Conference Paper, Published Version

Yazdi, Jafar; Zarrati, Amir Reza

Development of a Relationship for Air Vent Discharge in Bottom Outlets Using Numerical Simulation

DEVELOPMENT OF A RELATIONSHIP FOR AIR VENT DISCHARGE IN BOTTOM OUTLETS USING NUMERICAL SIMULATION

Jafar Yazdi[1] and Amir Reza Zarrati[2]

[1]MSc student, Department of Civil and Environmental Eng., Amirkabir University of Technology
Tehran, Iran, jr_yazdi@yahoo.com
[2] Associate Professor, Department of Civil and Environmental Eng., Amirkabir University of Technology, Tehran, Iran, zarrati@aut.ac.ir

ABSTRACT

High speed free surface flows in bottom outlets transfer a large amount of air towards downstream. If flow air demand is not supplied, negative pressure develops downstream of the tunnel gate which causes vibration and cavitation. Consequently air should be allowed to the tunnel via an air vent downstream of the tunnel gate. To design the air vent the air flow through the vent should be determined. The amount of vent air discharge is a function of flow characteristics and geometric parameters of the bottom outlet and the vent. In this paper, mechanism of air flow in the tunnel and effective factors on flow air demand was investigated employing a 3D numerical model. Simulation of flow is done using VOF method with Young's scheme and turbulence is simulated by standard $k-\varepsilon$ model. Using numerical model results, the vent air discharge "Q_{av}" was determined. Numerical model results were first validated by available experimental data and showed good agreement. In the next stage, the effect of flow discharge, tunnel length, relative flow depth, and air-vent losses on air-vent discharge was studied with the numerical model. A wide range of flow parameters were tested to study their effect on vent air discharge. To find a relationship between "Q_{av}" and dependent variables, dimensional analysis was employed. By Buckingham's theorem dimensionless arguments were derived. Subsequently using the numerical results and multiple-regression analysis a relationship was found for air vent discharge which can be used for vent design.

Keywords: air-water flow, bottom outlet, tunnel, 3D numerical modelling, Air demand, Buckingham's theorem

1. INTRODUCTION

Bottom outlets are gated tunnels in high dams that are used for different purposes. In these structures, high velocities cause sub-atmospheric pressures behind the gate. These negative pressures may lead to structural damage due to cavitation and vibration. To avoid sever negative pressures tunnel is connected to the atmosphere through a vent just after the gate. The purpose of the vent is to draw in air and thereby keep the pressures downstream of the gate at a safe level (Sharma, 1976). Estimating the air vent discharge Q_{av} is a fundamental parameter in design of an air vent. Campbell and Guyton (1953) presented the following relationship for air discharge ratio $\beta=\frac{Q_{av}}{Q_w}$, where Q_w is water flow discharge, using the Froude number at the contracted section Fr_c downstream of the gate:

$$\beta = 0.04(Fr_c - 1)^{0.85} \tag{1}$$

U.S. army corps of Engineers (USACE, 1964) extended the field data used by Cambell and Guyton and recommended the following equation for the case of free surface flow in the tunnel:

$$\beta = 0.03(Fr_c - 1)^{1.06} \tag{2}$$

Sharma (1976) classified three different patterns of flow in conduits: spray flow, free surface flow and hydraulic jump. By experiment in a rectangular tunnel, Sharma showed air demand in spray flow is more than free surface flow (with similar Froude Number) as is indicated below:

$$\text{spray flow:} \qquad \beta = 0.2Fr_c \tag{3}$$

$$\text{free surface flow:} \quad \beta = 0.09Fr_c \tag{4}$$

Speerli et al. (1997) studied air demand in a rectangular tunnel and related air-vent discharge to various parameters and presented the following relationship by analyzing experimental data:

$$q_{av} = \frac{Q_{av}}{\left(gB_u^2 H_E^3\right)^{0.5}} = 0.022\left(\frac{B_u}{H_E}\right)^{0.5}\left(\frac{L_u}{H_t}\right)^{1/6} S^{0.5}\xi_o^{-0.43} \tag{5}$$

Where B_u = tunnel width, H_E = upstream energy head, L_u =length of the tunnel, Ht = height of the tunnel, S =relative gate opening (defined as the ratio of opening height to gate height), ξ_o = total loss coefficient of air vent and q_{av} = dimensionless air discharge

Most of the previous works are based on physical model studies and are based on limited experimental data and therefore can not be generalized. The results of physical models are also affected by experimental limitations, scale effects and measurements accuracy. Conducting more experiments with larger facilities and wider range of flow parameters is also very expensive and time consuming. In this research, a 3D numerical model was first validated and then was employed to predicted vent discharge and to investigate the effect of various parameters on the air demand. A relationship was then developed for vent air discharge.

