JWSR

Model study reservoir flushing

Seyed Hossein Ghoreishi[1*], Mohammad Reza Majdzadeh Tabatabai[2]

[1,2]Assistant Professor of Water Engineering Department,
Shahid-Abbaspour University of Technology, Tehran, Iran.

ABSTRACT

   Sediment flushing of reservoirs is an operational technique, whereby previously accumulated sediments in the reservoirs are hydraulically removed by accelerated flow when the bottom desilting outlets of the dam are opened. In this research, the process of sediment flushing is simulated by a three dimensional numerical model in which sediment and flow interaction are reflected in the reservoirs. Reynolds Averaged Navier-Stokes (RANS) equations are solved numerically by Finite Volume on a three dimensional grid and a standard $k - \varepsilon$ turbulence model is used. The resulting flow analysis is used as an input data for the sediment model. The convection-diffusion equation for the sediment concentration is solved. The concentration equation derived by Van-Rijn is adopted as a boundary condition, resulting in a calculation of bed material load. The depth integrated mass balance equation is applied to find the bed changes. The results from the numerical model are compared favorably with the data from physical model studies available in the literature.

1. Introduction

   Reservoir sedimentation occurs worldwide at a rate of about 0.3 percent per year, but the sedimentation rate in many regions such as Asia is much higher estimated between 0.5 and 1.0 percent per year (Olesen, Basson 2004). Using an average rate, Palmieri estimated the loss to be approximately 45km^3 per year (Palmieri 2003). The cost of replacing the lost storage is significant; nearly US$13 billion per year would be needed, even without counting the environmental and social costs associated with new dams (Palmieri 2003). Hence, reservoir sedimentation and the corresponding loss of storage capacity is a common problem, which has attracted more and more attention in recent years.

   An engineering measure to reduce the sedimentation problem is to remove sediment hydraulically by flushing the reservoirs regularly; the effect of the flushing is highly varying in different reservoirs, and depends on the reservoir geometry, sediment size, magnitude of deposits, water discharge and water depth. In the planning of a dam and in dam operation it can be important to assess how much of the sediment can be removed, and how much flushing water is required. In most cases the effect of the flushing depends on the water discharge and the reservoir water level, which are functions of the capacity of the flushing gates. The design of the gates may include an optimization of the gate cost versus the economical value of the increased reservoir volume after flushing. Knowledge of the flushing process can therefore be important for the dam design.

[*] Corresponding Author Email: (ghoreishi@pwut.ac.ir)
Tel.: +98-9126226459

The flushing of the sediments involves several complex processes. Water flow field is three-dimensional, with complex recirculation zones and secondary currents. The turbulence is non-isotropic and varies throughout the computational domain. The erosion and sediment transport processes are also complex.

The flushing process can be studied in a physical model. This method is, however, relatively costly and time-consuming. The alternative is to adopt a numerical model. Then it is necessary to make some modeling simplifications because of the complexity of the flushing processes. Previously, a few one and two dimensional numerical models have developed to describe flow and sediment interaction during the flushing process. A one-dimensional numerical model has been used for studying flushing processes (Holly, Rahuel 1990, Basson, Rooseboom 1997). This will only give reasonable results as long as the reservoir is fairly long and narrow in shape. For a more complex geometry it is necessary to use a two or a three-dimensional numerical model. In the last few years the numerical models have increased the capability of assessing sedimentation problems for hydropower purposes (Lysne *et. al.* 1995). Ruland and Rouve used a two-dimensional finite element model to assess the risk of erosion in a reservoir in Germany (Ruland, Rouve 1992). Three-dimensional models have been used previously to calculate local scour (Olsen, Melaaen 1993), sediment deposition in a reservoir (Olsen *et. al.* 1994), and in a sand trap (Olsen, Skoglund 1994). A limited description of the two-dimensional numerical simulation of the flushing process was also given by Olsen (Olsen 1997), in another study the numerical model solved the depth-averaged Navier-Stokes equations on a two-dimensional grid and the resulting flow field was extrapolated to three-dimensions for solving the sediment

concentration in a flushing process (Olsen 1999).

