Numerical Simulation of Bubble Motions in a Gate Slot

Arman Raoufi, Mehrzad Shams, Reza Ebrahimi , and Goodarz Ahmadi

***Abstract*—A Lagrangian-Eulerian numerical scheme for the investigation of bubble motion in turbulent flow is developed. The flow is analyzed in the Eulerian reference frame while the bubble motion is simulated in the Lagrangian one. Finite volume scheme is used, and SIMPLEC algorithm is utilized for the pressure and velocity linkage. The Reynolds stresses are modeled by the RSTM model of Launder. Upwind scheme is used to model convective fluxes. The Results obtained of flow field simulation were compared with the experimental results and was shown good agreement with experimental data. The Gaussian Filter White Noise is incorporated to simulate the turbulent fluctuation velocities. The bubble diameter is found by the use of Rayleigh-Plesset equation. Various forces in the equation of motion of the bubble are considered. The Buoyancy, Saffman lift, drag, pressure, and change of volume forces are carefully applied. The effects of all of these forces on bubble path are also examined. The bubbles are created in the low pressure zones, and then traced in the flow field. It is observed that the bubble diameter is highly dependent on the mean stream pressure, and its location. The results are compared with the other published works, and have an acceptable accuracy.**

***Index Terms*—Two-phase flow, bubble, gate slot, Lagrangian-Eulerian approach, cavitation.**

I. INTRODUCTION

The first intensive study on outlets and gates was conducted by Kohler[1]. He reviewed criterions of gate selection at different condition. Use of Computational Fluid Dynamics (CFD) in hydraulic works was limited Because of difficulty of numerical simulation and common of experimental method and model test. CFD has a great potential to predict flow details, turbulent intensity and pressure disterbution in the gate. Among accomplished studies for numerical simulation of gate slot, can mention to Raoufi and shams [2]. Because of abrupt pressure drop in gate slots, probability of creation of cavitation is high.

In the liquid flow, cavitation generally occurs if the pressure in certain locations drops below the vapor pressure and consequently the negative pressure are relieved by the formation of gas-filled or gas and vapor-filled cavities. Cavitation occurs by the sudden expansion and the volumetric oscillation of bubble nuclei in the water due to the ambient pressure change. The size of bubble nuclei is in the order of O(10)[3]. Formation and collapse of bubble in liquids are used in many technical applications such as lithotripsy, ultrasonic cleaning, bubble chambers, and laser surgery [4]. Bubble dynamics has been the subject of intensive theoretical and experimental studies since Lord Raleigh (1917) found the well-known analytic solution of this problem for inviscid liquids. The advanced theory of cavitation developed by Plesset (1949) who found the differential Rayleigh – Plesset (RP) equation for the bubble radius [5]. The RP equation described the dynamics of a spherical void or gas bubble in viscous liquids and is also used as a first approximation in more complex problem such as cavitation near solid boundaries. Cavitation could be observed in a wide variety of propulsion and power systems like pumps, nozzles, injection, marine propellers, hydrofoils and underwater bodies.

The usual approach for predicting the cavitation inception pressure for a hydraulic device is to build and evaluate a small-scale model. While within certain restricted classes of turbomachines, the empirical scaling procedure may work satisfactorily, a physics-based predictive capability which integrates the known variables affecting the cavitation inception problem has not been developed.

Two-phase flow could be analyzed as two fluids in the Eulerian/Eulerian approach, or as a continuum phase and another bubble phase in the Eulerian/Lagrangian or trajectory approach. The bubble equation of motion is solved simultaneously with the RP equation to determine its trajectory (Eulerian/Lagrangian method) [6].

Meyer *et al* correlated a numerical simulation of the cavitation on a Schiebe headform. They developed a computer code to statistically model cavitation inception, consisting of a numerical solution to the RP equation coupled to a set of trajectory equations. Using the code, trajectories and growths were computed for bubble of varying initial sizes. An off-body distance was specified along the , and the bubble was free to follow an off-body trajectory. They also showed that the cavitation inception is sensitive to nuclei distribution [7].

