Development of a 2-D numerical model for simulation of air distribution in high speed air water flow

Javanbarg, M.B.[1]**, Zarrati, A.R.**[2]**, Jalili, M.R.**[3]**, Safavi, Kh**[4]**.**

[1]Post graduate Student, Dep. of Civil and Environmental Eng., Amirkabir University of Technology, Tehran, Iran, 032d842n@y05.kobe-u.ac.jp

[2]Associate Prof., Dep. of Civil and Environmental Eng., Amirkabir University of Technology, Tehran, Iran, Zarrati@aut.ac.ir, 64543002

[3]Assistant Prof., Dep. of Water Engineering, Power and Water University of Technology, Tehran, Iran, jalili@pwut.ac.ir

[4]Senior Research Engineer, Water Research Institute, Tehran, Iran, K.safavi@wri.ac.ir

Abstract: *In the present study a quasi 2-D numerical model is developed for calculating air concentration distribution in rapid flows. The model solves air continuity equation (convection diffusion equation) in the whole flow domain. This solution is then coupled with calculations of the free surface in which air content in the flow is also considered.*

To verify the model, its results are compared with an analytical solution as well as a 2-D, numerical model and close agreement was achieved. The model results were also compared with experimental data. This comparison showed that the decrease in air concentration near the channel bed in an aerated flow could be well predicted by the model. The present simple numerical model could therefore be used for engineering purposes.

Keywords: *High speed flow, Cavitation, Aeration, Numerical model, Air concentration.*

1. Introduction

Over chute spillways, as flow approaches a uniform state in downstream direction, flow velocity increases and flow depth decreases. These changes reduce cavitation index, and therefore increase the risk of cavitation attack. Experience shows that flow aeration is the most effective and economical way for prevention and/or minimizing the damage due to cavitation [4,6].

Defining air concentration in a small control volume as $c = \frac{V_a}{V_a + V_w}$, where V_a and V_w represent air volume and water volume respectively, Peterka [10] showed that about 8 % of air concentration near the channel bed is sufficient to protect concrete surfaces against cavitation.

Surface aeration from the free surface naturally occurs along chute spillways and is then diffused into the flow. However, to have air concentration of 8% near the chute surface, mean air concentration in a section should be more than 30% and chute slope should be greater than 22.5º. [6]

Therefore, protection of spillways from cavitation can not always be guaranteed with surface aeration, and so artificial aeration may be required by using an aerator on spillways [4]. The distance between aerators is a main point to be considered in designing aerators. Such a distance depends on the decrease in air concentration rate within deaeration zone downstream of an aerator [4]. Hence, estimating the distance in which air concentration near the bed is less than 8% would be a main factor in determining the location of a second aerator in order to increase again air concentration near the bed. It is therefore necessary to study air concentration distribution in air-water flows to be able to determine the number of aerators needed in a chute spillway.

Empirical equations based on hydraulic model tests have been used in many researches to evaluate reduction of air concentration downstream of aerators. However, due to insufficient experimental data and also scale effects in physical models, these equations are not very reliable. On the other hand physical model studies need long time and are also very expensive.

Based on physical model studies, Falvey [4] correlated air discharge detrainment form the flow to the distance from an aerator. Prusza [11] stated that the decrease rate of air concentration in deaeration zone is between 15% to 20% per 1 m along the chute. Minor [7] found this rate to be 10% to 15% per 1 m of chute length. May [6] studied air concentration distribution downstream of aerators and presented an exponential function between air concentration near the channel surface and distance from the aerator. Chanson [1] simulated air concentration variation along a chute spillway by simultaneously solving the energy equation and air continuity equation. Nokes [9] calculated air concentration distribution in flow by analytically solving mass conservation equation for air volume and supposing a uniform flow. In this solution turbulence diffusivity coefficient ε_d was assumed parabolic in depth. Using $K-\varepsilon$ turbulence model, Zarrati [15] presented a 2-D numerical model for air distribution in high speed free surface flows. The results of this model were compared with experimental data of air distribution downstream of an air injection slot in a channel and good agreement was obtained. Applying his model to downstream of aerators, Zarrati and Hardwick [17] evaluated air concentration variations in the dearation zone and compared them to experimental results. Despite previous studies, still a general and practical method is not presented for calculation of air distribution in free surface flows.

A quasi 2-D model is developed in the present research work for investigating air concentration distribution in rapid flows. The model consists of solving the air continuity equation (convection - diffusion equation of air) together with calculating the free surface location. Results of this model can be used to predict air distribution in air-water flows as well as reduction of air concentration within the deaeration zone, downstream of aerators.

2. Governing Equations and Basic Assumptions

Assuming a small control volume (Figure 1), continuity equation for air volume in a 2-D flow could be written as follows [15]:

$$u\frac{\partial c}{\partial x}+v\frac{\partial c}{\partial y}=\varepsilon_{(d)x}\frac{\partial^2 c}{\partial x^2}+\varepsilon_{(d)y}\frac{\partial^2 c}{\partial y^2}+u_s\frac{\partial c}{\partial x}+v_s\frac{\partial c}{\partial y} \tag{1}$$

Where, $\varepsilon_{(d)x}$ and $\varepsilon_{(d)y}$ are turbulence diffusivity coefficient, u and v are flow velocity, and u_s and v_s are air bubbles rising velocity in x and y directions, respectively. In deriving this equation air-water mixture is assumed as a single phase flow with air content in the mixture determined by the average density of the mixture ρ_{mix}, that is: $\rho_{mix}=\rho_w(1-c)$, where ρ_w is water density.