However, due to the complexity of the problem, researchers have been encouraged to conduct further studies for a better understanding of this sophisticated process three-dimensionally. In the present study a three-dimension hydrodynamic model was developed. In this model, the conservation equations are discretized in a three-dimensional Cartesian coordinates using a staggered grid in terms of primitive variables. The primitive variables are velocity components, pressure and components of the extra-stress tensor. Further on, a sediment transport function was added to the model to enable that to compute sediment transported during flushing processes.

2. Water Flow Governing Equations

The mean turbulent flow of a viscous, incompressible, Newtonian fluid is considered. To deal with the turbulent flow, the Reynolds decomposition is used, where the instantaneous variables for the velocity filed and the pressure filed are decomposed into the addition of mean and fluctuating parts. The equations governing the mean velocities $\overline{U_i}$ and pressure $\overline{P}$ are obtained from the Reynolds averaged Navier-Stokes (RANS) equations:

(i) Mass conservation

$$\frac{\partial \overline{U_i}}{\partial x_i} = 0 \tag{1}$$

(ii) Momentum conservation

$$\frac{\partial \overline{U_i}}{\partial t} + \frac{\partial}{\partial x_j}(\overline{U_j U_i}) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}(\nu \frac{\partial \overline{U_i}}{\partial x_j} - \overline{u_i u_j}) \tag{2}$$

where ν and ρ are the kinematics viscosity and the density of the fluid respectively.

The quantity $\overline{u_i u_j}$ is the Reynolds stress tensor. Since these equations are not closed,

additional equations have to be written for $\overline{u_i u_j}$. In order to develop such equations, most of the closure schemes assume the following functional form:

$$\overline{u_i u_j}(x,t) = P_{ij}\left(\overline{U_i}(x,t), K(x,t), \mathcal{E}(x,t), x, t\right) \qquad (3)$$

In which k and ε are the turbulent kinetic energy and the dissipation rate, respectively. This kind of closure implies that two additional transport equations are required, one for k and the other for ε. Standard k-ε model is selected for this study.

The idea of Yakhot and Orszag was based on the infinite scale expansion in η, which is defined by the ratio of the turbulent time scale to the mean strain scale (Yakhot, Orszag 1986):

$$\frac{\partial k}{\partial t} + \overline{U}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) - \overline{u_i u_j}\,\overline{D}_{ij} - \varepsilon \qquad (4)$$

$$\frac{\partial \varepsilon}{\partial t} + \overline{U}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}\left(-\overline{u_i u_j}\,\overline{D}_{ij}\right)$$
$$- C_{\varepsilon 2}\frac{\varepsilon^2}{k} \qquad (5)$$

where

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \qquad (6)$$

and

$$\overline{D}_{ij} = \left(\partial\overline{U}_i/\partial x_j + \partial\overline{U}_j/\partial x_i\right)/2 \qquad (7)$$

where the above coefficients are defined as:

$C_\mu = 0.09, \quad \sigma_k = 1.00, \quad \sigma_\varepsilon = 1.30,$
$C_{\varepsilon 1} = 1.44, \quad C_{\varepsilon 2} = 1.92$

3. Sediment Transport Governing Equations

Three modes of particle motion are distinguished: rolling and/or sliding particle motion, saltating or hopping particle motion and suspended particle motion. When the value of the bed shear velocity just exceeds the critical value for initiation of motion, the bed material particles will be rolling and/or sliding in continuous contact with the bed. For increasing values of the bed shear velocity the particles will be moving along the bed by more or less regular jumps, which are called saltation. When the value of the bed shear velocity begins to exceed the fall velocity of the particles, the sediment particles can be lifted to a level at which the upward turbulent forces will be of comparable or higher order than the submerged weight of particles and as a result the particles may go into suspension.

Usually, the transport of particles by rolling, sliding and saltating is called bed load transport, while the suspended particles are transported as suspended load transport. Sum of bed and suspended load is called bed material load (total load).