Chahine [8] has pursued a different approach to study the dynamics of traveling cavitation inception bubbles using inviscid potential flow, but including the modifications of the flow by the nucleus, and allowing the nucleus to deform. These

Manuscript received March 19, 2007.
Arman Raoufi is with Department of Mechanical Engineering, K.N.Toosi University of Technology, Tehran, Iran (e-mail: a_raoufi@sina.kntu.ac.ir).
Mehrzad Shams is with Department of Mechanical Engineering, K.N.Toosi University of Technology, Tehran, Iran (corresponding author to provide phone: +98-912-359-6236; fax: +98-021-733-4338; e-mail: shams@ kntu.ac.ir).
Reza Ebrahimi is with Department of Mechanical Engineering, K.N.Toosi University of Technology, Tehran, Iran (e-mail: a_raoufi@sina.kntu.ac.ir).
Goodarz Ahmadi is with Mech. and Aeronautical Engng. Dept., Clarkson University, NY 5725-13699, USA (e-mail: ahmadi@clarkson.edu).

calculations have provided detailed information about the capture, growth, and collapse of a single bubble in a Rankine vortex.

Hsiao and Pauley [9] completed a Reynolds-averaged Navier-Stokes computation of a tip vortex flow from a finite-span hydrofoil. The Rayleigh-Plesset equation for bubble growth was coupled with Johnson and Hsieh's [10] trajectory equation to track single microbubbles through the steady-state flow field and thereby infer cavitation inception. The larger bubbles controlled the cavitation index, and their likelihood of cavitation was less dependent on the release location than for the smaller bubbles. The present work builds on this type of soft coupling, yet includes a spectrum of nuclei sizes, the effect of turbulence on trajectory and pressure, and some additional forces.

Kevine J. Farrell developed an Eulerian/Lagrangian computational procedure for the prediction of the cavitation inception. The trajectories were computed using Newton's second law with models for various forces acting on the bubble. The growth was modeled using RP equation. Cavitation inception data show that inception indices generally decrease with increasing velocity, which is contrary to expectations based on the increased flux of nuclei to the minimum pressure region [11].

Can F. Delate and *et al* [12] considered quasi-one-dimensional steady-state cavitating nozzle flows by taking into account the effect of bubble nucleation. The nonlinear dynamics of cavitating bubbles is described by the classical RP equation where a polytropic law for the partial gas pressure is employed by taking into account the effect of damping mechanisms by an effective viscosity.

3D Reynolds-averaged Navier-Stokes equations were used to investigate flow around tip of a finite-span hydrofoil by Hsiao and Chahine (2004) [13]. They studied the behavior of a bubble inside tip vortex using coupling Navier-Stokes equations, Rayleigh-Plesset equation and trajectory equation of bubble. Rayleigh-Plesset equation with considering term of slip velocity was solved.

Zhang and Ahmadi recently computed an Eulerian-Lagrangian computational model for simulations of gas-liquid-solid flows in three phase slurry reactors. They used a two-way interaction between bubble-liquid and particle-liquid are included in the analysis. But, they did not consider the bubble growth in their analysis [14].

In this study, an Eulerian–Lagrangian method for liquid–gas flows in gate slot is developed. In this model, the liquid is the continuous phase and the bubbles are treated as the dispersed discrete phases. The microbubbles are assumed to remain spherical and their shape variations are modeled using the Rayleigh-Plesset equation. The volume-averaged, incompressible Navier–Stokes equation is solved for the liquid phase. A two dimensional model is create to simulate gate slot of bottom outlet of karoon3 dam located in Khuzestan region. A bottom outlet serves to regulate or release water impounded by a dam. The flow in the laboratory model of gate slot was studied for different flow rates. A physical model with a scale of 1:12 is built according to the Froude Law. The main purposes of the model tests are to measure pressure in the gate slote. Numerical simulation is done at flow rates of 75, 107 and 127 m3/s. The simulation results for flow field are compared with the experimental data and have good agreement. The Lagrangian method is used to investigate trajectories of bubbles with considering drag, pressure gradient, saffman lift and volume variation forces acting on bubble. Microbubbbles are released at the different locations in the gate slot. Effect flow rate on the bubble motion is studied.

