposition purely by simple geometrical improvement using numerical modeling. In this study, the main goal is to offer a reasonable solution based on basic hydraulic principles.

Sediment accumulation (as a result of deposition) is characterized by low stream velocity and also low turbulence kinetic energy. Based on the most recent suspended sediment theory, the developed local flow conditions under which the suspended particles start gravitating was identified and used as a criterion for controlling the flow conditions. The objective then was to hydraulically redesign the feeding channels for the clarification tank in an effort to increase the stream velocity and stop the early occurring sediment deposition. Low-velocity flow regions (at the downstream of the channel) were identified using CFD method. Also, initial conditions (water height) were identified and the CFD model was validated by a 1:10 scale physical model of the clarification tank. Consequently, by altering the geometry of the channel, these low power regions were activated in terms of suspension of sediment using contractions in the channel. The results are believed to be leading a low cost but effective solution to the problem which eliminates manual intervention during the treatment plant's operation.

Keywords: Clarification Tank, CFD, physical modelling, sedimentation, water treatment.

INTRODUCTION

This study aims to identify and resolve a hydraulic engineering problem with CFD method. The problem is about a sedimentation basin in a physical water treatment plant of ESKI (Eskisehir Water and Sewerage Administration) in Turkey.

The physical water treatment process has two integral parts; settling basin and clarification basin. The clarification tanks are widely used for water treatment plants. They are based on a physical process which suspends solid particles and not precipitate them as in a settling basin. Settling basins are ponds constructed for removing undesirable entrained solids by sedimentation. Clarifiers are tanks built with mechanical means for continuous removal of finer size solids being suspended by the water as shown in the upper left corner in Fig. 1. In this treatment plant, the problem is identified as the accumulation of sediment in distribution channels seen at the Fig. 1. The clarification tanks were generating operation-wise problems in the plant as the facility is entirely gravity driven and the flow conditions at various channels are difficult to control. Costly processes have been employed to remove the accumulated particles manually.

Figure 1. The clarification tank.

As a first step, the present system was examined to detect the source of the problem. In this stage, CFD method was performed for the clarification tank in the water treatment plant. ANSYS v.14 CFX was used as a solver. Initially, a computer aided drawing (CAD) model was generated representing the actual size of the clarification tank. Then, meshing process was applied to the flow domain for which a considerable time has been devoted in an effort to generate an accurate mesh in the entire domain. Following the meshing, proper boundary conditions were applied.

In this study, the main problem is low-velocity values at the downstream of the clarification tanks distribution channel. Low-velocity profiles cause sediment particles to settle. To solve this problem, velocity, as well as turbulence level, needs to be increased. In this model, the particle size is accepted as spherical with 0.1 mm diameter. The weight of solid particles is balanced by lift forces also directly related to stream velocity and particles settling velocity. Lift force must be equal to or greater than the weight of the solid particles which is supposed to be clarified completely from distribution channel to the clarifier. Lift force will be elaborated below.

To give background information, relevant studies in the literature will be reviewed here. Larsen (1977) is one of the researchers who employed a CFD model in studies involving clarification tanks [11]. Larsen (1977) identified the phenomenon which is also known as "density waterfall" that causes the fluid to fall down at bottom of the clarification tanks.

Shamber and Larock (1981) [13] used Finite Volume Method (VOF) with Navier-Stokes equations and they also used k-ε turbulence closure with the addition of solid concentration equation using a settling velocity. McCorquodale et al. (1991) [12] built a model using Finite Difference Methods (FDM) for the boundaries and Finite Element Methods (FEM) for flow functions. Krebs et al. (1995) [10] also investigated different inlet arrangements and assessed them as inlet baffle locations by Phoenics code. Deininger et al. (1998) [5] created the software, Champion3D, which is a numerical flow solver that could obtain velocity and solids distribution of a circular secondary clarifier system. Imam et al. (1983) [7] were able to determine vertical velocity profiles and an accurate model of vorticity transport flow functions with a constant eddy viscosity in a numerical simulation. Goula et al. (2008) investigated sedimentation tanks successfully using CFD method in potable water treatment [9]. A. Gkesouli et al. (2013) [6] formulated a validated CFD model to observe flow fields at tank inlet. They identified recirculation zones where the concentration of solid particles is uniformly distributed and concluded that as the flow rate increases, the concentrated regions shift to the outlet which causes a reduction in the efficiency of the tank.

