A simulation of air-lift artificial upwelling in vertical pipe

Qingsong Zhang[1,2], Li Xu[1*] ,Wen Yu[1]

[1]Engineering School, Zhejiang University City College,Hangzhou, Zhejiang, 310015, China

[2]School of Mechanical Engineering, Zhejiang University,Hangzhou, Zhejiang, 310027,China

e-mail: zhangqs@zju.edu.cn,e-mail: xuli@zucc.edu.cn, ,e-mail: yuw@zucc.edu.cn

Keywords:CFD;Two-phase;Air-lift; Artificial upwelling flow; Vertical pipe

Abstract. The development of Computational Fluid Dynamics (CFD) technology make people found the good point of compromise on the conflict of experiment and analysis,and air-lift artificial upwelling for deep ocean water efficiency is simulated with a simplified CFD model of two-phase flow in an upward pipe. The model is simulated in the two-dimensional space and based on the momentum transfer between the two phases. The influence of the pipe diameter and length on the lifting efficiency has been investigated numerically. It was shown that the increase of the pipe diameter leads to the increase of the lifting efficiency. However, with the increase of pipe length, the lifting efficiency of liquid was reduced slightly. The simulating results indicated that the parameter of pipe diameter is more sensitive than the pipe length.In addition, increasing the pipe length also led to the reduction of lifting efficiency due to the wall friction in stable full-blown flow field.

Introduction

The deep ocean water contains has feature of richness in nutrients and low temperature [1], which benefits the bottom end of a marine food chain, phytoplankton[2], so pumping nutrient-rich deep ocean water to the surface to feed them is a fundamental way toenhance fisheries productivity. This is an optimistic method for sustainable growth of global population [3].Generally, the natural upwelling sea area is a fishing ground of high production, but bringing a huge amount of deep seawater to the sea surface by an artificial method can also generate a new fishing ground, and some scientists have proved this assumption [4]. The ambitious project named Ocean Nutrient Enhancer (ONE) pumped deep seawaterdirectly and designed by Marino-Forum 21 (MF21) of Japan. The first ONE device called TAKUMI was equipped in2003[5].Liang et al in 2005[6]did a research to generate upwelling flow by injecting gas underwater, named air-lift method. The principle of this method is that the air is injected into the vertical pipe and the rising bubbles carry the liquid along due to the interfacial friction.As air fills part of the pipe, the pressure inside becomes less than the hydrostatic; therefore, additional upward force acting on liquid is produced, so deeper water will be sucked inside the pipe due to the drag exercised by the upper water. Differently from other upwelling concepts.

Two-phase flow that consists of liquid and gas is controlled in the upwelling pipe. It is necessary to obtain a better understanding of the material transport process and to establish an appropriate model to control thefluid and optimize the apparatus. Liang andPeng[7]improved the energy flux balance equations and empirical correlations for energy losses take into account the overall energy balance. They highlighted that the power demand associated with the density difference head and the sea surface rise are crucial parameters in the air-lift upwelling system.Although bubbles do not exhibit all of the attributes of a liquid, the two-fluid model, which is based on the Euler–Euler approach, is able to demonstrate the detailed distribution and profiles of the key parameters in two-phase flow system[8]. The influences of the pipe diameter and other parameters were discussed in the optimization of the system. Obviously, two-fluid models are time consuming and quite dependent on the computational resources. Hence, it is difficult to actually model the whole system.

In the present paper, a simplified CFD model is proposed to simulate the whole air-lift artificial upwelling system. The proposed model is expected to be efficient and accurate and is based on a clear physical background. The model has to be run repeatedly with a variety of parameters for the optimization of the parameters of the system. Numerical cases were simulated using different pipe diameter and pipe length.

Governing equation

The model describes the movement of the phasesaccording to the Euler approach: water is the continuous phase and air isdiscrete phase. The air bubbles are assumed to be round spheres, their diameter may change depending on the time and location. In order to simplifythe model, coalescence or breakagein bubbles is not considered, so the bubble diameter only associates with the change of pressure. It was also assumed that there is no phase change, no mixing, and no mass transfer between the two phases under any circumstances. Assuming that the flow in the tube is adiabatic, energy equations were not considered in the present study. The liquid phase was assumed to be an incompressible fluid.

It has a very complicated relationship between two-phase flow compared with one phase flow. In the process of flow motion, a place in flow field may be filled by only liquid phase, or only gas phase and or the two phase interfaces, so it is the key point to solve the data coupling and parameters of switching in two phases [9]. At present, multiphase flow model uses the Euler method to depict and handle the numerical relationship in various phases, i.e. Euler-Euler model. In the description of Euler-Euler model, different phase is regarded as interlock continuum, and in the same scope space at the same time, the sum of each phase volume fraction equals 1. In particular, every phase flow connects with one conservation equation [10].

