Proceedings of the 7th World Congress on Momentum, Heat and Mass Transfer (MHMT'22)
~~Lisbon, Portugal~~ Virtual Conference – April 10 – 12, 2022
Paper No. ICMFHT 135
DOI: 10.11159/icmfht22.135

CFD Simulation of Mixing Tank with Different Rushton Agitator Diameters and Constant Power Consumption

Luiza B. Fernandes[1], José R. Nunhez[2]
[1]Chemical Engineering Faculty/Unicamp
Cidade Universitária Zeferino Vaz, Campinas, Brazil
L162738@dac.unicamp.br; nunhez@unicamp.br
[2]Chemical Engineering Faculty/Unicamp
Cidade Universitária Zeferino Vaz, Campinas, Brazil

Abstract **–** The present work aims to evaluate the discharge flows generated in a 50 L mixing system induced by the rotation of Rushton turbines, which induces radial flow, and to evaluate quality of the CFD simulation of the system. Two agitator diameters were evaluated and compared. The rotation speeds were calculated so that the energy consumption of the two configurations remained the same, and both conditions were secured to be turbulent. The investigation was conducted by URANS equations, which was validated with experimental values reported in the literature for a similar system. Thus, the effects of increasing the diameter of the agitator were compared and evaluated in terms of mean velocities profiles along the blades discharge and in the bulk effects. The k-omega SST turbulence model were in good agreement with the average velocity reported in literature. It was concluded that to increase the diameter of the agitator, at the same energy consumption, led to similar normalized velocities peak values, however it decreases the average absolute velocity reached in the bulk, but it also decreases the variance of the velocity in the medium, which is considered advantageous depending on the system mixing goal.

Keywords**:** turbulent mixing, radial flow, k-omega SST, power consumption, pumping number.

1. Introduction

In mixing processes, Rushton-type agitators are used for generating radial flow in reactors. They are widely used in industry to keep the medium homogeneous or to generate emulsified media, e.g. In many cases, the agitator is positioned vertically in the centre of the stirred tank and four baffles positioned 90° apart on wall serve to prevent the formation of a central vortex with air intake, thus improving mixing efficiency [1]. The flow created by Rushton turbine are reported to create two recirculation zones, one above and another below de agitator, if inspected in a plane cut in z direction [1].

Agitated mixing systems are complex, and many investigations are carried out so that the mixing energy provided by the agitator is better distributed in the bulk and to better understand the flow generated. The present work aims to evaluate the effects of changing the agitator diameter, maintaining the power input constant, and analyse situations in which it may be advantageous to modify the diameter in use, to improve the mixing with no change in the energy consumption.

The system in study is represented in Fig. 1. The tank has a dished end bottom with a curvature of the same radius of the tank diameter (T) and with a circumscribed curvature of 0.1T, as depicted in Fig. 1 (A). The Rushton-type is represented in Fig. 1 (B), it has six blades equally separated by 60°. All the proportions and sizes are in Table 1.

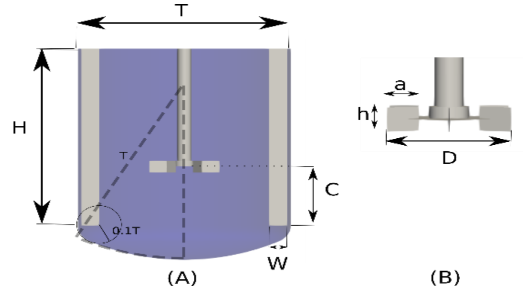


Fig. 1: Schematic drawing of the agitation system.

Table 1: Tank geometry proportions and value of tank diameter T [m].

T	H	C	W	D_1	D_2	A	h
0.38	T	T/3	T/12	T/3	2T/5	D/5	D/4

For validation of the CFD code, the system studied in the article by Wu and Patterson [2] was used for preliminaries simulations. This article [2] presents experimental results of velocity profiles for turbulent agitation system like the one studied here, but in smaller dimensions and with a flat-bottomed tank.

In many works reported in literature that URANS simulations provide good estimates of average velocity in the mixing system [3, 4, 5]. Thus, the k-omega SST turbulence model was used for all the simulations, as seen in the results section, it provided good estimates of average velocities. Also, for further investigation with more robust simulation, such as LES, URANS provides valuable initial values and information.

