As. J. Energy Env. **2005**, 6(03), 154-164

Asian Journal on Energy and Environment

ISSN 1513-4121

Available online at www.asian-energy-journal.info

CFD Modeling of a Gas-Liquid Mixing in T –junction

Wanchai Asvapoositkul, Jirapong Joraden and Somchai Wongwises

Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140

Abstract: The transport processes that are involved in the mixing of gas-liquid in a Tjunction mixer are investigated. This internal flow mixing is related to the process that occurs in T - mixer aerator. The study was limited to the effectiveness of branch tube depth and branch tube end angle. CFD code is used to predict the mixing performance of T -junction under those conditions. The results obtained by numerical simulations are verified with an experiment. The Dissolved Oxygen (DO) are measured and used to calculate the mass of oxygen transferred to the water. The amount of this mass transferred is found to be increased when the volume flow rate of air to that of water (Qa/Qw) is increased. Comparison of this value to the interfacial area per unit volume between air and water shows in the same trend for the variation Qa/Qw. The CFD can predict the flow patterns with adequate accuracy for the model. Good mixing is obtained in aT-junction if the branch tube is designed to penetrate deeply into the main tube while the branch tube end inlet angle does not have any significant to the flow field.

Keywords: Computational fluid dynamics, T-junction, gas-liquid flow, aerator

---

Introduction

In recent years, the numerical simulations employed the CFD to determine the flow characteristics of multi-phase flow and the mixing performance have studied by many authors since many industrial applications are involving with problems interacting fluids and the multi-fluid version of the Navier Stokes equations is extremely complex and represents a challenge to advanced numerical algorithms [1]. With CFD simulations for mixer design and analysis, engineers can investigate with a wide range of design options at a relatively low cost and a short time. Kok and van der Wal [2] focused on the mixing of natural gas and air in T-junctions. Gobby et al [3] used CFD to study the mixing characteristics of a T-type microscale mixer for gaseous. Their results showed that the flow patterns under a range of operating and design parameters indicated that the mixing length increases with the fluid speed and good mixing is obtained when the branch inlet flow penetrates to the opposite tube wall. CFD modeling for the mixing of a gas-liquid flow may be seen in Morchain, et al [4] for jet aerator and Deen, et al [5] for an aerated Rushton impeller.

Mixing of gas-liquid in T-junction or side-entry mixer with two inlets occurs in many applications. Comparisons and reviews dealing with a variety of types of aerators used can be seen in Boyd [6]. A good example is T-mixer (transverse jet in a pipe) of an aerator where bubbles of air injected radially into a fully developed turbulent pipe flow of water with the goal of efficiently mixing.

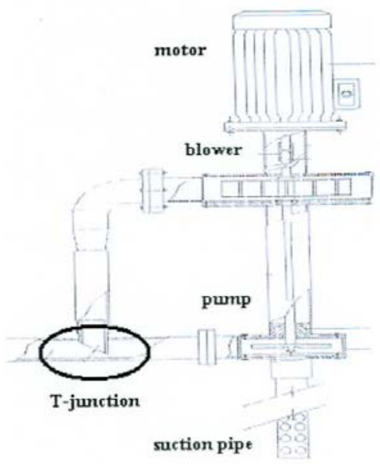


Fig 1 Schematic diagram ofT-mixer in aerator [7]

In the present work, we will refer to a particular type of aerator designed by His Majesty King Bhumibol Adulyadej (Model RX5C, [7]). The mixer model RX5C can be classified as diffused-air system aerator which has high efficiency for transfer oxygen from air bubbles to water. The rate of oxygen transfer from air to water is influenced by increasing turbulence and surface area of water in contact with air. The geometry of this aerator is shown in Fig 1. T-mixing in this study is part of the aerator where the transfer of oxygen occurs in this zone. The main objective of this study is to demonstrate of what can be accomplished through CFD, and at the same time, the conclusions regarding mixing mechanisms and their modeling with a view to design and prediction of performance may be achieved.