In the two-phase region, the flow characteristics to be required contain of velocity, volumetric fraction, pressure and density. The mass conservation equations which govern each phase in the gas-liquid two-phase flow region are given by

$$\frac{\partial(\rho_K\varepsilon_K)}{\partial t}+\frac{\partial(\rho_K\varepsilon_K u_K)}{\partial x}=0,\ K=G,L \qquad (1)$$

(1)

here, u is the velocity, ε the volumetric fraction, ρ the density and the subscripts G, L denote the gas-, liquid-, respectively. The momentum conservation equations which govern each phase in the gas-liquid two-phase flow region are given by

$$\frac{\partial(\rho_K\varepsilon_K u_K)}{\partial t}+\frac{\partial(\rho_K\varepsilon_K u^2{}_K)}{\partial x}=-\left(F_{iK}+\varepsilon_K\frac{\partial P}{\partial x}\right) \qquad (2)$$

in which P is the pressure t is the time, F_{iK} appearing in the momentum conservation equationsdenotes the drag force due to the interaction betweenphases. An equation for volumetric fraction is required to close the solving equations.

$$\varepsilon_G+\varepsilon_L=1 \qquad (3)$$

taking the incompressibility condition of water into consideration, the liquid-phase's mass conservation equation in Eq(2) is simplified to

$$\frac{\partial(\varepsilon_L u_L)}{\partial x}=0 \qquad (4)$$

It should be noted that mos tof the classic formulas for the drag coefficient are based on the assumption of a singl erigid spherical bubble in fresh water. The effect sof the inter actions between bubbles and different media, such as sea water and contaminated water, need to be addresse dinfurther studies.

Numerical formulation and results

Continuous differential numerical model based on fundamental equations (the conservation equations of mass, momentum and energy) must be discretized to discrete model and taking place the discrete solution on the grids to continuous solutionbefore using CFD software. In the paper, the finite difference method (FDM) is employed to discretize the flow. And around the entrance of air injection, because of the large change in velocity and turbulence, the grids must be small interval size. The grids are considered to be refined in the middle position of the upper pipe due to the formulation of gas phase and the interaction with the liquid phase. And below the air injection position, it is the single liquid phase, so the interval size of grids could be larger suitable on the purpose of reducing the computational time. Taking the pipe wall effect into account, it is certain to set the wall grids.

In this paper, the diameter of the pipe (1.5m, 2m, 2.5m) and the length (50m, 55m, 60m)are calculated to get the efficiency of the lifting water by controlling variable method. In the iteration, the models are designed for transient time solver and VOF Euler method and standard k-e turbulence. The solution method is for PISO scheme and pressure spatial discretization is for body force weighted. The compressible gas is regarded as the primary phase and the continuous liquid is the second phase which is patched into the flow area. It is an objective-oriented CFD simulation and the formulation data can be the guide for the further sea experiments.In the process of simulation of the tube diameter and length, the time interval is controlled in 0.0001 second to increase the convergence of calculation. So it can be estimated that a group of bubbles which arrives on the outlet at the top of pipe from 35 meters depth need approximately 10s relatively,however it needs more much time to achieve steady flow state. The results are illustrated in Fig. 1.

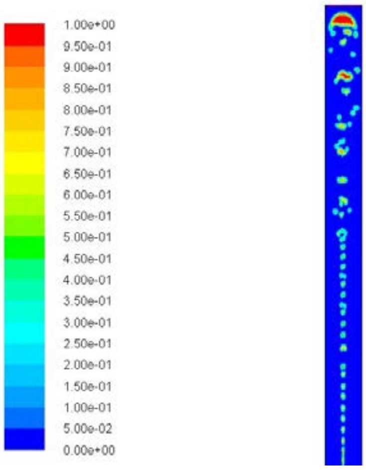
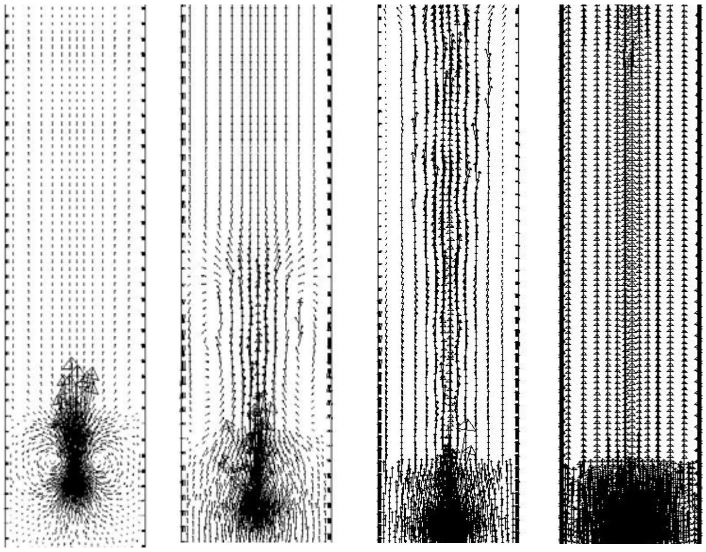


Figure 1. The distributed air-bubble state in the vertical pipe

As shown in Fig. 1, the first air bubble arrived the outlet about 10s larger than estimated value, the liquid resistance and friction result in the solution. With time went by, the gas phase volume fraction excluded, and as the energy input, the water phase along with the upwelling air moving up.When the time was 3.9 seconds, the obvious turbulence and backflow effect had generated. As is shown in fig 2.This effect in practice must be avoided.



