3D Simulation of Flow over Flip Buckets at Dams

M. Jorabloo[1], R. Maghsoodi[2], H. Sarkardeh[3]

[1] Assistant Professor, Garmsar Branch, Islamic Azad University, Garmsar, Iran.
[2] Department of Engineering, Sabzevar Branch, Islamic Azad University, Sabzevar, Iran.
[3] Hydraulic Structures Division, Water Research Institute (WRI), Tehran, Iran.
jorabloo.mehdi@yahoo.com

Abstract: In the present numerical study, by using the Fluent software, the ability of it to predict the complex flow conditions is presented. In this purpose, experimental data over a flip bucket in different hydraulic conditions were selected. To simulate the turbulence phenomenon, *k-ε* Standard turbulence model was selected. Moreover to predict jet surface the VOF free surface model was employed. Finally by comprising the numerical model and available experimental results, a good agreement was observed.

[M. Jorabloo, R. Maghsoodi, H. Sarkardeh. **3D Simulation of Flow over Flip Buckets at Dams.** Journal of American Science 2011;7(6):931-936]. (ISSN: 1545-1003). http://www.americanscience.org.

Keywords: Flip Bucket; Fluent Software; VOF Free Surface Model; k-ε Standard Turbulence Model.

1. Introduction

The energy dissipater structure for a spillway could be a flip bucket because it is generally the most economical solution. Improving in design parameters of a flip bucket will protect the tailrace from scouring of impacted jet. Analysis of flow with free surface which passes over a curved boundary by the gravity force is a challenging problem in hydrodynamics. From 1933 till 1954, United State Bureau of Reclamation (USBR) regarding to hydraulic model tests, covered a complete range of bucket sizes and tail water elevations, were conducted to verify the bucket dimensions and details and to establish general relations between bucket size, discharge capacity, height of fall, and the maximum and minimum tail water depth limits. Analytical studies on potential flows past spillway flip buckets were done by many researchers such as: Siao and Hubbard (1953), Tinney et al. (1961), Orlov (1968) Siao et al. (1988). Xie-Qing Xu and Xiao-Xia Sun (1990), by using the finite element method, obtained pressure distribution over spillway and flow bucket. Vischer and Hager (1998) proposed that flip buckets are used when energy has to be dissipated for a flow velocity larger than about 15–20 m/s. Roman Juon and Willi H. Hager (2000) and Valentin Heller et al. (2005) and Remo Steiner et al. (2008) performed some investigations on flip buckets, including scale effects in hydraulic models, bucket pressure distribution, and nappe trajectories with and without the presence of deflectors.

Regarding to the previous researches, in the present study by using the FLUENT Software, flow over the flip buckets at dams was numerically studied.

2. Experimental Data Collected

Selected experimental tests for this numerical study were conducted in a smooth channel by Roman Juon and Willi H. Hager (2000). The selected test cases include eight fixed-bed cases. All experimental tests were conducted in a flume with 499 mm wide and 700 mm deep with a total length of 7 m (Figure 1).

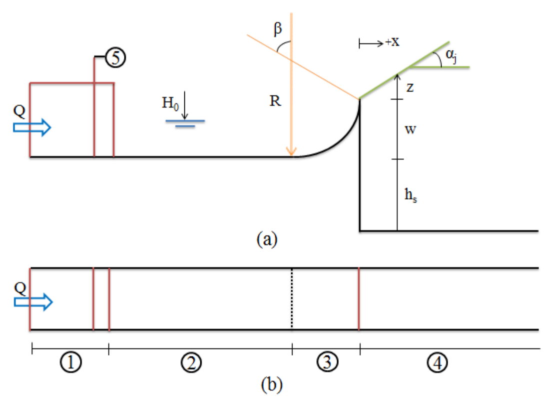


Figure 1. Schematic view of the selected experimental setup: (a) Side view (b) Plan view

In the Figure 1, notification numbers are as 1) Jet box, 2) Approach channel, 3) Flip bucket, 4) Downstream channel, 5) Start of chute. Other parameters are given in Table 1 ($F_0=V_0/(gH_0)^{1/2}$).