Numerical simulation

The equations used to describe the system are multi - phase version of the Navier Stokes equations

$$\frac{\partial(r_\alpha \rho_\alpha)}{\partial t} + \nabla \cdot r_\alpha(\rho_\alpha U_\alpha) = 0 \qquad (1)$$

and

$$\frac{\partial(r_\alpha \rho_\alpha U_\alpha)}{\partial t} + \nabla \cdot (r_\alpha(\rho_\alpha U_\alpha U_\alpha)) = -r_\alpha \nabla p_\alpha + \nabla \cdot \left(r_\alpha \mu_{\alpha eff}\left(\nabla U_\alpha + (\nabla U_\alpha)^T\right)\right) + S_{M\alpha} + M_{I,\alpha} \qquad (2)$$

where

$$\sum_{\alpha=1}^{N} r_\alpha = 1 \qquad (3)$$

and

$$\mu_{\alpha\, eff} = \mu_\alpha + \mu_{t\alpha} \qquad (4)$$

Simulations were performed using the 5.7 version of the CFD code CFX on Pc. The code uses the Finite Volume Method for the discretisation of the governing equations. The k-E model in multiphase flow was used to account for turbulence effects. The flow is steady three-dimension with symmetry plane at the half cross-section of the pipe. The geometry and grid used to simulate the mixer is shown in Fig 2. The T-mixer used in this calculation consists of the branch pipe (d = 19 mm.) injected air radially into the main pipe flow of water (D = 54mm.). The main pipe total length is 1 meter and the branch pipe joined the main pipe at 200 mm. from water inlet. The performance of the mixer when mixing water with air was determined for several flow rate ratios, branch tube depth and branch tube end angle. The flow rate ratio of air to water (Qa/Qw) is variable from 0.30-0.5 while the branch tube depth (H) is variable to 0.25D, 0.50D or 0.75D with branch tube end angle (8) of 0, 30 and 60 degree.

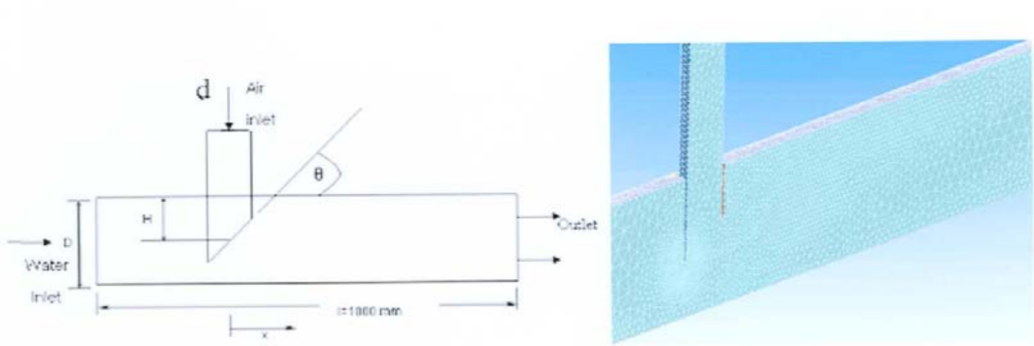


Fig 2 computational domain and computational mesh employed for the T-mixer.

The boundary conditions for this geometry are as follows. The mean velocity at the water inlet is set to a value which corresponds to the measured water flow rate passing through the main pipe (170-200 *Vmin).* The mean velocity at air inlet is prescribed according to the specified flow rate ratio of air to water. Hydrostatic pressure at the mixture outlet is 5 m of water which is the actual fluid head when the aerator is installed. The mass transfer between the fluids was not taken into consideration. A smooth wall with no-slip condition is imposed on both pipes wall. Refine grid in areas of high velocity gradients, this resulted in element sizes near the branch pipe tip as low as 0.5 mm. and final mesh sizes close to 460,000 elements (Fig 2).

Experimental set-up

The experiment was performed to determine the performance of the mixer when mixing water and air for several flow rates. The main inlet of the T -mixer was fed with water by a pump. Its flow rate was varied by motor speed which equipped with an inverter. Air was blown by a blower through the branch inlet. Each flow rate was measured by an orifice. This device was corresponded with the TISI 710-2530.

