Removal of Waste Volatile Solvents Using Hot Bubbles

T Srisurat\textsuperscript{1*}, R Kokoo\textsuperscript{2}, and N Swadchaipong\textsuperscript{3}

\textsuperscript{1} Faculty of Science, Energy and Environment, King Mongkut’s University of Technology North Bangkok, Rayong Campus, Rayong, Thailand
\textsuperscript{2} Department of Chemical Engineering, King Mongkut’s University of Technology North Bangkok, Bangkok, Thailand
\textsuperscript{3} The Sirindhorn International Thai-German Graduate School of Engineering, King Mongkut’s University of Technology North Bangkok, Bangkok, Thailand

*Corresponding author e-mail: thanarak.s@sciie.kmutnb.ac.th

Abstract. Water pollution is a serious problem affecting the quality of life and environment. Wastewater treatment is necessary for solving this problem. A bubble column or bubble gas reactor is one of the most widely used in the chemical industries. Volatile organic compounds can be absorbed via gas bubbles in the column. Furthermore, the advantage of the bubble column is highly mass transfer and heat transfer rates. The performance of bubble column depends on bubble size, flow regime, gas hold-up and operating conditions i.e., pressure and temperature. This research focused on hot air bubbles to remove volatile solvents in the solution. CFD simulation was used to study the temperature distribution of gas bubbles within the bubble column. The experiments were carried out to reduce the volatile solvents in contaminated solution using hot air bubbles. The simulation results can be used to predict the performance of the bubble column and the optimal operating conditions for laboratory or industrial scale.

1. Introduction
Gas bubbles are normally used in both environmental and industrial separation processes. For these systems, bubbles are generated and used in the treatment of drinkable water and wastewaters\textsuperscript{[1-2]} to remove volatile contaminants and to separate particulate materials from the aqueous phase\textsuperscript{[3-4]}. Depending on the treatment application, several methods are used to generate the gas bubbles. The diameter and size distribution of bubbles is important because smaller bubbles have more efficient in term of mass transfer rates\textsuperscript{[5]}.

Bubble column reactors or multiphase reactors consist of three main categories including the trickle bed reactor, fluidized bed reactor, and the bubble column reactor. The bubble column reactor is basically a cylindrical vessel equipped with a gas distributor at the bottom. Bubble columns are used as multiphase contactors and reactors\textsuperscript{[6]}. In recent years the design and scale-up of bubble columns have increased considerable attention due to complex hydrodynamics and its influence on transport characteristics, although the construction of bubble columns is simple. Industrial bubble columns normally operate with a length-to-diameter ratio, or aspect ratio of at least 5\textsuperscript{[6]}. In biochemical applications, this value generally varies between 2 and 5\textsuperscript{[7]}.

Microbubbles are currently used in water treatment process\textsuperscript{[8]}. Mixing efficiency can be increased by the movement of bubbles. The benefit of microbubbles is the surface area to volume ratio that smaller bubbles have higher surface area than larger bubbles at the same volume\textsuperscript{[9]}. So, microbubbles...
can be used in the process when large surface area between gas phase and liquid phase are required [10]. Huang et al., employed bubbles to remove heavy metals from aqueous solution in a treatment method known as adsorbing colloid flotation. Heavy metals attached bubbles and floats to the surface for removal [11]. Moreover, study of design parameters using computational fluid dynamics (CFDs) have led to better understanding of the hydrodynamic properties, heat and mass transfer mechanisms and flow regime in the bubble columns during the operation. Experimental studies are also supported by CFD simulations [7].

In this work, a hot air bubble was used to remove volatile solvents in the aqueous solution. Both simulation (CFD) and experiment were carried out. The temperature distribution and flow behavior of gas bubble were investigated. The evaporation rate of volatile solvents was also examined.

2. Simulations and Experiments

2.1. Governing Equations Applied in Simulations

COMSOL MULTIPHYSICS 3.5a was used to predict flow behavior of a single bubble and temperature distribution within the air (dispersed phase) - liquid (continuous phase) system. A symmetrical 2D axial symmetry model as shown in figure 1(a) was created with Governing equations, Cahn-Hilliard equation (Phase Field technique), Momentum Transport equations (Incompressible Navier-Stokes) and Energy Transport equations (Convection and Conduction). 2D asymmetric model of a column with a diameter of 10 mm and a length of 15 mm as shown in figure 1 were constructed.