Since v and $\varepsilon_{(d)x}$ have much smaller values than u and $\varepsilon_{(d)y}$ respectively in a uni-directional supercritical flow and also u_s is much smaller than v_s, the continuity equation for air volume could be simplified as follows:

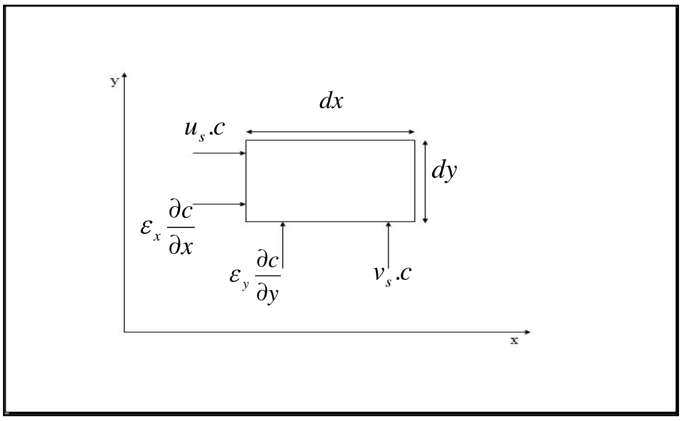

Fig. 1 A small control volume for developing air continuity equation

$$u\frac{\partial c}{\partial x} = \varepsilon_{(d)y}\frac{\partial^2 c}{\partial y^2} + v_s\frac{\partial c}{\partial y} \quad (2)$$

In a uni-directional flow in an open channel, distribution of turbulence viscosity coefficient in depth can be described by the following equation[8]:

$$\varepsilon_v = u_* \kappa y\,(1 - \frac{y}{H}) \quad (3)$$

where, H stands for flow depth, y is the distance of each point from the bed, κ is Von Karman constant coefficient, equal to 0.4 and $u_* = \sqrt{gRS_f}$ is bed shear velocity, R represents hydraulic radius and S_f is energy slope. The relationship between turbulence viscosity coefficient and turbulence diffusion coefficient can be represented as $\varepsilon_d = \varepsilon_v / \sigma_t$ in which σ_t represents Schmidt Number with a range of 0.5 to 2 [5]. In air-water mixtures, when air concentration near the bed is high, rising air bubbles intensify turbulence and as a result ε_d increases (or σ_t decreases) [15]. For this reason, σ_t was considered 0.7 in the deaeration zone to account for the effect of rising bubbles on increasing the intensity of turbulence.

Studies indicate that rising velocity of bubbles increases as air concentration increase, since bigger bubbles are formed at higher air contents. Therefore, a value of 12 cm/s was considered for v_s for air concentrations less than 5%, a value of 19 cm/s for air concentrations between 5% to 10% and a value of 25 cm/s for concentrations over 10%. [16]

Distribution of flow velocity in depth can be calculated by a logarithmic distribution for fully developed turbulent flow. Knowing absolute roughness of the bed K_s and shear velocity, the following relationships can be used for velocity calculations[12]:

If $\frac{u_* K_s}{\nu} < 5$ a smooth hydraulic bed

$$\frac{u}{u_*} = \frac{1}{\kappa}\ln(\frac{9 u_* y}{\nu}) \quad (4)$$

If $\frac{u_* K_s}{\nu} > 70$ a rough hydraulic bed

$$\frac{u}{u_*} = \frac{1}{\kappa}\ln(\frac{30 y}{K_s}) \quad (5)$$

Where, κ represents Von Karman constant

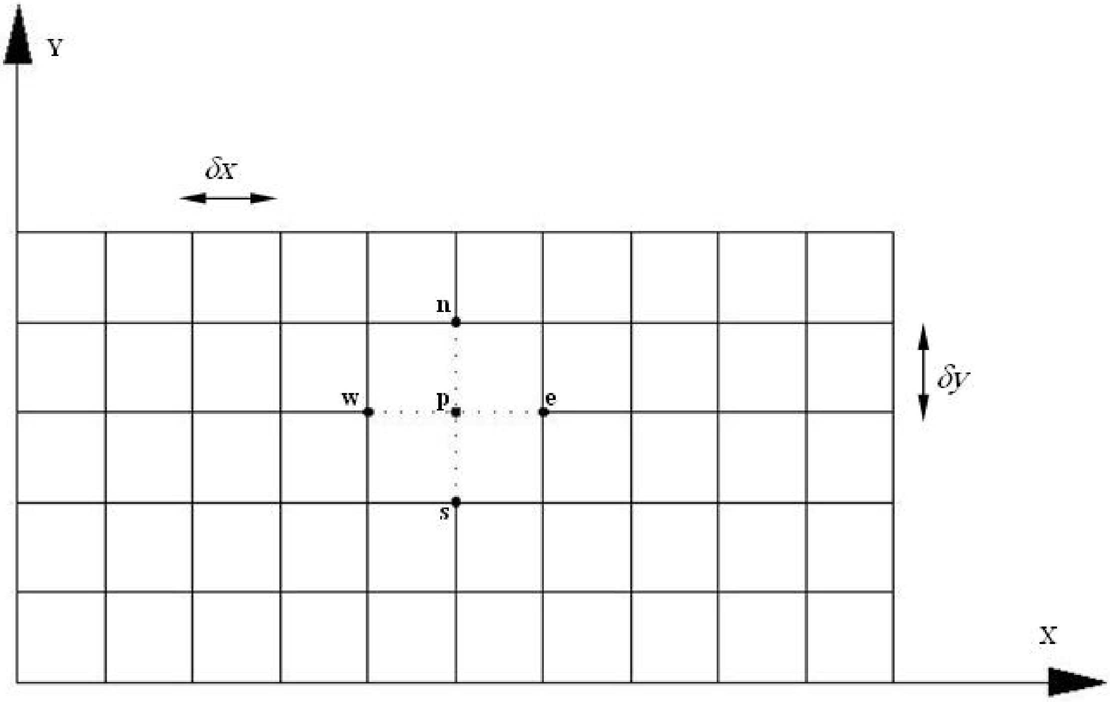

Fig. 2 Computational domain and Cartesian mesh

coefficient =0.4, y is distance from the bed and v is kinematic viscosity of water.