II. Flow Simulation

A. Mean-flow model

For an incompressible fluid flow, the equation of continuity and balance of momentum for the mean motion are given as:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \upsilon \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} R_{ij}$$

Where $\bar{u}_i$ is the mean velocity, x_i is the position, t is the time, $\bar{p}$ is the mean pressure, ρ is the constant mass density, υ is the kinematic viscosity, and $R_{ij} = \overline{u'_i u'_j}$ is the Reynolds stress tensor. Here, $u'_i = u_i - \bar{u}_i$ is the i th fluid fluctuation velocity component. The RSTM provides for differential transport equations for evaluation of the turbulence stress components.

$$\frac{\partial}{\partial t} R_{ij} + \bar{u}_k \frac{\partial}{\partial x_k} R_{ij} = \frac{\partial}{\partial x_k}\left(\frac{\upsilon_t}{\sigma^k}\frac{\partial}{\partial x_k} R_{ij}\right) - \left[R_{jk}\frac{\partial \bar{u}_j}{\partial x_k} + R_{jk}\frac{\partial \bar{u}_i}{\partial x_k}\right] \tag{2}$$
$$- C_1 \frac{\varepsilon}{k}\left[R_{ij} - \frac{2}{3}\delta_{ij} k\right] - C_2\left[P_{ij} - \frac{2}{3}\delta_{ij} P\right] - \frac{2}{3}\delta_{ij}\varepsilon$$

Where the turbulence production terms are defined as:

$$P_{ij} = -R_{jk}\frac{\partial \bar{u}_j}{\partial x_k} - R_{jk}\frac{\partial \bar{u}_i}{\partial x_k}, P = \frac{1}{2}P_{ij} \tag{3}$$

With P being the fluctuation kinetic energy production. Here is v_t the turbulent (eddy) viscosity; and $\sigma^k = 1.0, C_1 = 1.8, C_2 = 0.6$ are empirical constants.

The transport equating for the turbulence dissipation rat, ε , is given as:

$$\frac{\partial \varepsilon}{\partial t} + \bar{u}_j \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\upsilon + \frac{\upsilon_t}{\sigma^\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] - C^{\varepsilon 1}\frac{\varepsilon}{k}R_{ij}\frac{\partial \bar{u}_i}{\partial x_j} - C^{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{4}$$

In Eq. (1), $k = \frac{1}{2}\overline{u'_i u'_i}$ is the fluctuation kinetic energy, and ε is the turbulence dissipation. The values of constants are [15]:

$$\sigma^\varepsilon = 1.3, C^{\varepsilon 1} = 1.44, C^{\varepsilon 2} = 1.92. \tag{5}$$

B. Fluctuating velocity

The dispersion of small particles is strongly affected by the instantaneous fluctuation fluid velocity. The turbulence fluctuations are random functions of space and time. In this study, the continuous filter white-noise (CFWN) model described by Thomson (1987) is used to generate instantaneous fluctuating fluid velocity [16]. Accordingly, the ith component of instantaneous fluid velocity satisfies the following stochastic equation

$$\frac{du_i}{dt} = -\frac{u_i - \bar{u}_i}{T_i} + \left(\frac{2\overline{u_i'^2}}{T_i}\right)^{\frac{1}{2}} \zeta_i(t) \tag{6}$$

Here, $\overline{u}'^2_i$ is the mean-square of the ith fluctuation velocity, and the summation convention on the underlined indices is suspended.

In equation (6), T_1 is the particle integral time. For small particles that move with the fluid, the particle integral time may be approximated by the fluid point Lagrangian integral time. The latter is related to fluctuating kinetic energy and dissipation rat, i.e.

$$T_L = C_L \frac{k}{\varepsilon} \tag{7}$$

With the constant $C_L \approx 0.3$ [17]. Therefore,

$$T_I \approx T_L \approx 0.3\frac{k}{\varepsilon} \tag{8}$$

In equation (6), $\zeta_i(t)$ is a Gaussian vector. In the numerical simulation, the amplitude of $\zeta_i(t)$ at every time step is given as

$$\zeta_i(t) = \frac{G_i}{\sqrt{\Delta t}} \tag{9}$$

Where G_i is a zero-mean, unit variance independent Gaussian random number and Δt is the time step used in the simulation. Each entire time sample is then shifted randomly between 0 to Δt to generate an appropriate white-noise time history.

