Refrence

[1] Constantinescu. G.S, Patel, V.C, (1998) " A Numerical Model for Simulation of Pump Intake Flow and Vortices ", J. of Hyd. Eng., Vol. 124, PP 123-134

[2] Constantinescu. G.S, Patel, V.C, (2000) " Role of Turbulence Model in Prediction of Pump Bay Vortices ", J. of Hyd. Eng., Vol. 126, No 5, PP 387-391

[3] Forbes, L, Hocking, G. (1998) " Withdrawal from a two layer inviscid fluid in a duct" J. of Fluid Mech. Vol. 361 PP. 275-296

[4] Hocking. G. (1995) " Super critical Withdrawal from a two layer fluid through a line sink" J. of Fluid Mech. Vol. 297 PP 37-47

[5] Jain, A.K, Raju, K.G, Grade. R.J (1978), " Vortex Formation at Vertical Pipe Intakes ", J. of . Hyd. Eng., Vol. 104, No. 10, PP 1429-1445 1978

[6] Knauss, J., (1987), Swirling Flow Problems at Intakes, AIHR Hydraulic Structure Design Manual, Rotterdam, Netherlands

[7] Montazerin, N., Rahimzadeh, H., Hashemi Marghzar, Sh., (2001), "Prediction of Critical Condition at a Horizontal Intake Through Pressure Contours", Proceeding of the XIV IAHR Congress in Hydraulics, China, Beijing, PP 434-441-

[8] Nezu I., Nakagawa,H, (1993),Turbulence in Open-Channel Flows, IAHR-Monographs, Delft, Netherlands

[9] Niazmand H, Shaw, B. D., Dwyer, H. H., (1994), "Effect of Marangoni Convection on Transient Droplet Evaporation " Combust. Sci. Tech., Vol. 103, PP 219-233

[10] Odgard, A.J, (1986), " Free Surface Air Core Vortex", J. of Hyd. Eng., Vol. 112, No 7, PP 610-619

[11] Patankar S.V., (1980), Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington, D.C.

[12] Panton R, (1996), Incompressible Flow, 2nd edition, A Wiely Int. Pub. NY

[13] Rahimzadeh, H., Hashemi Marghzar Sh., Hajebrahim M., (1999) " An Experimental Investigation on Flow Behavior in a Vertically Downwards Intake", Proceeding of 2nd Iranian Conference on Hydraulic, Vol. 1, PP 56-63 (in Persian)

[14] Rajendran, V.P, Constantinescu G.S, Patel, V.C, (1999) " Experimental Validation of Numerical Model of Flow in Pump Intake Bay", J. of Hyd. Eng., Vol. 125, No 11, PP 1119-1125

[15] Rajendran, V. P, Patel, V.C, (2000) " Measurement of Vortices in Model Pump – Intake Bay by PIV", J. of Hyd. Eng., Vol. 126, No 5, PP 322-335

[16] Rodi.W (1993), Turbulence Models and Their Application Hydraulics, IAHR–Monographs, 3d edition, Delft , Netherlands.