1. EXPERIMENTAL SETUP

In this research, experimental set up of Safavi et al. (2007) was used to study air water flow in tunnels with circular cross-section. Flow equations were solved in the tunnel and then air vent discharge was calculated and validated by available measured data. The numerical model was then used to calculate vent air discharge in a wide range of flow parameters and vent size.

Experiments were conducted in a 17.5cm diameter Perspex pipe as a bottom outlet. A slide gate was installed upstream of the tunnel. Length of the tunnel after the gate was 7m. Considering usual area of the gates comparing with tunnel cross section area, full size of the gate opening was selected 10cm wide and 8cm high (Fig. 1). Energy head upstream of the gate was constant and equal to 2.2m. For any percentage of gate opening, $S = (w/h) \times 100$, flow discharge and relative flow depth downstream in the tunnel (*y/D*) were measured where *w* is the gate width, *h* is the gate height, *y* is the flow depth downstream in the tunnel and *D* is the tunnel diameter. The gate lip was inclined 45^0 towards upstream. To conform to prototype design a step was considered in the model after the gate so that a jet forms over this step. The step height was selected 2cm following design recommendations (Beichly et al.

1975). A 3cm diameter vent made from Perspex was installed just after the bottom outlet gate. A gate was also installed in the vent. By closing this gate, head losses in the vent increased and therefore air vents with smaller diameters could be simulated.

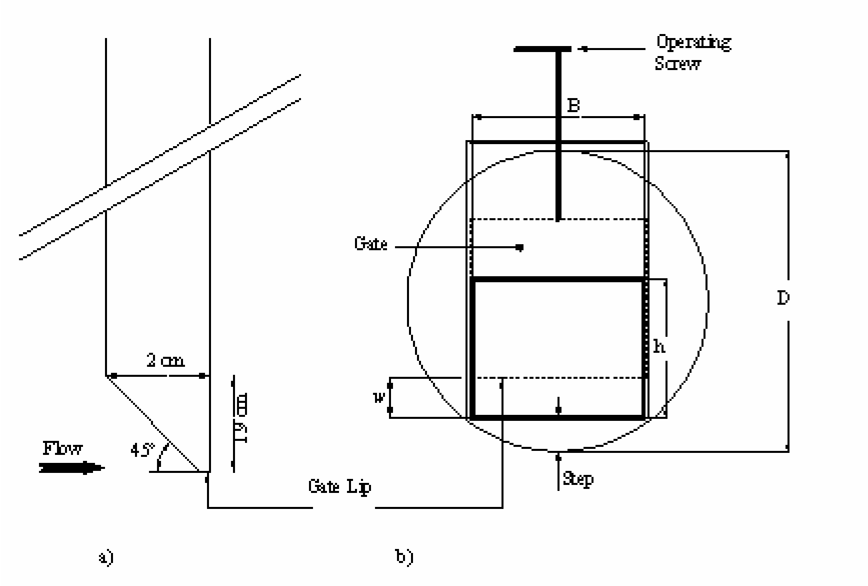

Fig. 1. Details of the gate in the experimental model, Safavi et al (2007)

2. NUMERICAL MODELLING

Fluent CFD code was used for numerical modelling. The numerical modelling involves the solution of the Navier-Stockes equations, which are based on the assumptions of conservation of mass and momentum within a moving fluid. In the absence of source terms, the conservation of mass is described by the following equation:

$$\frac{\partial}{\partial t}(\rho)+\nabla.(\rho\vec{v})=0 \tag{6}$$

In this equation $\vec{v}$ is the average velocity: $\vec{v}=\frac{\alpha_w \rho_w \vec{v}_w+\alpha_a \rho_a \vec{v}_a}{\rho}$ where $\vec{v}_w$ and $\vec{v}_a$ are velocity of water and air.