The channel had a PVC invert and right wall and a left glass wall. The flip bucket consisted of a 1 m long approach channel with a bucket of radius R and deflection angle β. The approach channel was inserted 250 mm above the original channel invert.

Table 1. Selected Experimental Parameters

	R (cm)	F_0	H_0
Test 1	20	5	5
Test 2	20	4	6
Test 3	20	4	5
Test 4	20	3	6
Test 5	20	3	5
Test 6	25	5	5
Test 7	25	4	5
Test 8	25	3	5

3. Numerical Modeling

In this section the turbulence model which used in the present research is described. In Reynolds averaging for the velocity components:

$$u_i = \bar{u}_i + u_i' \tag{1}$$

Where $\bar{u}_i$ and u_i' are the mean and fluctuating velocity components. Substituting expressions of this form for the flow variables into the instantaneous continuity and momentum equations and simplifying (and dropping the over bar on the mean velocity, $\bar{u}$):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) =$$

$$-\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \tag{3}$$

where equations 1 and 2 are called Reynolds-averaged Navier-Stokes (RANS) equations that $-\rho\overline{u_i' u_j'}$ is called Reynolds stresses, must be modeled. by using the Boussinesq hypothesis (Hinze, 1975) relate the Reynolds stresses to the mean velocity gradients:

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t \frac{\partial u_i}{\partial x_i}\right)\delta_{ij} \tag{4}$$

The Boussinesq hypothesis is used in the k-ε models. In the present work the Standard k-ε model (Launder and Spalding, 1972) was used to simulate the turbulence phenomenon. For Modeling the effective viscosity:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

where C_μ is a constant, k is the turbulence kinetic, and ε is the turbulence rate of dissipation. The transport equations for the Standard k-ε model are as follow:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) =$$

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) =$$

$$\frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M \tag{6}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) =$$

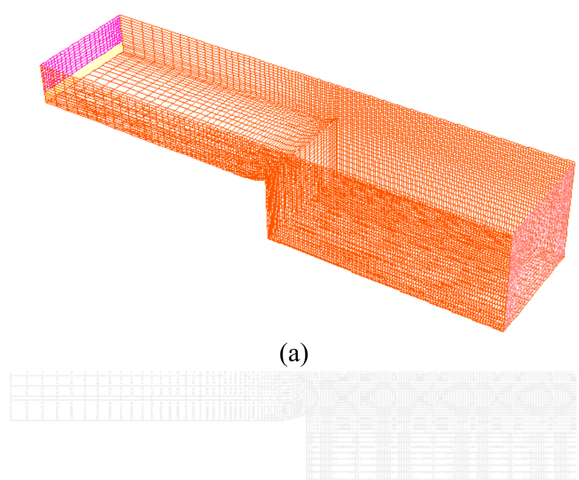
$$\frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k + C_{3\varepsilon}G_b\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{7}$$

Standard constants of k-ε model are listed in Table 2 and were used in the model.

Table 2. Standard k-ε turbulence model constants

	$C_{1\varepsilon}$	$C_{2\varepsilon}$	C_μ	σ_k	σ_ε
Standard k-ε	1.44	1.92	0.09	1.0	1.3

The volume of fluid (VOF) method was employed as a powerful computational tool for the analysis of free surface flow (Hirt and Nichols, 1981). The tracking of the interface(s) between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. In the present research, both structured and unstructured mesh was used (Figure 2).



(a)

(b)

Figure 2. Computational grid in the vicinity of flip bucket: (a) 3D view, (a) Side view.

Boundary conditions which were employed in this investigation are (Figure 3): Two different inlets were needed to define the water flow (Inlet 1) and air flow (Inlet 2) in the model domain. These inlets were defined as stream-wise velocity inlets that require the values of velocity. To estimate the effect of walls on the flow, empirical wall functions known as standard wall functions (Launder and Spalding, 1974) were used. The upper boundary above the air phase was specified as a symmetry condition, which enforces a zero normal velocity and a zero shear stress.