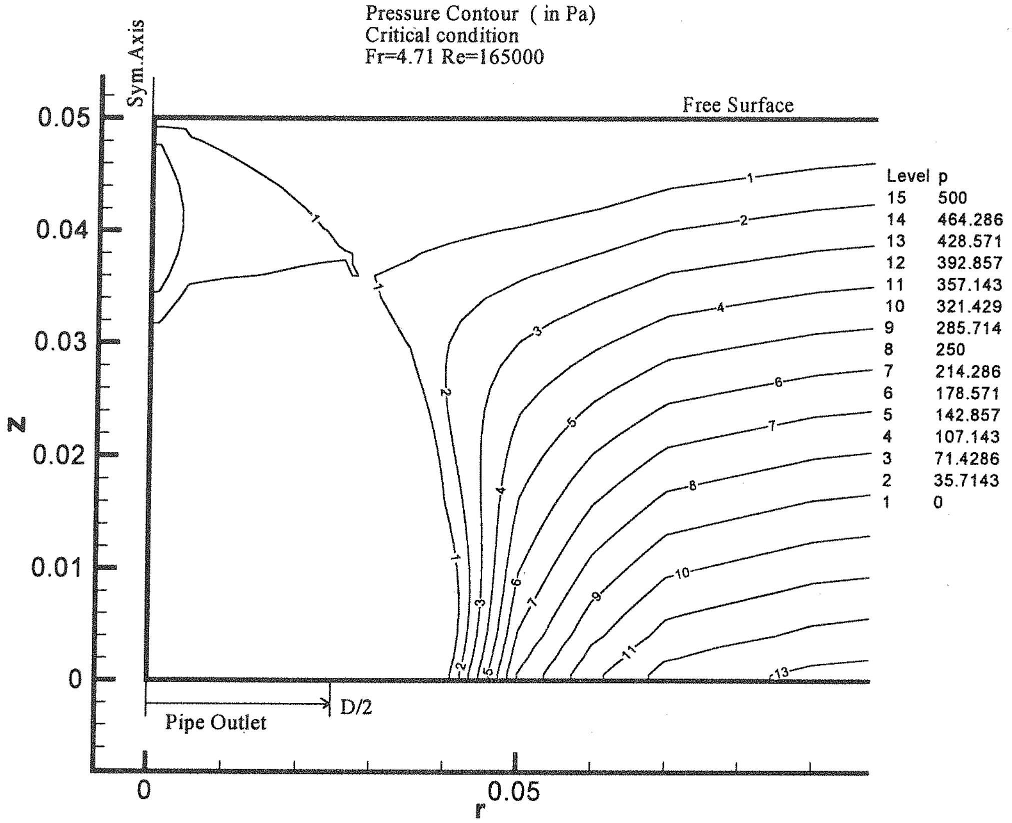

Figure (7) The pressure distribution in critical condition.

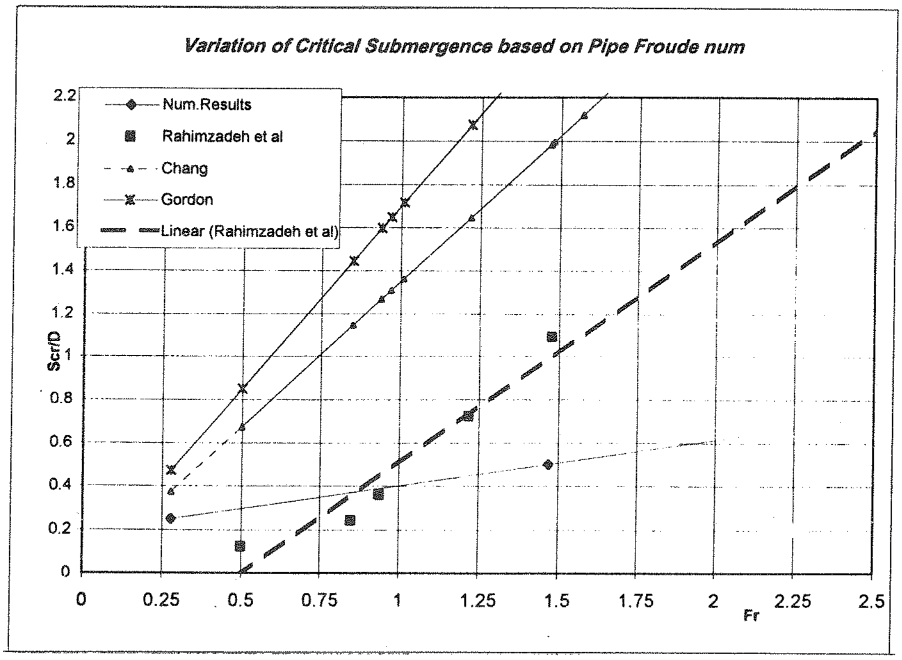

Figure (8) Comparison between results based on pipe Froude number.