C. Improved Spherical Bubble Dynamics Model

The behavior of spherical bubble in a pressure field is usually described with a relatively simple bubble dynamics model known as the Rayleigh-Plesset equation (Plesset 1948) [12]:

$$R\ddot{R} + \frac{3}{2}\dot{R}^2 = \frac{1}{\rho}\left[p_v + p_g - p - \frac{2\gamma}{R} - \frac{4\mu}{R}\dot{R}\right] \tag{10}$$

Where R is the time dependent bubble radius, ρ is the liquid density, p_v is the vapor pressure, p_g is the gas pressure inside the bubble, p is the ambient pressure local to the bubble, μ is the liquid viscosity, γ is the surface tension. If the gas is assumed to be perfect and to follow a polytrophic compression relation, then one has the following relationship between the gas pressure and the bubble radius:

$$p_g = p_{g0}\left(\frac{R_0}{R}\right)^{3k} \tag{11}$$

Where p_{g0} and R_0 are the initial gas pressure and bubble radius respectively and k is the polytropic gas constant. The internal process inside the bubble is assumed to be isentropic. Equation (10) does not consider the effect of slip velocity between the bubble and the carrying liquid. To account for this slip velocity, an additional pressure term $\rho\left(\vec{U} - \vec{U}_b\right)^2/4$ is added to the classical Rayleigh-Plesset equation [4].

$$R\ddot{R} + \frac{3}{2}\dot{R}^2 = \frac{1}{\rho}\left[p_v + p_g - p - \frac{2\gamma}{R} - \frac{4\mu}{R}\dot{R}\right] + \frac{\rho\left(\vec{U} - \vec{U}_b\right)^2}{4} \tag{12}$$

For initial conditions, it is usually appropriate to assume that the microbubble of radius Ro is in equilibrium at t = 0 in the fluid and dR/dt|t=0 = 0.

C. Bubble Motion Equation

Several prominent scientists such as Basset [18], Boussinesq [19] and Maxey [20] have derived the motion equation of a spherical particle subjected to the force of gravity in a fluid. Use of numerical simulation to studying particle dispersion and deposition in the turbulent flows was reported by Ahmadi *et al* [21] and Shams *et al* [22]. By considering the forces acting on the spherical bubble with radius R, the equation of motion is:

$$\begin{aligned}\rho_b V_b \frac{d\vec{U}_b}{dt} &= V_b(\rho_b - \rho)g\vec{j} + V_b\vec{\nabla}p + \frac{1}{2}\rho A_b C_D(\vec{U} - \vec{U}_b)\left|\vec{U} - \vec{U}_b\right| + \frac{1}{2}\rho V_b\left(\frac{d\vec{U}}{dt} - \frac{d\vec{U}_b}{dt}\right) + \\ &\frac{1}{2}\rho\left(\vec{U} - \vec{U}_b\right)\frac{dV_b}{dt} + 1.615\rho\upsilon^{\frac{1}{2}}d^2\left(\vec{U} - \vec{U}_b\right)\left|\frac{d\vec{U}}{dy}\right|^{\frac{1}{2}} Sgn\left(\frac{d\vec{U}}{dy}\right)\vec{j}\end{aligned} \tag{13}$$

Where terms with the subscript b are related to the bubble and those without a subscript are related to carrying fluid. V_b and A_b are the bubble volume and projected area, which are equal to $4/3\pi R^2$ and πR^2 respectively. The bubble drag coefficient C_D in equation (13) can be determined by using the empirical equation of Hagerman and Morton (1953) [9]:

$$\begin{aligned} C_D &= \frac{24}{\mathrm{Re}}\left(1 + 0.197\,\mathrm{Re}^{0.63} + 2.6\times10^{-4}\,\mathrm{Re}^{1.38}\right) \\ \mathrm{Re} &= \frac{2R\left|U - U_b\right|}{\upsilon}\end{aligned} \tag{19}$$