CFD is a useful tool in design or rehabilitation processes for physical water treatment plant facilities. For a typical distribution channel, upstream has a higher velocity than downstream. Therefore, the improvement on the channel geometry will be performed for the only downstream. For fast, reliable and fluid mechanics based decisions on the modifications in the geometry, CFD method is employed in this study.

The clarification tank is connected with seven outlet pipes at the bottom. A plane was fixed to obtain area averaged velocity (stream-wise direction) in z-direction near these outlets. The plane allows determining average local velocities. These velocity values are compared against a critical velocity value associated with lift forces sufficient to keep the particles suspended. This approach can be used for various geometry changes until a satisfactory result is obtained.

MATERIALS AND METHODS

CFD method consists of four steps. Firstly, known geometry is drawn on CAD model by considering interacting solid parts with fluids. Secondly, the numerical network was created by separating the fluid domain into the small cells. The other step is to define the initial and boundary conditions. Finally, the numerical model is run by means of a solver. These steps will be elaborated below.

Geometry of The Clarification Tank

The clarification tank is a rectangular box with a 66 cm width (-x direction), 112 cm height (-y direction) and 1150 cm depth (z-direction) as shown in Fig. 2. The channel has seven identical outlet pipes with 10 cm diameter at the bottom of clarification tank, each 100 cm apart. The first outlet pipe is located 100 cm away from the inlet. Top of the clarification tank is open to the free entrance and exit of the air to the system.

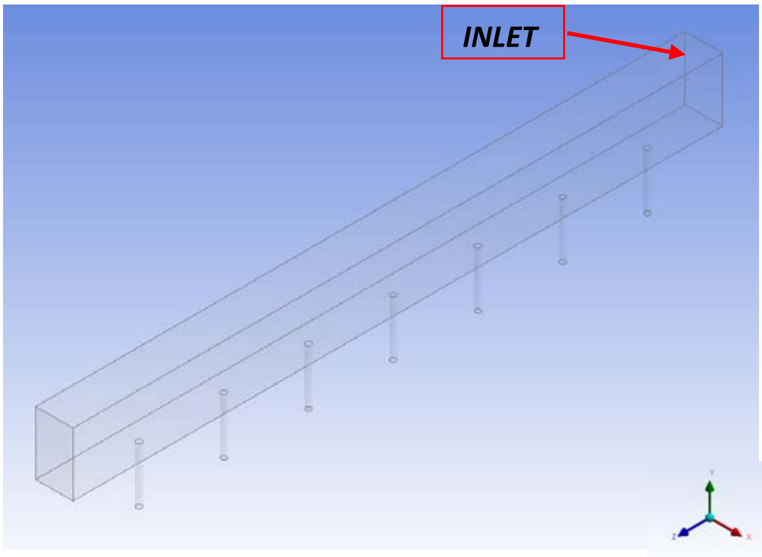

Figure 2. The CAD model of the clarification tank.

Meshing Process

One of the most critical points in CFD method is to obtain a proper mesh. The shape of the cells directly affects the accuracy of the solution. The mesh quality which is the key factor to reach to desired aim accurately as the important parameter. Related to this term, there are useful benchmark works such as the one performed by W.G.Habashi (2000) [8] regarding how the nodes in the mesh affect the result.

There are several factors that affect the mesh quality. "Skewness" is the major factor among the others. This study uses the Skewness parameter assessing the mesh quality. It is a rule of thumb in the literature now that the maximum Skewness shouldn't exceed 0.85 and also average value shouldn't exceed 0.25. The model meshed as shown in Fig.3 has 0.824 maximum and 0.204 average in Skewness respectively. In this study, the total number of elements and nodes are 4383872 and 1350712 respectively.