Generally, the first stage of the numerical flushing model consists of the analysis of the existing data. During the second stage, flow models are applied to determine the hydraulic conditions in the existing and the future situation. The results of these models are then used as input data for the morphological model. Two different models can be distinguished:

- initial or sediment transport models which compute the sediment transport rates and the bed level changes for one time step, resulting in a short-term prediction;

- dynamic models which compute the flow velocities, the sediment transport rates, the bed level changes and again the new flow velocities, etc. In a continuous sequence (loop) resulting in long-term predictions, that this model was applied in this study.

In this study, the total load is computed by coupling convection-diffusion equation and Van-Rijn reference concentration equation.

The suspended load is calculated by solving the convection-diffusion equation for the sediment concentration,

$$\frac{\partial c}{\partial t} + U_j \frac{\partial c}{\partial x_j} + \omega(\frac{\partial c}{\partial x_3}) = \frac{\partial}{\partial x_j}(\Gamma \frac{\partial c}{\partial x_j})$$

(8)

(j = 1,2,3)

The fall velocity of the sediment particles is denoted ω, the diffusion coefficient, Γ, is assumed to be equal to the eddy viscosity, v_t, described previously in equation (4), and for laminar flow it can assumed equal cinematic viscosity.

The first term on the left side of the top equation is the transient term. The next term is the convection of sediments, i.e. the sediment flux through the walls of the finite volume because of the velocity of the water at the wall. The third term on the left side is due to the fall velocity of the sediments. This is an extra convective term, added to the velocities in the vertical direction. The index 3 denotes the vertical direction. The right hand term in equation (3) is the diffusion of sediment. For this study, Γ is the diffusion coefficient due to the mixing by turbulence in the water. The term therefore tells how much sediments are transported through the sides of the finite volume because of turbulence and the concentration gradient.

By solving the convection-diffusion equation, sediment concentration for an individual cell is satisfied. Van-Rijn reference concentration equation was applied for all cells in the bed as a boundary condition. Therefore the convection-diffusion equation was not used for the cells in the bed. This was initially suggested by Einstein as an equilibrium concentration in the cells to the bed (Einstein 1950).

Hence Van-Rijn equation is (Van-Rijn 1987):

$$C_a = 0.015 \frac{d_{50}}{a} \frac{T^{1.5}}{D_*^{0.3}}$$

(9)

The parameter *a* is the reference level above the mean bed , the mean sediment par-

ticle diameter is denoted d_{50}, $T = (\tau - \tau_c)/\tau_c$, where τ is the shear stress, τ_c is the critical shear stress for movements of sediment particles according to Shield's formula, and D_* is given by:

$$D_* = d_{50}\left[(G_s - 1)g/v^2\right]^{\frac{1}{3}}$$

(10)

where, G_s is specific density and *v* is the kinematics viscosity of water.

Bed level changes were obtained from the depth-integrated mass-balance equation (Van-Rijn 1993), yielding:

$$(1-p)\frac{\partial Z_b}{\partial t} + \frac{\partial}{\partial x}(q_{b,x} + q_{s,x})$$
$$+ \frac{\partial}{\partial y}(q_{b,y} + q_{s,y}) = 0$$

(11)

Where Z_b is bed level with respect to a horizontal datum, *p* is porosity factor, $q_{b,x}$ and $q_{b,y}$ are volumetric bed load transport rates and $q_{s,x}$, $q_{s,y}$ are volumetric suspended load transport rates. The suspended load transport rates are given by:

$$q_{s,x} = \int_a^h (uc - \Gamma \frac{\partial c}{\partial x})dz$$
$$q_{s,y} = \int_a^h (vc - \Gamma \frac{\partial c}{\partial y})dz$$

(12)

Where *a* is the reference level above the mean bed and *h* is the water depth

4. Numerical Method

The method used to solve the set of equations is of the classical finite volume type. The conservation equations are integrated over a control volume, and then the Gauss theorem is used to transform the volume integrals into surface integrals. The method uses a staggered mesh; the pressure and the normal Reynolds stress components are treated in the center of the control volume; the velocities are computed in the center of

the faces and the cross components of the Reynolds tensor are attached to nodes located at the mid-edges. The quadratic upstream interpolation scheme for the convective kinematics (QUICK) from Leonard (1979) is used to evaluate the advection terms for the momentum equation.