III. RESULTS

A 2-dimensional model is used to simulate numerically. The finite volume method with unstructured meshes is used to solve the two dimensional incompressible Navier–Stokes equations. The velocity and pressure coupling used the SIMPLEC algorithm. RSTM model is used to model turbulence. Maximum relative error is considered smaller than 0.0005. A grid with about 60,000 cells is generated for analyzing the flow in the gate slot. The grid independency is checked in order to make sure that the grid numbers is sufficient and has enough accuracy. Fig.4 shows the geometry and the details of the computational grid for the gate slot model. The growth and radius change of bubbles are modeled by the Improved Rayleigh-Plesset equation. The velocity fluctuations have a significant effect on the bubble size and its trajectory. Some of the previous researches ignored this effect. Here, the CFWN model is use to calculate velocity components. The bubble motions are simulated by the Lagrangian trajectory analysis procedure. The radius of bubbles is small, therefore the effects of interaction between bubbles with carrier flow are neglected and one-way coupling is used. Forces acting on the bubble include drag, buoyancy, pressure gradient, Saffman lift and volume variation. Bubble with diameter about 50-micron release in the slot gate.

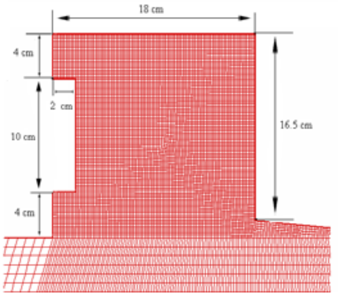


Fig.1 Geometry and generated grid for gate slot

Because pressure have strong effect on bubble growth and motion, confident of its precision cause that analysis is accurate. Fig.2 shows Comparison between the numerical simulation and the experimental data for the mean pressure in the gate slot. The numerical results have admissible agreement with the experimental data. Major reason error between CFD and experimental results is two-dimensional simulation, while nature of flow in the gate slot is three-dimensional.

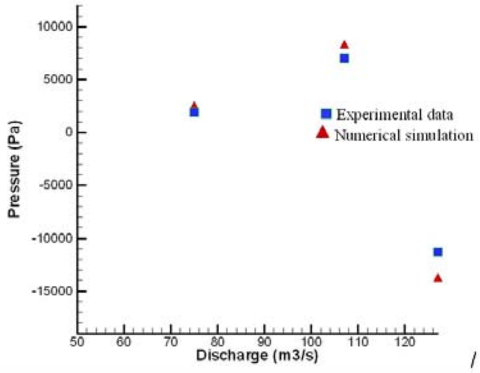


Fig.2 Comparison of numerical and experimental results for mean pressure at the different flow rate in the gate slot

The numerical simulation shows that two vortex in the gate slot is created (Fig.3). Then, pressure in these locations is strongly decreased. The counters of pressure at the different flow rate are indicated in Fig.4. It shows that with increasing the flow rate, pressure drop increase and vortex grow. With growing vortex, low speed regions in the gate slot increase and then drag force acting on bubble decrease in these regions. With decreasing drag force, forces of pressure gradient and volume variation are strongly effaced on the bubble trajectory. The pressure gradient transfers the bubble from high-pressure regions to low pressure (Fig.6). This affect is strong near vortex and tendency of the bubble to the vortex center (low-pressure region) increase. The bubble diameter in these regions is big because pressure is low. Inside vortex, this force also has little influence on bubble motion and force of volume variation is dominant. Influence of this force cause random motion of bubble in this region.