ρ is the average density: $\rho=\alpha_w \rho_w+\alpha_w \rho_w=\alpha_w \rho_w+(1-\alpha_w)\rho_a$ where α_w and α_a are the volume fraction of water and air phases. The value of α_w in a cell represents the fractional volume of the cell occupied by water phase. The volume fraction of air can be given as: $\alpha_a=1-\alpha_w$.

In the numerical model, a single momentum equation is solved throughout the entire air and water domain, and the resulting velocity field is shared among the phases. The momentum equation, shown below, is dependent on the volume fractions of air and water through the properties ρ and μ_0 (Fluent Manual, 2005).

$$\frac{\partial}{\partial t}(\rho\vec{v})+\nabla.(\rho\vec{v}\vec{v})=-\nabla p-\nabla.\left[\mu\left(\nabla\vec{v}+\nabla\vec{v}^T\right)\right]+\rho\vec{g}+\vec{F} \tag{7}$$

Where in this equation: p = pressure, $\vec{v}$ = average velocity and $\mu=\mu_0+\mu_t$. μ_0 is the average viscosity of air-water flow and is calculated similar to density. μ_t is turbulence viscosity and $\vec{F}$ is the body force. In this research, to determine μ_t, $k-\varepsilon$ standard model was used.

To treat the free surface, the VOF method of Hirt and Nichols (1981) with young's scheme (1982) was applied. When modelling the free surface between water and air, a transport equation can be solved for the water phase in the following form:

$$\frac{\partial}{\partial t}(\alpha_w) + \nabla.(\alpha_w \vec{v}) = 0 \tag{8}$$

Where α_w is the volume fraction of water. Only one such transport equation needs to be solved since the volume fraction of the other phase can be inferred from the constraint:

$$\alpha_a = 1 - \alpha_w \tag{9}$$

This equation is solved in the entire domain and volume fraction is computed for all cells. In each cell, if it contains only water, then $\alpha_w = 1$; if none, then $\alpha_w = 0$ and if $0 < \alpha_w < 1$ cell contains both water and air. In the present study, It was assumed that volume fraction of 0.5 shows the free surface. This was assumed since it was noticed that with distance from the bed in water flow the flow velocity increased to where $\alpha = 0.5$. Further away from the bed and in higher α velocity decrease showing that this area is out of water flow and in air flow.

To generate computational mesh, both structured and unstructured mesh was used. The gate area was meshed unstructured because of the complex geometry (Fig. 2). The size of mesh in this area was finer than the other regions. Structured mesh was used at other regions. Boundary conditions were as follows (Fig. 3):

In the upstream, the flow rate was introduced to the program since the measured flow rate was known. To estimate the effect of the wall on the flow, empirical wall functions were used. The Standard $k-\varepsilon$ turbulence model of Yakhot and Orsag (1986) was employed in computations. In the entrance of air vent that contacts with atmosphere, only air phase was defined and total pressure was assumed equal to zero. At the downstream section, the gradients of all variables was assumed zero.

