Numerical Study of Two-Phase Flow in Micro-/Nanobubble Generating Pump

Hilman Syaeful Alam¹, Bahrudin¹ and Anto Tri Sugiarto¹

¹Technical Implementation Unit for Instrumentation Development, Indonesian Institute of Sciences, Jl. SangkuriangKomplek LIPI Gedung 30 Bandung, 40135, Indonesia.
E-mail: hilm003@lipi.go.id, hilman.syaeful.alam@lipi.go.id,bahr003@lipi.go.id,anto001@lipi.go.id

Abstract. Gas-liquid mixing pump is one of the multiphase flow problem in industrial applications as a micro-/nanobubble generator. However, very few report that studied the two-phase flow for application of microbubble generation because of the analysis complexity. In this paper, a steady state numerical simulation of gas-liquid two-phase flow in the gas-liquid mixing pump was employed to predict a performance and characteristic of fluid flow. Based on simulation results, it is demonstrated that the pump can work in self suction, and generates a vortex flow pattern at every stage of the impeller as regenerative. Performance pump of the numerical simulation is slightly higher than the design specifications. Because of mechanical and volume losses was neglected. However, the evaluation method and simulation results from this work can be used as a reference for the design and improvement of the gas-liquid mixing pump.

1. Introduction

Multiphase flow is the most widely applied flow conditions used in various industrial applications. However, a reliable analysis tool for analyzing the problems of multiphase flow is still limited because of the analysis complexity. With the rapid development in the field of computer technology, multiphase flow problems can be solved by numerical analysis based on the computational fluid dynamics (CFD), therefore, it will be significantly reduced the time and cost in the design and production compared to experimental based approach [1]. CFD simulation has been used in previous studies to analyze the problems of multiphase flow, such as: multiphase pump-mixer [2], self-priming pump [3], biodiesel reactor [4], airlift reactor [5], mixed flow pump [6], gas-liquid multiphase scroll pump [7], pump-mix mixer [8] and others.

Gas-liquid mixing pump is one of multiphase flow applications in industries that used for mixing gas and liquid directly as an ultrafine bubble generator. Gas mixed with the liquid in the pump is formed into an ultrafine bubbles with a size <50 um (microbubbles) or <200 nm (nanobubbles), so the gas will easily dissolve in the liquid. In general, this type of generator using regenerative pump that has very simple and inexpensive construction [9]. The fluid enters a set of radial impeller teeth which guides the fluid from the inlet to the outlet, both separated by about 3000 annulus chamber producing a torroidal motion also known as regenerative flow [10]. Therefore, a regenerative pump can achieve lower mass fluxes but higher-pressure differences than other pumps at the same circumferential velocity [11]. Vortex flow pattern in regenerative pump impact on regenerative effects or impulses.
that recur the fluid particles at every level of vanes and volute pump channels. Gas fine bubbles would be obtained at the outlet of the impeller due to the super-saturated condition between gas and fluid. Design of microbubble generator still remains a challenge in generating an efficient design. Very few reports that studied the pattern of two-phase flow in a gas-liquid mixing pump for applications of microbubble generation. In recent years, much more research is carried out experimentally that require a huge costs and a long time. In this paper, the gas-liquid mixing pump was analyzed using CFD approach based on gas-liquid two-phase flow to predict the behavior and pattern of a fluid flow as reference in the design process.

2. Materials and Methods
The main part of gas-liquid mixing pump consists of impeller and housing pump. There are two suction or inlet fluids of mixing pump, i.e. gas and liquid which is mixed by the impeller component. The design parameter of gas liquid mixing pump was mass flow of 2.2 m$^3$/h, head of 20 m, rotation speed of 3000 r/min, and power of 0.55 kW.

2.1. Computational domains and meshing
According to Huang et al. [3], the conservation equations for each phase at a gas-liquid mixing pump are derived to obtain a set of equations with regard to momentum, continuity and energy. For CFD simulation in a rotary pump, k-$\varepsilon$ turbulence model and Eulerian multiphase approach had a satisfactory agreement between the experimental result and simulation. Therefore, their model were also used in this case and chosen to close the Reynolds average Navier-Stokes (RANS) equations.