3. Numerical Solution of Equations

The model presented in this work is a quasi 2-D model to calculate distribution of air concentration in rapid flows. It consists of solving a 2-D convection - diffusion equation, assuming logarithmic velocity distribution and parabolic turbulence diffusion coefficient in depth. Finite differences method with defined boundary conditions is used for numerical solution of the convection diffusion equation. In a flow field shown in figure 2, equation (2) can be discretized as follows:

$$u_p \frac{c_p - c_w}{\delta x} = (\varepsilon_d)_p \frac{c_n - 2c_p + c_s}{\delta y^2} + v_s \frac{c_n - c_s}{2\delta y} \qquad (6)$$

Evaluating variables in equation 6 yields the following Algebraic equation :

$$a_p c_p = a_n c_n + a_s c_s + a_w c_w \qquad (7)$$

Where a_p, a_n, a_s and a_w are known coefficients and are defined as:

$$a_p = \left(\frac{u_p}{\delta x} + \frac{2\varepsilon_{(d)p}}{\delta y^2}\right), \qquad a_n = \left(\frac{\varepsilon_{(d)p}}{\delta y^2} + \frac{v_s}{2\delta y}\right),$$

$$a_s = \left(\frac{\varepsilon_{(d)p}}{\delta y^2} - \frac{v_s}{2\delta y}\right), \qquad a_w = \frac{u_p}{\delta x}$$

Knowing air concentration distribution at the inlet section, above equation can be solved at each section using Three Diagonal Matrix Algorithm (*TDMA*) [13] method and air concentration can be calculated at different points in the computational domain by marching technique. In this technique, air concentration is calculated in each section referring to known values in the upstream section. After air concentration is calculated

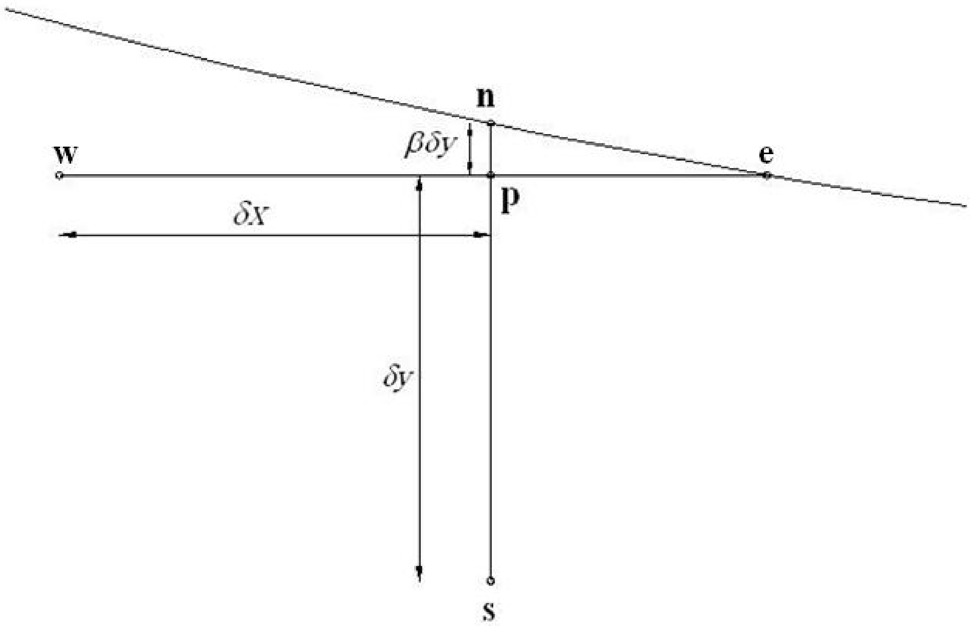

Fig. 3 Computational mesh at free surface

in each section, solution progresses towards downstream sections. Flow velocities at each section are calculated using equations 4 or 5 and ε_v using equation 3.

Backward difference is used for the left-hand side term of equation 6 (i.e. in *x* direction) and central difference approximation is used for the terms on the right (i.e. in *y* direction). Therefore, first order and second order approximation is used in *x* and *y* directions respectively. In this way, marching technique is possible for computation. Calculation errors can be minimuzed if mesh sizes δx and δy are considered small enough. This algorithm requires minimum of computer memory owing to application of marching technique and time of computation will be short. In this condition reducing the mesh size will not increase the time of computation considerably.