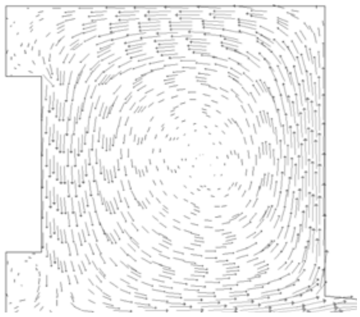
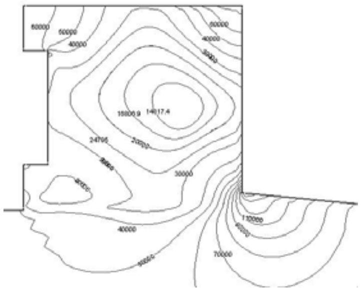
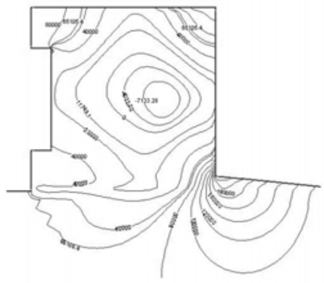


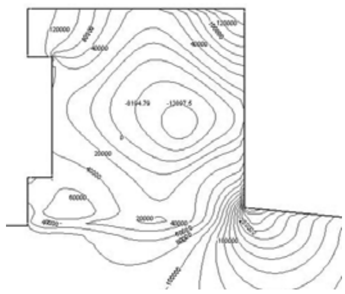
Fig.3 Creation the vortexes in the gate slot



a)



b)



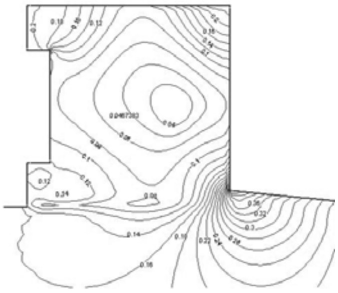
c)

Fig.4 Contours of pressure in the gate slot at flow rate a) 75 b) 107 c) 125 m^3/s

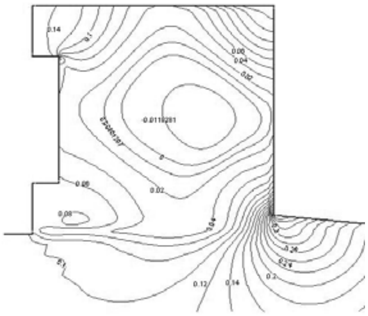
For low flow rate, the bubble follows an iterative rotating motion because drag force acting on the bubble is stronger than pressure gradient force. Therefore, bubble follows from flow pattern.

For flow rate 125 m3/s, if the bubble release at away regions from vortex, drag force is dominant. Behavior of the bubble in these regions is same flow pattern (Fig.7).

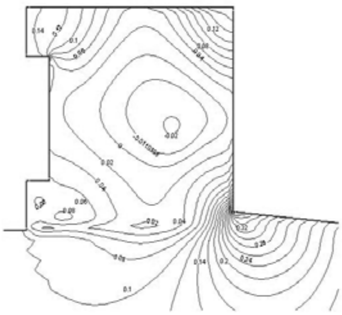
It shows that the bubble trajectory in away regions of vortex is independent of flow rate. Because force of pressure gradient is weak, it cannot transfer the bubble to vortex. Therefore bubble follow flow pattern (Fig. 7, 8 and 9).



a)



b)



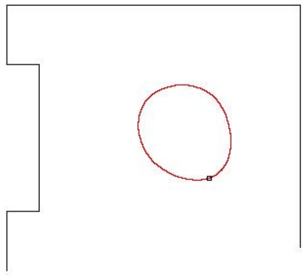
c)

Fig.5 Contours of cavitation index in the gate slot at flow rate a) 75 b) 107 c) 125 m^3/s

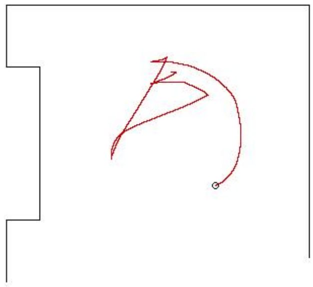
Microbubbles are initial nuclei for formation hydrodynamic cavitation[3]. Possibility of existence these nuclei (microbubbles) are in whole of fluid and its boundaries. These microbubbles convert to cavitation bubble till they attain to low-pressure regions. In these regions, abrupt growth is created and bubbles radius enlarges suddenly. Then, cavitation can predicate with scrutiny of microbubbles trajectory.

As shown in the results, microbubbles, which release near the gate boundaries, move close to boundaries. They enter to low-pressure regions, therefore cannot growth and convert to cavitation bubbles.

