Optimization of the Mixing Flow in an Agitated Tank

Dal-Hyun Yoo, Si-Young Yang and Youn-Kyu Choi

Abstract

In the chemical, mineral and electronics industries, mechanically stirred tanks are widely used for complex liquid and particle mixing processes. In order to understand the complex phenomena that occur in such tanks, it is necessary to investigate flow field in the vessel.

Most difficulty on the numerical analysis of stirred tank flow field focused particularly on free surface analysis. In order to decrease the dead zone and improve the flow efficiency of a system with free surface, this paper presents a new method that overcomes free surface effects by properly combining the benefits of using experiment and 3-D CFD. This method is applied to study the mixing flow in an agitated tank. From the results of experimental studies using the PIV (particle image velocimetry) system, the distribution of mixing flow including free surface are obtained. And these values that are expressed as a velocity vector field have been patched for simulating the free surface. The results of velocity distribution obtained by 3-D CFD are compared with those of experimental results. The experimental data and the simulation results are in good agreement.

Keyword

Agitated Tank, Free Surface, CFD, PIV System

1. INTRODUCTION

During the last ten years, computational fluid dynamics has become a very powerful tool in the process industry not only for the research and development of a new process but also for the understanding and optimization of exciting ones. In the chemical, mineral industries, mechanically stirred tanks are widely used for complex liquid mixing processes. In order to understand the complex phenomena that occur in such tanks, it is necessary to investigate flow field in the vessel, as well as turbulence characteristics. Advanced experimental method, such as Particle image velocimetry (PIV) and laser doppler velocimetry (LDV), have been shown to give detailed information on the turbulence flow field in agitated tank for single and two phase applications.

These experimental methods require that the vessel and the operating liquid are translucent in order for laser light transmission, that a transparent rectangular vessel is placed around cylindrical vessels to minimize light refraction. Most often in industrial situations, however, vessels are made of nontransparent materials, the operating liquids are opaque, all of which suggest difficulties for experimental measurement of turbulent flow fields in such vessel types. For these reasons, computational fluid dynamics (CFD) has become an important tool for prediction of flow fields in industrial vessels to better understand flow phenomena and to optimize the processes. The growing importance of the application of CFD to industrial problems raises a major question. That is the numerical solutions physical, and correctly predicted. More and more published works on CFD in stirred vessels consist of combined experimental and numerical studies, using LDV or PIV results to validate the numerical solutions. Most other CFD studies use previously reported experimental results in order to compare their predictions. As a general conclusion, the authors claim that CFD satisfactorily predicts, qualitatively and quantitatively, the axial and radial mean flow patterns but under or over predicts the tangential velocity component and turbulent quantities, such as the turbulent kinetic energy, and the turbulent energy dissipation rate. As CFD is based on the Navier-Stokes equations and mathematical methods with simplifying assumptions, there are many sources for such discrepancies. Amongst these are the type of modeling approach employed, such as many type of boundary conditions, steady state or transient models, turbulent models and discretization schemes. A few reported works compare the effect of different modeling approaches. Brucato et al. [3] compare simulation results for Rushton turbines and down pumping axial impeller that use multiple reference frame and sliding mesh with the k-ε turbulence model and either the hybrid upwind differencing or the QUICK discretization scheme. They showed that the results of the method are very sensitive to the imposed boundary conditions themselves and satisfactory predictions reply completely on the availability and accuracy of these for a specific stirred vessel geometry. The authors found that the method yielded the vest agreement with experimental data for the mean flow field, although it tended to under predict k values. The same group authors carried out a similar study on dual Rushton turbines and came to the same conclusions. The effect of discretization schemes on the simulation of flow stirred vessels has been dealt with in very few works. Sahu and Joshi [4] used the impeller boundary conditions technique to simulate the flow. Several studies have focused on the effect of various turbulence models on the final numerical solution. Most commonly, a comparison between the standard k-ε and RNG k-ε models has been made. Jaworski et al. [5] studied the flow produced by a Rushton turbine using a sliding mesh and reported that the type of turbulence model did not have much effect on the mean velocities. These results showed good agreement with experimental results except for in the trailing vortex region. The turbulence quantities were found to be largely under predicted, although better agreement with experimental data was found for the standard k-ε model than the RNG k-ε model. The k-ε models, which make the isotropic eddy viscosity assumption, are known to be insensitive to streamline curvature and system rotation. These limitations have led some researchers to investigate the use of anisotropic Reynolds stress models on the calculated turbulent flow field. Amenante and chou [6] studied the turbulent flow produced by down pumping single and dual six blade pitched blade turbines using the impeller boundary condition method with LDV data. They compared results employing the Algebraic stress model and the standard k-ε model and reported that the results of the Algebraic stress model were marginally but consistently in better agreement with the experimental data than the standard k-ε model. Although these preceding studies are extensive in themselves, they appear to have some weakness with respect to the capabilities of CFD today.