Regarding the curvature of the flow free surface, computational mesh will be cut at the free surface (Figure 3). To deal with this problem first and second terms on the right side of equation 2 should be descreticized close to the free surface as follows:

$$\left.\frac{\partial c}{\partial x}\right|_p = \frac{c_p - c_w}{\delta x} \tag{8}$$

$$\left.\frac{\partial c}{\partial y}\right|_p = \frac{c_n - c_s}{(1+\beta)\delta y} \tag{9}$$

$$\left.\frac{\partial^2 c}{\partial y^2}\right|_p \frac{2}{\delta y\,(1+\beta)}\left(\frac{c_n - c_p}{\beta\delta y} - \frac{c_p - c_s}{\delta y}\right) \tag{10}$$

Hence, coefficients of equation 7 take the following forms at the free surface:

$$a_p = \left(\frac{u_p}{\delta x} + \frac{2\varepsilon_{(d)p}}{\beta\delta y^2}\right),\quad a_n = \left(\frac{2\varepsilon_{(d)p}}{\beta(1+\beta)\delta y^2} + \frac{v_s}{(1+\beta)\delta y}\right)$$

$$a_s = \left(\frac{2\varepsilon_{(d)p}}{(1+\beta)\delta y^2} - \frac{v_s}{(1+\beta)\delta y}\right),\qquad a_w = \frac{u_p}{\delta x} \tag{11}$$

Where β is a value less than 1.

4. Boundary Conditions and Solution Algorithm

The present numerical model is employed to calculate air reduction rate downstream of an injection slot installed at the bed of a channel and also in deaeration zone downstream of aerators. Boundary conditions for free surface and bed are similar in both of these cases and will be described in this section. Boundary condition at the inlet section will be given in describing each case separatelly in the following sections.

Free surface is assumed as a symmetric line and therefore concentration gradients normal to free surface was assumed zero. Since the slope of the free surface is small this condition could be considered as $\partial c/\partial y=0$. At the channel bed also $\partial c/\partial y=0$. Since equation 6 is parabolic, no boundary condition at the downstream section is requied.

To run the numerical model, at the first step geometry of the flow field including the length of the channel, channel width and slope, mesh sizes as well as hydraulic characteristics of the flow such as discharge and bed Roughness should be introduced to the program. Solution should then follow the following steps:

1. Assume a water surface profile at the beginning of calculations. This profile can be calculated using step by step method [3] and assuming no air content.

2. Solve equation 7 in the whole computational domain by marching technique and calculate air concentration at all mesh points considering boundary conditions.

3. Correct flow depth at each section based on calculated mean air concentration at each section (see Section 5) and return to step 2.

4. Repeat steps 2 and 3 up to the point that the convergence criteria is met. Such a criteria is defined as follows for the present numerical model:

$$Max\left|\frac{y^{k+1}-y^{k}}{y^{k+1}}\right|<\varepsilon$$

Where y^{k+1} and y^k represent calculated flow depth at two successive iterations and ε is a small value.

5. Calculation of the Free Surface

At the first stage of calculation as described in the previous section step by step method can be used to calculate the free surface assuming no air in water. Alternativelly water surface level can be considered parallel to the bed at the beginning of calculations. In the next step, sloving equation 11, air concentration is calculated in the whole flow field. Knowing air concentration at all sections, free surface of air-water mixture could be calculated using Wood method as follows [14].

To calculate water surface position in the presence of air, bed friction coefficient should be evaluated first. Previous researches have shown that existance of air in the flow decreases the friction coefficient. Using experimental data of aerated flows in equilibrium zone (where gradient of air concentration in flow direction is zero) Wood [14] developed a relationship between mean air concentration in depth $\bar{c}_e$ and Darcy-

Weisbach friction Coefficient f_e . This relationship can also be used in gradullay varied flows with air concentration changing in flow direction.

Wood showed that mean specific energy of air water flow in unit weight at each section can be written in the following form:

$$\overline{SE} = d \cos\theta + \overline{E}\frac{u_w^2}{2g} \tag{12}$$

where d represents depth of flow with no air content, $u_w = q_w / d$ is mean flow velocity of the air-water mixture, q_w is unit discharge, θ is the angle between channel bed and the horizon. Measured values on Aviemore Dam show that $\bar{E}$ is independent of c and has a value of 1.05. Total energy at each section can therefore be written as: [14]

$$H = z + d \cos\theta + \frac{1.05}{2g}(\frac{q_w}{d})^2 \tag{13}$$

Where H represents total energy and z is the bed elevation. Differentiating Equation 13 and simplifying it, dynamic equation of gradually varied flow in air-water mixture can be found without considering the bulking effect due to air entrainment. This equation can be written as:

$$\frac{d}{dx}d = \frac{\sin\theta - S_f}{\cos\theta - 1.05 Fr^2} \tag{14}$$

Where $Fr^2 = \frac{q_w^2}{gd^3}$, $S_f = (\frac{q_w^2 f_e}{8gd^3})$ and $\frac{d}{dx}d$ represent depth variations along the flow.

Assuming a flow depth at the beginning of computation, Equation 11 is solved for calculation of air concentration in the flow domain. Equation 14 will then be solved and flow depth without air will be obtained. Knowing d and mean air concentration at each section, $\bar{c}$, flow depth of air-water mixture can be evaluated using the following equation[2,14]:

$$y = \frac{d}{1-\bar{c}} \tag{15}$$

Where y represents flow depth in the air-water mixture. Flow depths calculated from equation 15 can then be used in the next iteration for calculation of air concentration in the flow domain using Equation 11. After this stage Darcy-Weisbach friction Coefficient f_e is updated and iteration continues till convergence is achieved.

6. Model Verification

Results of an analytical model as well as physical model studies downstream of an air injection slot and deaeration zone downstream of an aerator are used to verify the accuracy of the numerical model.