Changes in the agitator diameter or in the tank dimensions are not much discussed beyond empirical scale-up relationships. One which is used in this work is the relation to keep the power consumption constant, which is, that the diameter of the agitator (D, m) and the rotation speed (N, rev/s) must have the same N^3D^5 product value [6], the rotational speeds N are table at Table 2.

2. Methodology

The geometries described in Fig.1 and the geometry depicted in paper [2] were built in the software program Salome version 9.2.1. All meshes were hexahedral and built in this software. The CFD code was then carried out in the open-source program OpenFoam version 1812 and post processed with Paraview 5.6.0. All softwares are open-source and free available online.

At first, the case described in the article [2] was simulated to validate the code along with the choices on the turbulence model and the boundaries conditions, e.g. The turbulence model used was the k-omega SST. The walls were set to no-slip condition for velocity, wall functions for turbulence parameters and of zero gradient for pressure. At the top of the tank, at the liquid surface, the symmetry boundary condition was chosen to simulate surface with zero shear stress surface [7]. Numerical schemes were all of second order. The simulations were carried out with the sliding mesh technique for the impeller rotation, as reported by to provide good results for time dependent flows [8]. The data reported in this work, in the Results section, corresponds to the average value in time of the flow after it reaches a periodic behavior for velocity value, this periodicity is cause by the blades permanent motion.

Also, to validate the simulation, values such as pumping number (Nq) and power number (Np), which are dimensionless numbers inherent of the agitator in turbulent mixing conditions, were also calculated from the simulation results to compare with the values reported in literature. The pumping number is calculated by Eq. (1) [1, 2, 6], being the volumetric flux Q[m^3/s] a result from the simulation, and the rotation speed N[rev/s] and the agitator diameter D [m] known for each configuration, as seen in Table 2. Experimental values are reported in Wu and Patterson article [2] shown in Fig. 3. To calculate Q during the simulation, an area must be determined in the simulation setup to calculate the flux through it. The areas in the simulation were positioned in agreement with the ones described in the referenced paper [2] in which is reported the Nq experimental results used.

$$\mathrm{Nq} = \frac{Q}{ND^3} \tag{1}$$

And Eq. (2) [1, 6] is used to calculate the power number, in which the torque (τ, [N.m]) is resulted from simulation. As the fluid is water, the density ρ [kg/m^3] is 998 and the kinematic viscosity ν [m^2/s] of 10^{-6}, approximately.

$$\mathrm{Np} = \frac{\tau N}{\rho N^3 D^5} \tag{2}$$

For the Rushton agitator, reported values varies on the value of Np between 5 and 6 [1, 9]. All simulations were held under turbulent conditions, that is, the mixing Re number (Re_{mixing}) should be higher than 10.000 [1, 6], which is calculated by Eq. (3).

$$Re_{mixing} = \frac{ND^2}{\nu} \tag{3}$$

The different simulations were configured as in Table 2. Simulation #0 is the one from the system reported in the article [2] to validate the CFD code, as mentioned previously, and Simulation #1 and #2 are the simulations with the geometry represented in Fig. 1, each one with a respective diameter size (Table 1).

Table 2: Simulation's configurations.

Simulation	Cells in Mesh	D/T	N [rev/s]	Re_{mixing}
#0	1.534.336	1/3	3.33	2.9×10^4
#1	1.996.816	1/3	5.54	8.7×10^4
#2	2.908.732	2/5	4.00	9.3×10^4

