CFD Simulation of a Rushton Turbine in a Baffled Tank

G.L. Lane and P.T.L. Koh

CSIRO Division of Minerals, P.O.Box 312, Clayton South, Victoria 3169, Australia.

ABSTRACT

The fluid flow in a baffled tank stirred by a Rushton turbine was simulated using computational fluid dynamics. A time-dependent simulation was carried out using a sliding mesh technique, where the flow domain is divided into an inner rotating domain and an outer stationary domain. This is a fully predictive method which models the impeller directly without recourse to experimental data. Geometry of the tank and operating conditions were chosen to match those for which experimental data is available in the literature. A detailed comparison of the values of the mean velocity components indicates reasonable agreement between the simulation and experimental data. The impeller flow number and power number are also calculated and are found to be in close agreement with values reported in the literature.

NOMENCLATURE

A	area (m^2)
D	impeller diameter (m)
N	impeller rotational speed (s^{-1})
p	pressure (Pa)
P	power (J/s)
r	radius (m)
T	tank diameter (m)
Q	flow rate (m^3/s)
v	radial velocity (m/s)
V	volume (m^3)

Greek symbols

Γ	torque (Nm)
ε	turbulent energy dissipation (m^2/s^3)
ρ	density (kg/ m^3)
τ	shear stress (Pa)

Subscripts

i	control cell number
j	control cell number

1. INTRODUCTION

Impeller-stirred tanks are widely used in the process industries to carry out many different operations. These vessels exhibit complex three-dimensional and periodically unsteady flow, leading to considerable uncertainty in design and scale-up. Advances in computational fluid dynamics (CFD) have resulted in increased interest in numerical simulation of the fluid flow in stirred tanks and CFD studies have been reported in the literature since the 1980s.

The simulation of stirred tanks with baffles is complicated by the relative motion between the impeller and the baffles, so that a single frame of reference is not available for carrying out computations. Various approaches have been taken to account for the effect of the impeller. One typical method has been to model the impeller by prescribing experimentally measured quantities as a boundary condition[1,2], or alternatively, the impeller is modelled as an empirically determined body force[3]. Such methods have several disadvantages: the simulations may only be valid for the conditions under which the impeller data were obtained; experimental data may not be available for a particular impeller; and the simulation method fails to capture the full details of the flow within the impeller and the inherently unsteady flow due to the interaction of impeller blades and baffles.

Recent advances in CFD have led to the development of fully predictive methods for modelling baffled stirred tanks. In several published papers, various methods are demonstrated which directly simulate the impeller geometry, but make a steady-state approximation of the flow. These methods include the "multiple reference frame" method[4,5] and the "snapshot" method[6]. In other papers, a sliding mesh technique is adopted which allows for a time-dependent simulation. This approach has been reported in the literature for the pitched blade turbine[7] and the Rushton turbine[8,9,10]. The time-dependent method is also presented here. The simulation was achieved using the sliding mesh facility incorporated into the CFD code CFX4. The flow field is divided into two domains, an inner domain which rotates with the impeller and an outer domain which is stationary. The calculation procedes by taking into account at each time step the additional velocity component due to the motion of the mesh. At the sliding interface, an interpolation method is used for the conservation of mass and momentum.

This paper presents the simulation results for single-phase liquid flow in a baffled tank stirred by a Rushton turbine. The Rushton turbine, a six-flat-bladed disc impeller, is a commonly used impeller type for which extensive experimental data is available, so that comparisons between simulation and experimental data can be made, so as to assess the accuracy of CFD simulations. The geometry and operating conditions are matched to those of a laboratory tank for which detailed measurements of velocity have been reported[11]. The paper presents a comparison between experimental and simulation results for radial profiles of axial, radial and tangential components of the velocity, for a representative set of axial positions over the height of the tank. The simulation is also used to calculate the impeller flow number and power number.