6.1 Comparing Results of the Present model with Nokes Analytical Model

Analytically solution of air continuity equation (Equation 2) is presented by Nokes (1985) [9]. To solve this equation, Nokes considered following distributions for flow velocity and turbulence diffusion coefficient:

$$u = \bar{u}(1+\alpha)(\frac{y}{H})^{\alpha} \tag{16}$$

$$\varepsilon_v = \varepsilon_d = u_* \kappa y(1 - \frac{y}{H}) \tag{17}$$

where, α = 0.1852, H represents flow depth and $\bar{u}$ is mean flow velocity. A channel 4 m long with a slope of 45º and a mean flow velocity of 30 m/s and constant flow depth of 4 cm was assumed in these studies. Air concentration of 50% was introduced at the inlet section from 0.02 to 0.12 of flow depth and rising velocity of bubbles was assumed

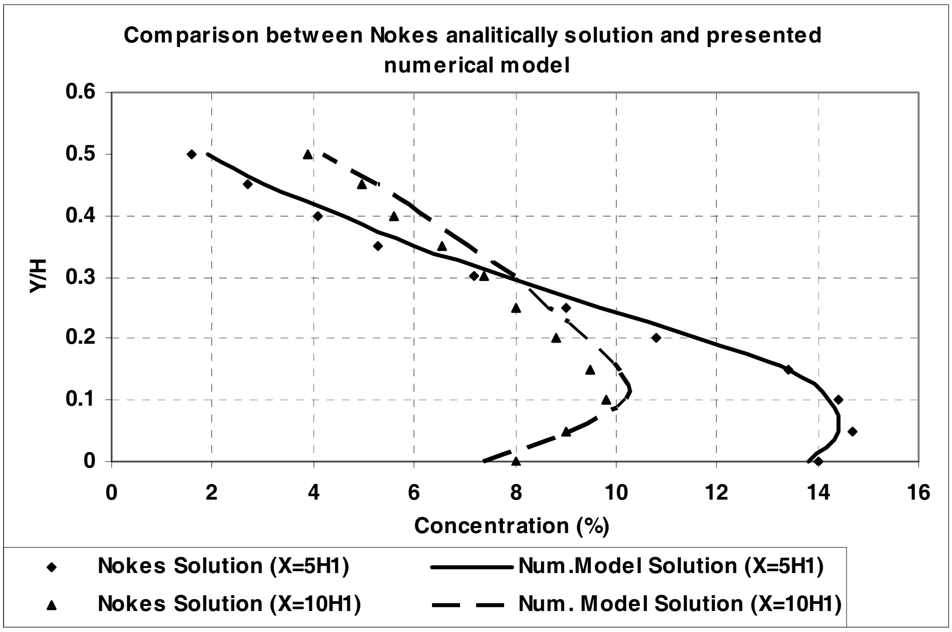

Fig. 4 Comparison between Nokes analytical solution and the present numerical model

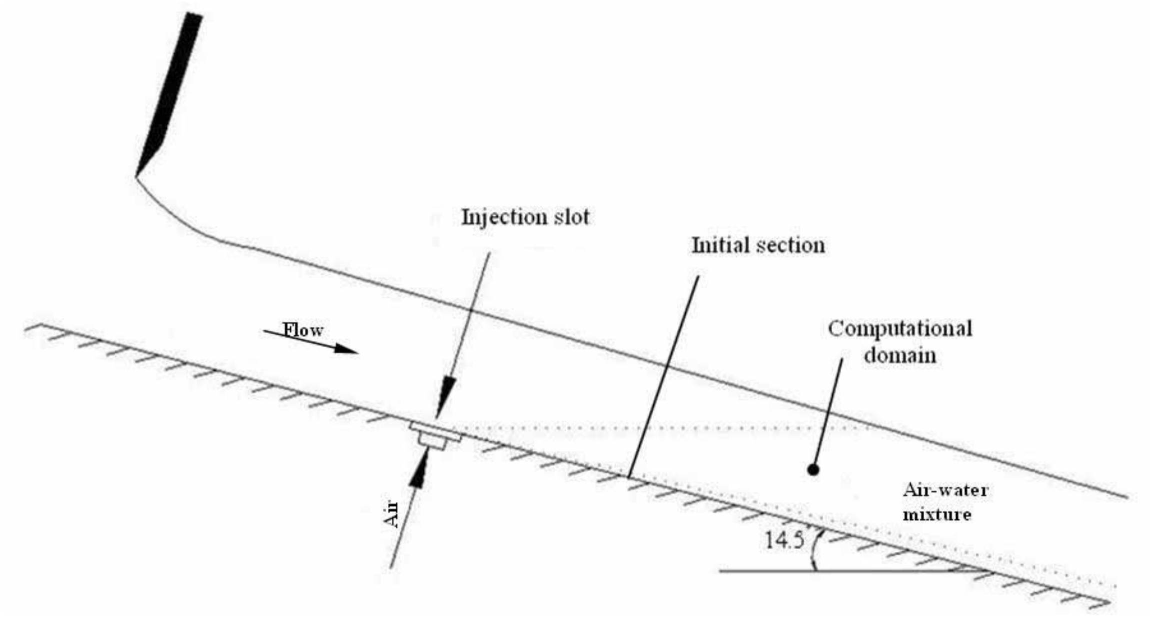

Fig. 5 Air injection from bed

20 cm/s.

Present model was solved in combination with equations 16 and 17 for the same problem and concentration distribution profiles were calculated at different sections. Comparison of the calculated concentration profiles with analytical results of Nokes in two sections at 5, and 10 times the flow depth downstream of the inlet section are shown in Figure 4.

As it is shown in Figure 4, only a slight discrepancy is seen between two models which could be due to errors of reading numbers of iso-concentration lines presented by Nokes (1985) [9]. So, it can be concluded that results of the present model well agrees with the analytical solution.

