Criteria for Analyzing the Quality Estimate shake using two different impellers, “Rushton Turbine” and “PBT 45”

2016 J. Phys.: Conf. Ser. 738 012024

(http://iopscience.iop.org/1742-6596/738/1/012024)

Criteria for Analyzing the Quality Estimate shake using two different impellers, “Rushton Turbine” and “PBT 45”

E C M Oliveira[1], M O M Silva[1], J R Nunhez[2] e L C L J Ribeiro[1]
[1] Faculdade de Tecnologia, Universidade Estadual de Campinas – Limeira, Brasil
[2] Faculdade de Engenharia Química, Universidade Estadual de Campinas – Campinas, Brasil.

E-mail: evacarolline.mo@gmail.com, mayarmaia@gmail.com, nunhez@feq.unicamp.br, lubi@ft.unicamp.br.

Abstract. The Computational Fluid Dynamics (CFD) is used to numerically simulate the behavior of fluid flow, and all the laws that rule the study of fluids, both the energy and mass transfer, chemical reactions, hydraulic behavior, and other applications. Among these innumerable applications, the local behavior of a mixing tank can be estimated using the CFD techniques. In order to follow-up studies in the literature giving them increased credibility, this project aimed to determine whether both the scheduling Potency/Volume and Torque/Volume maintain the necessary level of agitation in the mix tank to extrapolate the volume 10 times and 100 times, using an axial impeller type blades inclined at 45 degrees (PBT 45), and a radial impeller (Rushton turbine) with the use of the CFD. With the results, was analyzed if a criterion is superior to the other in terms of maintaining the level of agitation in scheduling. The case study made it possible to achieve the main objective, deepen studies in Computational Fluid Dynamics. Therefore, managed to the largest domain in tools offered by CFX-ANSYS 14.0 software, allowing future work to be conducted.

1. Introduction

Much of the production processes in industries, requires the stirring of the products at any stage. Agitation systems vary depending on the geometry of the tank and the type and number of impellers. In addition to geometry, other important variables are the properties of the fluid and the rotation speed of impeller [3]. These variables are used to calculate the power (P) and torque (Tq) responsible for the operational cost of the process and fixed costs of the tank, respectively. Such factors must be taken into account to provide the best conditions for the process considered and the greatest possible operational savings. In the case of agitation, the torque is the relationship between the power and the speed of rotation of the impellers.

The hardest part in the design of a impeller is the estimate of the size of the engine, what influences sharply in price from a stirring [3], usually this estimate is less when using the torque/volume criterion, however, this does not indicate that the level of agitation is maintained. Agitating systems are estimated from the data of number of power, used typically for turbulent flow, which can be understood as a "coefficient of drag" of impeller. From the use of computational fluid dynamics, this paper proposed studying the scale-up in mixing tanks from P/V criteria and Tq/V. The project allows estimating the power consumed by a mixing tank extrapolated from a tank pilot, analyzing if both escalation methods maintain the level of unrest as the initial tank, checking if a criterion is superior to another in terms of maintaining the level of unrest in the staggering. It permits the decision-making on the operating costs of a process to be carried out with greater reliability.

2. Methodological procedures

It was employed as pilot tank mixing tank with the impeller type PBT 45 [6], corresponding to a square tank of 0.24 m, with an approximate volume of 10.86 liters. The computational modeling was performed to extrapolate the tank 10 times and 100 times. With the same characteristics, was made a tank with the impeller type Rushton turbine, repeating the same extrapolations. Scale-up was carried out in the initial model. From the criterion P/V and Tq/V scaling has been increased by 10 times, using two levels of agitation for the tests, P/V = 0.2 kW/m ^{3} and P/V = 2.0 kW/m ^{3}. In sequence, increased the scale in 100 times, using the same two levels of agitation for the tests doing it to both impeller, PBT 45 and Rushton turbine. Adding to the final twenty simulations with different rotation speeds.

Fluid dynamic behavior of the fluid in turbulent regime mixing tanks was analyzed by developing a three-dimensional model. The numerical results have been obtained through the use of the software CFX-14.0 the ANSYS, was discredited by a mesh that aims at obtaining the variables, such as: speed, temperature, viscosity, and others, at specific points within the entire tank [1]. Here the interest was at inside of the tanks. To simulate the tank was with two domains: rotating and stationary. The stationary domain comprises the geometry of the tank, while the Rotary field is constituted by the geometry of the region around the impeller. It is observed that the geometry of the tank has the space corresponding to the rotation of the impeller domain.