The stationary solution was obtained by a time-marching algorithm. The convective, diffusive, production and dissipation terms of the different transport equations are treated by an explicit Euler scheme. The advection terms in the K-ε equation are discretized in space using the first order upwind scheme. The diffusion terms are discretized with a second-order cell centered scheme. The decoupling procedure for the pressure is derived from the Mark and Cell (MAC) algorithm proposed in Harlow and Welch (Harlow, Welch 1965).

The method of solution consists in substituting the equation for the velocity at the new time within the discretized equation for mass conservation at the same time level. With this method, the pressure at the time   is obtained by the resolution of the discretized Poisson equation. Then, the velocities are calculated from the momentum equation. The solution consists in velocity and pressure fields that respect the conservation equations. The linear system for the pressure is solved using the Cholesky procedure because the matrix of this system is positive definite and symmetric.

For the first time-step, at the inlet of the duct, a constant profile was given to U, K and ε. The secondary velocities were initialized as nil ($V = 0$ and $W = 0$) all over the domain. The K and ε inlet values were obtained from the DNS data using the u_{rms} and the eddy viscosity, that was about four times the molecular viscosity.

At the cutlet, a homogeneous Neumann boundary condition was used for all variables. In the next time-step, the calculated

outlet values were used for the inlet condition. The same procedure is used for the following time-steps up to the convergence.

The boundary condition values for K and ε, at the first grid point near the wall, were calculated taking into account the fact that this point was in the viscous sub layer. Also, due to the use of a staggered grid, the value of K and ε are not defined on the wall. At the wall, the boundary conditions for the equations of K and ε were (Patel *et. al.* 1984):

$$\frac{\partial k}{\partial y} = 0 \quad \text{or} \quad \frac{\partial k}{\partial z} = 0,$$

$$\varepsilon = v\frac{\partial^2 k}{\partial y^2} \quad \text{or} \quad \varepsilon = v\frac{\partial^2 k}{\partial z^2}. \tag{13}$$

Initial water surface was used as a rigid lid, and it is an assumption where water level at the reservoir is lowered during flushing process in present study. Any way this lid was adjusted after each time step based outflow of sluice gate(s), inflow and actual volume of reservoir.

The numerical method used to solve the convection-diffusion sediment concentration equation is a finite volume technique and the method uses a staggered mesh; the sediment concentration components are computed in the center of the control volume and the extra-stress components and the velocities that results of flow model, are attached similar above for water flow method .

To discrete the equation, the hybrid differencing and the fully implicit schemes are used. The hybrid differencing scheme is based on a combination of central and upwind differencing scheme Spalding (Spalding 1972). The central differencing scheme, which is accurate to second-order, is employed for small Peclet numbers (Pe<2) and the upwind scheme, which is accurate to first order but accounts for transportiveness is employed for large Peclet numbers (Pe≥2).

For discretization the depth-integrated mass-balance equation by finite volume me-

thod is used the integrated form of this equation over a tow-dimensional scalar control volume .

To specify the boundary conditions of the transport model, information of the bathymetry, water depths and sediment characteristics (size, density etc.) is required. The most fundamental boundary condition is the process that controls the exchange of sediment particles at the bed. In the present study Van-Rijn equation was used (equation 9).

Sediment concentrations at inlet boundary are given and the other boundary conditions that were applied are (Van-Rijn 1993):
Outlet and solids boundary :

$$\frac{\partial}{\partial n}\left(\Gamma\frac{\partial C}{\partial n}\right) = 0 \qquad (14)$$

Water surface:

$$\omega_s C + \Gamma\frac{\partial C}{\partial z} = 0 \qquad (15)$$

5. Numerical model verification

The accuracy of the numerical model was assessed by comparing with experimental data obtained from a physical model study by Lai and Shen (Lai, Shen 1995). The experiments were conducted in the Hydraulic Laboratory at the Richmond Field Station of the University of California at Berkeley. A concrete rectangular flume with dimensions 50 m long, 2.44m wide, and 1.52m high was designed to model a reservoir. Part of the bottom of the flume was elevated up to 0.6m to avoid submerging the sluicing outlet. Sediment was paved in a 9m reach upstream from the dam. Three sluice gates (15cm in width) and eight sluice pipes were installed on the dam. In the present study results of two sluice gates were only used.