The steady-state test was conducted, measured flow volume and DO concentration before and after aeration. The difference in the mass of DO between the inflow and the outflow represented the mass of oxygen transferred to the water by the aerator [6].

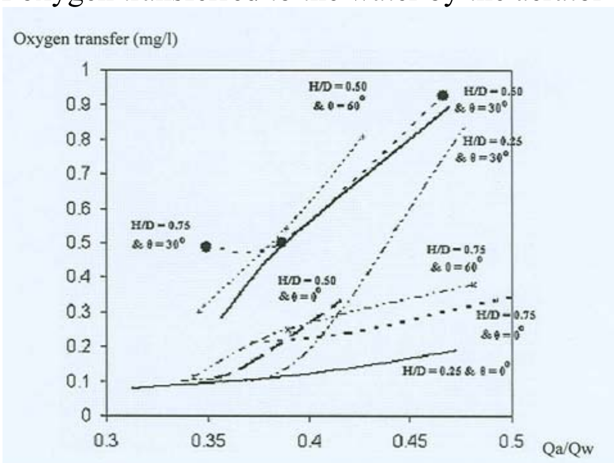


Fig 3 Oxygen transfer with respect to QalQw, for given e and HID (Experiment result)

Results and Discussion

Oxygen transfer and Interfacial area density

The steady-state oxygen transfer test is shown in Fig 3. The figure shows variation of oxygen transfer with respect to QJQw, for given 8 and HID. Its' value

increases as QJQw increases. For branch tube end angle of 0 degree (8 = 0°), the rate of oxygen transfer increase is low at the branch tube depth of 0.250 (HID = 0.25) while that of HID = 0.5 is high. When Qa/Qw > 0.4, the amount of oxygen transfer is high for HID = 0.5. Compare to the other branch tube end angles (8 = 30°& 60°), the configuration of 8 = 60° and HID = 0.5 provides high value of oxygen transfer when Q.IQw> 0.38. These results demonstrate that the designs with HID <: 0.5, in general, were more efficient than other designs, except with 8 = 0° and 60°.

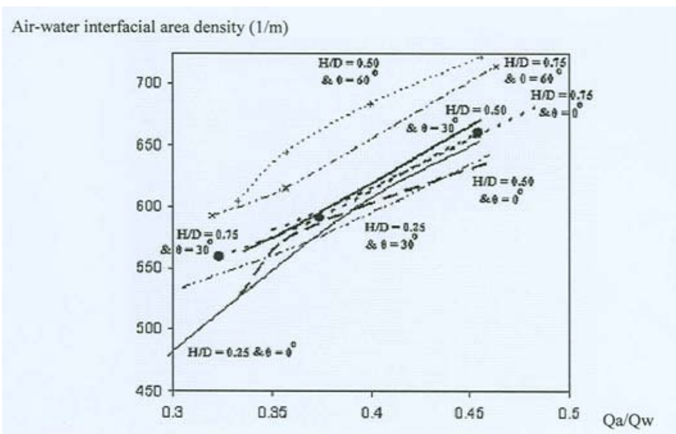


Fig 4 Air-water interfacial area density with respect to Qa/Qw, for given e and HID (CFD calculation)

The predicted value of DO cannot calculate directly from the code since no mass transfer of air to water is involved. However, multiphase modeling is assumed that each phase is present in each control volume occupied by that phase. Thus the volume occupied by air in a small volume around a point is assigned as air volume fraction. The contact surface area between air and water influence interfacial transfer of momentum, heat and mass.This is characterized by the interfacial area per unit volume between air and water, known as air-water interfacial area density. It has dimensions of one over length. Calculated interfacial area density of various investigations is given in Fig 4. It can be seen that for e = 0°, high value of interfacial can be expected when HID = 0.75. For e = 30°, the interfacial is high when HID = 0.5. The design ofe = 60° and HID = 0.5 gives the highest value. In general, the high values are among the design with HID ~ 0.5. By visual comparative, one can see that both plots are in similar trend except that the design for HID = 0.75 of both e = 0° & e = 60° show low oxygen transfer in Fig 3.

