A NEW FORMULATION TO ESTIMATE INITIAL DILUTION OF DENSE JET FLOW

A.H.AZIMI[1], F. AZIZI[2] and A. ETEMAD-SHAHIDI[3]

[1]Research Associate, College of Civil Eng., Iran Uni. of Sci. & Tech.,
Tehran, Iran, P.O.Box 16675-163
(Tel: +98-21-7896623, Fax: +98-21-7454053, e-mail: a_h_azimi@iust.ac.ir)
[2]Research Associate, College of Civil Eng., Iran Uni. of Sci. & Tech.,
Tehran, Iran, P.O.Box 16675-163
(Tel: +98-21-7896623, Fax: +98-21-7454053, e-mail: fatemeh_2000_azizi@yahoo.com)
[3]Assistant Professor, College of Civil Eng., Iran Uni. of Sci. & Tech.,
Tehran, Iran, P.O.Box 16675-163
(Tel: +98-21-7896623, Fax: +98-21-7454053, e-mail: etemad@iust.ac.ir)

The behavior of plume trajectory and its dilution are commonly experienced with physical models and the laboratory results are compared with the field observation. However, behavior of discharged flow in the ambient water is complex and exact simulation for optimization and prediction of plume dilution is difficult. In the presence of computer technology, numerical simulation of discharged flow and their corresponding parameter results in saving in time and expenses compared to using laboratory experiments.

The aim of this study is to analysis the behavior of dense jet flow and fined a new formulation for outfall design in dense receiving water. The proposed equation is verified and compared with an expert system (CORMIX).

Any dependent variable denoted by Φ can be characterized as a function of the jet and ambient variables:

$$\Phi = f(Q, M, B, H, u_a) \tag{1}$$

For example, the initial dilution, S_m, can be expressed as:

$$S_m = f(Q, M, B, H, u_a) = f(u_o, u_a, H, D, \frac{\Delta\rho}{\rho_a}) \tag{2}$$

Where Q is kinematics mass, M is momentum, B is buoyancy fluxes, D is port diameter, u_o and ρ_o are jet discharge velocity and density, respectively. ρ_a is the ambient density, g is gravitational acceleration and H is depth of discharge (Fischer et al., 1979).

If the functional relationship in Eq. (1) is expressed in nondimensional parameters formed from the various above parameters, one possible non-dimensional number is (Azimi and Etemad-Shahidi, 2003):

$$R_V = \frac{u_a.H}{u_o.D} \tag{3}$$

R_V is the ratio of ambient volume flux to the discharge volume flux.

The propose equation is combination of the above variables.

$$S_p \propto c(H / L_b)^a.R_V^b \tag{4}$$

L_b represents the vertical distance at which the velocity induced by the buoyancy has decayed to the ambient velocity, and is expressed as:

$$L_b = \frac{B}{u_a^3} \tag{5}$$

Where a, b, c are model coefficients to be determined based on laboratory experiment and field measurement from Richards Bay experiment and S_p is the semi empirical dilution.

A series of experiment was conducted to simulation of a dense mixture of gypsum, which is discharged into the Indian Ocean at Richards Bay, (Roberts and Toms, 1987). In this experiment, the nozzles discharged the effluent inclined upwards at 60°. Most tests were for the inclined jets for which the angle of the crossflow to the vertical plane through the jet (ϕ) was 0° (jet opposing the crossflow), 30°, 60°, 90°, 120°, 150°, and 180° (coflowing). The model coefficients of equation (4) were determined by using laboratory experiments of Roberts and Toms (1987).

That yield $a = 1$, $b = -2.5$ and $c = 2.8$. These values in Equation (4), gives equation (5). Equation 5 is then used to obtained semi empirical dilution for our outfall discharges.

$$S_p = 2.8(H / L_b).R_V^{-2.5} \tag{5}$$

Three types of numerical models are presently available to simulate the discharged flow characteristics. Three dimensional finite element or finite difference models solve the system of complex equations that require large computational time. The second type of numerical models is called integral type model. Integral type models are based on ordinary differential equations. These models are developed by integrating the equations of motion over a control volume. The last group of numerical models is called length scale models. Length scale models use dimensionless parameters to classify the flow regime and predict the initial dilution. Initial dilution obtained from laboratory experiments and field observations have been extensively verified and compared with the last group of numerical models. Tsanis et al. (1994) compared the results of initial dilutions for three types of multi-port diffusers with five integral type models and a length scale model (CORMIX) they found that CORMIX predictions compare well with laboratory data and very limited field data.Cornell Mixing Zone Expert System (CORMIX) uses length scales for classification of flow regimes and predicting the initial dilution. CORMIX consists of three separate submodels, CORMIX1, CORMIX2 and CORMIX3 for analysis and prediction of outfall behavior. CORMIX simplifies the characteristics of each stage in the steady-state condition and predicts the plume dilution by using some semi empirical equations. in order to evaluate the performance of the suggested prediction formula, the dilutions substituting obtained from laboratory experiment of Roberts and Toms (1987) were numerically simulated with a robust mixing zone models i.e. CORMIX.

Comparison between measured and predicted dilutions shows that both proposed equation and CORMIX model performs well and the results have good agreement with the observations. The predicted results of CORMIX model and proposed formula are well supported regarding ±50% error stipulated by the CORMIX model. The dilution obtained from the proposed equation had on average +2.8% error and the standard deviation of the results is 26.6%. Dilution predicted by CORMIX model had on average +15.1% error and the standard deviation of the results is 33.8%.

REFERENCES

Azimi, A.H. and Etemad-Shahidi, A. 2003, Performance of an integral type model applied to ocean outfalls, 7^{th} IWA Conf. on diffuse pollution and basin management (DIPCON), Dublin, pp. 134-139.

Fischer, H. B., List, E. J., Koh, R. C. H., Imberger, J. and Brooks, N. H. 1979. "Mixing in inland and Coastal Waters." Academic Press, New York, U.S.A. 340p.

Roberts, P. J. W., and Toms, G. 1987. "Inclined dense jets in a flowing current." Journal of Hydraulic Engineering., ASCE, 114(3), 323-341.

Tsanis, I. K., Valeo, C., and Diao, Y., 1994. "Comparison of near-field mixing zone models for multi port diffusers in the Great Lakes" Can. J. Civ. Eng., Vol. 21, 141-155.