Most difficulty on the numerical analysis of stirred tank flow field focused particularly on free surface analysis. In order to decrease the dead zone and improve the flow efficiency of a system with free surface, this paper presents a new method that overcomes free surface effects by using experiment and 3-D CFD. The method is applied to study the mixing flow in agitated tank. From the experimental studies using the particle image velocimetry (PIV) system, the distribution of mixing flow including free surface are obtained. And these values are used as patch condition of 3-D CFD modeling. The results of velocity distribution obtained by 3-D CFD are compared with those of experimental results. The experimental data and the simulation results are in good agreement. And, From the results of this three dimensional turbulence flow field analysis for an agitated tank, it was verified that the flow dead zone is decreased and volume flow rate is raised by using a baffle, streamline shape guide and radial fan

2. Flow Analysis

2.1. Geometric shape

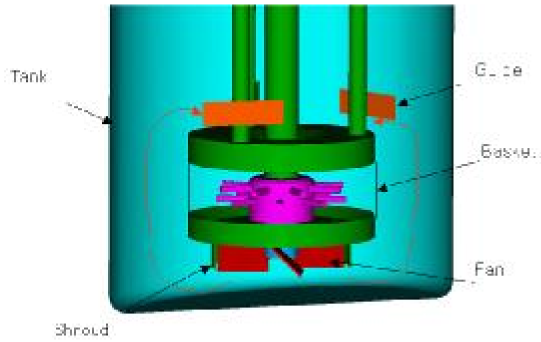


Fig. 1 Agitated tank configurations

TABLE 1. AGITATED TANK SPECIFICATIONS

Designation	Size
Tank radius	250mm
Tank height	610mm
Fan type	Axial flow fan (blade number : 4)
Basket radius	136mm
Basket height	149mm
Rotating axis radius	25mm
rpm	600

The agitated cylindrical tank geometry used in this work is shown in figure 1 and table 1. It employs an axial fan, three supports and straight guide.

2.2. Experimental analysis

Although many flow visualization techniques have been employed to study the dynamics and topology of coherent structures in turbulent flow, none of the currently available techniques can provide instantaneous vorticity fields. Vorticity is one of the fundamental characteristics that is important for the understanding of the physics of turbulence phenomena. While numerical simulation allows us to explore sophisticated aspects of vortex dynamics, it is generally limited to simple geometries and low Reynolds numbers. The new approach with particle image velocimetry system (PIV)(Figure 2) has shown promise in the measurement of vorticity fields. In recent years, PIV has been used very extensively in fluid flow research. The most salient characteristic of the technique is its ability to measure flow in a two dimensional plane, providing the two components of velocity vectors. The technique also makes it possible to capture instantaneous turbulent flow fields for the purpose of identifying fundamental mechanisms of turbulence and extracting their spatial structure. Due to its capability of getting on