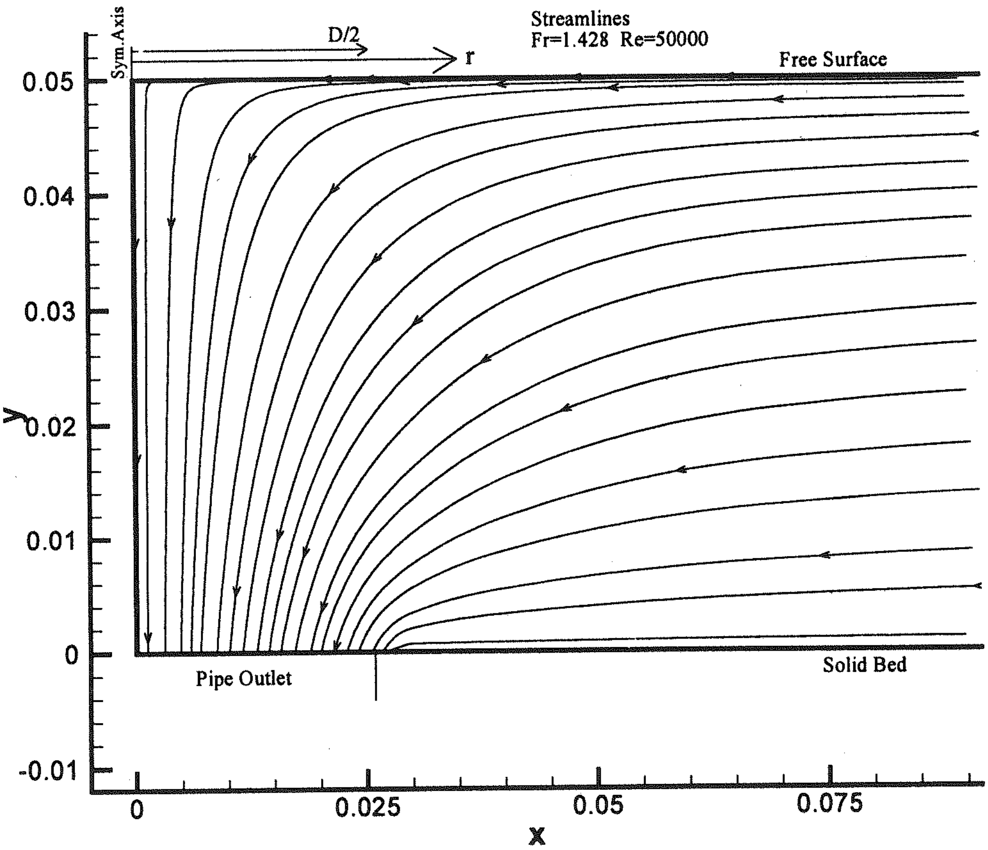

Figure (5) The streamlines for flow description.

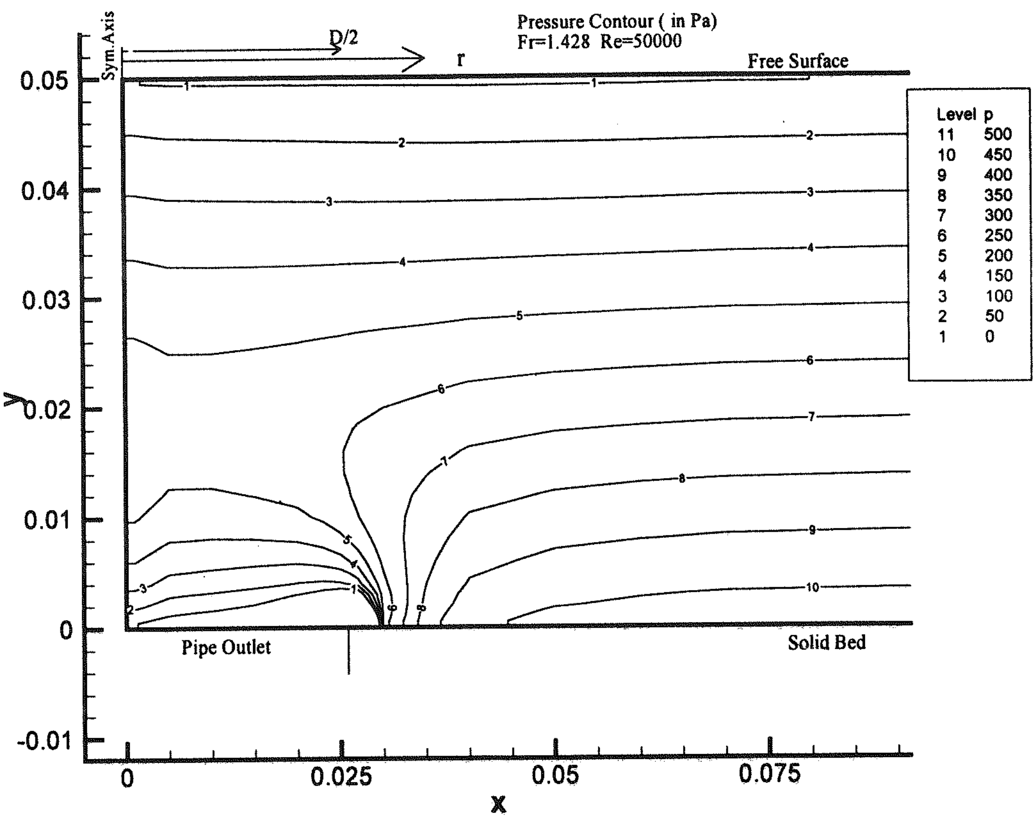

Figure (6) The pressure distribution in sub-critical condition.