6.2 Prediction of Air Concentration Distribution Downstream of an Air Injection Slot.

In this section the present model is used to simulate air diffusion downstream of an injection slot.

Zarrati [15] injected air through a porous slot installed at the bed of a flume and measured air distribution at downstream sections, using a needle probe [18]. The first section immediately downstream of the slot was used as the inlet section in the numerical model and measured air concentrations at this section were used as the initial boundary condition. (Figure 5).

Experimental flume had a bed slope of 14.5°, a bed width of 15 cm, a flow depth of 2.4 cm, a mean flow velocity of 3.4 m/s and Darcy-Weisbach Coefficient of the flume was 0.024. Air was injected to the high speed flow with different discharges. Rising velocity of bubbles was considered 12 cm/s and 19 cm/s in the numerical model for air concentration rates smaller than 5% and greater than 5%(and less than 10%), respectively based on previous experiences [16]. Equations 4 and 5 were used to calculate flow velocity and equation 3 was employed to evaluate turbulence diffusion coefficient, assuming a Schmidt Number value of 0.7. As was mentioned before, to protect concrete surface from cavitation attack air concentration near the channel invert should not be less than 8% [10]. It is therefore important to predict the air content near the channel surface. Longitudinal profiles of concentration at 3 mm above the bed, for two low and high air injection rates are compared with experimental data and numerical model of Zarrati [15] in Figure 6 and 7. Sensitivity analysis was also performed to investigate the effect of computational mesh size. It was concluded that for values of 0.001 m and 0.0001 m for length and height of the computational mesh, results would not be affected by mesh dimensions.

As it is shown in figures 6 and 7, there is a good agreement between results of the present model and experimental measurements. Comparing present model results with 2-D K–ε numerical model of Zarrati [15] indicates accuracy of the present quasi 2-D model in the unidirectional model tested here. It should be noted that the present model solves only few equations with the capability of free surface calculation and can be developed much faster.

6.3 Predicting Concentration Distribution Downstream of an Aerator

Zarrati [17] measured air concentration and velocity at different sections downstream of an aerator by a double needle probe. Slope of the flow channel was 14.5° and flow discharge was 23.24 lit/s. To simulate air distribution in the deaeration zone

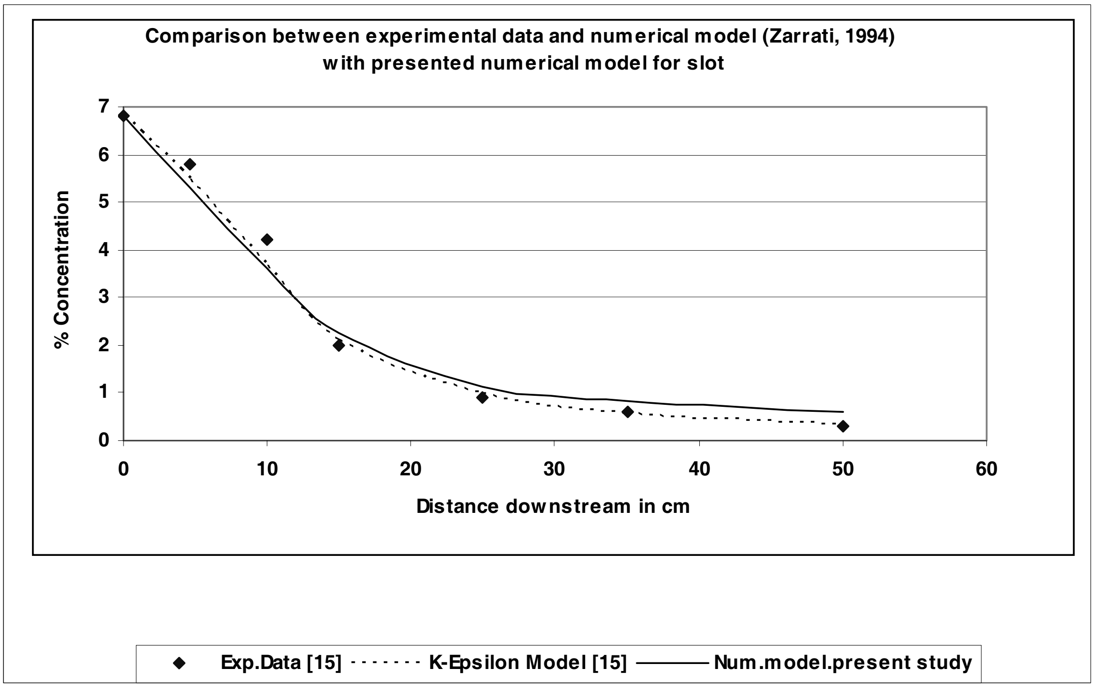

Fig. 6 Longitudinal profiles of air concentration at 3 mm from the bed at low air discharge