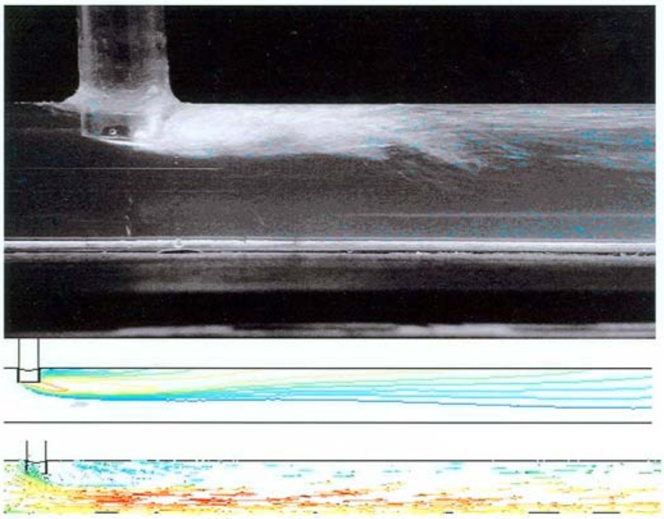


Fig 5A *HID* = 0.25 & 8 = 0°

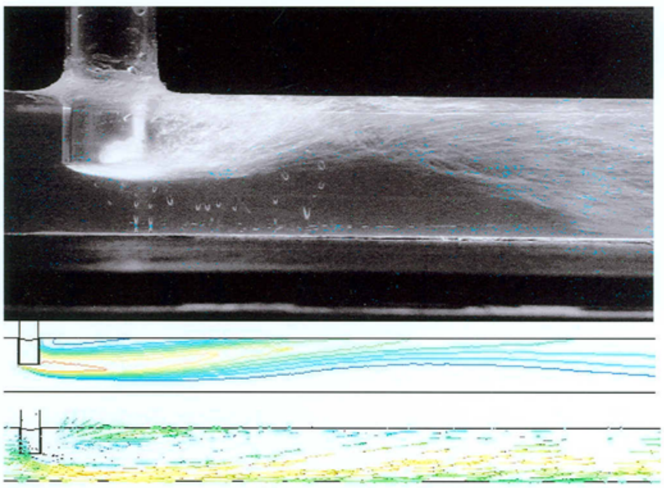


Fig 5B *HID* = 0.50 & 8 = 0°

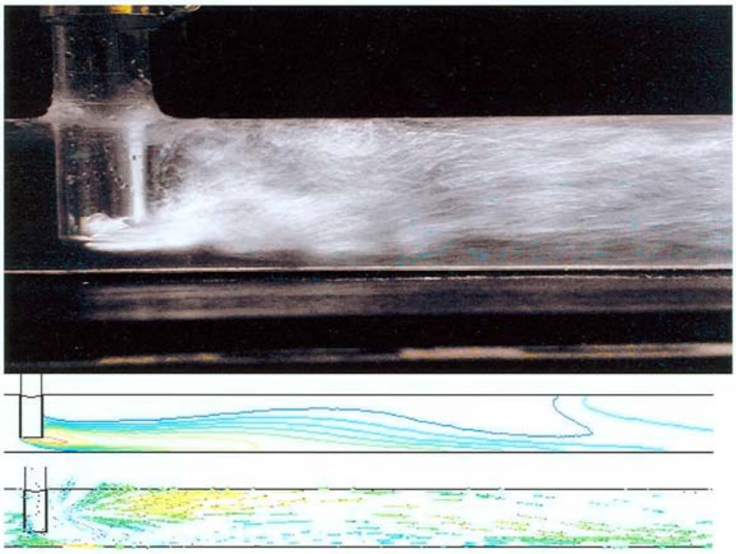


Fig 5 C H/D = 0.75 & θ = 0°