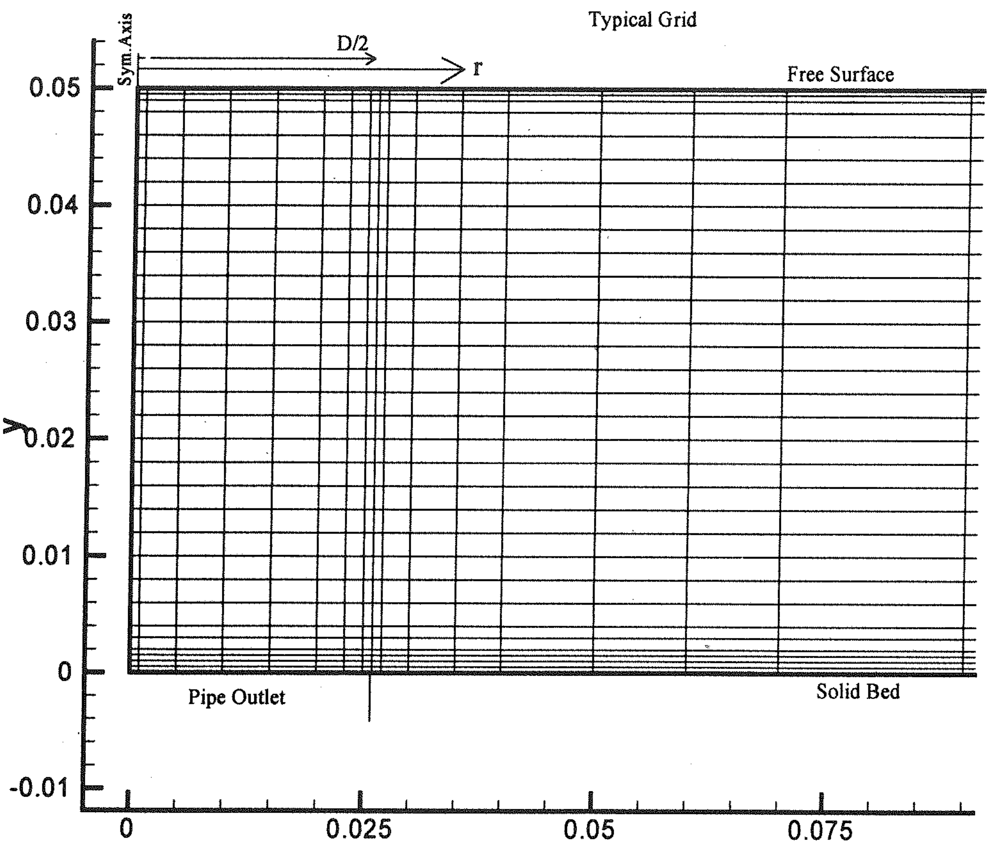

Figure (3) Typical grid for modeling.

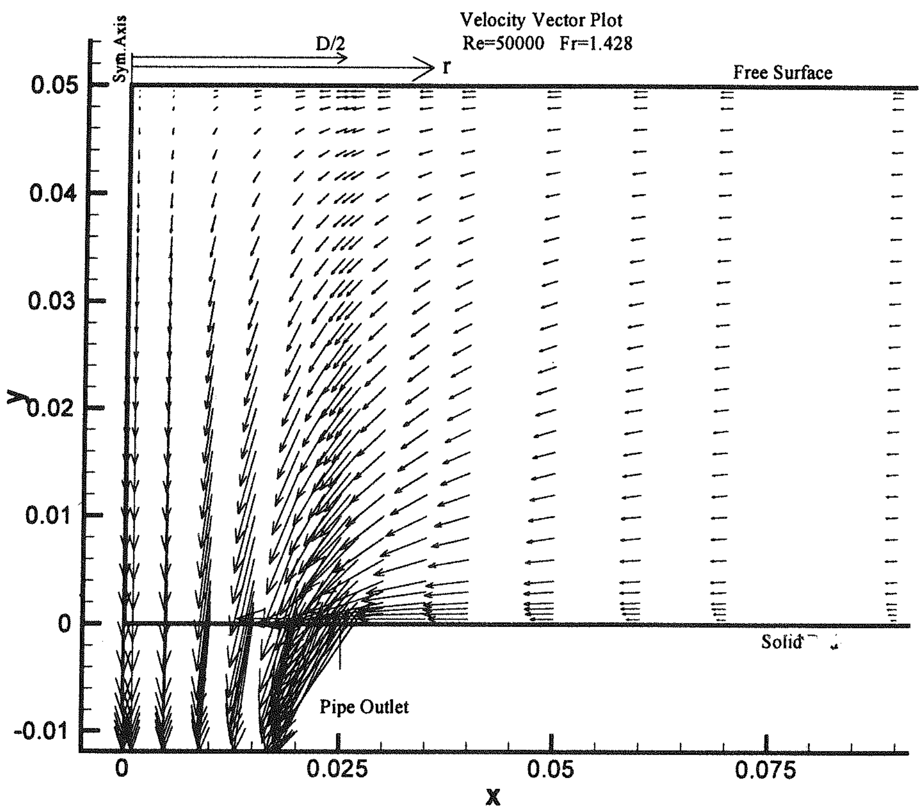

Figure (4) Velocity vectors plot.