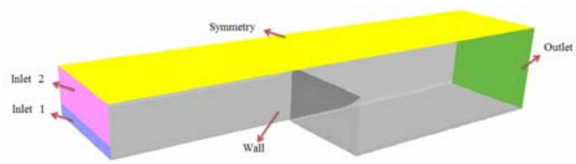


Figure 3. Solution domain and boundaries for modeled flip bucket

Other considerations in this simulation in this section are presented. The PRESTO pressure discretization was selected because this scheme was showed the best convergence in this simulation. The momentum and turbulent kinetic energy equations were discredited by first order upwind. The PISO pressure-velocity coupling algorithm was also used. Using unsteady and free surface equations required fine grid spacing and small initial time steps. To do so, a sensitivity analysis was performed on grid spacing and finally a number of meshes equal to 200000 were selected as the best result. Time steps were selected equal to 0.001 to 0.01. Due to model runs, solution convergence and water-surface profiles were monitored. The value of VOF parameter was selected equal to 0.5 which is a common practice for volume fraction results (Fluent Manual, 2005 and Dargahi, 2006).

4. Verification

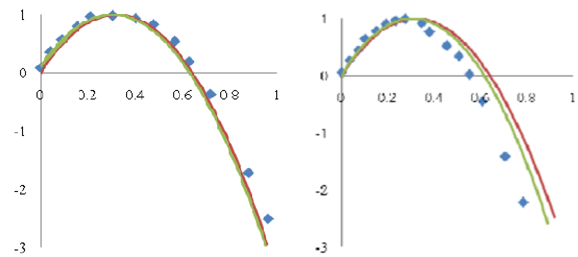
Before employing the numerical model, it is necessary to ensure about the accuracy of the numerical model. For this purpose, experimental cases which were mentioned in the previous section were employed. To evaluate the free surface, the first case was selected regarding the available flume data. Existing experimental results to validate the numerical simulation predictions included water surface profiles and pressure distributions. To calculate the jet trajectory by defining α_j as the takeoff angle and V_j as the takeoff velocity, the trajectory geometry $z(x)$ may be described for free jet flow as

$$z = z_0 + \tan\alpha_j x - gx^2 \big/ \left(2V_j^2 \cos^2\alpha_j\right) \qquad (9)$$

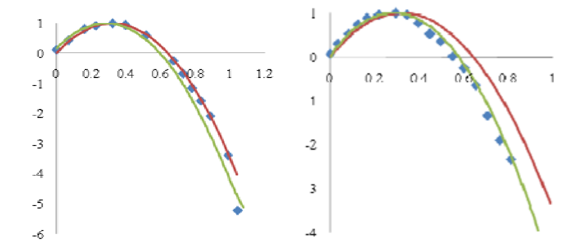
By considering the flow depth across the flip bucket remains constant, takeoff flow depth at $x=0$ and so $z_o=h_o$ for the upper nappe and $z_o=0$ for the lower nappe. The takeoff velocity V_j is V_o at $h_o \geq 5$ *cm*. By introducing the normalized coordinates relating to the upper (subscript *O*) nappe profile as $Z_O = (z_O - h_o)/(z_M = h_o)$ and $X = x\big/\left(h_0 F_0^2\right)$ where z_M, is the maximum (subscript M) nappe elevation above the takeoff elevation, results in

$$Z_0 = \tan\alpha_j X - \frac{1}{2}\frac{X^2}{\cos^2\alpha_j} \qquad (10)$$

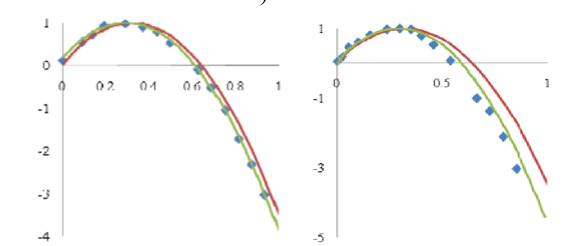
Figure 4 shows the numerical results in comparison with experimental data $Z_O(X)$ for various flow configurations and produces agreement with equation 10 provided $\alpha_j=20^o$ is fitted. The data for the lower (subscript *U*) nappe trajectory were analyzed correspondingly using $Z_U=z_U/z_M$ and $X = x\big/\left(h_0 F_0^2\right)$.