![Figure 1. 2D asymmetric models of (a) bubble column geometry and (b) discretization and mesh size.](image)

The diameter of bubble was set at 3 mm. In the simulations, the following assumptions were made:
- Flow is in a time dependent (Transient 0.5 s)
- Flow is asymmetric and laminar i.e, $Re_D = O(1)$
- Flow and temperature and pressure are uniform at the inlet
- Transport properties of the system are: $\rho$, $\eta$, $C_p$ and $k$ which function of temperature
The transport phenomena and thermodynamics of air-liquid (water) system were set at various times and temperature of 100°C, 125°C, 150°C, 175°C, and 200°C.

The Phase Field application mode describes the two-phase flow dynamics using a Cahn-Hilliard equation. The equation tracks a diffuse interface separating the immiscible phases. The diffuse interface is defined as the region where the dimensionless phase field variable $\phi$ goes from $-1$ to $1$.

$$\left( \frac{\partial \phi}{\partial t} \right) + u \cdot \nabla \phi = \nabla \cdot \left( \frac{\lambda}{\varepsilon} \nabla \psi \right)$$  

(1)

In the above equations $u$ is the fluid velocity (m/s), $\gamma$ is the mobility (m$^3$·s/kg), $\varepsilon$ is the interface thickness parameter (m), $\lambda$ is the mixing energy density (N) and $\psi$ variable is defined as the phase field help variable.

Momentum Transport: In both the Two-Phase Flow, Laminar, Level Set and the Two-Phase Flow, Laminar, Phase Field application modes, the transport of mass and momentum is governed by the incompressible Navier-Stokes equations, including surface tension:

$$\rho \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) = -\nabla p + \nabla \cdot \eta \left( \nabla u + \nabla u^T \right) + \rho g + F_{st}, \nabla u = 0$$  

(2)

where $\rho$ (kg/m$^3$) denotes the density, $u$ is the velocity (m/s), $t$ equals time (s), $p$ is the pressure (Pa), and $\eta$ denotes the viscosity (Pa·s). The momentum equations contain gravity, $\rho g$ and surface tension force components, denoted by $F_{st}$.

2.2. Energy Transport equations (Convection and Conduction)

$$\rho C_p \left( \frac{\partial T}{\partial t} + u \cdot \nabla T \right) = \nabla \cdot (k \nabla T) + Q$$  

(3)

where $T$ = temperature (K), $C_p$ = heat capacity (J/kg·K), $k$ = thermal conductivity of the mixed gas (W/m·K), and $Q$ = heat generation rate (W/m$^3$).

2.3. Experimental approach

The solution containing acetone of 50% by volume and water was filled in the bubble column (diameter of 7.5 cm). The volume of solution was fixed at 200 ml. Flow rate of hot air were set at 5, 7.5, 10, 12.5 and 15 L/min. Temperatures of hot air were also set at 85°C and 100°C (depending on the simulation result). Hot air bubbles were generated while air was fed through the column. Bubble-sizes were varied depending on the flow rate of hot air. Here, the diffuser with a diameter of 50 mm and a pore size of 10-16 µm was used that can generate bubbles with the diameter of 3 mm.

As shown in figure 2, the compressed air flow was fed through a regulator and heated before entering the bubble column. The temperatures were controlled using the controller and the type-K thermocouple was installed inside the column’s center. The amount of evaporated liquid was recorded at 10 minutes after feeding.
3. Results and discussion

3.1. Simulation Results

Figure 3 showed the bubble moving up through liquid and merges with air represented by 2D asymmetric model of air-liquid system in cases of a surface and contour plot of the phase field function. The snapshots also showed the interface prior to and just after the bubble hits the surface (at \( t = 0 \), the velocity is zero).

As the bubble rising, its shape remains spherical due to the surface tension and the high viscosity of the bubble. After the bubble hitting the water surface, it merges with the air above and creates waves on the surface as indicated by velocity vectors (green arrows) as shown in figure 4.