2. CFD METHOD

The CFX4 package was used to numerically solve the equations for conservation of mass and momentum for an incompressible fluid using a finite volume mesh. Since the operating conditions were turbulent, the exact equations are replaced by the Reynolds-averaged equations. Solution of these equations requires closure by a turbulence model, which in this case was specified as the standard k-ε model[12]. The tank geometry is the same as that used in the simulation by Luo et al.[8], for which Laser Doppler measurements of the velocity components are reported by Hockey[11]. This represents a typical "standard" configuration. The Rushton turbine has six equally spaced blades and there are four equally spaced baffles at the tank wall. The tank diameter, T, is 0.294 m, and other dimensions may be related to T as follows:

Impeller clearance = $1/3T$;
Impeller diameter, $D = 1/3T$;
Blade height = $1/5D$;
Blade length = $1/4D$;
Disc diameter = $3/4D$;
Baffle width = $1/10T$.

For the CFD simulation, this geometry is approximated by a finite volume grid, as shown in Figure 1. A sliding mesh is implemented, where the inner rotating block extends from the centreline to a radius of 0.06 m and the outer stationary block extends from 0.06 m to the wall. Overall, there are 48, 38 and 60 cell divisions in the axial, radial and azimuthal directions respectively (109440 cells), with the grid resolution being finest in the region of the impeller. To reduce the computational time required, a 180° section of the tank is modelled and flow is assumed to be symmetrical for the other half of the tank. The baffles, impeller disc, and impeller blades are treated as zero thickness walls and the impeller shaft is treated as a solid zone. The liquid is specified as water and the impeller rotational speed is 300 rpm, corresponding to a Reynolds number of 48000. The simulation was run as a transient problem with an initial condition of zero velocity at all grid nodes. The simulation was run until the developed flow pattern became periodically repeatable, indicating that a "steady-state" was reached. This was carried out in two stages. In the first stage, an approximate solution is obtained using coarse time steps corresponding to 30° rotation for 10 revolutions of the impeller. The solution was then refined by calculations with steps of 12° for a further 3 revolutions.

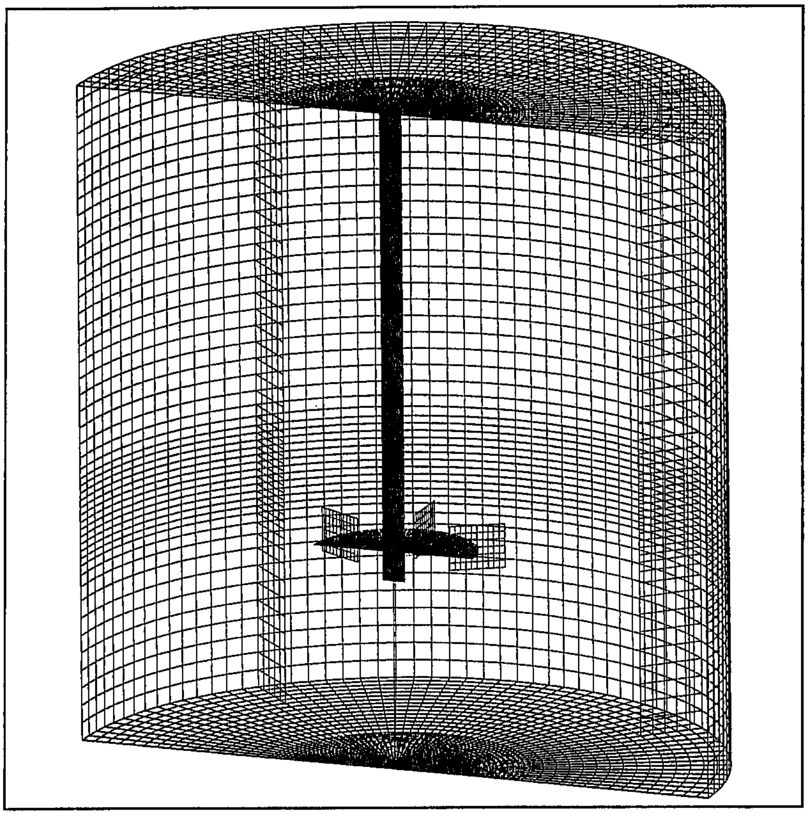

Figure 1. Finite volume grid for stirred tank with Rushton turbine.