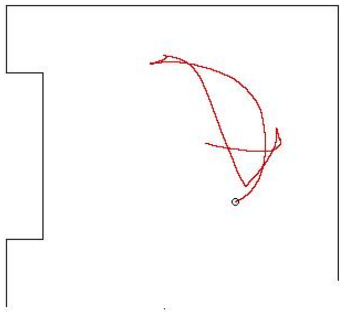
It is obvious that microbubbles, which are near vortex inside gate, move inside gate because of influence of pressure gradient force. These microbubbles grow and convert to cavitation bubbles. By increasing flow rate, vortex enlarges and then, many microbubbles can change to cavitation bubble. Then, it is shown that probability of occurrence of cavitation increase by increasing flow rate. Contours of cavitation index that is criterion for cavitation indicate rectitude of this claim (Fig.5). From this method can used to predicate probability of occurrence of cavitation.



a)



b)



c)

Fig.6 Bubble trajectory in the gate slot at flow rate a) 75 b) 107 c) 125 m^3/s

IV. CONCLUSION

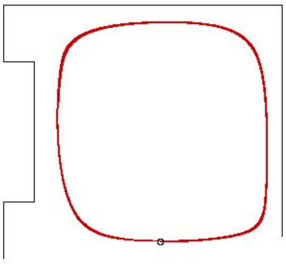
In this paper, the bubble trajectory in a gate slot is studied. The Eulerian – Lagrangian approach is used to analyze two-phase flow. A two-dimensional is used to simulate flow field. The RSTM model models the Reynolds stresses. The components of instantaneous fluctuation velocity are simulated using the Continuous Filter White-Noise model. The simulation results with experimental model data are compared. The numerical results have good agreement with the experimental data.

Trajectories of bubbles are simulated using the Lagrangian method with considering drag, pressure gradient, saffman lift and volume variation forces. The growth and radius change of bubbles are modeled by the Rayleigh-Plesset equation. Based on the results presented, the following conclusions are drawn:

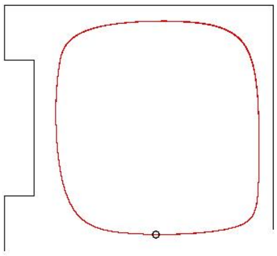
In away regions of vortex, drag force have most effect whereas force of pressure gradient is dominant in near vortex. However, inside vortex, this force also has little influence on bubble motion and force of volume variation is dominant. Saftman force has negligible effect on bubble trajectory.

Microbubbles near vortex, affecting force pressure gradient, have tendency to vortex center where pressure is low. These microbubbles grow. Therefore, probability of conversion these bubbles to cavitation bubble increase. Microbubbles, which are at a far distance from vortex, this probability is lower because they do not enter low-pressure regions.

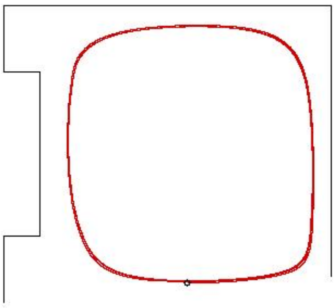
With increasing flow rate, vortex enlarges. Therefore, probability of conversion microbubbles to cavitation bubble and cavitation risk increase.



a)



b)



c)

Fig.7. Bubble trajectory in the gate slot at flow rate a) 75 b) 107 c) 125 m^3/s

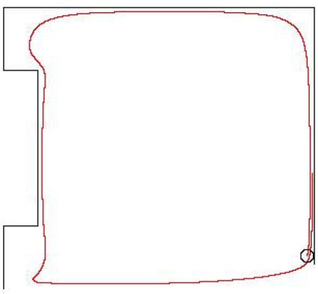


Fig.8 Bubbles trajectory near boundary in the gate slot is same at the different flow rate

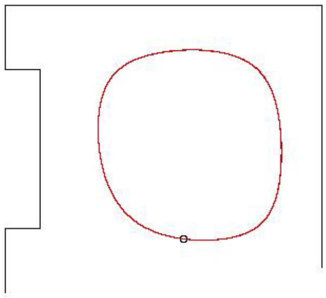


Fig.9 Bubbles trajectory in center of the gate slot is same at the different flow rate

V. References

[1] W.H. Kohler, 1969,"Selection of outlet works, gates and valves",ASCE Annual and Environmental Meeting. Chicago.Illinois.USBR.