Fig 5 Flow visualization with Qa/Qw = 0.4 & 8 = 00

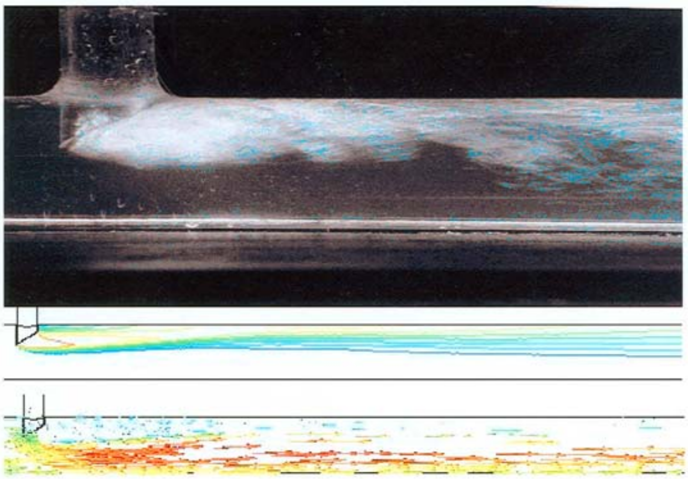


Fig 6A H/D = 0.25 & θ = 30°

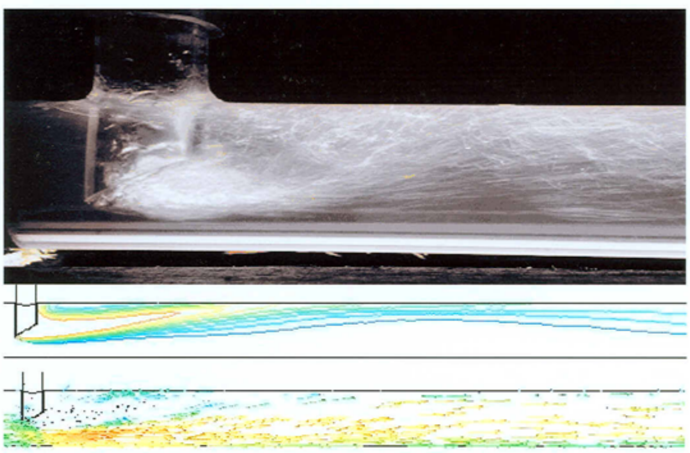


Fig 6B H/D = 0.50 & θ = 30°

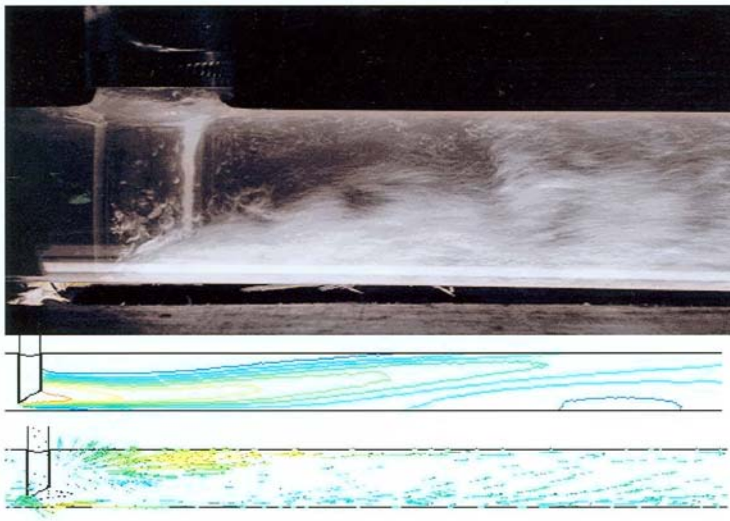