3. SIMULATION RESULTS

The results of the simulation are illustrated by Figures 2 and 3. Figure 2 shows the pressure distribution in a horizontal plane through the impeller, showing the regions of high pressure in front and low pressure behind each blade. Figure 3 is a vector plot of the velocities for a vertical cross-section of the tank, through the middle of the tank, half way between baffles and half way between impeller blades. The flow field exhibits the characteristic pattern of a Rushton turbine, with radial discharge from the impeller, which splits into upper and lower circulation zones, with liquid returning axially to the top and bottom of the impeller.

The final solution of the simulation represents an instantaneous solution for a particular orientation of the impeller with respect to the baffles, and velocities vary according to impeller position. For the experimental Laser Doppler measurements, however, the velocity measurements represent a time-average for all impeller positions. To make the simulation results comparable, "time-averaged" mean velocity data were obtained by averaging over 24 positions of the impeller relative to the baffles.

A comparison with experimental data has been made for a range of axial positions over the height of the tank. Figures 4, 5 and 6 show the simulated and experimental radial profiles, from the tank centre to the wall, for the time-averaged axial, radial and tangential velocities at 0.02 m (near the tank bottom), 0.065 m (just below the impeller), 0.12 m (just above the impeller) and 0.18 m (about 2/3 of the tank height). All velocities are in a plane half way between baffles. The tangential velocity components at 0.12 m and 0.18 m were not reported in the experimental data and so comparison is not possible for these cases. Although there are some discrepancies, the results show reasonably good agreement for all velocity profiles.

Another approach to validating the simulation is the calculation of global parameters such as the flow number and power number. The

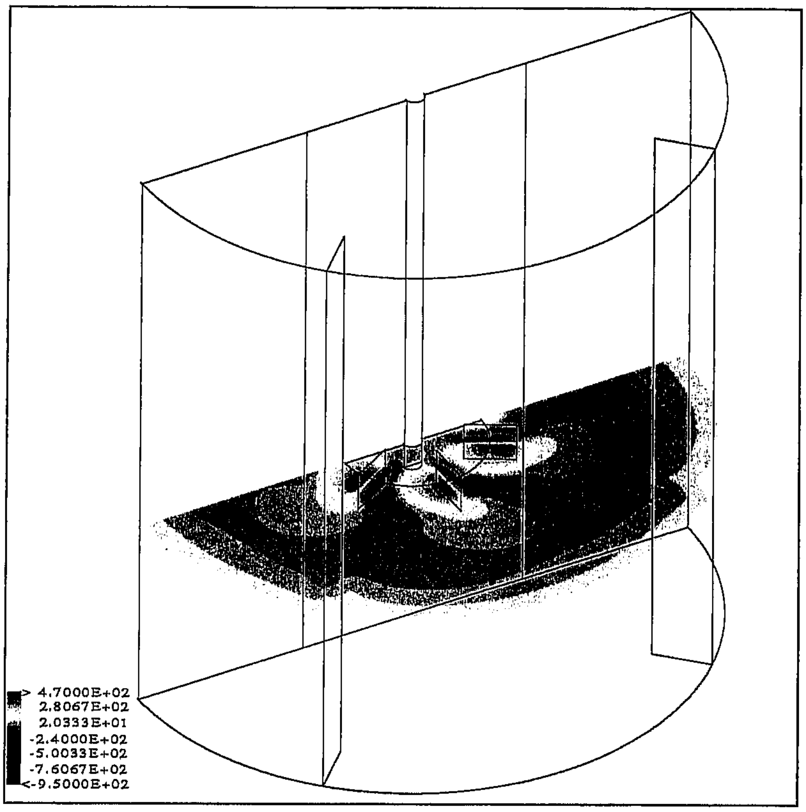

Figure 2. Pressure distribution in the plane of the impeller centre.