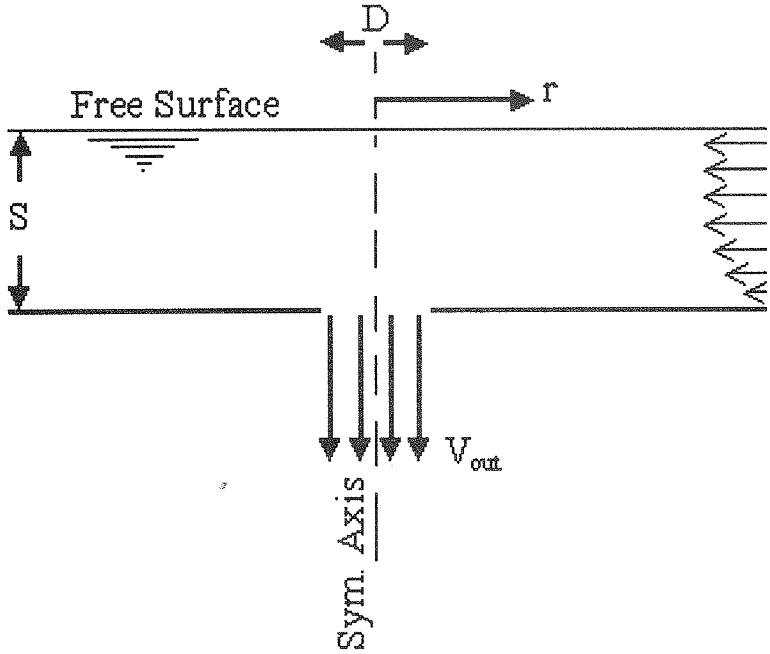

Figure (1) Problem description.

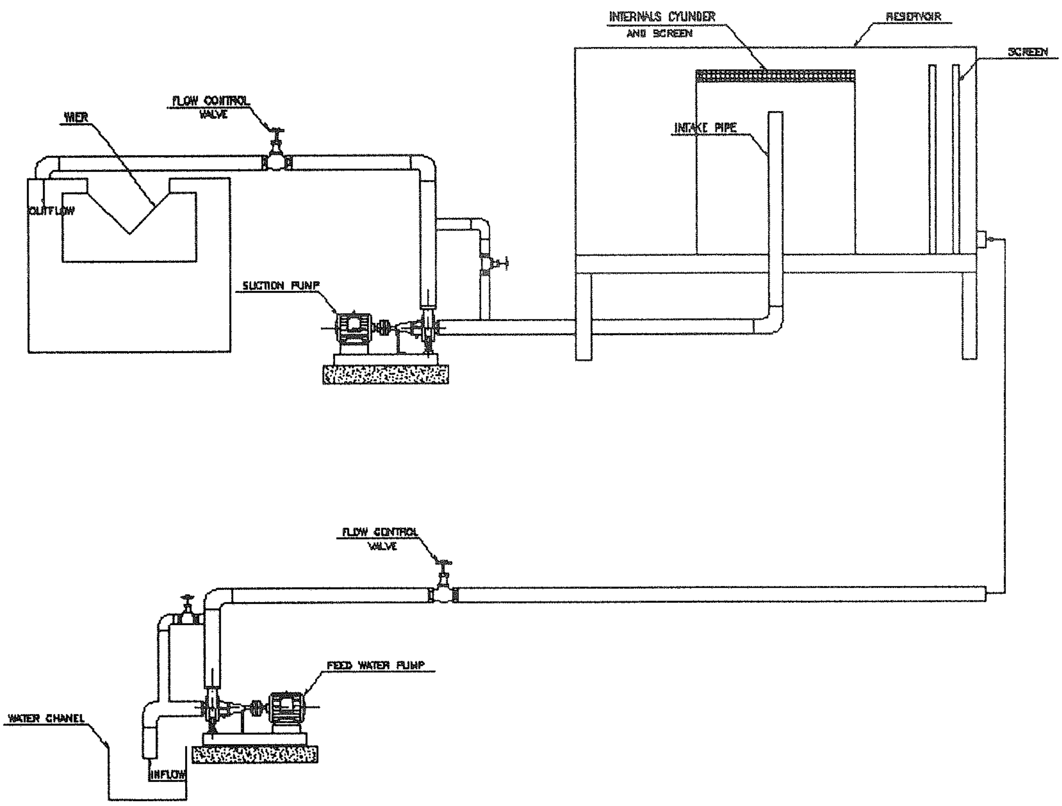

Figure (2) Physical model.

model show the critical submergence should be increased when the velocity (or Froude number) increases in the system. It is expected from the both experimental and analytical works. It is evident that the results from experimental works should show more critical condition because the experimental results were extracted from a rotational basement.

In fact the vorticity generation by inherent non-uniform roughness in the system or related turbo machinery, such as a pump, can not be controlled. Both asymmetries in the geometry and non uniformity in the flow entrance can create free surface vortices without using any especial treatment. It is necessary to mention that special treatments such as trainers can make strong swirl condition, then results from these type of experiments are so far from results obtained under random circulation tests and no surface swirl condition [6] [7].

8-Conclusion

A numerical method based on finite volume and SIMPLE algorithm is developed for turbulent and gravitational flows and modeled a vertically downward intake to predict the critical condition. The method solves the Navier-Stokes equations in conjunction with using k-ε turbulence model in cylindrical coordinate system.

The results are compared with the data from experimental investigation to validate numerical method and show rotation importance in large Froude numbers. Results from numerical modeling are in a close agreement with experimental data in the range of Froude number less than unity. They show a difference with experimental results for the region of Froude number is larger than unity. It shows the rotational effects are much more important in critical condition and vortex formation in the region Froude greater than unity. The fact that is the predicted values of critical submergence are similar to values that are extracted from the experiments is very important because the values can be used as the minimum value for intake design criterion. The rotational effects from any other sources such as geometry should be added to the predicted value when it is used for intake design. The model is being developed for three dimensional and stratified flows.