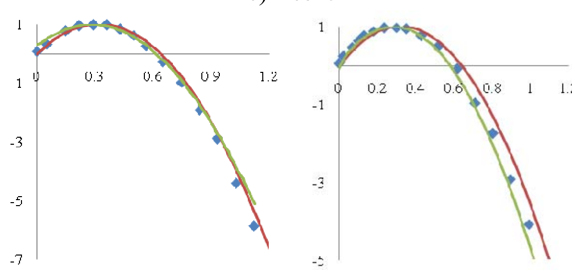
a) Test 1



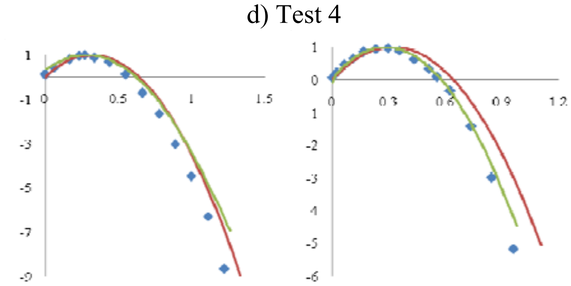
b) Test 2



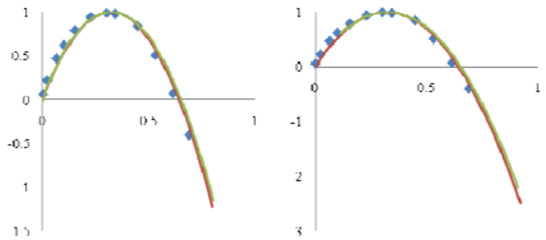
c) Test 3



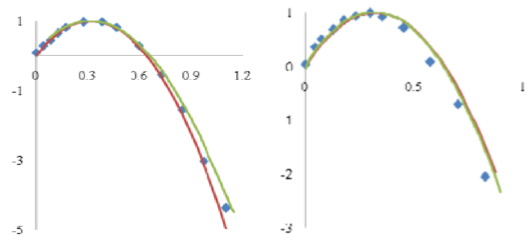
d) Test 4



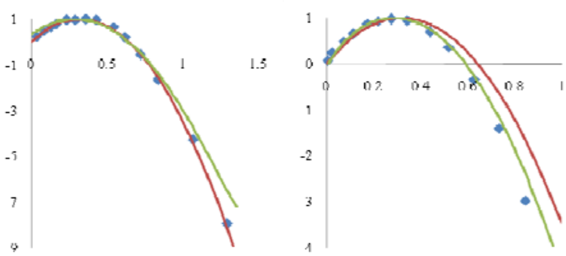
e) Test 5



f) Test 6



g) Test 7

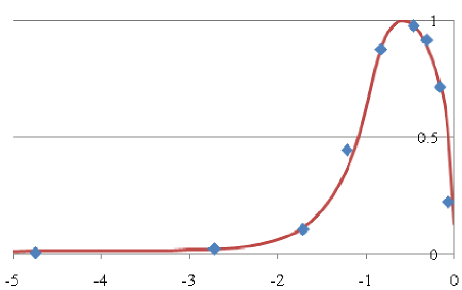


h) Test 8

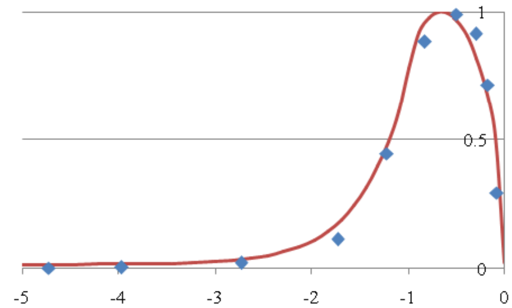
◆ **Experimental** **Jet Trajectories** **Numerical**

Figure 4. Water surface profiles for flow over Flip-Bucket (left diagram upper nappe profile, right diagram lower nappe profile)