[2] A. Raoufi, M. Shams, R. Ebrahimi, G. Ahmadi, 2006,"The Eulerian–Lagrangian analysis of the bubble motion in a gate slot", 14th International Mech. Engng. Conf., Esfehan, Iran.

[3] B. Lertnuwat, K. Sugiyama, 2001, "Modeling of Thermal Behavior Inside a Bubble". CAV2001.

[4] A. Vladislav, 1999, "Differential Criterion of a Bubble Collapse in Viscous Liquids", Physical Review E, Volume 60, Number 1, JULY 1999.

[5] M . S. Plesset, 1948, "Dynamics of Cavitation Bubbles", Journal of Applied Mechanics, 16, 228-231.

[6] C. E. Brennen, 1995, "Cavitation and Bubble Dynamics", Oxford University.

[7] R. S. Meyer, M. L. Billet, and J. W. Holl, 1992, ''Freestream Nuclei and Traveling Bubble Cavitation", J. Fluids Eng., 114, 672-779.

[8] G. L. Chahine, 1997, "Numerical Simulation of Cavitation Dynamics", Dynaflow, inc. 7210 Pindell School Road. Fulton, Maryland 20759, USA.

[9] C. T. Hsiao, and L. L. Pauley, 1999, ''Study of Tip Vortex Cavitation Inception using Navier-Stokes Computation and Bubble Dynamics Model,'' ASME J. Fluids Eng., 121(1), pp. 198–204.

[10] J. E. Johnson, T. Hsieh, 1966, ''The Influence of the Trajectories of the Gas Nuclei on Cavitation Inception,'' Sixth Naval Hydrodynamics Symposium, pp. 163–182.

[11] K. J. Farrell, 2001, "Eulerian/Lagrangian analysis for the prediction of cavitation inception", CAV2001.

[12] C. F. Delale, 2003, "Steady – State Cavitating Nozzle Flows With Nucleation". Fifth International Symposium on Cavitation (Cav2003).

[13] C. T. Hsiao, G. Chahine, 2004, "Prediction of Vortex Cavitation Inception Using Coupled Spherical and Non – Spherical Models and UnRANS Computations", Symposium on Naval Hydrodynamics Fukooka, JAPAN, 8-13 July 2004.

[14] X. Zhang, G. Ahmadi, 2005, "Eulerian – Lagraungian Simulations of Liquid – Gas – Solid Flows in Three – Phase Slurry Reactors", Chemical Engineering Science 60 (2005) 5091 – 5106.

[15] B.E. Launder, G.J. Reece, W. Rodi, 1975," Progress in the development of a Reynolds stress turbulent closure", J Fluid Mech;68:537–8.

[16] D. J. Thomson, 1987, "Criteria for the Selection of Stochastic Models of Particle Trajectories in Turbulent Flows", J. Fluid Mech. 180, 529 – 556.

[17] B. J. Daly, F. H. Harlow, 1970, "Transport Equations in Turbulence", Phys. Fluids, No.13,2634-2649.

[18] A. B. Basset, 1888, "A Treatise on Hydrodynamics", Vol2. ch. 5, Deighton, Bell and Co., Cambridge, England.

[19] J. Boussinesq, 1903, "Theorie Analytique de la Chaleur", Vol2, P. 224, Gauthier – Villars, Paris.

[20] M. R. Maxey, J. J. Riley, 1983, "Equation of Motion for a Small Rigid Sphere in a Nonuniform Flow", Phys. Fluids, Vol. 26, No. 4, pp. 883 – 889.

[21] G. Ahmadi, 1996, "Introduction to Mechanics of Aerosol With a Review of Recent Momputational Methods", Clarkson University,Potsdam.NY.

[22] M. Shams, G. Ahmadi, And H. Rahimzadeh., 2000, "Nano Particle Deposition in a Turbulent Duct", J.Chem.Engn.Sci, 2000.