Acknowledgement

Amirkabir University of Technology supported this work. Authors wish to thank Prof. Jorg Imberger and Dr. Graeme Hocking for their helpful discussion and valuable comments on numerical modelling.

Notation

$C_{\varepsilon 1}$ = Constant in the turbulent transport equations;

$C_{\varepsilon 2}$ = Constant in the turbulent transport equations;

C_μ = Kolmogorov constant;

D = Intake pipe diameter;

$Fr = \dfrac{V_{out}}{\sqrt{gD}}$ = Pipe Froude number;

g = Gravitational acceleration;

K = Turbulent kinetic energy;

n = Normal direction;

P = Pressure

P_k = Production rate;

$Re = \dfrac{\rho V_{out} D}{\mu}$ = Reynolds number;

S = submergence;

Scr = Critical submergence;

t =Time;

V_{out} = Mean velocity at pipe outlet;

V_r = Velocity in r direction;

V_z = Velocity in Z direction;

Greek Symbols

ε = Rate of turbulent energy dissipation;

∂ = Differential operator;

μ = Water viscosity

μ_t = Eddy viscosity

ρ = Water density

σ_k = Constant in turbulent transport equations;

σ_ε = Constant in turbulent transport equations;

the solid bed therefore mentioned treatment will be active near the basin bed.

5-Pressure algorithm for recognition of critical condition

The critical condition is an important problem in intake modelling. In absence of surface tension effects, a balance between inertial and gravitational forces at the free surface makes the surface deformation. This condition makes a very complicated moving boundary problem. Then, the solution is limited to inviscid and potential flow problem solution [3], [4].

Here instead of using a moving boundary mechanism, a pressure algorithm is used to show the surface drop and critical condition. In critical condition, continuous zero (gauge) pressure line will be recognized from the free surface to the pipe. The critical condition is searched by a tracing method with using a try and error procedure. For each submergence depth, a value for critical velocity at intake outlet is guessed. The governing equations with selected velocity will be solved. Then, if the pressure contours show the sub critical condition the chosen velocity will be increased. Similarly for super critical condition, the selected initial value for outlet velocity should be reduced. The procedure will be repeated to achieve the critical condition.

6-Results and discussion

The velocity vectors are shown in fig 3. In this case, the values of Froude and Reynolds numbers are 1.428 and 50000, respectively. The distribution of velocity vectors shows the flow accelerates near the pipe. Fig 3 express that velocity should increase when the projected area becomes small in the vicinity of pipe. It is also seen that flow direction will change from radial to vertical near the pipe.

The distribution of velocity vectors shows a flat velocity profile, except near the bed. Radial velocity will be increasing far from to the pipe. Both mass conservation rule and axisymmetric geometry make this enforcement. The bed effects show that the near wall turbulence mechanism is affective and its height is limited to the maximum 5 percent of submergence depth. It is seen that the velocity vectors change their direction near the intake.

The streamlines usually show a good picture of fluid behavior. They are shown in fig 5. It is seen that the flow structure consists of three different regions including the uniform accelerating flow in radial direction from far field to the pipe outlet region, uniform flow at pipe outlet and a transition region close to the pipe intake. It is expected from the continuity for radial flow and flow through a contraction in cylindrical coordinate system.

The pressure distribution is shown in fig 6. In this case, the flow condition is sub-critical. From the figure, it is observed that the hydrostatic pressure distribution is existing in the basin except outlet zone, because the pressure gradient is required to flow leave the pipe. It is evident that the pressure gradient is the main source to flow changes its direction. From the velocity vectors plot, fig 4, and pressure distribution, fig 6, it is observed clearly.

It is explained that the critical condition is a result of observation of zero or negative pressure from the free surface to the pipe. The pressure distribution in critical condition is shown in fig 7. For this condition, the Reynolds and Froude numbers (based on pipe diameter) are 165000, 4.71 respectively.

7-Comparison between numerical and experimental results

Here it is interested to compare results from numerical modeling in absence of rotation effects with experimental data. The variation of critical submergence based on Froude number from numerical and experimental results are shown in fig 8. The results from other investigations [6] are also shown in mentioned figure to clarify the difference between experimental results where extracted under different random circulation condition and model test geometry.

