JSCS-info@shd.org.rs • www.shd.org.rs/JSCS

J. Serb. Chem. Soc. *90* (11) 1369–1382 (2025) *Original scientific paper*
JSCS–5459 Published 13 November 2025

A CFD investigation of the performance of stirred tanks

KAMLA YOUCEF[1]*, ZIED DRISS[2], MOHAMMED FOUKRACH[1], SERGIO ROSA[3] and TOUHAMI BAKI[4]

[1]Laboratory of Mechanics and Energy, Faculty of Technology, University Hassiba Ben Bouali of Chlef (UHBC), Algeria, [2]Laboratory of Electro-Mechanic Systems (LASEM), National School of Engineers of Sfax (ENIS), University of Sfax (US), Tunisia, [3]Polytechnic Institute of Bragança (IPB), Santa Apolonia Campus, Portugal and [4]Mechanical Faculty, Gaseous Fuels and Environment Laboratory, University of Sciences and Technology of Oran Mohamed Boudiaf (USTO-MB), BP 1505, El-M'Naouer, Oran 31000, Algeria

(Received 15 April, revised 2 June, accepted 12 August 2025)

Abstract: Rushton turbine was employed in this study to numerically analyze the fluid flow it generates within a stirred tank. The topology of the resulting flow was found to be strongly dependent on several parameters, including the geometric configurations of the system and the properties of the moving fluids. The governing equations, based on the k–ε model, were solved using the finite volume method. Velocity field profiles, streamlines and vortex sizes were analyzed for several geometries, varying the number of blades from 6 to 12 and others. A comparison was also conducted to evaluate the effect of the number of stirring mobiles used to mix the fluid (single stage, two stages and three stages), as well as the influence of the spacing ratio between the different stirrers. Finally, our numerical simulation procedure was validated through comparing the results obtained with experimental work available in the literature, showing good agreement between the different approaches.

Keywords: Rushton turbine; turbulent flow field; power number; mixing; simulation.

INTRODUCTION

Mechanical agitation is a process used in many industrial sectors; its field of application is extremely wide. The choice of the stirring system depends on the operation to be performed (*e.g.*, homogenization, heat transfer, dispersion of a gaseous phase in a liquid or emulsion, *etc.*). For this reason, many researchers have focused on optimizing mixing processes and reducing energy consumption. By combining experimental studies and numerical modeling, they seek to improve

* Corresponding author. E-mail: y.kamla@univ-uhbc.dz
https://doi.org/10.2298/JSC250415066Y

mixing performance while limiting energy consumption, an essential aspect for ensuring the sustainability of industrial processes.

The aim of this work is to improve the mixing efficiency of a tank agitated by a Rushton turbine by modifying important geometric parameters, such as the number of blades and the spacing between impellers. The objective is to identify the most effective configuration that promotes vortex formation without disturbing the free surface, while significantly reducing energy consumption compared to the standard geometry.