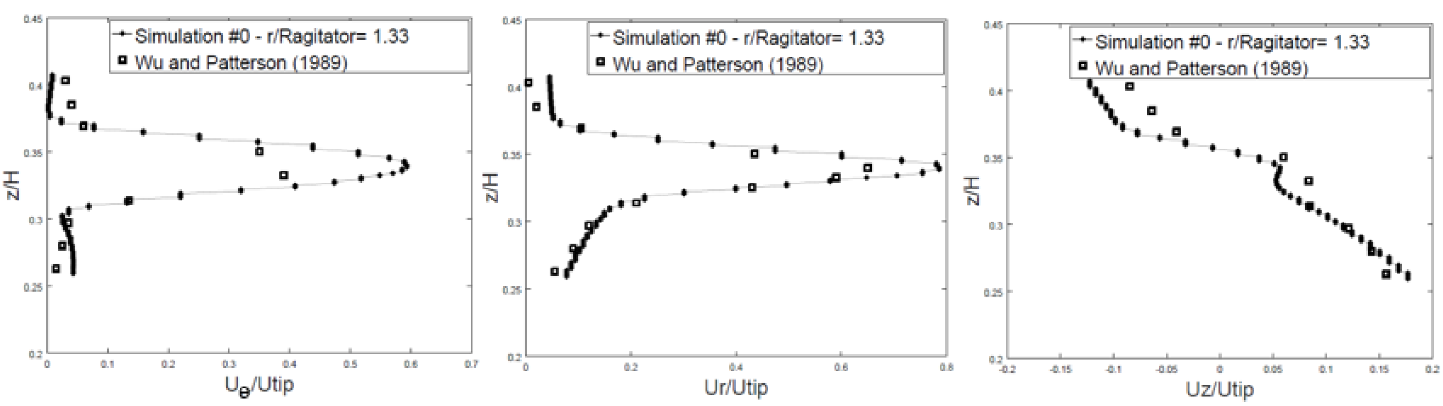
It is important to note that there were simulations conducted to test the mesh independency but will not be discussed here. Further refinement in each mesh did not significantly change the results. Simulation #1 and #2 were set to have the same power consumption, that is, as mentioned, they have the same N^3D^5 product value.

3. Results

The mean velocities components normalized by the velocity at the blade tip are depicted in Fig. 2, at the normalized position r/Ragitator of 1.33, in which Ragitador is half the agitator diameter size of the respective simulation configuration. The velocity at the tip of the blade is calculated by Eq. (4), for Simulation #1 and #2, the values were of 2.17 m/s and 1.91 m/s. The smaller diameter rotates faster with the same energy consumption.

$$\mathrm{Utip} = \pi \mathrm{DN} \tag{4}$$

All the components calculated by the simulation were in good agreement with the experimental result reported by Wu and Patterson (1989) [2]. The CFD code was also good for other r/Ragitator values, such as 1.11, 1.55, 1.71 and 2, not shown. Thus, the present work assumed that the k-omega SST turbulence model with the URANS equations, and the boundary conditions used, were valid and a good choice to simulate mixing mean velocities profiles in similar systems. The results from the simulation, Fig. 2, were averaged in time once the simulation achieved a quasi-permanent behavior in which the velocity calculated became periodic when sampled at a point (specially near the blade motion).



(A) (B) (C)

Fig. 2: Normalized velocities profiles values along the z direction for Simulation #0 and experimental results reported in [2]. (A) tangential velocity; (B) radial velocity; (C) axial velocity.

In Fig. 3, the pumping number calculated from Simulation #1 and #2 were plotted against the value reported by the experimental results of Nq [2], calculated as Eq. (1). It was expected that at each position, the pumping number would agree with the experimental for each configuration simulation, since pump number is inherent of the Rushton turbine. It is realized that both simulations slightly overestimated the pumping number near the agitator discharge position (in dotted circle in Figure 3), however, it presents the same behavior, increasing with the radius and good general agreement with the literature.

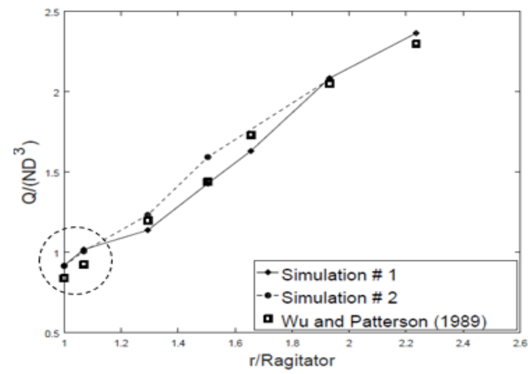


Fig. 3: Pumping number values along the radius for Simulation #1 and #2 and experimental results reported in [2] for Rushton agitator.

The simulated results for power Np, using Eq (2), was of 5.75 and of 5.68, both values agree with the range of 5 to 6 often reported. These simulated values should not be treated as absolute, once the width of the agitator is not represented in the geometry built and it affects the Np value [10], however, they were able to give good predictions of power consumption. Power consumption was of 29,69 W calculated by simulation #1 and of 29,44 W simulation #2.