In the range of Froude number less than unity, the predicted critical submergence depth from numerical model is in close agreement with the data from the experiments. Fig 8 also shows that the circulation effects are much more important for the region of Frroude number more than unity. The predicted results from numerical

$$\rho\left(\frac{\partial V_r}{\partial t}+V_r\frac{\partial V_r}{\partial r}+V_z\frac{\partial V_r}{\partial z}\right)=(\mu+\mu_t)\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial (rV_r)}{\partial r}\right)+\frac{\partial^2 V_r}{\partial z^2}\right)-\frac{\partial P}{\partial r} \tag{4}$$

$$\rho\left(\frac{\partial V_z}{\partial t}+V_r\frac{\partial V_z}{\partial r}+V_z\frac{\partial V_z}{\partial z}\right)=(\mu+\mu_t)\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V_z}{\partial r}\right)+\frac{\partial^2 V_z}{\partial z^2}\right)-\frac{\partial P}{\partial z}+\rho g \tag{5}$$

The turbulent viscosity μ_t is found to be function of the turbulent energy K and its dissipation rate ε as follow:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \tag{6}$$

Closure turbulence equations, including K and ε equations are:

$$\rho\left(\frac{\partial K}{\partial t}+\frac{1}{r}\frac{\partial (KV_r)}{\partial r}+\frac{\partial (KV_z)}{\partial z}\right)=\left(\mu+\frac{\mu_t}{\sigma_k}\right)\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial K}{\partial r}\right)+\frac{\partial^2 K}{\partial z^2}\right)+P_k-\varepsilon \tag{7}$$

$$\rho\left(\frac{\partial \varepsilon}{\partial t}+\frac{1}{r}\frac{\partial (V_r\varepsilon)}{\partial r}+\frac{\partial (V_z\varepsilon)}{\partial z}\right)=\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial \varepsilon}{\partial r}\right)+\frac{\partial^2 \varepsilon}{\partial z^2}\right)+\frac{\varepsilon}{K}\left(C_{\varepsilon 1}P_k-C_{\varepsilon 2}\varepsilon\right) \tag{8}$$

Where the production rate P_k and the constants are:

$$P_k=\mu_t\left(2\left[\left(\frac{\partial V_r}{\partial r}\right)^2+\left(\frac{\partial V_z}{\partial z}\right)^2+\left(\frac{V_r}{r}\right)^2\right]+\left[r\frac{\partial}{\partial r}\left(\frac{V_r}{r}\right)\right]^2+\left[\left(\frac{\partial V_r}{\partial r}+\frac{\partial V_z}{\partial z}\right)^2\right]\right) \tag{9}$$

$C_\mu=0.09$, $C_{\varepsilon 1}=1.44$, $C_{\varepsilon 2}=1.92$, $\sigma_k=1.0$ and $\sigma_\varepsilon=1.3$ [16], [8].

3-Numerical method and validation

The numerical model is based on finite volume method [9]. It is developed to solve the Navier–Stokes equations for turbulent and gravitational flow. It uses a structured and non-uniform grid. A hybrid-upwind central differencing scheme is used to discretize the convection diffusion terms. A semi implicit method for pressure and velocity correction is selected based on SIMPLE algorithm for staggered grid [11]. It is shown that corrector scheme is independent of gravitational forces [12]. The numerical model is applied on different test cases such as flow in a sudden expansions and cavity driven for code validation. The results are in close agreement with similar results from mentioned standard test cases.

3-1-Initial condition and boundary condition

A summary of initial and boundary conditions for this study are listed here.

Initial conditions, $t=0$
$V_r=V_z=0$, $P=P_{Hydrostatic}$

Solid boundaries
$V_r=V_z=0$ $\quad \frac{\partial P}{\partial n}=0$, $K=\varepsilon=0$

Free Surface
$V_z=0$ $\quad \frac{\partial U}{\partial n}=\frac{\partial K}{\partial n}=\frac{\partial \varepsilon}{\partial n}=0$

Outlet condition, $V_z=V_{out}$

4-Grid characteristics and near wall modelling

A typical grid for the present study is shown in Fig.3. It is an algebraic grid and includes the fine grid close to solid walls and free surface and coarse grid on the other parts. The calculations were made for different grids such as 32*24 and 70*40 to specify the code grid dependency.

K−ε model is designed for isotropic and highly turbulent flows. Due to the viscous influences near wall, the local Reynolds number becomes small, and mentioned model is inadequate. To overcome this, an especial treatment should be applied. A treatment based on the logarithmic law of the wall is applied to parallel the velocity component in the region nearby solid wall. In this procedure μ_t will be calculated by using the shear velocity definition. The Von Karman constant (κ) is selected for this study [8]. It ensures that the main turbulence model with mentioned coefficients is compatible with the logarithmic law velocity distribution near walls. Here the solid wall is limited to the initial circulation. Readers are refereed to reference [6] for more details about model tests, coefficients and suggested formulas.

To find the critical submergence, analytical solutions of the problem were based on specified vortex at free surface and neglecting of the kinematic viscosity [10]. Also an analytical–numerical formulation of the problem predicts the critical and super critical withdrawals for two layers model through a line sink [4]. It is based on an inviscid and potential type of flow.

A numerical modeling provides a powerful tool for the problem modeling when employed in conjunction with laboratory experiments. The Numerical results for intake simulation is limited to the pump sump modeling [1],[2]. They employed a finite difference method to solve the Reynolds average Navier–Stokes equations in conjunction with the k-ε model for a vertically upward intake. They assumed that free surface remained constant and used a pressure algorithm at the free surface for water drop calculation. Their model was applied for moderate Reynolds numbers and didn't predict any critical condition due to small Froude number. The numerical model was validated by laboratory tests [14], [15].