Fig 6C H/D = 0.75 & θ = 30°

Fig 6 Flow visualization with *QalQw* = 0.4 & e = 30°

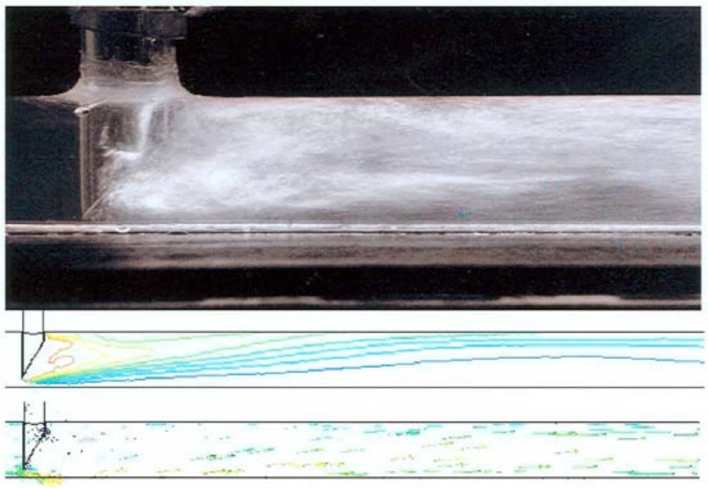


Fig 7A H/D = 0.50 & θ = 60°

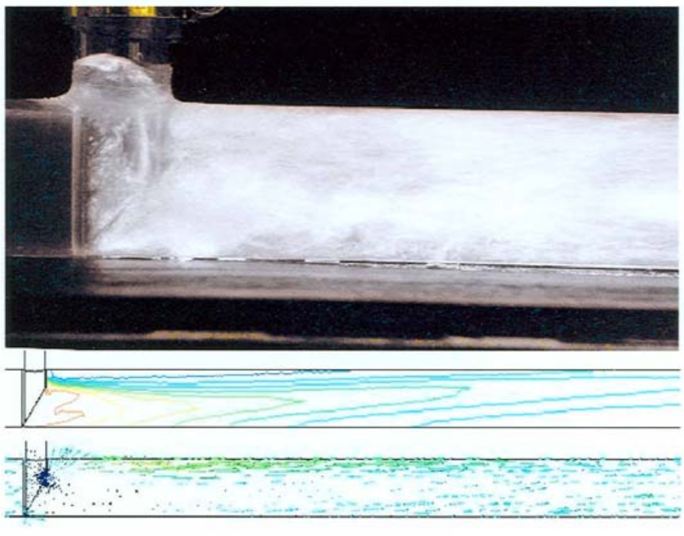
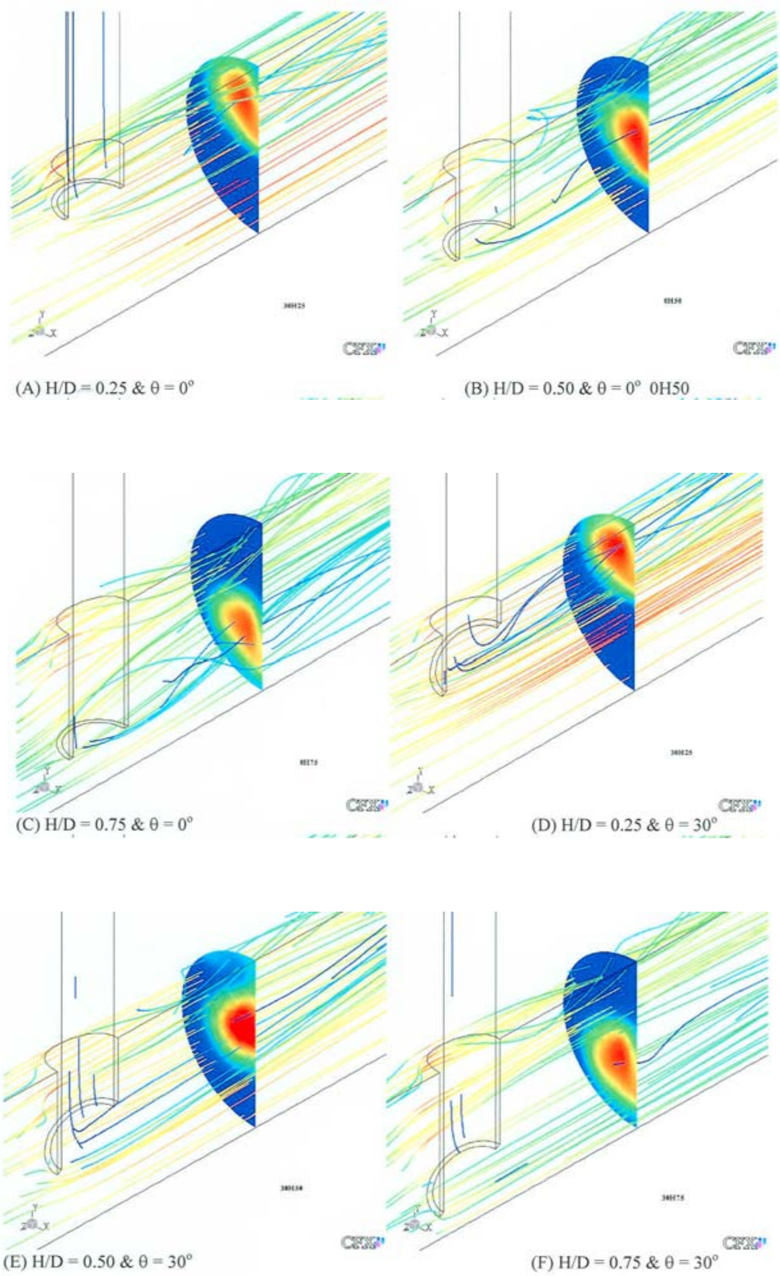