![Figure 3](image)

**Figure 3.** The interface prior to and just after the bubble hits the surface of air-liquid system at 100°C (a) \( t = 0 \) s (b) \( t = 0.02 \) s (c) \( t = 0.04 \) s (d) \( t = 0.05 \) s (e) \( t = 0.06 \) s (f) \( t = 0.07 \) s (g) \( t = 0.08 \) s and (h) \( t = 0.11 \) s, respectively.

The transport phenomena and thermodynamics of air-liquid system at various times and temperatures were illustrated in figure 5. Heat transfer within the column resulted from temperature distributions among the hot air bubble and interface prior to and just after the bubble hits the surface. The local convective heat transfer might be influenced by the bubble hydrodynamics and air-liquid interactions. Figure 4 and 5 showed the simulated temperature fields and streamwise velocity vectors and temperature profiles of each temperature, respectively. The convective fluxes resulted from shearing local velocity of the air-liquid system due to the change of flow direction when the bubble moves up through the liquid and merges with the air above, as previously observed in figure 3. The simulation results of the transport phenomena and thermodynamics of air-liquid system at various times might be applied to completely remove volatile solvents at 100°C.
3.2. Preliminary Experimental Results

The simulation results confirmed that volatile solvent can be removed at the temperature above 100°C. Hot air with various flow rates at the temperatures of 85°C and 100°C were set. At the inlet air temperature of 85°C, the volume of solution decreases from 200 ml to 175 ml, 156 ml, 147 ml, 136 ml, and 104 ml with the air flow rate of 5, 7.5, 10, 12.5 and 15 L/min, respectively. At the inlet air temperature of 100°C, the volume of solution decreases from 200 ml to 158 ml, 148 ml, 140 ml, 123 ml, and 98 ml with the air flow rate of 5, 7.5, 10, 12.5 and 15 L/min, respectively. This means that volatile solvents can evaporate at the inlet air temperature below 100°C without the increase of solution temperature. Moreover, the increase of inlet air temperature and volume metric flow rate result in the increase of liquid evaporation rate. This can reduce the treatment cost due to the small amount of solvent remained.
4. Conclusion
In the present study, the CFD was applied to investigate the hydrodynamics and heat transfer of the air-liquid system in the bubble column. It was also applied to predict the inlet air temperature that volatile solvent can be evaporated. Both simulation and experiment results are identical that the inlet air temperature below 100°C can remove the volatile solvent without the increase of solution temperature. This research results might be used to improve and design the bubble column for industrial scale.

Acknowledgment
This research was funded by King Mongkut’s University of Technology North Bangkok. Contract no. KMUTNB-NEW-60-33. We would like to thank the Department of Chemical Engineering, KMUTNB for both workstation computer and COMSOL Multiphysics 3.5a software support.

References
[1] Zabel T 1985 J. Am. Water Works Assoc. 77 42–6
[2] de Rijk S, Graaf J and den Blanken J 1994 Water Res. 28 465–73
[3] Ahmed N and Jameson G J 1985 Int. J. Miner. Process. 14 195–215
[4] Ketkar D R, Mallikarjunan R and Venkatachalam S 1991 Int. J. Miner. Process. 31 127–38
[5] Burns S E, Yiacoumi S and Tsouris C 1997 Sep. Purif. Technol. 11 221–32
[6] Degaleesaa S, Dudukovic M and Pan Y 2001 AIChE J. 47 1913–31
[7] Kantarci N, Borak F and Ulgen K O 2005 Process Biochem. 40 2263–83
[8] Elbing B R, Still A L and Ghajar A J 2016 Ind. Eng. Chem. Res. 55 385–403
[9] Takahashi M, Chiba K and Li P 2007 J. Phys. Chem. 111 1343–47
[10] Maeda Y, Hosokawa S, Baba Y, Tomiyama A and Ito Y 2015 Exp. Therm. Fluid Sci. 60 201–7
[11] Huang S, Ho H, Li Y and Lin C 1995 Environ. Sci. Technol. 29 1802–07