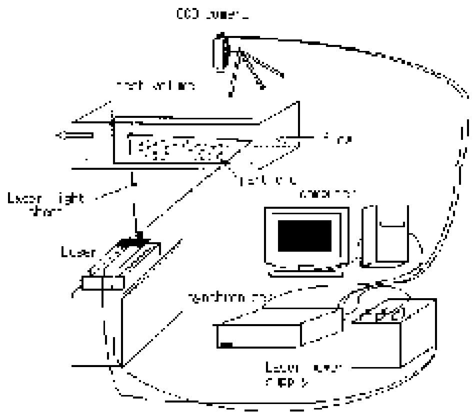


Fig. 2 PIV system

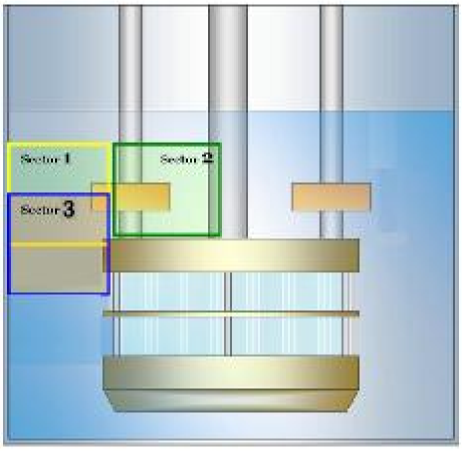


Fig. 3 Measurement position for Agitated tank

instantaneous measurement of velocity field accurately, PIV can also provide the vorticity and rate of strain fields. The fundamental principle of PIV is simple. It measures the displacements of the particle images. The displacements must be small enough so that the data is a good approximation of velocity. The PIV system is comprised of two systems. Image capturing and image analysis. In an image capturing system, a pulsed laser is generally used for illumination due to the high energy in each laser pulse. Either a photographic camera or a CCD video camera can be used as recording medium.

From the experimental studies using PIV system for the visualization of the mixing flow in an agitated tank, the shape of free surface was investigated. These values which are expressed as a velocity vector field has been patched for simulating the free surface. Also the experimental data for full flow fields were compared with the 3-D CFD results. The measurement position and velocity vectors are shown in figure 3-6. Figure 4 shows the velocity vectors near the wall of a vessel. In this area, the rising of free surface by means of the centrifugal force shapes the vortex near the corner of vessel. Figure 5 shows the velocity vectors at the entrance of a basket. The main flow turns toward the entry. The velocity vectors in Figure 6 are formed at the side of a basket. Some of the flow that goes inside of a basket comes out, and the upstream flow exists near the wall of a vessel.

2.3. Numerical analysis

The governing equations of three dimensional incompressible steady flow are as follows.