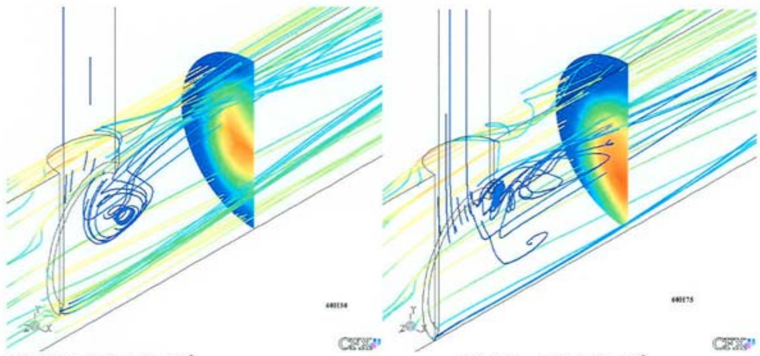
Fig 7B H/D = 0.75 & θ = 60°

Fig 7 Flow visualization with QalQw = 0.4 & e = 60°

Flow visualization

The flow of air and its influence on the motion of water can be presented in pictures as shown in Fig 5-7. These pictures allow us to observe the mixing characteristics. The structure of the air-water flow was shown to be emulsion like with bubbles following the water path. The presence of bubbles does not significantly the velocities of the liquid phase. From the pictures, it can be seen that the release of air bubbles near the bottom of pipe wall create a large surface contact area between air bubbles and surrounding water. As HID is increased, bubbles rise upward and suspend beneath the top of pipe wall. Rising bubbles also create turbulence within a body of water. Streamwise roll-up of the flow behind the branch tube is clearly visualized (Fig 5C).