Tetrahedral volumes formats were used and prisms for the creation of the meshes. It was considered the minimum size of the element, the rate of glineth and refinement located mainly in regions near the surfaces of the walls of the tank and of the impeller where if you expect high-speed gradients or turbulent kinetic energy. In figures 1, 2 and 3 it can be observed the meshes created. According to [5], for the great performance of turbulence models one of the key factors is the exact solution of the boundary layer velocity profile. That way, according to [1], one of the methods to control the size of the mesh in the boundary layer is the specification of the dimensionless distance y+ wall and the Reynolds number.

In the definition of the simulation in CFX-Pre, the characteristics about the setup were defined, where was the coupling of different domains. Physical and chemical data were provided, defined the mathematical models employed, as well as what the boundary conditions of the simulation. On Computational solution step with CFX-Solver Manager, during the simulation accompanied by graphics error waste and predefined monitoring points, the answers were generated in files simulation. In the last step, the post processing through the analyses do the results in CFD-Post, if the behavior of the fluid dynamics of the problem at any stage of the solution. Maximum values were rated first and of y+ in the impeller and on the wall of the tank, and then raised the speed profiles at different points in the tank and then analyze and generate the results.

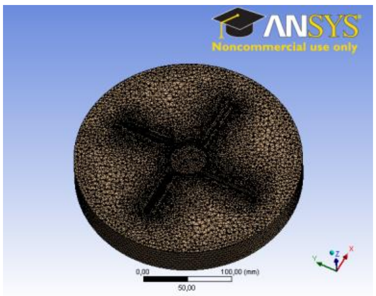

Figure 1. Mesh Impeller with straight blades inclined at 45 degrees (PBT 45).

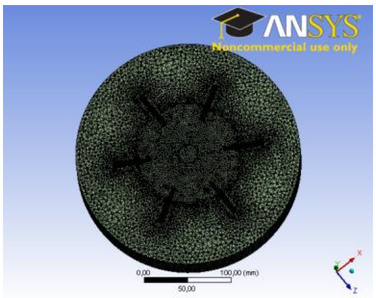

Figure 2. Mesh radial impeller (Rushton turbine).

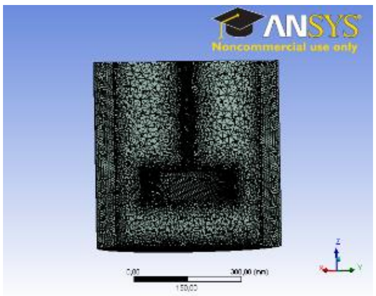

Figure 3. Mesh tank.

Journal of Physics: Conference Series **738** (2016) 012024 doi:10.1088/1742-6596/738/1/012024

3. Results and discussion analysis

The first analysis refers to the validation result in CFD pilot tank with the experimental results. Experimental data was obtained from the University of Alberta -Canada, carried out by researchers Suzanne Kresta and Márcio Bezerra Machado [4]. Figure 4 shows the comparison of the experimental points with the results in CFD with the SST turbulence model used. This graph in Figure 4 demonstrates that the CFD model was able to capture the variations in speed with enough precision, proving the model validation in terms of profiling speed. The dimensionless speed is the relationship of local speed and speed at the end of the impeller.

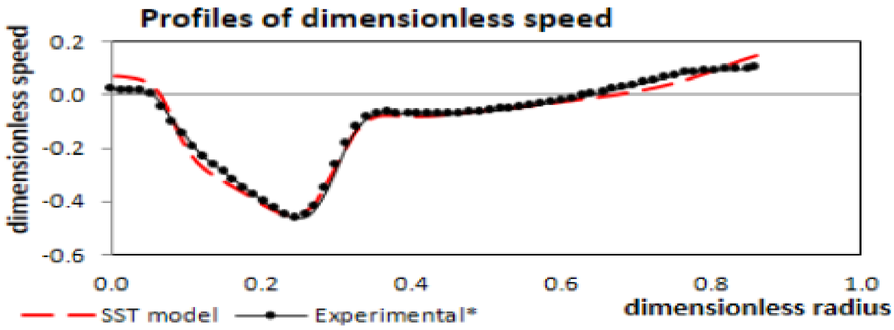

Figure 4. Comparison of the experimental model with the SST turbulence model using the tank pilot [6].