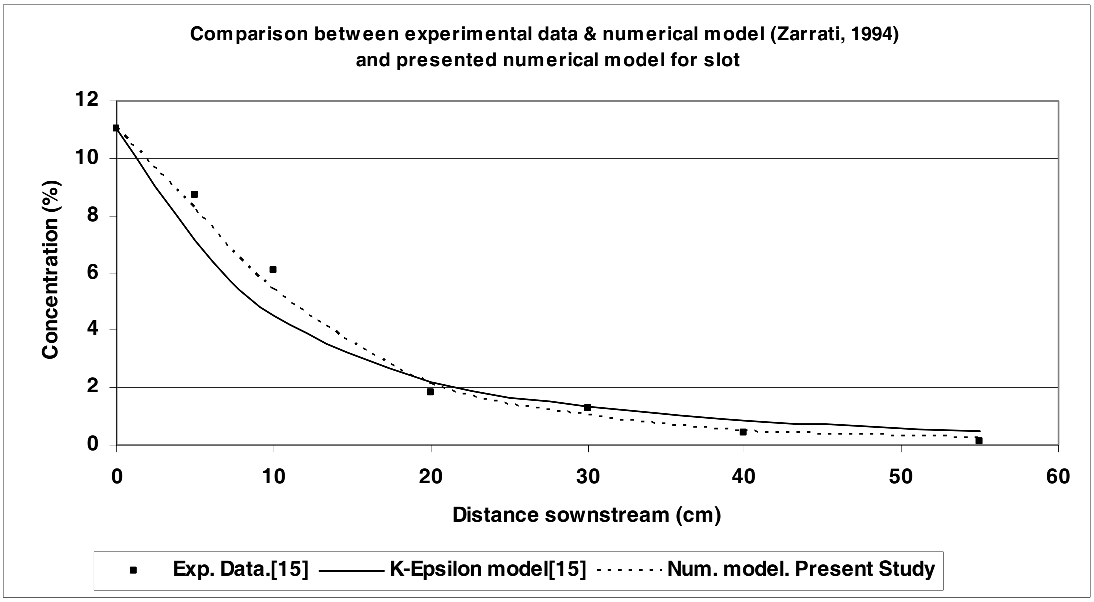

Fig. 7 Longitudinal profiles of air concentration at 3 mm from the bed at high air discharge

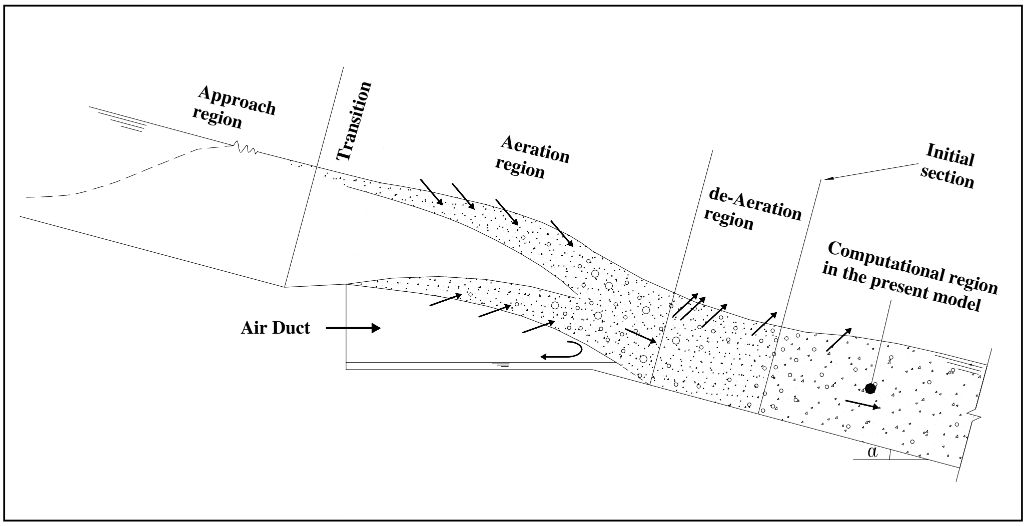

Fig. 8 Flow pattern over an aerator

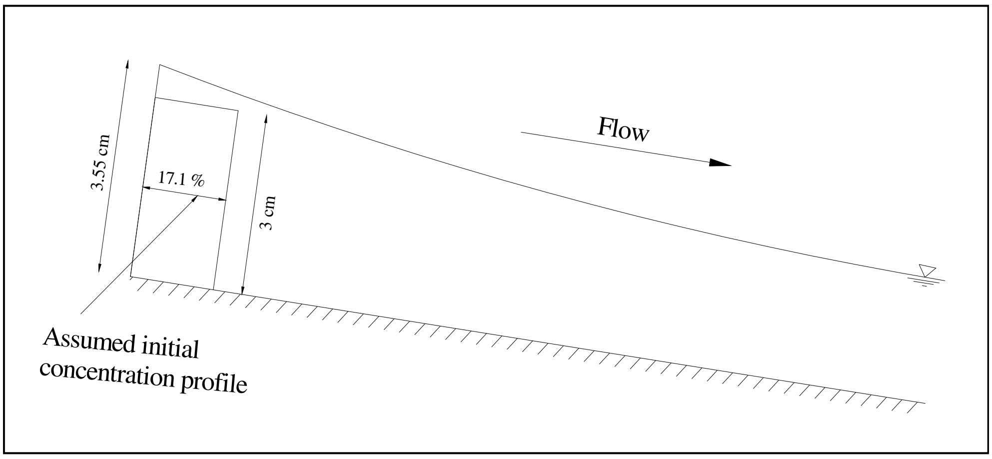

Fig. 9 Initial condition for simulation of deaeration zone

downstream of the jet impact a computational domain was assumed starting from a section after the impact zone where pressure in depth was again hydrostatic (Figure 8). Zarrati and Hardwick [17] measurements showed that immediately after the impact zone, air concentration was constant and equal to 17.1% in a 3cm layer from the bed. This section was therefore selected as the inlet section for the numerical model. Knowing flow discharge and velocity total depth of the flow at this section was calculated and is shown in Figure 9.

Longitudinal profiles of air concentration at 3 mm above the bed as calculated by the numerical model are compared with experimental data in Figure 10.