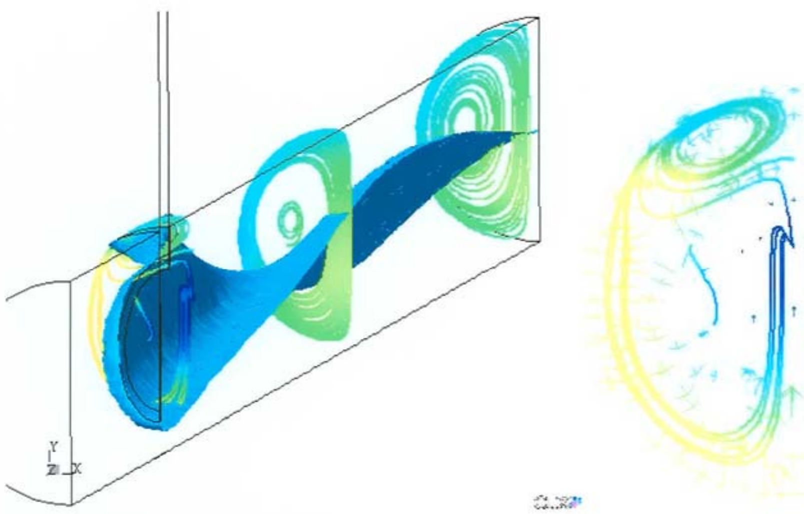




(G) H/D = 0.50 & θ = 60° (H) H/D = 0.75 & θ = 60°

Fig 8 Streamline and concentration of air volume fraction with Qa/Qw = 0.4

Base on these side-view pictures, the calculated results from CFD were also presented. The effect of HID on the opposite pipe wall can be observed. The contours of air volume fraction show similar distribution as that of flow visualization in all designs (Fig 5-7). It should be noted that concentration of air volume fraction move downward as HID is increased. Combining this with the associated streamline plot in Fig 8, it can be concluded that air from the branch inlet is very well dispersed over the cross-section when e = 60° and HID = 0.5. This cross-section contour was taken at jet momentum length (d Ua/Uw). This length parameter is interpreted as the expanded diameter of the jet air after the air-velocity is aligned with the pipe flow [8].



(A) air volume fraction iso-surface at 0.20 and streamline contour plot (B) section view with velocity vector

Fig 9 T-junction with Qa/Qw = 0.4, *HID* = 0.50 & e = 60°

The combined plot of streamline and velocity vector at this cross-section was also presented in Fig 9. It is confirmed the existence of counter-rotating vortex pair. Shortly downstream, these vortices break down and give rise to the bubbles.

Conclusions

A numerical and experimental investigation of air-water mixing in T -junction was conducted. The oxygen transfer for various designs was measured. This parameter was not available for CFD since no mass transfer was assumed. The other parameters like interfacial area per unit volume between air-water and air volume fraction were adequate for comparison of mixing performance.

Base on flow visualization, it was demonstrated that when air bubbles released near bottom of pipe wall, the flow was subsequently deflected and air bubbles rise upward. This enhanced the mixing where air could enter a given volume of water in a specified time interval.

As a conclusion with the determination of the flow characteristics and oxygen transfer in the T -junction it can be state that the mixing performance has been shown to be depend on the branch inlet position. Fig 3 & 8 give a good illustration of this result.

Acknowledgement

The present study was partially supported by the Thailand Research Fund and the staff of R-CAR Laboratory, Mechanical Engineering Department, King Mongkut's University of Technology Thonburi. Their guidance and assistance are gratefully acknowledged.

Nomenclature

d = branch tube diameter

D = main pipe diameter

H = branch tube depth

$M_{I,\alpha}$ = interphase force acting on phase α

p = static pressure

Q = volume flow rate

r = volume fraction

$S_{M\alpha}$ = momentum source e.g. body force

U = velocity vector

θ = branch tube end angle

ρ = density

μ_{α} = dynamic viscosity of fluid phase α

$\mu_{t\alpha}$ = turbulent eddy viscosity of fluid phase α

$\mu_{\alpha\ eff}$ = effective viscosity

Subscript

a = air

w = water

α = fluid property of continuous phase

References

[1.] Thompson c.P. and Lezeau, P., "Application of the full approximation storage method to the numerical simulation of two-dimensional steady incompressible viscous multiphase flows", International Journal for Numerical Methods in Fluids, Vol. 28, No.8, (1998), pp 1217-39.

[2.] Kok J.B.W. and van der Wal S., "Mixing in T-junction", Applied Mathematical Modelling, Vol. 20, No.3, (1996), 232-43

[3.] Gobby D., Angeli P. and Gavriilidis A., "Mixing characteristics of T-type microfluidic Mixers ", Journal of Micromechanics and Microengineering, Vol.14, No.9, (2004), pp.l23-7

[4.] Morchain J., Maranges C. and Fonade c., "CFD modelling of a two- phase jet aerator Under influence of a cross flow ", Water Resource, Vol. 34, No. 13, (2000), pp. 3460 3472

[5.] Deen N.G., Solberg T. and Hjertager B.H., " Flow generated by an aerated Rushton Impeller: two – phase PIV experiments and numerical simulations" , The Canadian Journal of Chemical Engineering, (2002)