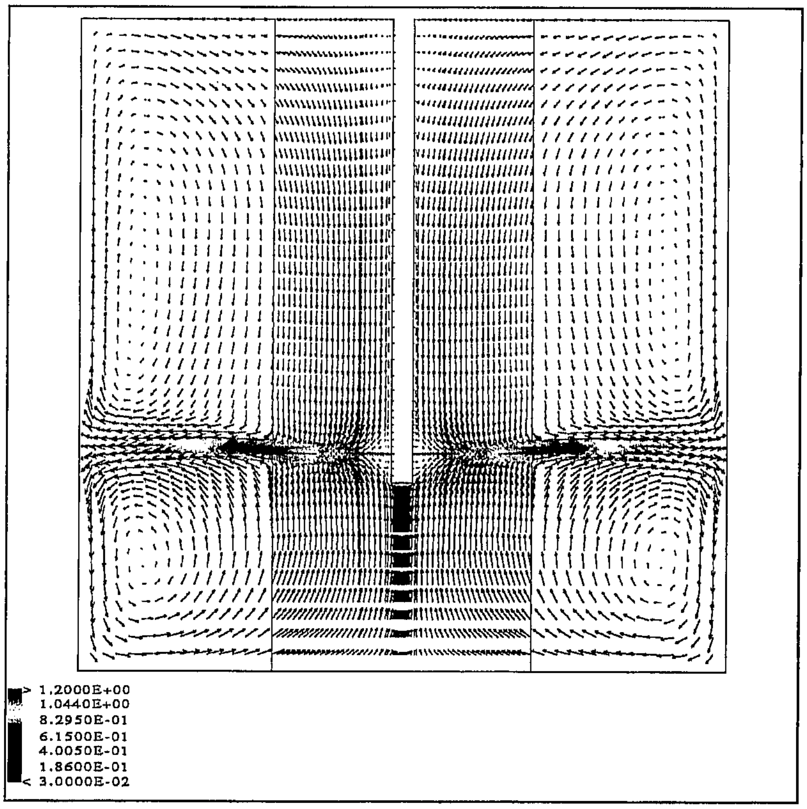

Figure 3. Vertical profile of velocity vectors through centre of tank.