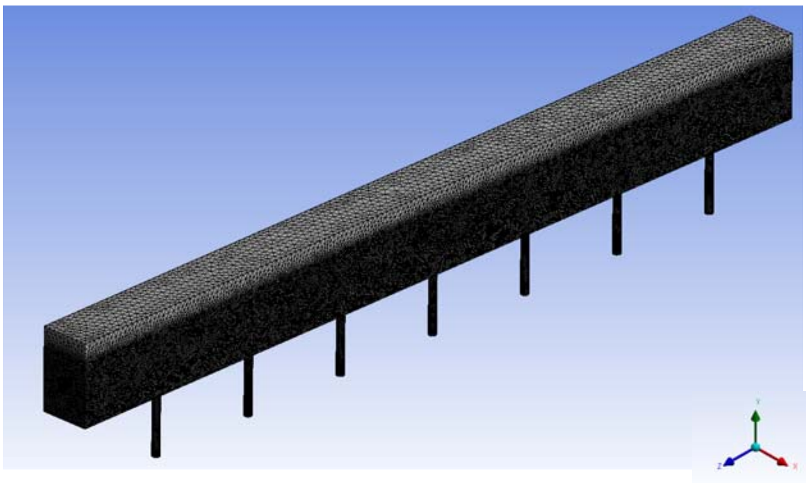

Figure 3. The view of the numerical grid (mesh) of the clarification tank.

Boundary Conditions

In the present work, as an initial condition, water height and operating discharge were identified by experiments on a 1:10 scale physical model of the clarification tank as shown in Fig. 4. With this approach, CFD model was also validated in terms of gross flow parameters. "Froude Model" assumes that the acceleration due to gravity is identical in both the model and the prototype [4]. 0.1, 0.2, 0.3, 0.4, 0.5, 0.6 lt/s of discharges were tried in the physical model. At 0.5 lt/s flow depths near outlet pipes became stable at 85 mm. At 0.6 lt/s of discharge, the model overflows. After the experiment, results were transformed to prototype values by appropriate scale factors which are listed as 157.92 lt/s of discharge and 85 cm of flow depth to be used in the CFD model.

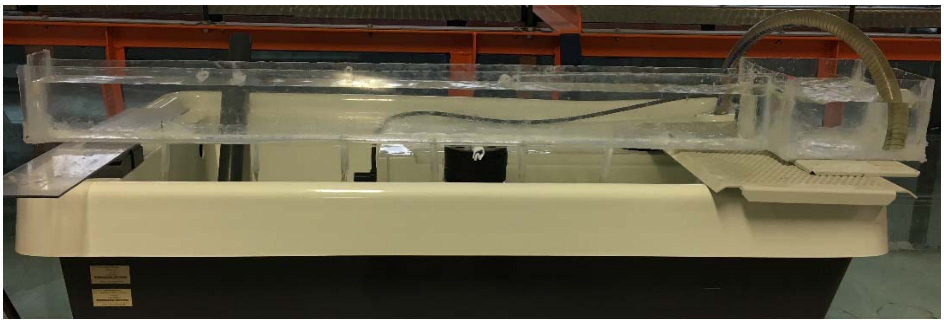

Figure 4. The 1:10 scale physical model of the clarification tank.

Top of the clarification tank is open to the atmosphere (1 atm pressure). Namely an "opening" type boundary is considered here. For observing the free surface, surface tension model is necessary to define fluid pair (water/air) interaction. In this numerical model, surface tension model was selected as continuum surface model. The continuum method eliminates the need for interface reconstruction, simplifies the calculation of surface tension, enables accurate modeling of two- and three- dimensional fluid flows driven by surface forces and does not impose any modeling restrictions on the number, complexity, or dynamic evolution of fluid interfaces having surface tension [3]. The surface tension coefficient is defined as 0.072 N/m to represent the interaction between water and air.

K-epsilon turbulence model was used as a closure in this study. "k" is the turbulence kinetic energy which means the variance of the fluctuations in velocity and epsilon (ε) is the turbulence eddy dissipation which is defined as the rate at which the velocity fluctuations dissipate. As described by J.E Bardina et al. (1997) [1] this closure is generally useful for free-shear layer streams which have small pressure gradients. Similarly, for wall-bounded and internal streams, if mean pressure gradients are small, k-epsilon gives reasonably well results.