As shown in this figure, decreasing air concentration near the bed is very well predicted by the present model. It can

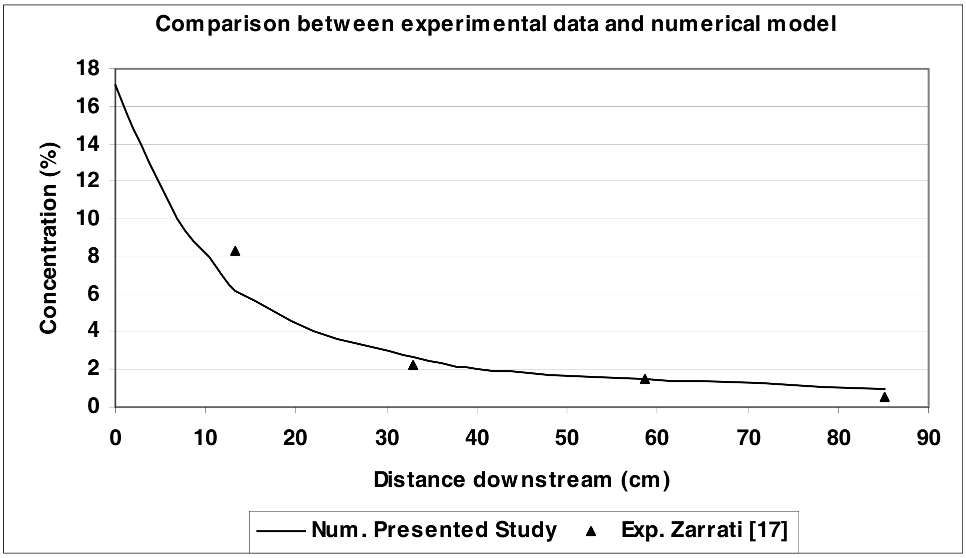

Fig. 10 Longitudinal profiles of air concentration, 3 mm from the bed, at deaeration zone downstream of an aerator

therefore be concluded that the model can be used to simulate distribution and diffusion of air in high speed air water flows.

7. Summary and Conclusions

To prevent cavitation, air is forced into the flow employing aerators. A numerical, quasi 2-D model is developed in the present research work to simulate air distribution in high speed air water flows. In this model air continuity equation is solved in the flow domain. Water surface level is determined using step by step method considering the amount of air content in the flow. Flow velocity and turbulent diffusion coefficients are calculated based on well known equations for open channel flow. The developed numerical model results were compared with 1- an analytical solution of air continuity equation downstream of an air injecting point, 2- experimental as well as a numerical model of air distribution downstream of an injection slot at the invert of a channel and 3- deaeration zone downstream of an aerator. All results showed the ability of the present model for calculation of air distribution in high speed air-water flows.

8. References

[1] Chanson, H.: 1989, Flow downstream of an aerator-aerator spacing, IAHR, Vol.27, No.4, 519-536.

[2] Chanson, H.: 1996, Air Bubble Entrainment in Free – Surface Turbulent Sheer Flows, Academic Press, London, UK, 127-130.

[3] Chow, V.T.: 1959, Open Channel Hydraulics, McGraw-Hill Publisher.

[4] Falvey, H.T.: 1990, Cavitation in Chute and Spillways, Eng. Monograph 42, USBR, Denver, USA.

[5] Gibson, M.M., Launder, B.E.: 1978, Ground effects on pressure fluctuations in the atmospheric boundary layers, J.

of Fluid Mech, Vol. 86, Part 3, 491-511.

[6] May, R.W.P.: 1987, Cavitation in hydraulic structures : Occurrence and preventation, Hydraulic Research Report SR 79, Wallingford, England.

[7] Minor, H.E.: 2000, Cavitation preventation on the Alicura and the Restruction spillways, Personal Communication.

[8] Nezu, I., and Nakagawa, H.: 1993, Turbulence in Open Channel Flows, IAHR Monograph Series, Balkema Publ.

[9] Nokes, R.I.: 1985, Dispersion of buoyant particles in two-dimensional turbulent open channel flow, 21st IAHR Congress, 200-204.

[10] Peterka, A.J.: 1953, The effect of entrained air on cavitation pitting, IAHR–ASCE Joint Conference, Minnesota, Minneapolis, USA, 507-518.

[11] Prusza, V.: 1983, Remedial measures against spillway cavitation, 20th IAHR Congress, Moscow, USSR, 468-476.

[12] Schlichting, H.: 1979, Boundary layer theory, Seventh ed. Mc Graw-Hill Classic Text Book Series, New York.

[13] Versteeg, H.K. and Malalasekera, W.: 1995, An Introduction to Computational Fluid Dynamics, The Finite Volume Method, Longman Book Publisher.

[14] Wood, I. R.: 1985, Air water flows, 21st IAHR Congress, Brisbourne, Australia.

[15] Zarrati, A.R.: 1994, Mathematical modeling of air-water mixtures in open channels, J. of Hydraulic Research, Vol.32, No.5, 707-720.

[16] Zarrati, A.R., Hardwick J.D.: 1991, Rising velocity of air bubbles in high speed free surface flows, Int. Symposium on Environmental Hydraulics, University of Hong Kong, Hong Kong.

[17] Zarrati, A.R.: 1995, Mathematical modeling of deaeration zone downstream of aerators, HYDRA 2000, Vol. 2, 220-225.

[18] Zarrati, A.R. et al.: 1998, Development of a Needle Probe for Measurement in Air-Water Flow, Int. Conf. on Water Res. Eng. ASCE, USA.