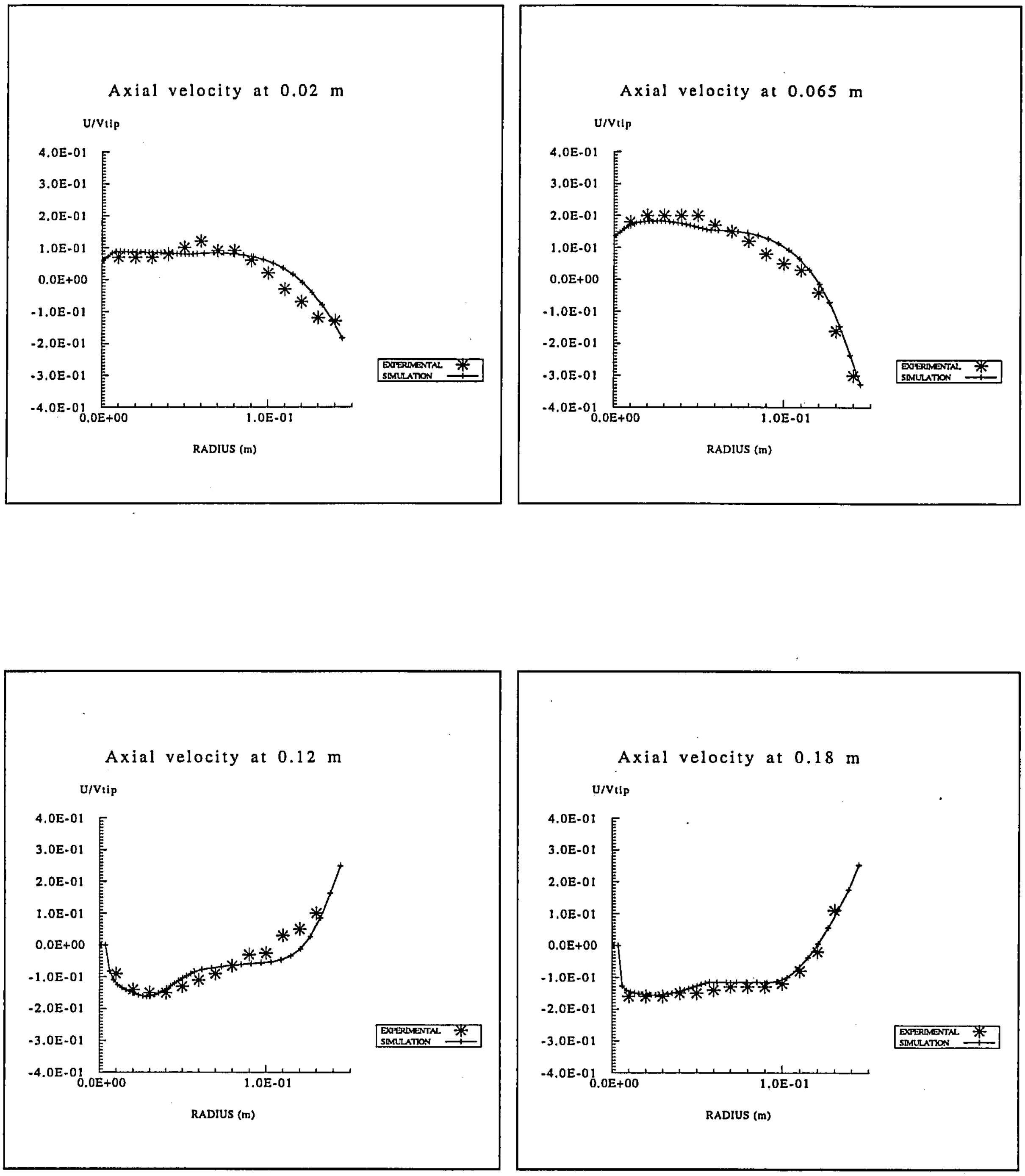

Figure 4: Radial profiles of the time averaged axial velocity component: comparison between simulation and experimental data.