To complete the boundary conditions on the model, outlet pipes were identified as an outlet with 1 atm pressure. This is also necessary to prevent a redundant solution because the inlet is defined as normal speed (value of which was obtained from the discharge and cross-sectional area from the physical model). Rest of the flow domain was assigned as a wall with sand grain roughness of 0.1 mm to reflect the concrete walls in the actual channel.

RESULTS AND DISCUSSION

At each outlet pipe in -xy plane, an intersecting plane was assigned to determine the water velocity profile. As mentioned above, velocity values must be equal or higher than threshold velocity values which ensure lift forces sufficient for sedimentation of particles as shown in equation 1 below.

$$F_{LIFT} = 1/2 \ \rho_W \ C_L \ U^2 A \tag{1}$$

In equation 1, "ρ_w" is the density of the water which equals to 1000 kg/m^3 "A" is the projected area of the solid particles and "C_L" is the lift coefficient (assumed 0.35 in this work). "U" is the superficial water velocity in z-direction acting on the projected area, A. Weight of those particles opposing the lift force can be computed by equation 2 below. In equation 2 below, "g" indicates the acceleration due to gravity, "r" is the radius of the spherical particle. According to situ investigations, the particles were reported to be silt whose density (ρ_s) equals to 1100 kg/m^3 [2].

$$W = 4/3 \ \pi \ r^3 \rho_S \ g \tag{2}$$

The approach was to model the channel as it is (Fig. 5 left image) and to verify the problematic zones. That is, before the modifications, the zones near the exit holes were investigated to determine flow velocities below the critical value. This way, the modification regions would also be determined. Subsequently, the modification was implemented on the geometry to increase the flow velocity in the zones where the deposition of sediment would potentially occur.

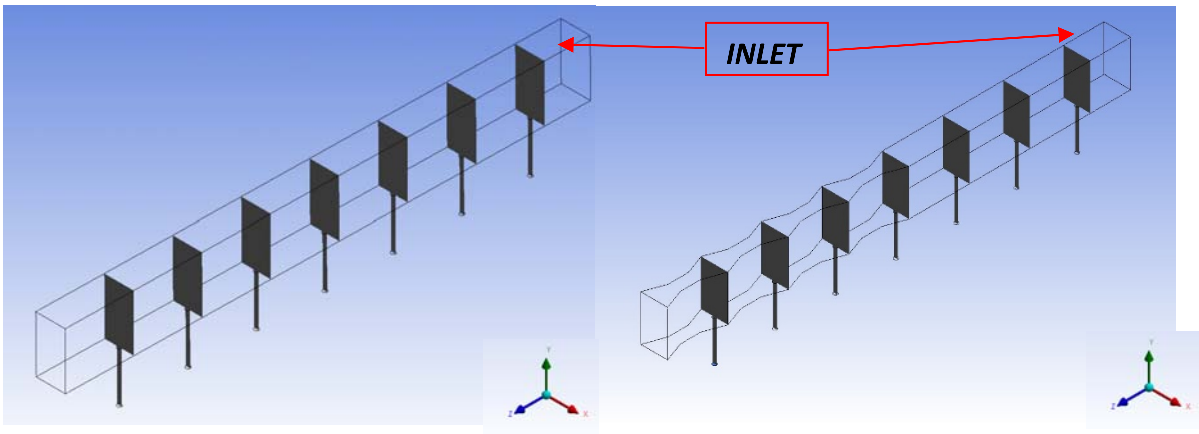

Figure 5. a) Planes in the x-y direction at each outlet pipes for unmodified geometry, b) Planes in the x-y direction at each outlet pipes for modified geometry.

Table 1 below summarizes the results for all 7 sections seen in Fig. 5 for both the original and the modified channel geometries. It is not surprising to see that the low-velocity regions are near planes 4-7. This is where the deposition actually occurs at the plant. In an effort to offer an effective and affordable solution, geometry near planes 4-7 was modified. Contractions as shown in Fig. 5 (right image) were applied at the channel walls to reduce the cross-sectional area and manipulate the flow. Table 1 summarizes the modified channel results as well and suggests even the most critical section, plane 7 has an average flow velocity above the critical. The effect of the modification on the flow is also given in Fig. 6. The contraction and the resulting increase in the flow velocity near the downstream of the channel is seen on the velocity contour figure.