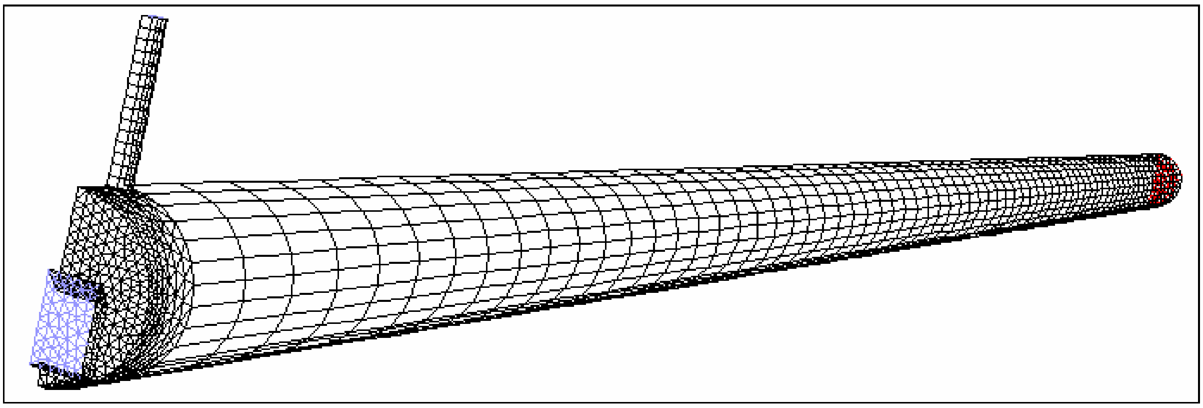

Fig. 2. Computational grid in the numerical model

The PRESTO pressure discretization scheme was used because this scheme showed the best convergence in this model together with PISO pressure-velocity coupling algorithm.

The unsteady free surface calculations required small initial time step. A time step from 0.001 up to 0.01 was used in different conditions. During the calculations, solution convergence and the water surface profiles were monitored. Convergence was achieved when the normalized residual of each variable was on the order of 1×10^{-3}.

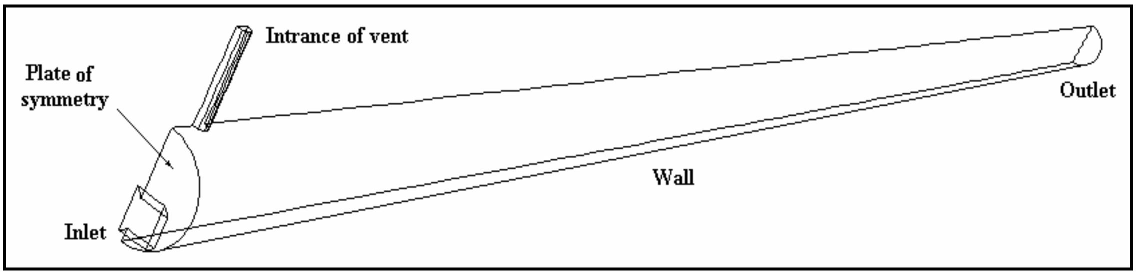

Fig. 3. Domain of solutions and boundaries

Computations were conducted by a Pentium IV system with an AMD 3800_{+} processor. Each run of the numerical model took 6 to 18 hours depending on tunnel length, mesh numbers and flow characteristics.

3. Verification of the numerical model

To evaluate the model's applicability, measured data from the experimental studies of Safavi et al. 2007 were used. Pressure and velocity fields were calculated by the numerical model in various gate openings or in another word relative depths *y/D*, air vent discharges at various *y/D* were calculated and compared with experimental data in Figure 3.

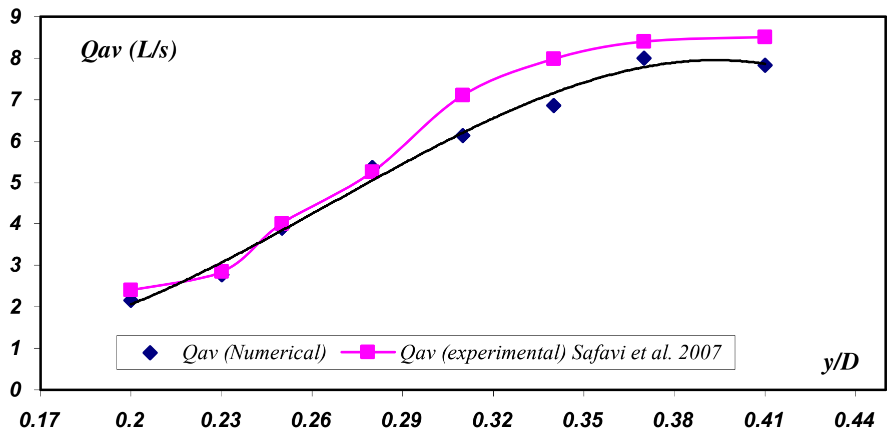

Fig. 4. Compared results of numerical and experimental data

As can be seen in this figure, results show a good agreement. In average the numerical model underestimates the vent discharge by about 7%. This difference can be due to air entrainment to the flow which can not be simulated by the computer model.