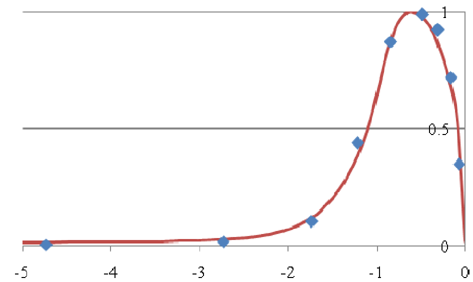
The distribution of pressures at the bottom along the flip bucket is an important design parameter for static purposes. It is equal to the sum of the static approach pressure head h_o plus a dynamic portion. Figure 5 shows the comparison between experimental and numerical results. Normalized parameter $H_P = (h_P - h_o)/(h_{PM} - h_o)$ where h_P and h_{PM} are the total and maximum pressure heads, respectively, plotted along the normalized stream wise coordinate $X_P = x/(R\sin\beta)$, where $x=0$ is located at the takeoff point, and $R\sin\beta$ is the stream wise flip-bucket length.



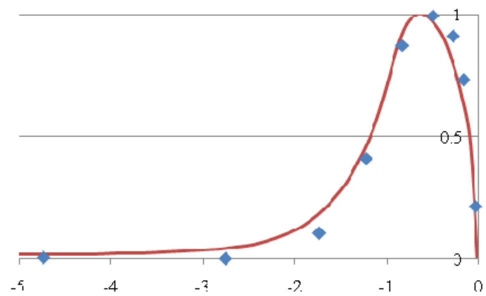
a) Test 1



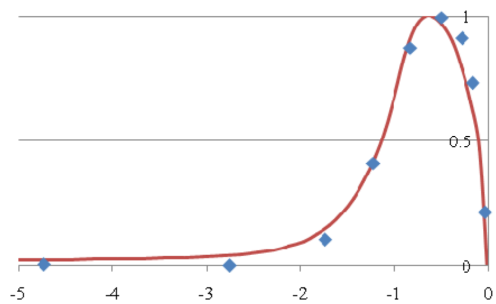
b) Test 2



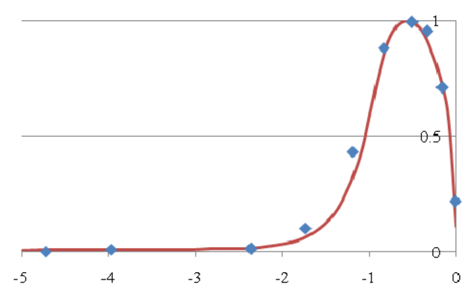
c) Test 3



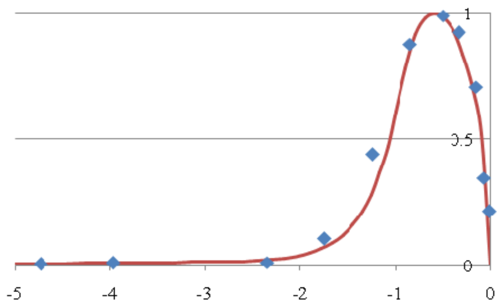
d) Test 4



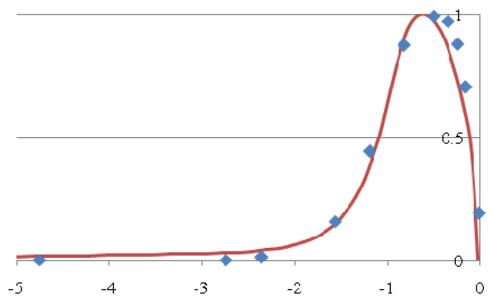
e) Test 5



f) Test 6



g) Test 7



h) Test 8

Numerical ◆ **Experimental**

Figure 5. Comparison between computed and measured pressure head distributions at bottom

Conclusions:

One way to dissipate of the energy in large dams is using flip bucket at the terminal of over fall spillways. Improving in design parameters of a flip bucket will protect the tailrace from scouring of impacted jet. Numerical modeling and analysis of flow with free surface which passes over a curved boundary by the gravity force is a challenging problem in hydrodynamics. In the present research by using Fluent software, a flip bucket structure was numerically simulated. To simulate the geometry and verify the results, experimental tests which were conducted in a smooth channel by Roman Juon and Willi H. Hager (2000) were selected. k-ε Standard turbulence model and VOF free surface model were employed in the model. The results jet trajectory properties and pressures in the bottom and its good agreement with analytical and experimental data in eight cases showed the ability of Fluent numerical model in modeling the flow over the flip buckets.