Table 1. Distances from the inlet and superficial velocity at each plane for unmodified and modified geometry.

	Distance from The Inlet	**Superficial Velocity Z direction For unmodified model**	**Superficial Velocity Z direction For modified model**
	(cm)	**(m/s)**	**(m/s)**
Plane 1	100	0.196	0.271
Plane 2	250	0.159	0.243
Plane 3	400	0.124	0.212
Plane 4	550	0.091	0.172
Plane 5	700	0.061	0.148
Plane 6	850	0.034	0.112
Plane 7	1000	0.011	0.102

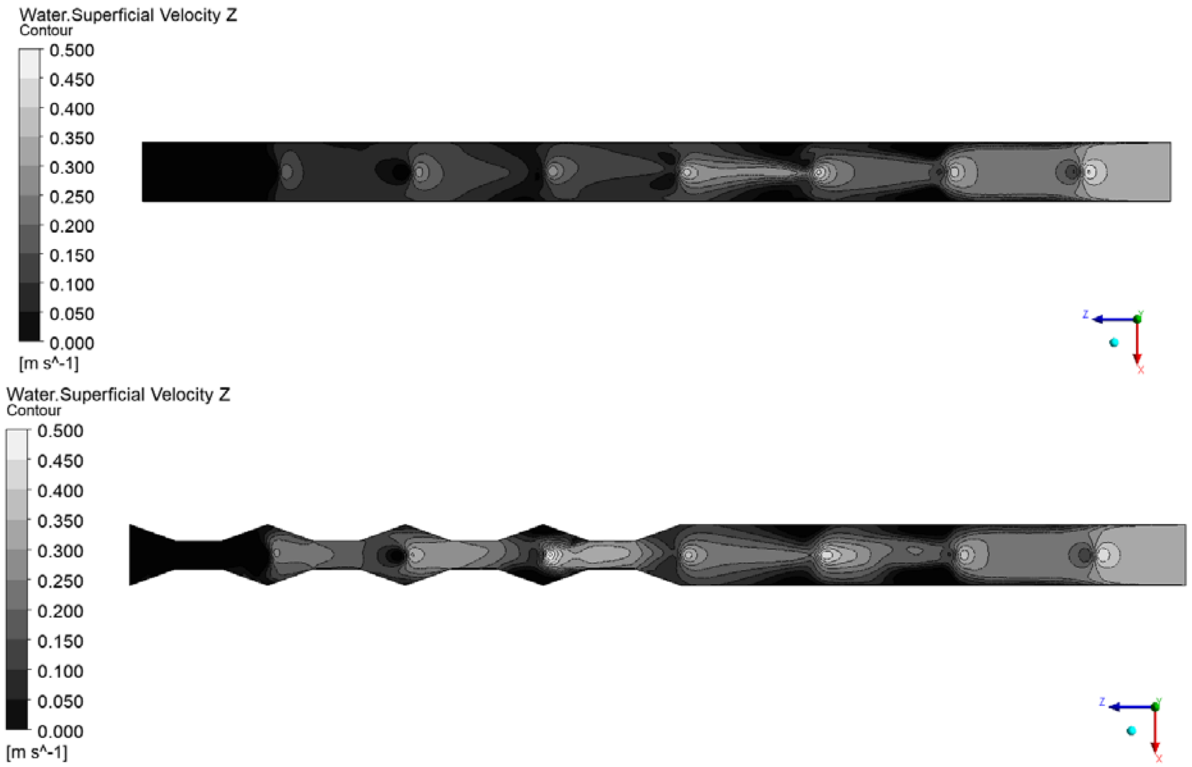

Figure 6. a) Superficial Velocity in Z direction at the x-z direction for existing geometry, b) Superficial Velocity in Z direction at the x-z direction for modified geometry.