As the tanks grossed in 10 and 100 times the same computational model used and respected the same parameters of the tank pilot, it is considered that the model in 10 times and 100 extrapolated times also represented with details the speed profiles. The graphics were generated from the P/V criterion and the criterion Tq/V. Considering the comparisons to the two levels of agitation, P/V=0.2 $\mathrm{kW/m^3}$ and P/V=2.0 $\mathrm{kW/m^3}$, which are widely used in the chemical industry. In the next sequence analysis was to compare the dimensionless speed profiles in several lines in the tanks, both to the tank pilot, extrapolated in 10 times and the extrapolated in 100 times, ensuring the proportionality of the positions of the lines. In each line, compared the graphics of the tank pilot speed profiles with the extrapolated in 10 and 100 times. If the criterion for extrapolation holds the same conditions of the tank pilot, the three profiles (pilot, 10 and 100 times) must match. These processes were performed also for the two impellers studied, type PBT 45 and Rushton turbine. Figure 5 and 6 show the lines where the results were compared for each of the respective impellers. It is observed that the impeller type Rushton has a line (line 9), the main interest was just to show that at the end of the impeller the speed is greater, closer to the value 1, which was to be expected.

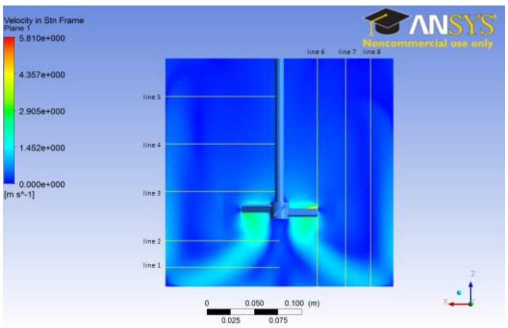

Figure 5. Comparison lines to the tank with the impeller straight blades inclined at 45 degrees (PBT 45).

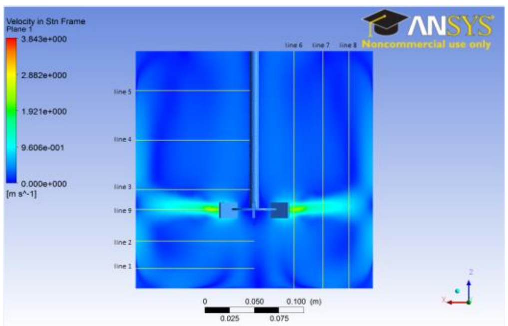

Figure 6. Comparison lines to the tank with the impeller type Rushton turbine.

In all graphs generated compared to each line the variation of dimensionless speed in relation to the dimensionless radius. Generally the speed profiles in the three cases were similar escalation. However, there is a coincidence between the profiles; this is was expected, because you know that is impossible in scale-up keep geometric and kinematic similarity at the same time [2]. Note that the speeds for each tank, despite showing peaks displaced, have similar magnitudes. Therefore, it can be said that the agitation level indicates a similar condition in terms of intensity of agitation. It is observed that both criteria P/V and Tq/V enabled extrapolated very similar conditions. The graphs of figures 7 and 8, for example, prove it.

After comparing both criteria of extrapolation, P/V and Tq/V, notice some variations in the profiles of dimensionless speed between methods, mainly the lines went more of impeller, and near the impeller speeds are greatest. However, in General, both the P/V criterion as Tq/V, kept on extrapolation of dimensionless similar speed profiles.