As shown in the study by Driss,[1] the inclination angle of the blades significantly affects energy consumption, with lower angles resulting in reduced power numbers. These results provide clear evidence that mixers which generate a mixed flow (inclined blades: 45, 75 and 60°) consume less energy than mixers that generate a radial flow (90°), which can be explained by the character of the fluid generated by the mobile and the contact surface between the stirred system and the fluid. According to investigations conducted by Li *et al*. and Cui *et al.*,[2,3] the effect of blade pitch on the recirculation zone structure and power consumption was examined, revealing that power consumption increases as the blade pitch angle becomes larger. As reported by Jaszczur *et al.*,[4] an experimental and numerical study analyzed the effect of stirrer mobile design on mixing efficiency. The analysis was mainly based on the distribution of water in the stirred tank. Three configurations were studied: one with a tank fitted with a six-bladed Rushton turbine, and the other two mixers designed as scaled-down versions of a standard industrial mixer. In the numerical study by Yang,[5] the geometry of a two--blade stirrer was examined for its impact on flow distribution and power requirements required for mixing an incompressible Newtonian fluid in the laminar regime. The results showed that the new shapes of stirrers exhibit characteristics similar to turbines, generating not only tangential but also increased axial velocities, which lead to the formation of secondary flows around the blades. In another simulation by Liangchao *et al.*,[6] the hydrodynamic characteristics of the fluid in a tank stirred by a Rushton turbine in the laminar regime were investigated. It was found that the power number decreases when the impeller is larger. Furthermore, in a baffled tank with a small impeller, the power number was almost identical to that in a tank without baffles. As shown in the experimental study conducted by Kordas *et al.*,[7] the influence of a new stirrer on mixing time was assessed. It was found that the type of motion (rotary or reciprocating) significantly affected mixing time and energy consumption. The geometry of an anchor-type stirrer was also analyzed with respect to mixing improvement. In a comparative study by Ameur *et al.* and Kamla *et al.*,[8,9] three modified impeller geometries were evaluated to improve fluid circulation throughout the tank, particularly in the lower part. The proposed configurations were also aimed at minimizing energy consumption. In the study carried out by Dang *et al.*,[10] the geometry of a two-blade stirrer was analyzed using numerical simulation to determine its effect on flow distribution and the power required to mix an incompressible Newtonian fluid in the laminar regime. The stirrer shapes demonstrated turbine-like behavior, enhancing both tangential and axial velocities, leading to the formation of secondary flows around the blades. As reported in the experiments by Youcefi,[11] the mass transfer performance of the cup blade mixer was evaluated and found to be superior to that of the 45° blade and the Rushton turbine. Additionally, the design of this new stirrer was shown to enable efficient mixing in multiphase flow systems. In the studies by Bonnot *et al.*, Hachemi and Yang,[12–14] the effects of rotation mode, impeller inner diameter, gas flow rate and viscosity were examined in a vessel agitated by a Rushton and anchor-type coaxial stirrer. The findings demonstrated that the co-rotation mode provided an advantage in reducing energy consumption compared to counter-rotation. According to research using particle image velocimetry (PIV) by Mortensen *et al.*,[15] the effect of stirrer slot width on flow and velocity profiles in a key mixing zone was explored. The study showed that flow decreases as slot width increases, and the velocity profile and reflux proportion vary with the number of flows, an essential parameter for describing the hydrodynamics of batch SMR. Finally, the studies summarized in the following table. Mahmoudi *et al.* and Rutherford[16,17] investigated the effect of wheel clearance on the transition between flow regimes in single and dual Rushton turbine systems. It was also found that different combinations of $C1$ and $C2$ values led to parallel, merging, and diverging flow regimes. In this work, $C1$ represents the distance between the tank bottom and the impeller, while $C2$ represents the distance between the impellers. A Rushton-type impeller was used in a tank with a diameter T (Table I).

TABLE I. Summary of the range of clearances corresponding to steady flow development in the literature

Flow model	Space between the agitators	Experimental date
Parallel flow	$C1 > 0.19T$, $\Delta C > 0.4T$	1992.[16]
	$C1 > 0.20T$, $\Delta C > 0.385T$	1996.[17]
	$C2 > 0.415T$	
Merging flow	$C1 > 0.17T$, $\Delta C < 0.34T$	1992.[16]
	$C1 > 0.20T$, $\Delta C < 0.385T$	1996.[17]
diverging flow	$C1 < 0.17 – 0.19T$	1992.[16]
	$C1 < 0.17T$, $\Delta C > 0.385T$	1996.[17]
Single loop flow	$C1 < 0.167T$	1997.[18]
pattern	$C1 < 0.15 – 0.20T$	1992.[19]
		2009.[20]

EXPERIMENTAL

Mathematical formulation

The basic equations used in the simulation of stirred vessels are the continuity equation and the momentum equations (Navier–Stokes). These equations describe the flow behavior of a fluid in the vessel and are used to analyze the hydrodynamics generated by the stirrer.