CONCLUSIONS

This paper studied an engineering problem observed at a water treatment plant and offered a sound solution using CFD method. The existing tank was observed to have very low velocities near downstream end. A 1:10 scale physical model was set up to help determine the model initial and steady conditions. Boundary conditions were also determined from the experimental data and in situ investigation of the water treatment plant. The numerical model was also validated using the flow depths and discharge values from the physical tests. For each plane near outlet pipes, velocity profiles were investigated. Necessary geometry changes were done and their effect was assessed using again the validated CFD model. The results indicate that the suggested simple changes in the geometry create flow regions with higher average velocities which prevent deposition in the tank. If implemented, the plant operation will not be affected by the sediment deposition as the particles will remain suspended and discharged into the settling basin.

ACKNOWLEDGEMENT

The authors would also like to thank Senay Subasi and ESKI (Eskisehir Water and Sewerage Administration) for their assistance and guidance in acquiring the plant data.

REFERENCES

[1] Bardina, J.E., Huang P.G., Coakley T.J. 1997. *Turbulence Modeling Validation, Testing, And Development*, NASA Technical Memorandum 110446.

[2] Bayazit M., Avci I. 2010. *Akarsularda Akım ve Sediment Taşınımı (River Flow and Sediment Transportation)*, Birsen, Istanbul, Turkey.

[3] Brackbill J.U., Kothe D.B. and Zemach C. 1992. A Continuum Method for Modeling Surface Tension, *Journal of Computational Physics*, 100, 335-354. DOI: 10.1016/0021-9991(92)90240-Y.

[4] Chanson H. 1999. The Hydraulics of Open Channel Flow by Hubert Chanson, Published by Arnold, 338 Euston Road, London NW1 3BH, UK.

[5] Deininger A., Holthausen E., Wilderer P.A., 1998. Velocity and Solids Distribution In Circular Secondary Clarifiers: Full-Scale Measurements And Numerical Modeling, *Water Research*, 32, 2951–2958, DOI: 10.1016/S0043-1354(98)00072-4.

[6] Gkesouli A., Stamou A. I., Xanthaki M. and Georgiadis S. 2013. Validation of a CFD Model in Rectangular Settling Tanks, *Proceedings of the 13th International Conference on Environmental Science and Technology*, Athens, Greece.

[7] Goula A. M., Kostoglou M., Karapantsios T. D., Zouboulis A. I. 2008. A CFD Methodology for The Design of Sedimentation Tanks in Potable Water Treatment Case Study: The Influence of a Feed Flow Control Baffle, *Chemical Engineering Journal*, 140, 110–121, DOI: 10.1016/j.cej.2007.09.022.

[8] Habashia W. G., Dompierrea J., Bourgaulta Y., Ait-Ali-Yahiaa D., Fortinband M. and Vallet M. 2000. Anisotropic Mesh Adaptation: Towards User-Independent, Mesh-Independent and Solver-Independent CFD Part I: General Principles, *International Journal For Numerical Methods In Fluids*, 32, 725-744, DOI: 10.1002/(SICI)1097-0363(20000330)32:6<725::AID-FLD935>3.0.CO;2-4.

[9] Imam E., McCorquodale J.A., Bewtra J.K. 1983. Numerical Modeling of Sedimentation Tanks, *Journal of the Hydraulics Division*, ASCE 109, 1740–1754, DOI:10.1061/(ASCE)0733-9429(1983)109:12(1740).

[10] Krebs P., Vischer D., Gujer W. 1995. Inlet-Structure Design for Final Clarifiers, *Journal of Environmental Engineering*, ASCE 121, 558–564, DOI: 10.1061/(ASCE)0733-9372(1995)121:8(558).

[11] Larsen P. 1977. *On The Hydraulics of Rectangular Settling Basins*, Report No. 1001, Department of Water Research Engineering, Lund Institute of Technology, Lund, Sweden.

[12] McCorquodale J.A., Yuen E.M., Vitasovic Z., Samstag R. 1991. Numerical Simulation of Unsteady Conditions in Clarifiers, *Water Pollution Research Journal of Canada*, 26, 201–222.

[13] Shamber D.R., Larock B.E. 1981. Numerical Analysis of Flow in Sedimentation Basins, *Journal of the Hydraulics Division*, ASCE 107, 575–591.