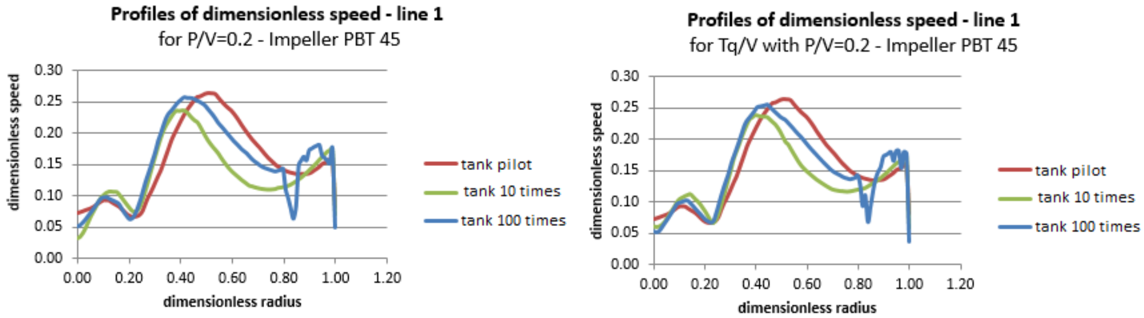

Figure 7. Comparison of dimensionless speed profiles in 1 line (Figure 5) for pilot tanks, extrapolated in 10 times and extrapolated in 100 times, with the impeller straight blades inclined at 45 degrees (PBT 45) and level of agitation for the P/V=0.2 kW/m^3.

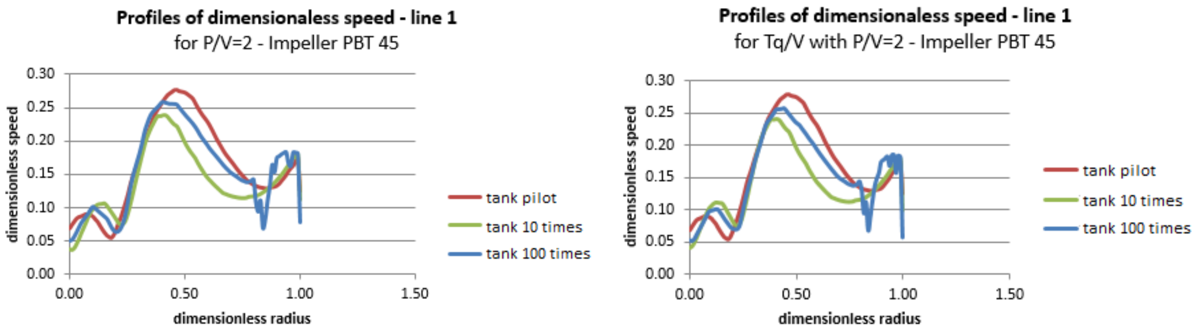

Figure 8. Comparison of dimensionless speed profiles in 1 line (Figure 5) for pilot tanks, extrapolated in 10 times and extrapolated in 100 times, with the impeller straight blades inclined at 45 degrees (PBT 45) and level of agitation for the P/V=2.0 kW/m^3.

4. Conclusion

The analysis done for the tank with the impeller type PBT 45 and tank with the impeller type Rushton turbine is similar, since the conclusions concerning comparisons of two extrapolation criteria studied, P/V and Tq/V, are the same for both types of impellers. The criteria of extrapolation managed to maintain in a similar manner to the level of agitation when extrapolated.

It is concluded that the criterion Tq/V is the most appropriate, since it leads to impellers projects with a lower cost installation, as well as maintenance.

Acknowledgements
The authors gratefully acknowledge the support provided by FAPESP process number 2012/12298-1 and 2012/12296-9 and the PRP/UNICAMP.

References
[1] ANSYS, Inc. User Manual. ANSYS CFX 14.0, 2011.
[2] HASSAN, Y. A. Multiphase Flow Measurements Using Particule Image Velocimetry. The Handbook of Fluid Dynamics, (ed.) Johnson, R. W., Press- Springer, Boca Raton FL, 1998.
[3] JOAQUIM JUNIOR, C. F.; CEKINSKI, E.; NUNHEZ, J. R.; URENHA, L. C. Agitação e mistura na indústria. LTC. Rio de Janeiro, 2007.
[4] MACHADO, M. B.; NUNHEZ,J. R.; NOBES, D.; KRESTA, S. M. Impeller characterization and selection: Balancing efficient hydrodynamics with process mixing requirements. AIChE Journal, Volume 58, ed 8, 2012, p. 2573-2588.
[5] SPOGIS, N. Metodologia para determinação de curvas de potência e fluxos característicos para impelidores axiais, radiais e tangenciais utilizando a fluidodinâmica computacional. Campinas, SP, 2002.
[6] VERGEL, J. L. G. Estudo da influência da malha computacional, modelos de turbulência e aspectos numéricos da modelagem CFD em impelidores PBT usando malhas não-estruturadas. Campinas, SP, 2013.