T=0.75s T=3.9s Y=8.0s Y=15.0s

Figure 2. The velocity vector in various time and the obvious turbulence and backflow effect

It can be seen that the state two-phase flow appeared about in 15s, and it was larger than estimated value, the liquid resistance and friction result in the solution. The outlet velocity is shown in Fig. 3.

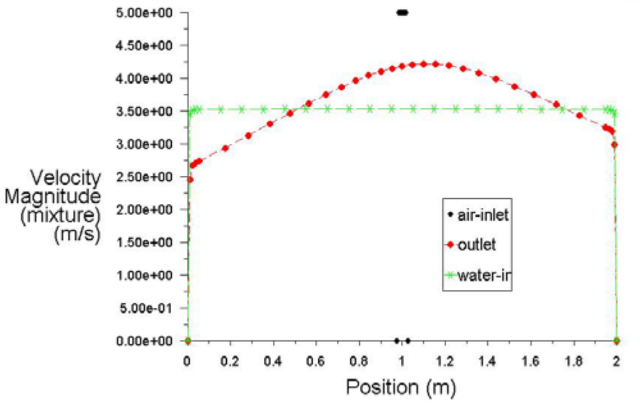


Figure 3.Water velocity in different position

As depicted in section 2.2, the pipe is equipped in 10m underwater, so it can lift the sea water from 60m to 10m and the 0.0098cms input volumetric air flow rate in 2m diameter and 50m length of the tube can get 6.28cms water volumetric flow rate.

Conclusion

The influence of the pipe diameter and length on the lifting efficiency has been investigated numerically. It was shown that the increase of the pipe diameter leads to the increase of the lifting efficiency. However, with the increase of pipe length, the lifting efficiency of liquid was reduced slightly. The simulating results indicated that the parameter of pipe diameter is more sensitive than the pipe length. It was found that there was a transient fluctuation in low length to diameterratio pipe due to the destabilizing effect of the wall friction on the flow; in addition, increasing the pipe length also led to the reduction of lifting efficiency due to the wall friction in stable full-blown flow field. One can obtain preliminary performance data by applying a simplified model along the axial coordinate and these preliminary data can guide the design and establish ranges for which for exploration should be followed up.

Acknowledgement

This work was financially supported by the Zhejiang Province Natural Science Foundation (LY13E090006).

References

[1] H., Guldberg, and J. F., Bruno.The Impact of Climate Change on the world's Marine Ecosystems. Science, 2010, 328(5985):1523-1528.

[2] A. V., Geen, R. K., Takesue, J. Goddard, T., Takahashi. Carbon and nutrient dynamics during coastal upwelling off Cape Blanco, Oregon[J]. Deep-Sea Research II, 2000, 47, 975-1002.

[3] J. P., Jeffrey, H., Evan, R. K., Donald. The transition zone chlorophyll front, a dynamic global feature defining migration and forage habitat for marine resources[J]. Progress in Oceanography, 2001, 49(3): 469-483.

[4] Cheung,S.C.P.,Yeoh,G.H.,Tu,J.Y.,2007. On the modelling of population balance in isothermal vertical bubbly flows—average bubble number density approach. Chemical Engineering and Processing 46(8), 742–756.

[5] O., Kazuyuki, J. M., Alan, Real sea experiment of ocean nutrient enhancer "TAKUMI" upwelling deep ocean water [J]. Oceans Proceedings, 2003:881-885.

[6] Chen,J.W.,Huang,H.C.,Ge,H.,Meng,Q.C.,2011. The Air-lift Artificial Upwelling Field Tial Report of Zhejiang University. (17).

[7] N.K., Liang. Concept design of artificial upwelling induced by natural forces. Proceedings of Oceans’91 Conference, Hawaii, 1991,1: 391–393.

[8] Davidson,J.,Schuler,B.,1960. Bubble formation at an orifice in a viscous liquid. Chemical Engineering Research and Design 38(a), 144–154.

[9] Hsieh,C.T.,Huang,P.A.,Li,D.J.,Chu,L.L.,Wu,C.L.,Liang,N.K.,1978. Artificial upwelling induced by ocean currents—theory and experiment. Ocean Engineering 5(2), 83–94.

[10] Kulkarni,A.A.,Joshi,J.B.,2005.Bubble formation and bubble rise velocity in gas-liquid systems: areview. Industrial and Engineering Chemistry Research 44(16), 5873–5931.