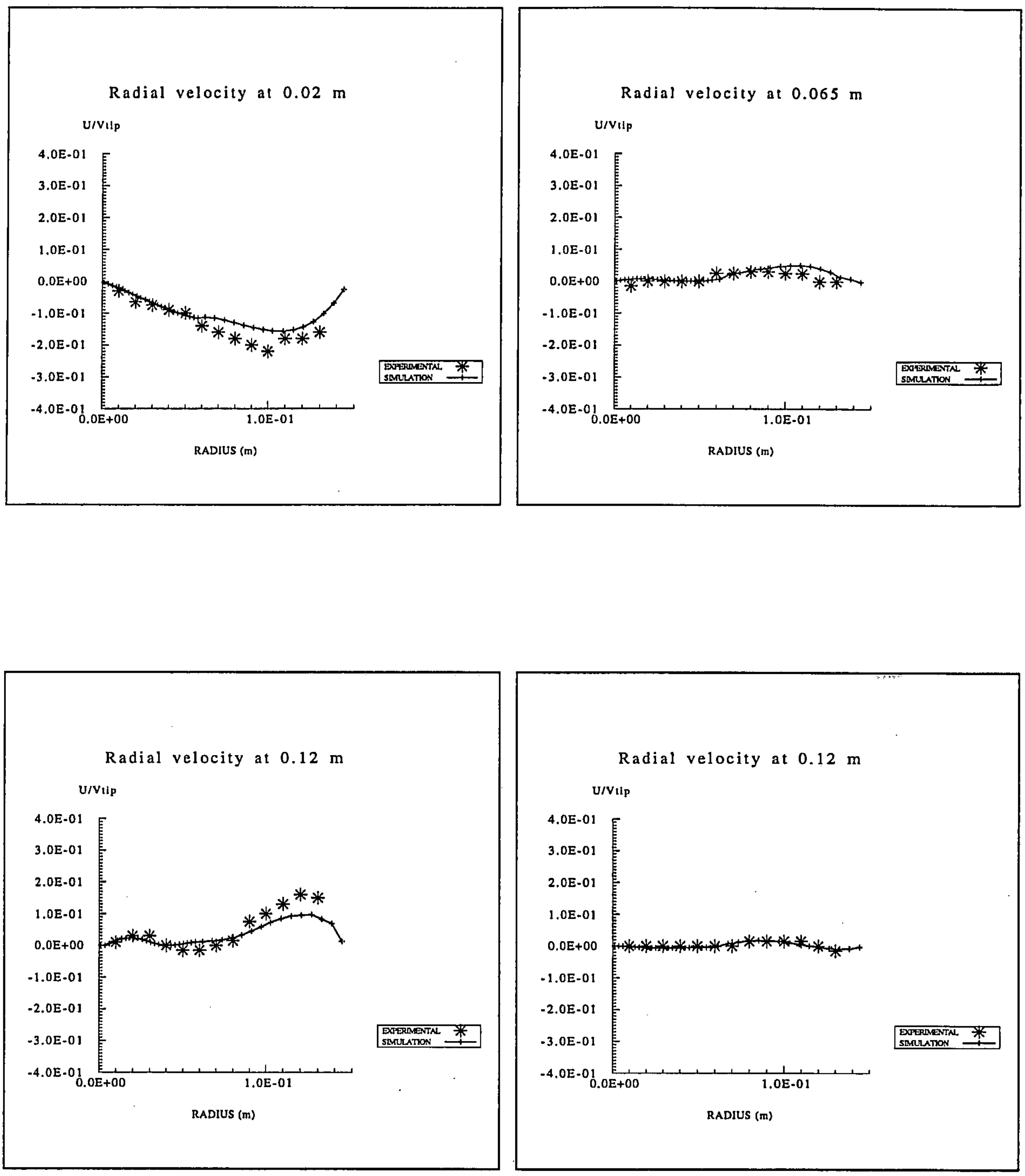

Figure 5: Radial profiles of the time averaged radial velocity component: comparison between simulation and experimental data.