3D model of gas-liquid mixing pump was built using ANSYS Design Modeler was imported into the ANSYS CFX software to generate the computational domain in CFD simulation, as shown in Figure 1(a). The 3-D computational domains of gas liquid mixing pump consist of solid part i.e. impeller and fluid part i.e. volute pump. The volute pump was connected with the liquid and gas suction inlet and pump outlet. The impeller was located in volute pump where the rotation was coupled by the electric motor shaft. The solid and fluid parts were meshed using tetrahedral elements to adjust to the complex geometry of each part. The finer the size of the elements then the numerical accuracy will increase but will also increase the computational time of the simulation. The mesh generation of solid part or impeller produces 50,736 elements and 10,890 nodes as shown in Figure 1(b), while fluid part produces 971,284 cells and 190,944 nodes as shown in Figure 1(c). Finally there were totally 1,022,020 cells and 201,834 nodes. According to Huang et al [3], when cell number achieved 1 million, the variation of pump head under steady simulation could be within 1%.

![Figure 1](image.png)

Figure 1. (a) 3-D computational domains, mesh generation for (a) solid domain and (c) fluid domain.
2.2. Boundary conditions
The next stage of CFD simulation was to set the boundary condition. Water and air at 25 °C were set as the liquid-phase and the gas-phase respectively. The design pump head was used to set the pressure at the inlet and the outlet of gas liquid mixing pump. If the inlet water pressure is set at 1.0 atm, the pressure at the pump outlet is set around 3.0 atm for the pump head of 20 m. Gas liquid mixing pump is designed to be self-suction gas, so that the air inlet pressure is set at 1.0 atm. According to the governing equations in 2.1, the two-phase flow and the standard k-ε turbulence models were selected in the governing equations and the steady state simulation was selected in the analysis type. The impeller part was chosen as a continuous solid with aluminum material and rotates on the X coordinate with the rotation of 3000 rpm. The second order upwind scheme was set on the discrete scheme of two phase flow model, the turbulence kinetic energy and the turbulent dissipation rate.

3. Results and Discussion
The distribution of gas liquid two-phase flow on the gas liquid mixing pump as the water volume fraction can be seen in Figure 2(a). The void fraction, α = 1 means pure water and α = 0 means pure liquid. There is an interesting phenomenon in the analysis results of the figure which are the following: 1. The gas in the gas suction pipe seems inhaled into a liquid storage chamber and impeller indicating that the pump is worked by self-suction mechanisms. 2. The gas is concentrated along the impeller blade and almost uniform from a suction blade to the blade outlet with a concentration above of 75%, it shows if the impeller works well in mixing the gas with liquid. 3. Distribution of gas around the outlet pump looks almost evenly with the liquid gas concentration average was 70%.

Figure 2(b) shows the streamline of the liquid superficial velocity where in the impeller area, it seems that the fluid velocity is lower than the inlet and outlet areas, because the kinetics energy is converted to the pressure energy. Vortex flow pattern is formed in the impeller area which give regenerative effects or impulses that repeatedly on the fluid particles at every level of the vanes and volute pump channels. The magnitude of the fluid velocity is below 18 m/s. Figure 2(c) shows the pressure distribution in the gas-liquid mixing pump on vertical planes. From these images, the pressure at the pump is gradually increased from the blade suction to a peak at the pump outlet. This suggests that the kinetic energy of the blade rotation is gradually converted into pressure energy at each stage of the impeller blade. With a vortex flow pattern and pressure in each stage of the blade, then the fine gas bubbles will be formed at the outlet of the impeller due to the super-saturated condition between gas and fluid. The pressure in the pump outlet was an average of 0.323 MPa and reach a maximum of 0.395 Mpa. It is higher than the design specifications, because CFD simulations did not take into account the factor of both the mechanical and the volume losses caused by bearings and seals.