The results for the velocity profiles for Simulation #1 and #2, geometry in Fig. 1, are resented in Fig.4. Comparing Fig. 2 and Fig.4, it is subtly realizable that, at the same normalized position (z/H and r/Ragitator), the shape of the velocity profile changed in its values and peak position in z direction (peak positions in Fig. 2 are at higher position in z than in Fig. 4). That is assumed to be due the change in the tank geometry in which the flow interaction with the tank walls is different. That is, this paper assumes that the resulting shape of the flow generated by the agitator is dependent also on the interaction with the surrounding walls, though it does not significantly change the results in the studied cases of this paper.

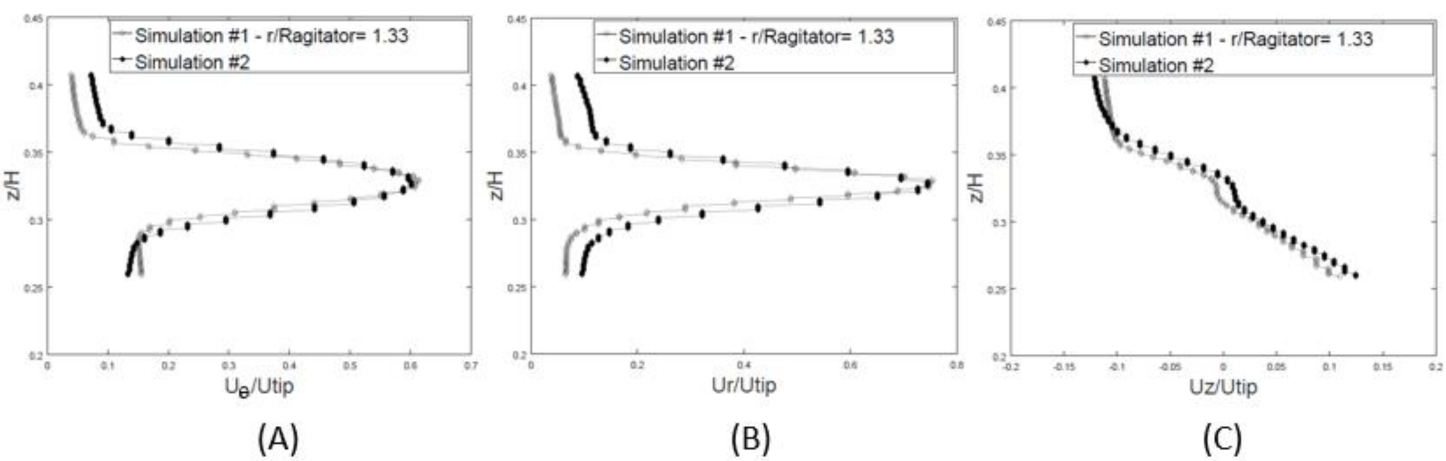


Fig. 4: Normalized velocities profiles values along the radius for Simulation #1 and #2 with different diameters sizes at r/Ragitator= 1.33. (A) tangential velocity; (B) radial velocity; (C) axial velocity.

In Fig.4, it is possible to note that the peak position of the normalized velocity profiles and its value practically remains the same when changing the diameter of the Rushton turbine. That may indicate that the system may be fairly predicted if the system does not change much. However, since the Utip value is smaller in the Simulation #2, the velocities achieved with the bigger diameter are smaller in absolute values. Similar conclusions are valid to others r/Ragitator positions, such as 1.11, 1.55 and 1.71. The velocities profiles at r/Ragitator of 1.71 are shown in Fig. 5.

From Fig. 4 and Fig.5, though it can be concluded that the maximum velocity achieved is smaller with the bigger diameter (same normalized peak value), it is seen that the velocity profile shape in the discharge flow, is more opened, which means that the velocity increases more gradually to its peak. It is observed that the velocity is more equally distributed in Simulation #2.