The sediment material used in the experiments was uniform size walnut shell grit. This non-cohesive lightweight material can satisfactorily simulate the relatively fine se-

diment particles transported in reservoirs and are light enough to be moved with slow velocities. The walnut shell grit has a specific gravity of 1.39, a median diameter of 1.25 mm, and its porosity is 0.55. In addition, the gradation coefficient is 1.18, and the angle of repose for submerged walnut shell grits is about 35 degrees. Eight runs were conducted by Lai and Shen (Lai, Shen 1995). 7 runs out of 8 were conducted with the central gate while the last one was undertaken with two central and right gates with 0.1m opening, 0.59lit/sec inflow discharge, 0.122m initial water stage above dam and the gates were kept open for 30 minutes. However, herein this verification the latest was used.

6. Comparison of experimental and predicted results

In order to compare the results of the numerical model with the described model, a reservoir by a computational domain of $35\times 31\times 11$ grid points in longitudinal (x), lateral (y) and vertical (z) directions was used, respectively. The characteristics of the simulated reservoir as well as the sediment and flow conditions were nearly the same as the physical model (Fig. 1). Flushing duration of the intended physical model was 30 minutes with the first 7-10 minutes of the flushing duration was under pressure flushing. Then, the flushing condition approached the free flow or empty condition where water level drops below the apex of the deposition in the vicinity of the dam. In fact, water surface elevation is variable along the profile, while the bottom outlets are kept consistently submerging (Lai, Shen 1995). Since the numerical model assumes the water surface as a rigid lid, it was impossible to truly simulate the free flow flushing. To simulate free flow flushing, while keeping the outlets submerged, the water level was assumed to be over the sediment deposition by a small depth. Fig.(1) show the computed and meas-

ured values for sediment outflow in numerical and physical models .

In an experimental model, the phenomenon of mass movement of the sediments is highlighted in the early stages of the physical and numerical models. In fact, there is a reasonable agreement between observed and predicted outflow sediment discharge throughout the experiment (Fig. 1).

Bed level variations were displayed by drawing contour lines for observed and predicted results (Fig. 2). In these figures the predicted contour lines were not exactly the same as the observed ones. This is reflected by the simulation of free flow flushing. As free flow flushing occurs in nature and can easily be simulated in laboratories, however, its numerical simulation dose not appears just as easy. Nevertheless, is free flow flushing numerical simulated by decreasing water surface elevation to small depth above the sediment deposition and water surface slope variations were considered negligible and assumed to be rigid lid. As it is noticed in Fig. (2), the observed flushed eroded channel developed longitudinally upstream by head cutting as a result of retrogressive erosion, whereas the predicted flushed channel developed laterally more and longitudinally less than that of the experimental. This may be justified by taking water surface as a rigid lid in the numerical model. In Fig. (2), this appears approximately the same as before for the observed results and skewed towards second bottom outlet for the predicted results which seems to be more realistic. Another reason for results discrepancy is due to the fact that in the experimental model retrogressive erosion was initialed in the small channel that was cut through the sediment deposited in the flume before the start of the experiments .

7. Conclusion

The numerical model was able to simulate the flushing of a reservoir. This included
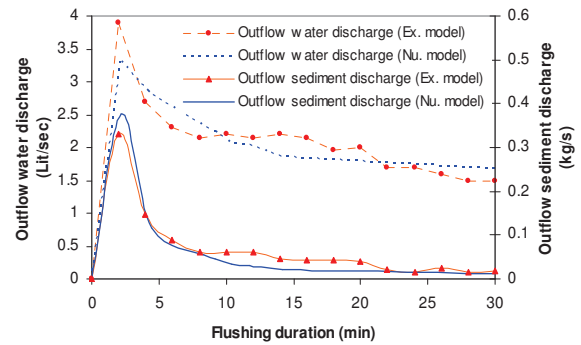