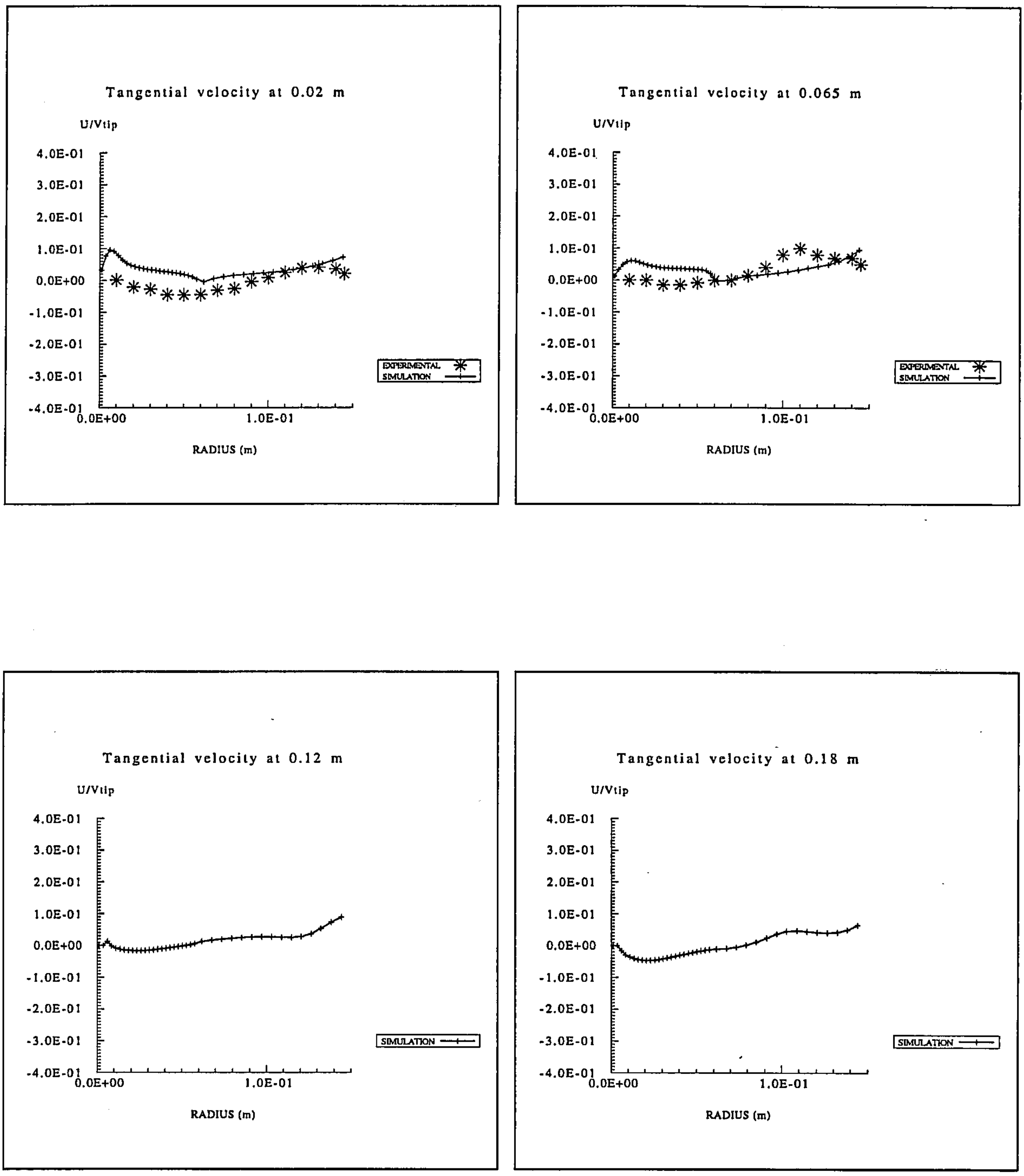

Figure 6: Radial profiles of the time averaged tangential velocity component: comparison between simulation and experimental data.

impeller flow number, N_Q (also known as the pumping number), is defined as:

$$N_Q = \frac{Q}{ND^3}, \qquad (1)$$

where Q is the discharge flow from the impeller. The discharge flow was estimated by calculating a summation over a cylindrical area spanning the width of the impeller blades at a diameter of $1.02D$. The summation is carried out over all the grid cells at this location from one blade to the next, and the total discharge flow, Q, is calculated according to:

$$Q = 6\sum_i v_i\,\delta A_i, \qquad (2)$$

where v_i is the radial velocity at the ith cell node and δA_i is the area of radial flow of the ith cell. Using this method, the estimated flow number, N_Q, is 0.72. This agrees well with the value of 0.73 reported in the literature[13].

The power number, N_P, is defined as:

$$N_P = \frac{P}{\rho N^3 D^5}, \qquad (3)$$

where P is the power input. The power may be calculated from the torque, Γ, on the impeller shaft[10], and this was estimated from the pressure differential on the blades and the shear stress on the disc according to:

$$\Gamma = \sum_i (p_1 - p_2)_i . r_i . \delta A_i + \sum_j \tau_j . r_j . \delta A_j, \qquad (4)$$

where the summation is over the control cells i corresponding to each blade and j corresponding to the disc. Power is then calculated according to:

$$P = 2\pi N \Gamma . \qquad (5)$$

Using this method, the power number is calculated to be 4.5, which agrees reasonably well with the experimentally determined value of 4.67 reported by Hockey[11].

An alternative method of calculating the power is to carry out a summation of the turbulent energy dissipation rate, ε, over the whole tank[6], so that power is estimated as:

$$P = \int \rho\varepsilon\, dV. \qquad (6)$$

However, using this method the calculated power number was only 2.5. The calculation may underestimate the power because ε accounts only for the turbulent eddy dissipation. The value would increase if the viscous dissipation is included in the calculation.