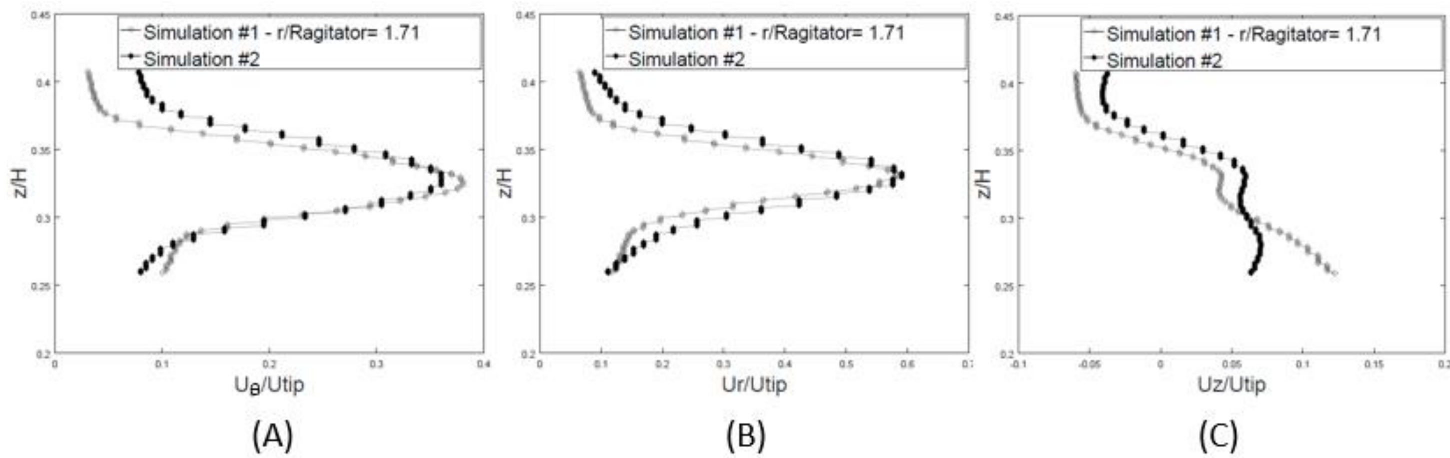


Fig. 5: Normalized velocities profiles values along the radius for Simulation #1 and #2 with different diameters sizes at r/Ragitator= 1.71. (A) tangential velocity; (B) radial velocity; (C) axial velocity.

In fact, by post processing the simulation bulk with statistical tool available in Paraview 5.6.0, the maximum absolute value averaged in time (after achieving quasi-permanent state) in the bulk is 2.0 m/s for Simulation #1 and of 1.8 m/s for #2. Fig. 6 represents the magnitude of velocity calculated, at an instant in time, for Simulation #1 (A) and Simulation #2 (B).

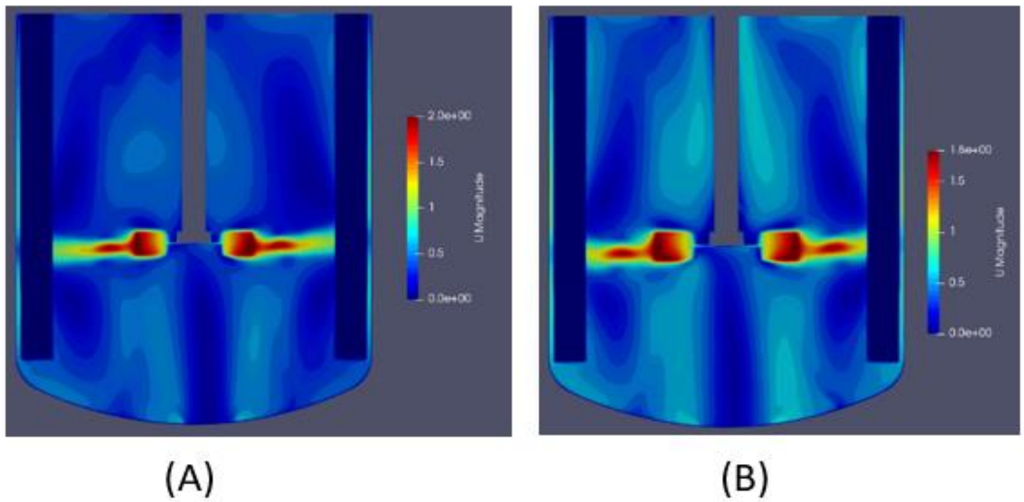


Fig. 6: Bulk velocity magnitude at an instant in time. (A) Simulation #1; (B) Simulation #2.

