**Computational fluid dynamics analysis of 3D flow patterns in a stirring blade turning space in a wide range of Reynolds number**

T. Hakamada1, R. Misumi\*1, M. Kaminoyama1

*1 Yokohama National University, 79-5, Tokiwadai, Hodogaya-ku, Yokohama-shi, Kanagawa,*

*240-8501, Japan*

*\*misumi-ryuta-zm@ynu.ac.jp*

**Highlights**

* We used CFD to qualify 3D flow patterns in the stirring blade turning space.
* We validated the CFD results from PIV measurements in the blade turning space.
* Effects of viscosity on the flow pattern were ascertained for a wide range of *Re*.

**1. Introduction**

During mixing operations, the flow pattern in the stirring blade turning space affects the circulation flow in the tank. The pattern is linked directly to the success or failure of the operation. However, because it is difficult to measure the flow pattern in the turning space, the flow pattern in the turning space has not been revealed sufficiently. Especially in the transitional regime, the effects of the fluid viscosity, vessel size, and impeller speed on the flow pattern have remained unclear. Computational fluid dynamics (CFD) is a very useful technique to predict fluid flow. However, when using CFD, validation with reliable experimental data is fundamentally important. For this study, the three-dimensional fluid velocity distribution in the turning space of a six-blade paddle in a non-baffled vessel was quantified by CFD from the laminar, transitional, and then to turbulent regime. Furthermore, horizontal two-dimensional particle image velocimetry (2D-PIV) in the turning space was performed in the rotational coordinate system. The CFD analysis was validated based on the PIV data.

**2. Methods**

Software (Fluent 18.0; ANSYS Inc.) was used for CFD analysis. For this study, a non-baffled cylindrical tank with a flat bottom of inner diameter *T* = 0.20 m was used. The blade diameter and width were, respectively, *D* = 0.10 m and *b* = 0.02 m. The rotational speed was *N* = 2 s-1. The fluid depth was *H* = 0.25 m. The blade height was *h* = *H*/2. Water or glycerin aqueous solution was assumed as the working fluid. We performed CFD analysis at *Re* = 24, 177, 511, and 22,400 by changing the glycerin solution viscosity. Laminar, transition-SST, DES, and LES models were adapted for each flow regime. Polyhedral mesh was adapted, and two conditions of 390,000 and 890,000 cell number were prepared. The radial, tangential, and axial velocities in the rotational coordinate system were calculated. Time-averaged velocities were quantified for 4 s under a steady state.

2D-PIV measurements were taken under the same conditions as those used for CFD. The tracer particle motion was captured continuously for 2 s using a high-speed video camera. Fluid velocity distributions in the Cartesian coordinate system were analyzed from 2000 images using PIV software (Flow Expert 2D2C; Kato Koken). Then, the radial velocity *u*r,ro and the tangential velocity *u*θ,ro synchronized with the impeller rotation were calculated using the original MATLAB® program.

**3. Results and discussion**

**3.1 Radial velocity distribution between two blades in the rotational coordinate system**

Fig. 1 shows the relation between time-averaged radial velocity normalized with the blade tip velocity and circumferential angle [rad] at the center height of blade at *Re* = 177. Laminar, transition-SST, and LES were used with 390,000 cells at *Re* = 177. Fig. 2 shows the parity plot of between PIV and CFD by the transition SST model. Fig. 1 and Fig. 2 show the PIV measurements can be predicted most accurately by transition-SST model. Their accuracy is confirmed as approximately ±16%. Similar procedures were used for other *Re* number conditions.

**3.2 Quantification of three-dimensional flow pattern**

Fig. 3 shows the isosurface of = 0.3 of the CFD result in the wide range of *Re*. The isosurface shows the distribution of discharge flow from blades. At *Re* = 24, the fluid is pushed out by the blade and is discharged from the front side of the blade. At *Re* = 177, the fluid is discharged from both the front and back sides of the blade. At *Re* = 511, the discharge flow is mainly induced around the back sides, and at *Re* = 22400, the radial flow velocity from the back sides decreases. In other words, results show that the discharge flow shifts from the front side to the back side of the blade as the flow regime changes from laminar, transitional, and then to turbulence.

**Figure 1.** Relation between / and **Figure 2.** Parity plot of between PIV and CFD

(*Re* = 177, at the blade center height) (*Re* = 177 with transition-SST model)



**Figure 3.** Isosurface of = 0.3 ((a) *Re* = 24, (b) *Re* = 177, (c) *Re* = 511, (d) *Re* = 22400)

**4. Conclusions**

This study quantified three-dimensional flow patterns in the turning space of the stirring blades from laminar, transitional, and to a turbulent regime by CFD. Moreover, to confirm the validity of CFD, the discharge flow velocity distribution in the turning space was quantified using horizontal 2D-PIV. The accuracy of CFD results with various analytical models was evaluated for the respective *Re* number conditions. The most accurate CFD results showed that the discharge flow shifts from the front side to the back side of the blade as the flow regime change from the laminar, transitional, and then to turbulence.

This study was supported by JSPS KAKENHI (Grant Nos. 19K05118 and 17K06886).