Corresponding Author:
Dr. Mehdi Jorabloo
Department of Engineering
Islamic Azad University, Garmsar Branch,
Garmsar, Iran.
E-mail: jorabloo.mehdi@yahoo.com

References

1. B. E. Launder and D. B. Spalding., 1972, “Lectures in Mathematical Models of Turbulence”, Academic Press, London, England.
2. B. E. Launder and D. B. Spalding.,1974, “The numerical computation of turbulent flows.”, Computer Methods in Applied Mechanics and Engineering, 3, pp. 269-289.
3. Dargahi, B., 2006, “Experimental study and 3D numerical simulations for a free-overflow spillway”, J. Hydraulic Engineering, 132 (9), 899, 10.1061/(ASCE), 0733–9429.
4. Fluent Manual, 2005, “Manual and user guide of Fluent Software.”, Fluent Inc.
5. Hirt, C.W. and Nichols, B.D., 1981, “Volume of fluid methods for the dynamics of free boundaries”, Journal of Computational Physics, 39, pp. 201–225.
6. J.O. Hinze., 1975, “Turbulence”, McGraw-Hill Publishing Co., New York.
7. K. C. Karki and S. V. Patankar., 1989, “Pressure-based calculation procedure for viscous flows at all speeds in arbitrary configurations.”, AIAA Journal, 27, pp.1167-1174.
8. Li Shuguarig and Liang Zhengxiang, 1988, ‘‘Gravity-affected potential flows past spillway flip buckets.’’,Journal of Hydraulic Engineering, Paper No. 22362, Vol. 114, No. 4.
9. Orlov, V. T., 1968, ‘‘Calculation of free surface profile and pressure distribution along a circular arc connecting sloping and horizontal segments.’’, Transactions of All Union Scientific Research Institute of Hydraulic Engineering, No. 87 (Russian).
10. Peterka, A. J., 1983, ‘‘Hydraulic design of stilling basins and energy dissipators.’’, 7th Ed., Engrg. Monograph 25, Bureau of Reclamation, U.S. Department of the Interior, Denver.
11. R. I. Issa., 1986, “Solution of implicitly discretized fluid flow equations by operator splitting.”, J. Comput. Phys., 62, pp. 40-65.
12. Remo Steiner & Valentin Heller & Willi H. Hager and Hans-Erwin Minor, 2008, “Deflector Ski Jump Hydraulics”, ASCE, Journal of hydraulic engineering, Vol. 134, No. 5, 347-355.
13. Siao, T. T., and Hubbard, P. G., 1953, ‘‘Deflection of jets, Pt I: symmetrically placed V-shaped obstacle Free streamline analysis of transitional flow and jet deflection.’’, J. S. McNown and C. S. Yih, eds., State Univ. of Iowa, Iowa City, Iowa.
14. Tinney, R. E .et al, 1961, ‘‘Free streamline theory for segmental jet deflector’’, J. Hydr. Div., ASCE, 87(HY5), 135-146.

15. Valentin Heller &Willi H. Hager and Hans-Erwin Minor 2005, “Ski Jump Hydraulics”, ASCE, Journal of hydraulic engineering, Vol. 131, No. 5, 347-355.
16. Valentin Heller & Willi H. Hager, 2000, “Flip bucket without and with deflectors.”, ASCE, Journal of hydraulic engineering, Vol. 126, No. 11, 837-845.
17. Vischer, D. L., and Hager, W. H., 1998, ‘‘Dam hydraulics’’, Wiley, Chichester, England, and New York.
18. Xie-Qing Xu and Xiao-Xia Sun, 1990, “Flow in spillways with gradually varied curvature”, Journal of Engineering Mechanics, Paper No. 24365, Vol. 116, No. 2.
19. Young D.L., 1982, “Time-dependent multi-material flow with large fluid distortion”, In: K.W. Morton and M.J. Baines, eds. Numerical methods for fluid dynamics. New York: Academic Press, pp. 273–285.

5/31/2011