In the next stage**,** simulated air discharge ratio β was compared with some of empirical equations in Figure 5. This figure shows that the numerical model results agree better with the experimental measurements. Also numerical model results has a good agreement with Campbell and Guyton (1953) and USACE (1964) equations (average error of 10.44% and 19.32%, respectively). These equations are based on prototype data. Sharma equation overestimates the air discharge ratio. Overestimation of Sharma was also reported by Safavi et al. (2007).

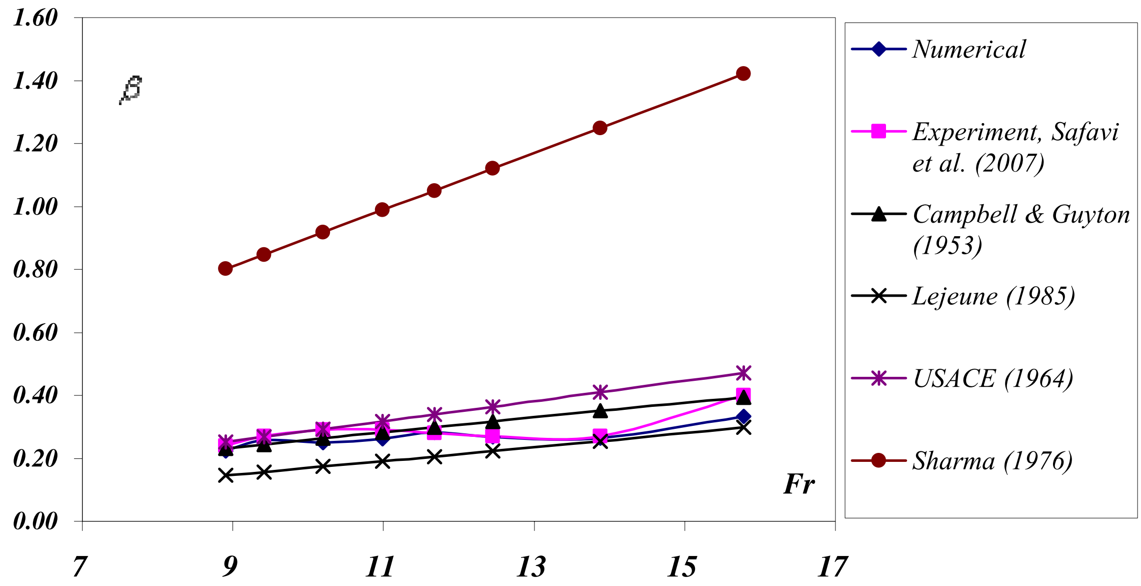

Fig. 5. Comparison of computed β of the present work with some of empirical equations

From Figs. 4 and 5, it can be seen that the numerical model results are acceptable. By changing various parameters including flow depth, discharge, tunnel length and diameter of vent, the numerical model was used to calculate air discharge. Results of these computations were then used to study the parameters that affect vent discharge. In the following section these results are discussed.

4. ANALYSIS AND DISCUSSION

Air vent discharge is a function of flow characteristics and geometric parameters of the tunnel. Changing all these variables is very difficult in physical model studies whereas it can be easily done in a numerical model. A wide range of flow parameters were tested in the numerical model to study their effect on air vent discharge. Fig. 6 shows the effect of various parameters on the air vent discharge. In this figure, variations of Q_{av} versus each of these parameter is shown when other parameters were assumed constant. For example in Fig. 6-a, $Q_{av} - y/D$ diagram was drawn for *L/D*=40 and *d/D*=0.17. Similar diagrams were obtained for other *L/D* and *d/D*. As shown in Fig. 6-a, by increasing *y/D*, Q_{av} increases because of increasing surface flow velocity and therefore the dragged air along the free surface. However rate of increase in Q_{av} decreases at higher y/D due to reduction of air passage area above the free surface.