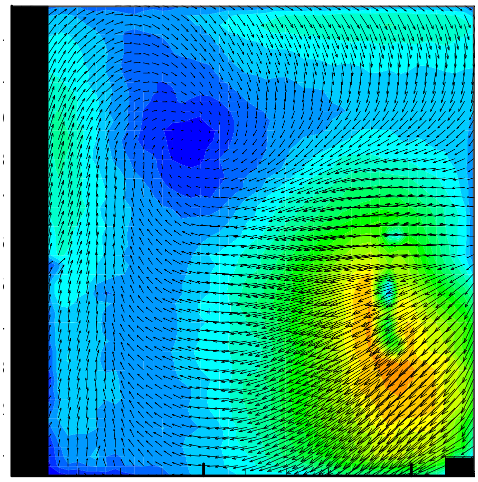


Fig. 4 Velocity vectors at the section 1

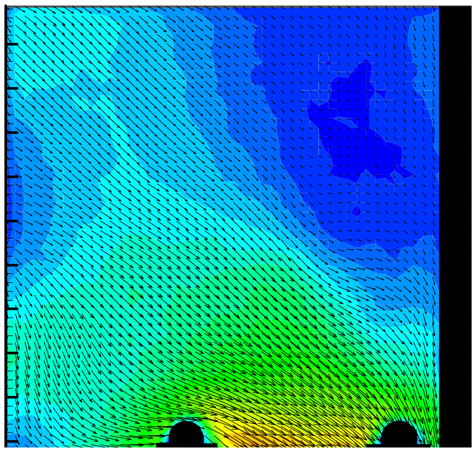


Fig. 5 Velocity vectors at the section 2

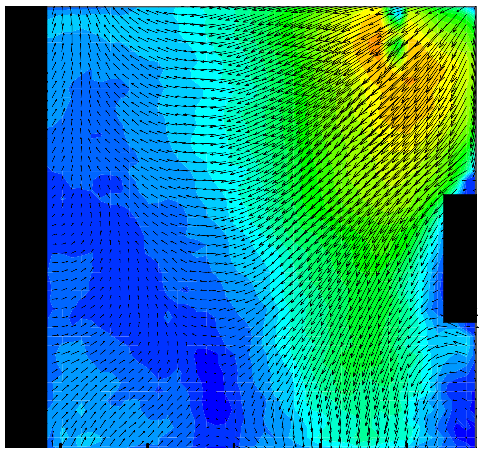


Fig. 6 Velocity vectors at the section 3

$$\nabla \cdot V = 0 \qquad (1)$$

$$V(\nabla \cdot V) = \nabla P + \nabla \cdot (m \nabla \cdot V) \qquad (2)$$

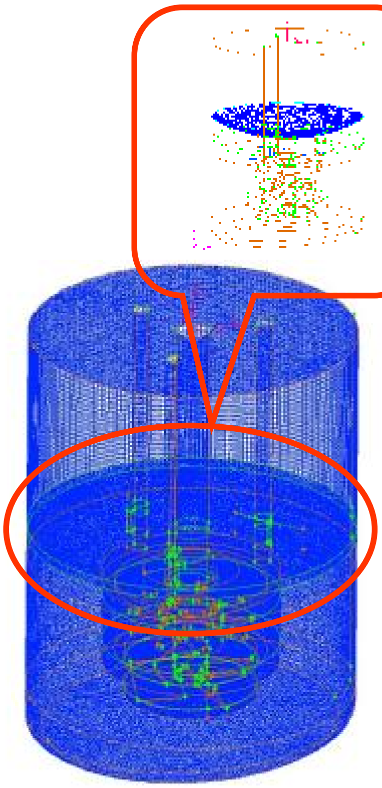


Fig. 7 Computational mesh

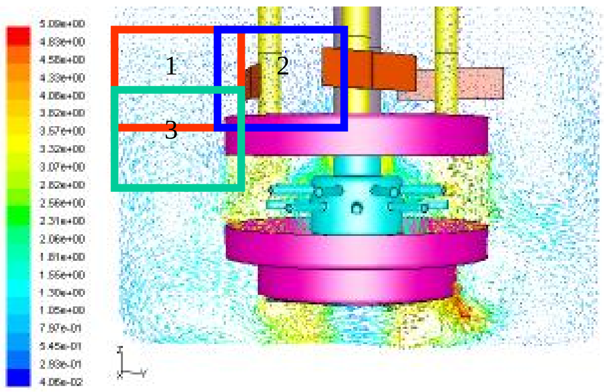


Fig. 8 Velocity vectors at the full domain by 3-D CFD

A multi-block approach was applied to the computational mesh for flow analyze because of the complicated shape. The flow field was divided into 4 regions, and a 2,000,000 unstructured grid was generated for each case, as shown in Figure 7. The governing equations were solved by the finite volume method with the SIMPLE algorithm. The second-order upwind scheme was applied to the convection term, the first-order upwind scheme was applied to the turbulence equation, and standard k-ε model was applied to the turbulence model.

2.4. Boundary conditions

From the results of experimental studies using the PIV system, the distribution of mixing flow including free surface are obtained. These value which are expressed as a velocity vector fields have been patched for simulating the