Figure 2. (a) Gas-liquid two phase-countours on vertical plane (b) Streamline of the liquid superficial velocity, (c) pressure distribution in gas-liquid mixing pump on vertical planes.
4. Conclusion

CFD simulation is able to display the characteristics of gas-liquid two-phase flow on micro-/nanobubble pump. Based on the analysis, the pump can self-suction work, where the gas is successfully sucked into the impeller. The gas is concentrated along the impeller blade uniformly from the suction to the impeller outlet. From the results of streamline analysis, vortex flow pattern proved to be formed in each stage of the impeller that may impact on regenerative effects or impulses thus increasing the pressure in regenerative required to produce ultrafine bubbles at the impeller outlet. The fluid pressure of the numerical simulation results are higher than the design specifications, because both the mechanical and the volume losses caused by bearings and seals was neglected.

Acknowledgement

This work was financed by Kegiatan Unggulan, Indonesian Institute of Sciences (LIPI). The computational facilities were extended by the Research Center of Electrical Power and Mechatronics, Indonesian Institute of Sciences, which is thankfully acknowledged.

References

[1] C. Li, G. H. Yeoh, S. C. P. Cheung and J. Tu, “Modelling Horizontal Gas-Liquid Flow using Averaged Bubble Number Density Approach”, Journal of Computational Multiphase Flows, 2(2), pp. 89-99, 2010.

[2] M. V Tabib, G. Lane, W. Yang, and M. P. Schwarz, “CFD study of single phase and multiphase (liquid–liquid) pump-mixer: Analyzing design parameters, flow structures and turbulence,” Chem. Eng. Sci., 80, pp. 55–69, 2012.

[3] S. Huang, X. Su, J. Guo, and L. Yue, “Unsteady numerical simulation for gas-liquid two-phase flow in self-priming process of centrifugal pump,” Energy Convers. Manag., 85, pp. 694–700, 2014.

[4] K. De Boer and P. A. Bahri, “Investigation of Liquid-Liquid Two Phase Flow In Biodiesel,” 7th Int. Conf. on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia, pp. 1–6, 2009.

[5] J. Karez, M. Musia, M. Bitenc, and M. Doma, “CFD Modelling of the Fluid Flow Characteristics in an External-Loop Air-Lift Reactor,” Chemical Engineering Transactions, 32, pp. 1435–1440, 2013.

[6] S. Srivastava, A. Kumar, and K. Kumar, “Design analysis of Mixed Flow Pump Impeller Blades Using ANSYS and Prediction of its Parameters using Artificial Neural Network,” Procedia Eng., 97, pp. 2022–2031, 2014.

[7] J. Wang, H. Zha, J. M. McDonough, and D. Zhang, “Analysis and numerical simulation of a novel gas-liquid multiphase scroll pump,” Int. J. Heat Mass Transf., 91, pp. 27–36, 2015.

[8] Y. Zou, S. Ye, Y. Wang, and W. Fei, “Journal of the Taiwan Institute of Chemical Engineers CFD simulation and PIV measurement of liquid – liquid two-phase flow in pump-mix mixer,” J. Taiwan Inst. Chem. Eng., 60, pp. 15–25, 2016.

[9] H. S. Alam & Bahrudin, “Integrity Evaluation of Micro- / Nanobubble Pump Impeller using Fluid-Structure Coupled Simulation,” International Journal of Engineering and Technology (IJET), 8, no. 6, pp. 2594–2600, 2017.

[10] K. V. Karanth and N. Y. Sharma, “CFD Analysis of a Regenerative Pump for Performance Enhancement,” 3rd World Conf. on Applied Sciences, Engineering & Technology, Kathmandu, Nepal, pp. 3–8, 2014.

[11] T. A. Meakhail, “Numerical Study Of Unsteady Flow Characteristics In Regenerative Pump,” Journal of Engineering Sciences, 35, no. 4, pp. 933–943, 2007.