On the other hand Q_w increase linearly with the reservoir head as shown in Fig. 6-b

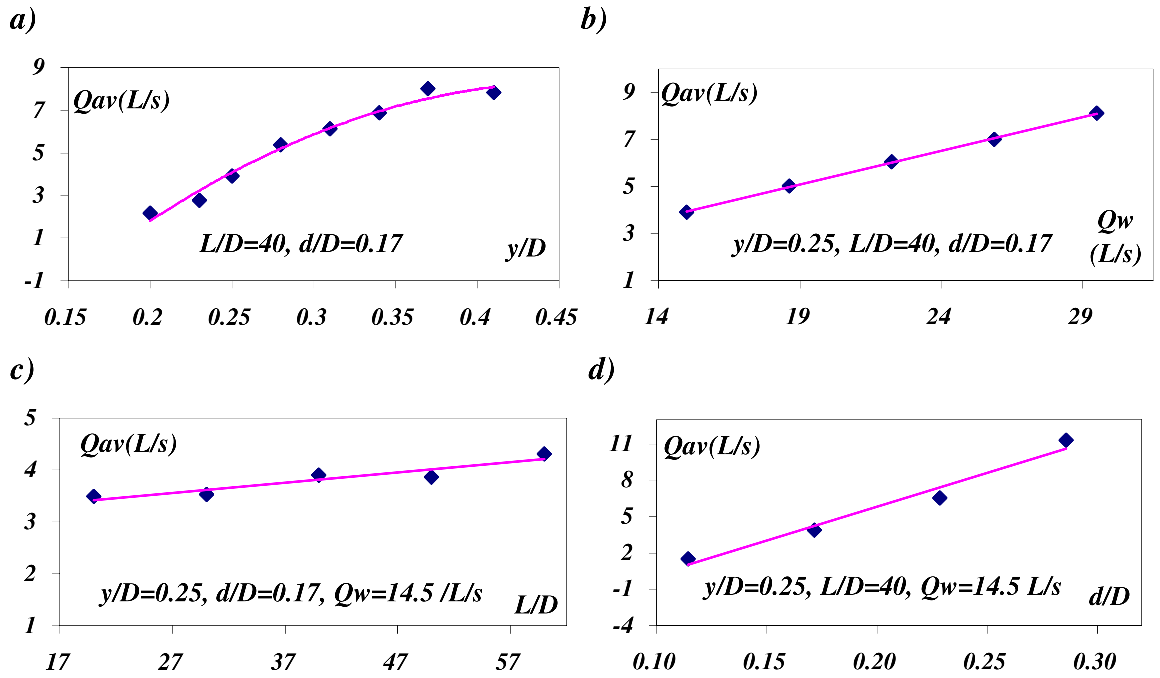

Fig. 6. Effect of various parameters on the Q_{av} : $a)\, Q_{av} - y/D$, $b)\, Q_{av} - Q_w$, $c)\, Q_{av} - L/D$, $d)\, Q_{av} - d/D$

In general, air demand can be supplied from both air vent and the tunnel outlet. As shown in Fig. 6-c, Q_{av} increases in higher *L/D*. In higher *L/D*, higher frictional losses reduces air flow from the tunnel outlet and this in turn increases the vent air discharge to compensate for that.

As shown in Fig. 6-d, with increasing the diameter of the vent and therefore reduction of losses, air vent discharge can be supplied easier to the tunnel and therefore Q_{av} increases.

Considering the factors affecting the vent discharge as discussed above Q_{av} is a function of y, D, L, Q_w and ξ_0. Following dimensional analysis therefore one can write:

$$\frac{Q_{av}^2}{gD^5} = \phi\left(\frac{Q_w^2}{gD^5}, \frac{y}{D}, \frac{L}{D}, \xi_0\right) \tag{10}$$

Using the numerical results and multiple-regression analysis the following relationship was found for air vent discharge:

$$\frac{Q_{av}^2}{gD^5} = 0.026\left(\frac{Q_w^2}{gD^5}\right).\left(\frac{y}{D}\right)^{-1} + .00033\left(\frac{L}{D}\right) + .823\left(e^{-64.94(\xi_0 - 0.5)}\right) - 0.159 \tag{11}$$

This equation can be used for circular cross-section tunnels. From equation (11) air vent discharge in the physical model of Safavi et al. 2007 was calculated and compared with measured values in Fig. (7). Agreement is good and the average error is 8.5%.