However, the average in time for standard deviation of the absolute velocity value for Simulation #1 is 15% bigger than the value in Simulation #2, which means, that though with a smaller diameter it reaches higher velocities, the velocity distribution in the bulk is less efficient. That is, the velocity of the bulk is more homogeneous with a higher diameter of agitator, though achieving smaller absolute values. That is, if the bulk velocity is at least above the required for the system, it is easier to control and monitor the Simulation #2 condition.

From Fig. 6, both profiles are as expected by literature: they create a radial discharge flow which is divided into two recirculation zones, one above and one below the agitator position. As discussed, zones with lower velocities are bigger in Simulation #1, with smaller agitator.

4. Conclusion

The CFD code was in good agreement with the reported values in literature, thus concluded to be a good tool to analyse average velocities in mixing systems. Moreover, it was achieved with open-source and free softwares programs available online to analyse velocities profiles in complex system such as mixing stirred tank with relatively flexibility in changing the geometry and the operations configurations. It was concluded that bigger diameter is advantageous when it desirable to achieve more equally distributed velocity in the bulk. The profiles of normalized velocity change slightly when the geometry of tank is changed, and it was assumed that the interactions of the flow generated by the blades of the moving agitator with its surrounding walls are the reason. Velocity profiles remain with peak values and positions constant with the change in diameter which is good for parameterize the system. The CFD code was proved to be a powerful tool to analyse mixing system in general, leading good estimation of power consumption, pumping, and mean bulk and discharge velocity profiles.

References

[1] E. L. Paul, V. A. Atiemo-Obeng and S. M. Kresta, Handbook of Industrial Mixing, Hoboken, New Jersey: JOHN WILEY & SONS, INC., 2004.

[2] H. Wu and G. K. Patterson, "Laser-doppler measurements of turbulent-flow parameters in a stirred tank," *Chemical Engineering Science,* vol. 44, pp. 2207-2221, 1989.

[3] A. Delafosse, A. Line, J. Morchain and P. Guiraud, "LES and URANS simulations of hydrodynamics in mixing tank: Comparison to PIV experiments," *Chemical Engineering Research and Design,* vol. 86, no. 12, pp. 1322-1330, 2008.

[4] D. Deglon and C. Meyer, "CFD modelling of stirred tanks: Numerical considerations," *Minerals Engineering,* vol. 19,

no. 10, pp. 1059-1068, 2006.

[5] J. B. Joshi, N. K. Nere, C. V. Rane, B. N. Murthy, C. S. Mathpati, A. W. Patwardhan and V. V. Ranade, "CFD simulation of stirred tanks: Comparison of turbulence models. Part I: Radial flow impellers," *Canadian Journal of Chemical Engineering,* vol. 89, no. 1, pp. 23-82, 2011.

[6] C. F. J. Junior, E. Cekinski, J. R. Nunhez and L. c. Urenha, Agitação e Mistura na Indústria, LTC, 2007.

[7] F. L. Yang, S. J. Zhou, C. X. Zhang, G. M. Evans and G. C. Wang, "Study of the turbulent flow in an unbaffled stirred tank by Detached eddy simulation," *Chemical Engineering Communications,* vol. 200, no. 10, pp. 1347-1365, 2015.

[8] J. B. Joshi, N. K. Nere, C. V. Rane, B. N. Murthy, C. S. Mathpati, A. W. Patwardhan and V. V. Ranade, "CFD Simulation of Stirred Tanks: Comparison of Turbulence Models (PART II: Axial Flow Impellers, Multiple Impellers and Multiphase Dispersions)," *The Canadian Journal of Chemical Engineering,* vol. 89, no. 4, pp. 754-816, 2011.

[9] J. H. Rushton, E. W. Costich and H. J. Everett, "Power Characteristics of mixing impellers," *Chemical Engineering Progress,* vol. 46, no. 8, pp. 395-404, 1950.

[10] H. Singh, D. F. Fletcher and J. J. Nijdam, "An Assessment of different turbulence models of predicting flow in baffled in a baffled tank stirred with a Rushton turbine," *Chemical Engineering Science,* pp. 5976-5988, 2011.