Fig.1: Numerical results in comparison with experimental results
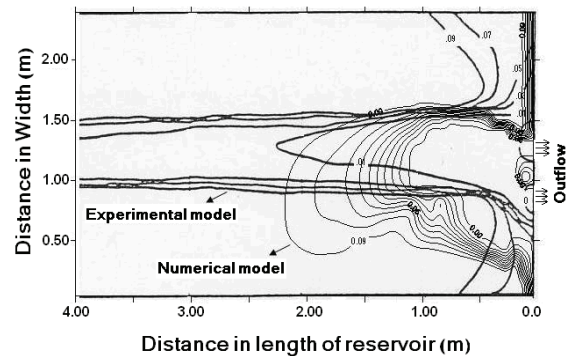

Fig.2: Contour maps of final reservoir bed profile for experimental and numerical models

complex phenomena such as drawdown of water surface duration of flushing. Comparison with the physical model study showed that sediment flushed in two models are same. The main features of the erosion pattern had been reproduced, chiefly in pressure flushing. The main deviation between measured and calculated in bed level profiles was most likely due to rigid-lid assume for water surface. Future research on sediment flushing from reservoirs with complex geometries should therefore include an actual water surface profiles.

References

Basson GR, Rooseboom A (1997) Dealing with reservoir sedimentation. ICOLD report.

Einstein HA (1950) The bed-load function for sediment transportation in open-channel flow.Technical Bulletin 1026, US Department of Agriculture, Soil Conservation Service, Washington DC.

Harlow FH, Welch JE (1965) Numerical calculation of time-dependent viscous incompressible flow or fluid with free surface. Physics of Fluids 8: 2182-2189.

Holly FM, Rahuel JL (1990) New numerical/physical framework for mobile-bed modeling.IAHR Journal of Hydraulic Research 28(5).

Lai JS, Shen HW (1995) An experiment study on reservoir drawdown flushing. International Journal of Sediment Research 10(3): 19-36.

Lysne DK, Olsen NRB, Stole H, Jacobsen T (1995) Withdrawal of water from sediment carrying rivers.Recent developments in planning and operation of headworks. International Journal on Hydropower and Dams, March.

Olesen KW, Basson G (2004) A 2D model for reservoir sedimentation and flushing. Conference of Hydraulics of Dams in Iran.

Olsen NRB (1997) Computational fluid dynamics as a tool for prediction of sedimentation and erosion in reservoirs. ICOLD Conference, Florence, Italy.

Olsen NRB (1999) Two-dimensional numerical modeling of flushing processes in water re servoirs. IAHR Journal of Hydraulic Research 37(1).

Olsen NRB, Melaaen M C (1993) Three-dimensional Numerical modeling of scour around cylinders. ASCE Journal of Hydraulic Engineering 119(9).

Olsen NRB, Skoglund M (1994) Three-dimensional Numerical modeling of water and sediment flow in a sand trap. IAHR Journal of Hydraulic Research 6.

Olsen, NRB., Jimenez O, Lovvoll A, Abrahumsen L (1994) Calculation of water and sediment flow in hydropower reservoirs. 1st International Conference on Modeling, Testing and Monitoring for Hydro Powerplants, Hungary.

Palmieri A (2003) Social and economic aspects of reservoir conservation. World Water Forum, Kyoto, Japan.

Patel CV, Rodi W, Scheuerer G (1984) Turbulence models for near-wall and low Reynolds number flows: a review. AIAA Journal 23: 1308-1319.

Ruland P, Rouve G (1992) Risk assessment of sediment erosion in river basins. Fifth International Symposium on River Sedimentation, Karlsruhe, Germany.

Spalding DB (1972) A novel finite-difference formulation for differential expressions involving both first and second derivatives. Int. J. Number. Methods Eng. 4: 551-560

Van-Rijn LC (1987) Matematical modelling of morphological processes in the case of suspended sediment transport. Delft Techn Uni, The Netherlands, Delft Hydraulics Communication 382.

Van-Rijn LC (1993) Principles of sediment transport in river: estuaries and coastal seas. Aqua Publication I11.

Yakhot V, Orszag SA (1986) Renormalization group analysis of turbulence. I. Basic theory. Journal of Scientific Computations 1: 3-51