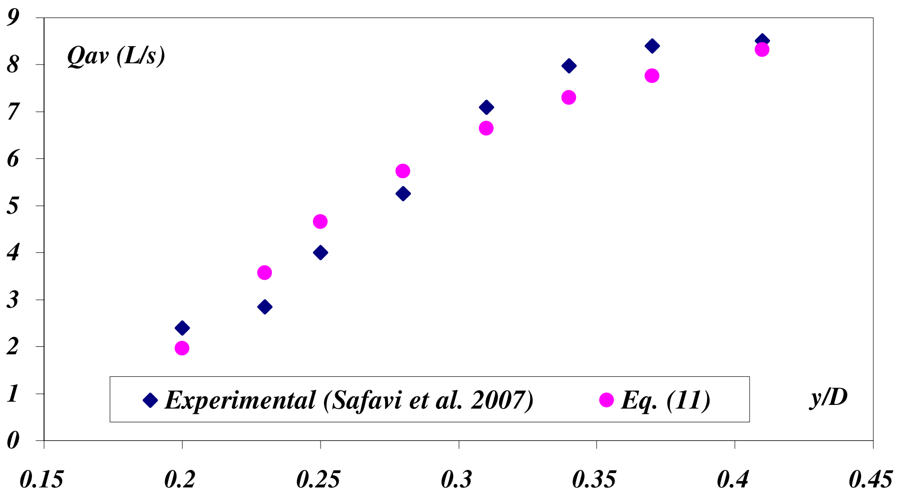

Fig. 7. Comparison of Equation (13) with the measured data

5. CONCLUTION

In this research work, a 3D numerical model was employed to study the characteristics of air-water flow in bottom outlets with a circular cross section. Vent discharge calculated with the numerical model showed a good agreement with experiments. Variables such as flow depth, head over the gate, tunnel length and vent diameter were then widely changed and their effect on vent air discharge was studied. Based on dimensional analysis and using the numerical results a relationship was then found for vent air discharge. This relationship was then checked against the measured data and the agreement was good.

REFERENCES

Beichly, G.L. and King, D.L. (1975), Cavitation control by aeration of high-velocity jets, *Journal of Hydraulics Divisions*, ASCE, HY7, 829-846.

Campbell, F.B. and Guyton, B. (1953), Air-demand in gated outlet works, *Proceedings 5th IAHR congress*, Minnessota, U.S.A., 529-533.

Fluent Manual (2005), Manual and user guide of Fluent Software, *Fluent Inc*.

Garcia, J. and Fuentes, R. (1984), Influence of the tunnel length on the hydraulic modeling of the air entrainment in the flow downstream of a high head gate, *Symposium on scale effects in modeling hydraulic structures*, Esslingen, Germany, 4-14-1 – 4-14-2.

Hirt C.W. and Nichols B.D. (1981), Volume of fluid methods for the dynamics of free boundaries, *J. Comp. Phys*. 39.

Lejeune A. and Ouazar D. (1985), Theoretical and Experimental Study of Cavitation Preventation by Ventilation, *21st IAHR Congress*, Melbourne, Australia,1985

Launder and spalding

Safavi, kh. , Zarrati A.R. and Attari J. (2007), Experimental study of air-demand in high head gated tunnels, *Journal of water management*, April 2007, Water Management 161 Issue WM2.

Sharma, H.R. (1976), Air entrainment in high head gated Conduits, *J. Hydraulics division , ASCE* , Vol. 102 , No. 11

Speerli, J. and Volkart, P.U. (1997), Air entrainment in bottom outlet tailrace tunnels, Proc.*27th IAHR congress*, SanFrancisco. Theme D, pp.613-618.

U.S. Army Corps of Engineers (1964), Air-demand regulated outlet works, *Hydraulic Design Criteria*, chart 050-1

Yakhot V. and Orsag S. (1986), Renormalization group analysis of turbulence, *Journal of science and Computing* 1:3-51.

Young, D.L. (1982), Time-Dependent Multi-Material Flow with Large Fluid Distortion, In K. W. Morton and M. J. Baines, editors, *Numerical Methods for Fluid Dynamics*. Academic Press, 1982.