The simulation could possibly be further improved by the use of a finer grid, in particular better resolution in the region of the impeller, and by the use of smaller time increments. The k-ε model is not the most appropriate turbulence model since it assumes that the turbulence is isotropic, whereas it is known that turbulence in stirred tanks is anisotropic. The use of a different turbulence model, such as the Reynolds stress model, could further improve the results of the simulation.

4. CONCLUSIONS

Fluid flow in a baffled tank stirred by a Rushton turbine has been simulated by computational fluid dynamics using a sliding mesh method. The flow domain was divided into an inner rotating and outer stationary domain, and by carrying out computations in a time-dependent manner, the effect of the impeller is computed directly. The tank geometry and operating conditions were chosen to be the same as those for which experimental measurements have been made using Laser Doppler anemometry. A detailed comparison with experimental measurements is presented for time-averaged radial profiles of the axial, radial and tangential velocity components. There is reasonable agreement overall between the simulation and experimental results.

Good agreement is also found between calculated and experimental values of the impeller flow number and power number. Agreement with experimental results would be expected to be improved further by the use of a finer finite volume mesh and smaller time steps. Further improvement might also be obtained by a turbulence model which takes into account the anisotropic nature of the turbulence in stirred tanks.

REFERENCES

1. Bakker, A. & Van den Akker, H.E.A., 1994, "Single-phase flow in stirred reactors", *Trans. I. Chem.E.*, Vol.72, No. A4.

2. Kresta, S.M. & Wood, P.E., 1991, "Prediction of the Three-Dimensional Flow in Stirred Tanks", *AIChEJ*, Vol.37 No.3.

3. Middleton, J.C., Pierce, F., & Lynch, P.M., 1984, "Computation of Flow Fields and Complex Reaction Yields in Turbulent Stirred Reactors and Comparison with Experimental Data", *I.Chem.E. Symposium Series*, Vol. 87.

4. Brucato, A., Ciofalo, M., Grisafi, F. & Micale, G., "Complete Numerical Simulation of Flow Fields in Baffled Stirred Vessels: the Inner-Outer Approach", *I.Chem.E. Symposium Series*, No.136.

5. Luo, J.Y., Issa, R.I. & Gosman, A.D., 1994, "Prediction of Impeller Induced Flows in Mixing Vessels using Multiple Frames of Reference", *I.Chem.E. Symposium Series*, No.136.

6. Ranade, V.V. & Dommeti, S.M.S., 1996, "Computational Snapshot of Flow Generated by Axial Impellers in Baffled Stirred Vessels", *Trans. I. Chem. E.*, Vol. 74, Part A.

7. Bakker, A., Laroche, R.D., Wang, M.H., & Calabrese, R.V., 1997, "Sliding Mesh Simulation of Laminar Flow in Stirred Reactors", *Trans. I. Chem.E.*, Vol.75, Part A.

8. Luo J.Y., Gosman, A.D., Issa, R.I., Middleton, J.C., & Fitzgerald, M.K., 1993, "Full Flow Field Computation of Mixing in Baffled Stirred Vessels", *Trans. I. Chem.E.*, Vol.71, Part A.

9. Tabor, G., Gosman, A.D., & Issa, R.I., 1996, "Numerical Simulation of the Flow in a Mixing Vessel Stirred by a Rushton Turbine", *Fluid Mixing V, I.Chem.E. Symposium Series,* No. 140, pp. 25 - 34.

10. Harvey, A.D. & Rogers, S.E., 1996, "Steady and Unsteady Computation of Impeller-Stirred Reactors", *AIChE Journal,* Vol. 42, No. 10, pp. 2701 - 2712.

11. Hockey, R.M., 1990, *PhD Thesis* (London University).

12. Launder, B.E. & Spalding, D.B., 1974, "The Numerical Computation of Turbulent Flow", *Comp. Meth. in Appl. Mech. & Eng.*, Vol. 3, pp. 269 - 289.

13. Costes, J.& Couderc, J.P., 1988, "Study by Laser Doppler Anemometry of the Turbulent Flow Induced by a Rushton Turbine in a Stirred Tank: Influence of the Size of the Units-I. Mean Flow and Turbulence", *Chem Eng Sci*, Vol.43, No. 10.