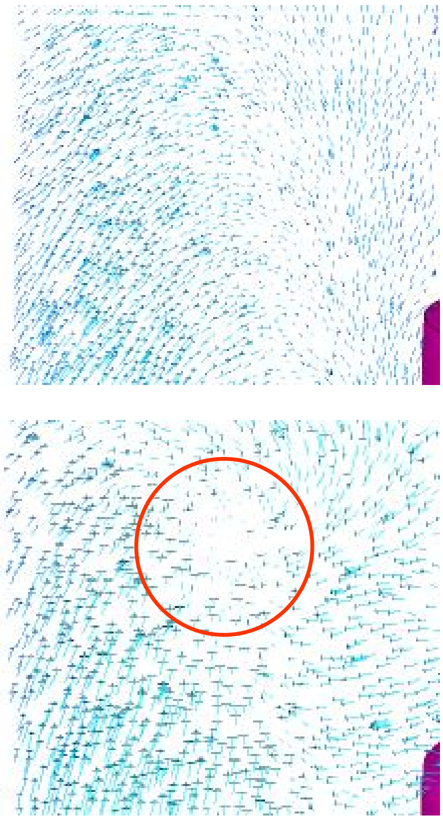


Fig. 9 Velocity vectors comparison at the section 1

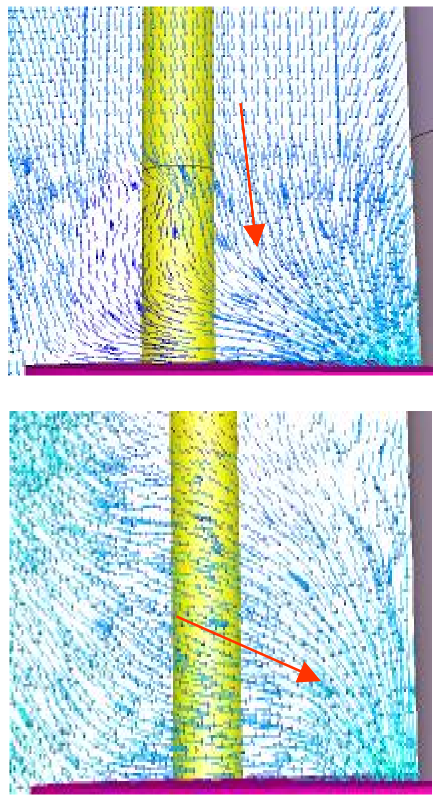
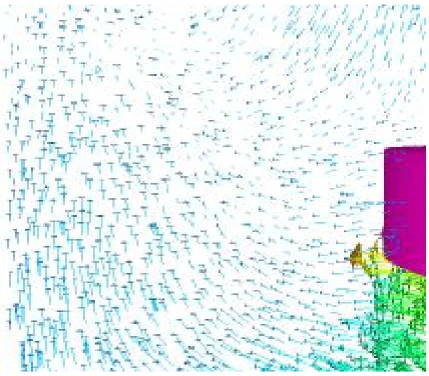


Fig. 10 Velocity vectors comparison at the section 2

free surface. The Boundary conditions at the rotating axis were given the rotating wall condition, and the moving reference frame(MRF) conditions were endowed



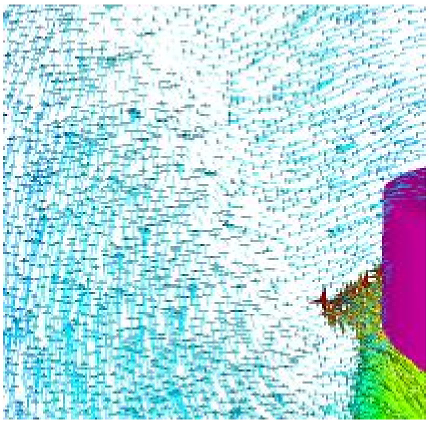


Fig. 11 Velocity vectors comparison at the section 3

at the circumference area. At the free surface, the slip wall condition was given.

2.5. Result of numerical analysis

In order to examine whether the results of the three-dimensional CFD analysis patched with free surfacecan be applied to a agitated tank, the velocity vectors were compared for each sections. Figure 8 is the velocity vectors at the full domain. As the axis rotates, fluid enters into the axial direction of a rotating axis and goes out to the axial direction of a fan. In this process, some of flow rate goes out the side of basket. Figure 9-11 shows the comparison of velocity vectors between the original model and the free surface model. At the section 1, the rising of free surface by means of the centrifugal force shapes the vortex near the corner of vessel. The free surface model predicts this physical phenomenon very well. At the section 2, the angle of velocity vectors near the entrance of basket is different from each other. The free surface model resembles the experimental data. This result is due to the shape of free surface that is a steep grade. At the section 3, some of the flow that goes inside of a basket comes out, and the upstream flow exists near the wall of a vessel.

3. Shape improvement

A rotating axis and basket are the core part of an agitated tank. The flow field of the inside of basket becomes very complicated because of the characteristic