A two dimensional numerical model based on finite volume is applied in Cartesian system to simulate critical condition in a horizontal intake [7]. They used a free surface algorithm based on pressure contour to predict critical condition. The model results were in accordance with experimental data [7].

In the present work, it is interested to simulate the flow behavior in a symmetrical vertically downward intake to predict the critical condition in no surface swirl condition. The results carried out from both numerical model and laboratory tests show a close agreement for the range Froude less than unity. This helps to study the problem under more controlled condition to evaluate the non-rotational effects on critical condition formation. Results can be considered as the minimum possible value in order to predict the critical condition occurrence margin for this type of intake.

1-Model description, governing factors and assumptions

Since this study is based on laboratory experiments, the main model and governing factors are introduced below. A schematic drawing of the problem is depicted in Fig.1. It shows the basic flow configuration in a vertically downward intake under symmetry condition. It includes a circular basin, which is fed up radial and flow leaves through a pipe at the basin center. The pipe outlet diameter (D) is selected equal to 5 cm. The submergence depth (S) is chosen variable.

The experimental setup is shown in fig 2. It consists of a circular basin located in the reservoir with a hole at the basin center. Also this rig is equipped with pipes, isolating valves, feeding pump, suction pump and a weir for flow measurement. The flow enters into the circular basin radial as described in flow configuration. A simple mechanical probe carefully used to measure the depth of air core and indicate critical condition occurrence. The testes were done under forced and free discharge condition by using a suction pump. During tests no artificial circulation was made by any especial devices e.g. trainers and random circulation were made just by non-uniform roughness at solid boundaries and pump effects [13].

The geometrical parameter is limited to submergence ratio (S/D) for the problem formulation. In absence of surface swirl, main parameters are Froude and Reynolds numbers based on outlet pipe diameter. In this study it is assumed that the surface tension in turbulent flow is negligible and flow is studied in absence of circulation effects. It is also assumed that the flow leaves the pipe with a constant velocity. Since the flow is turbulent, this approximation is very close to nature of phenomenon.

2-Governing equations

The Reynolds averaged continuity and Navier-Stokes equations in cylindrical coordinates system for axisymmetric condition are as follows.

$$\frac{1}{r}\frac{\partial(rV_r)}{\partial r}+\frac{\partial(V_z)}{\partial z}=0 \qquad (3)$$

A Joint Experimental and Numerical Investigation on Flow Structure and Critical Condition at A Vertically Downward Intake

Sh. Hashemi Marghzar
Ph. D Candidate

H. Rahimzadeh
Associate Professor

N. Montazerin
Associate Professor

School of Mechanical Engineering,
Amirkabir University of Technology

Abstract

This paper presents a joint numerical-experimental study of the flow behaviour in a vertically downward intake in absence of any rotational effects. A numerical method based on finite volume is applied on governing equations in a cylindrical coordinate system to simulate the gravitational flow with fixed free surface grid. Results introduce critical condition based on Froude number. In this study, it is assumed that the critical condition is due to existing of continuous zero pressure line from the free surface to the intake pipe. In the region Fr<1.0, a fairly good agreement between numerical results and experimental data are observed. Predicted submergence depth from numerical approach is less than values from the experimental data for Fr>1.0. It expresses that the critical condition in high flow rate highly depends on random circulation due to non-uniformity in the flow field rather than flow regime and hydraulic parameters.

Keywords

Critical Condition, Vertically downward intake numerical simulation, Turbulent flow, Free Surface

Introduction

The symmetrical condition of flow field is often interested, especially in hydropower plant intakes. Frequently an uneven distribution of velocity occurs when the flow is being reversed in the approach channel or part of reservoir. Usually vortices and swirling phenomena are a result of non-uniform and asymmetrical condition in the flow field. These vortices are created and controlled by different factors such as solid boundaries, bed roughness, submergence depth and geometry ratios. It can reflect difficulties such as additional head loss, draw down of floating debris, reduction of efficiency in hydraulic machinery [6]. Prediction of critical condition has been of interest for engineering application and water quality management [6], [3].

The usual solution of the vortices problem and prediction of critical submergence is to conduct laboratory experiments on scaled model to identifying the source of particular and field practical solution to eliminate them. The results form experimental studies usually are presented with an exponential formula based on pipe Froude number.

$$(S/D)_{cr} = B\ .Fr^{x} \qquad (1)$$

The suggested value for exponent x and constant B are different respect to model tests. The experimental results show that the exponent x will be increased when circulation strength in the system increases [6]. It is suggested that in the range of practical Froude numbers (0.3<Fr<2.1) and for a normally given 'moderate' vortex formation, critical submergence is calculated by a linear formula [6].

$$(S/D)_{cr} = \text{constant}\ .Fr \qquad (2)$$

The constant in mentioned equation also depends on the geometry and the strength of