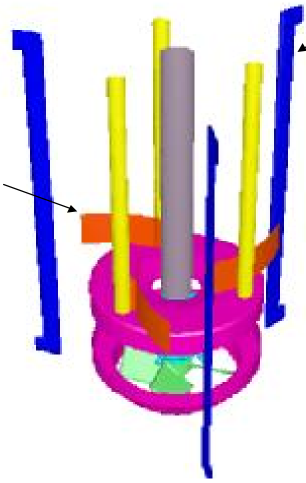


Fig. 12 Agitated tank with baffle and streamline shape guide

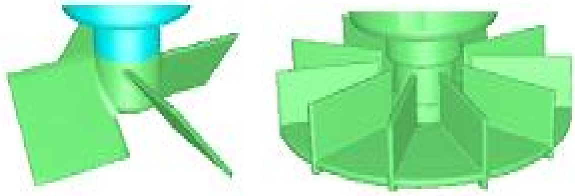


Fig. 13 Axial and radial fan

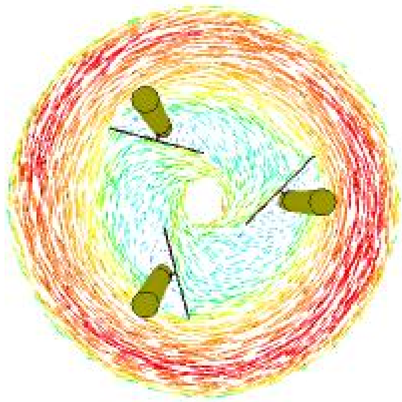


Fig. 14 Velocity vectors at free surface of the original model

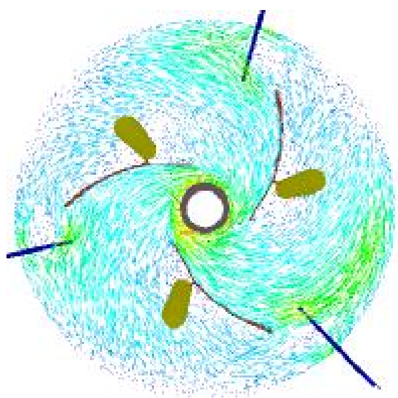


Fig. 15 Velocity vectors at free surface of the improvement model

geometry shape. As the axis rotates, fluid enters into the axial direction of a rotating axis. But the flow near the wall of vessel rotates continuously. Therefore the flow dead zone is generated and its size is not uniform in both the

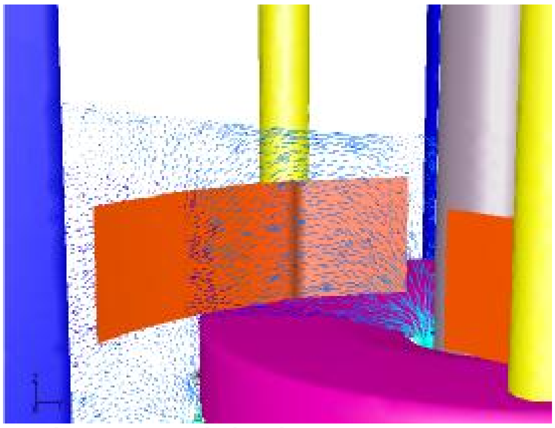


Fig. 16 Velocity vectors at a basket inlet

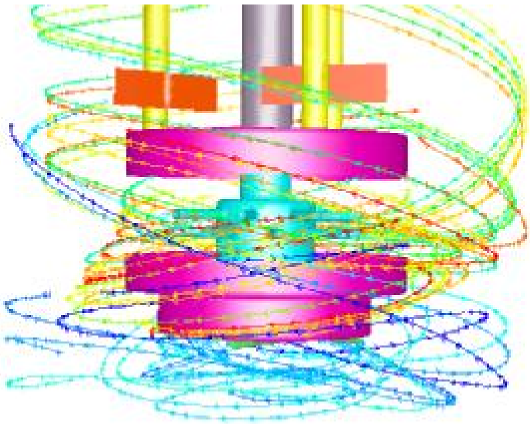


Fig. 17 Streamline distribution for original model

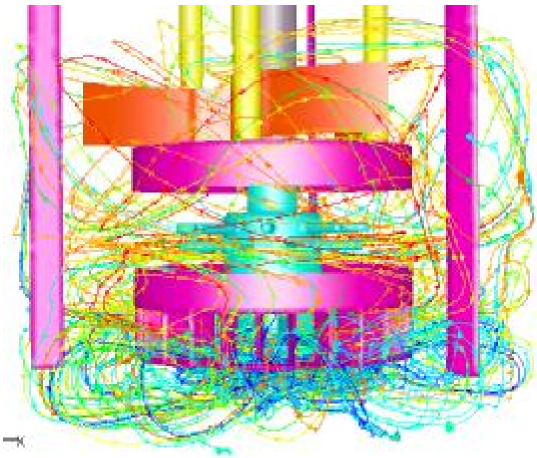


Fig. 18 Streamline distribution for improved model

axial and circumferential directions. In order to minimize the flow dead zone occurring at the wall of a vessel, the baffles are affixed, and the straight guide are exchanged with the streamline shaped guide(Fig. 12). Also, to transform the whole flow fields that have rotating main streams into the flow fields of the up-down pattern, the axial fan was replaced with the radial fan(Fig. 13). The numbers of baffle was accorded with that of guide. Figure 14 is the velocity vectors at free surface of the original model, and figure 15-16 show the velocity vectors when the baffle and the streamline shaped guide are used. Compared with Figure 14, it is verified that the flow dead zone of free surface is completely diminished and the flow into a basket inlet became efficient by the baffle and the streamline shaped guide.

Figure 17 shows the streamline of a system with an axial fan. The direction of main flow is the tangential direction of a rotating axis, and this flow pattern caused a flow dead zone. The flow field with a radial fan is showed in figure 18. A baffle and streamline shape guide are added in this system. Considerable flow patterns are exchanged into the flow fields of the up and down. Actually, the volume flow rate of this system increased about 10% of its original value.

4. CONCLUSIONS

In order to decrease the dead zone and improve the flow efficiency of a system with free surface, this paper presents a new method that overcomes free surface effects by properly combining the benefits of using experiment and 3-D CFD. From the results of the computational fluid analysis, the following conclusions were obtained.

1. The experimental data using the PIV have been patched for simulating the free surface. The results of this simulation method are in good agreement with the experimental results.
2. From the results of three dimensional turbulence flow field analysis for an agitated tank, it was verified that the flow dead zone is decreased and volume flow rate is raised by using a baffle, streamline shape guide and radial fan.
3. By combing the benefits of using the three dimensional CFD and experiment for a complicated turbulent flow field with free surface as an agitated tank, it is shown that the accurate flow analysis is possible.

REFERENCE

[1] C. M. Chew, R. I. Ristic, G. K. Reynolds and R. C. Ooi, Characterisation of Impeller Driven and Oscillatory Mixing by Spatial and Temporal Shear Rate Distribution, Chemical Engineering Science 59, 1557-1568 (2004).

[2] J. Aubin, D. F. Fletcher and C. Xuereb, Modeling Turbulent Flow in Stirred Tanks with CFD: the Influence of the Modeling Approach, Turbulence Model and Numerical Scheme, Experimental Thermal and Fluid Science 28, 431-445 (2004).

[3] A. Brucato, M. Ciofalo, F. Grisafi and G. Micale, Numerical Prediction of Flow Fields in Baffles Stirred Vessels: A Comparison of Alternative Modeling Approaches, Chemical Engineering Science 53, 3653-3684 (1998).

[4] A. K. Sahu and J. B. Joshi, Simulation of Flow in Stirred Vessels with Axial Flow Impellers: Effects of Various Numerical Schemes and Turbulence Model Parameters, Ind. Engineering Chemical Res. 34, 626-639 (1995).

[5] Z. Jaworski, K. N. Dyster, I. P. T. Moore, A. W. Nienow and M. L. Wyszynski, The Use of Angle Resolved LDA Data to Compare Two Difference Turbulence Models Applied to Sliding Mesh CFD Flow Simulation in a Stirred Tank, Recents Proges en Genic des Procedes 11, 187-194 (1997).

[6] P.M. Armenante and C. C. Chou, Velocity Profiles in a Baffled Vessel with Single or Double Pitched Blade Turbines, Journal of AIChE 42, 42-54 (1996).

[7] M. Rahimi and R. Mann, Macro-Mixing, Partial Segregation and 3-D Selectivity Fields Inside a Semi-Batch Stirred Reactor, Chemical Engineering Science 56, 763-769 (2001).
[8] J. Aubin, N. Le Sauze, J. Bertrand, D. F. Fletcher and C. Xuereb, PIV Measurements of Flow in an Aerated Tank Stirred by a Down-and an Up-Pumping Axial Flow Impeller, Experimental Thermal and Fluid Science 28, 447-456 (2004).
[9] Y. S. Lee, H. S. Kim, J. H. Lee, Y. W. Kim and S. H. Ko, Journal of KSME (A), Vol. 27, No. 12, 2875-2886 (2003).
[10] G. A. Bokkers, M. Van Sint Annaland and J. A. M. Kuipers, Transitional Mixing in Multiple-Turbine Agitated Tanks, the Journal of Chemical Engineering 63, 53-58 (1996).