Continuity equation (conservation of mass). The continuity equation expresses the conservation of mass in an incompressible flow:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0 \tag{1}$$

Momentum equation. The Navier–Stokes equations describe the conservation of momentum taking into account the forces acting on the fluid (such as viscous forces, pressure and external forces). The governing equation is given as follows:

$$\rho(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z})=-\frac{\partial p}{\partial x}+\mu(u\frac{\partial^2 u}{\partial x^2}+v\frac{\partial^2 u}{\partial y^2}+w\frac{\partial^2 u}{\partial z^2}) \tag{2}$$

$$\rho(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z})=-\frac{\partial p}{\partial y}+\mu(u\frac{\partial^2 v}{\partial x^2}+v\frac{\partial^2 v}{\partial y^2}+w\frac{\partial^2 v}{\partial z^2}) \tag{3}$$

$$\rho(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z})=-\frac{\partial p}{\partial z}+\mu(u\frac{\partial^2 w}{\partial x^2}+v\frac{\partial^2 w}{\partial y^2}+w\frac{\partial^2 w}{\partial z^2}) \tag{4}$$

Power number. The power number, also known as the power coefficient, is a dimensionless parameter that characterizes the energy consumption of a stirrer in a stirred tank. It is defined by the following equation:

$$Np=\frac{P}{\rho N^3 D^5} \tag{5}$$

where P denotes the power consumption of the stirred system, which can be determined using the viscous dissipation function and is given by the following formula:

$$P=\mu\iiint Q_{\mathrm{v}}\,\mathrm{d}x\mathrm{d}y\mathrm{d}z \tag{6}$$

where Q_{v} represents the viscous dissipation function, which is calculated based on the fluid flow velocities (u, v, w) inside the stirred vessel. The volumetric flow rate (Qv) used in this study is based on the formulation presented by Hadjeb *et al.*[21] The Reynolds number (Re) is a dimensionless parameter used to determine the flow regime (laminar, transitional or turbulent) in a stirred tank. It is defined by the formula:

$$\mathrm{Re}=\frac{ND^2\rho}{\mu} \tag{7}$$

Mixing system description

Fig. 1 shows the geometric configuration of the stirring system, which consists of a flat-bottomed cylindrical tank of diameter D and height H. The mechanical stirring system is equipped with a six-bladed Rushton turbine mounted on a disk of diameter d. To avoid contact with the tank bottom, the stirrer is positioned at a distance c and mounted on a central cylindrical shaft of diameter da. The fluid used in this study is water at 25 °C, which behaves as a Newtonian fluid with a density of ρ = 997 kg m^{-3} and a dynamic viscosity of μ = 8.9×10^{-4} Pa s. In this study, the effect of the spacing between the stirring mobiles was analysed. The different spacing configurations used are shown in Fig. 1 and Table II.

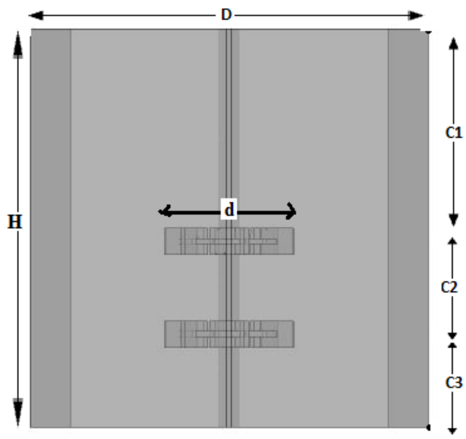


Fig. 1. Geometric configuration.

TABLE II. The spacing *C* values between stirrers for different geometries

Case	*C1*	*C2*	*C3*
Case 1	0.45*D*	0.70*D*	0.35*D*
Case 2	0.80*D*	0.35*D*	0.35*D*
Case 3	0.45*D*	0.35*D*	0.70*D*
Standard turbine	–	–	0.50*D*

The multiple reference frame (MRF) approach was used in this part of this research. It is commonly used to simulate flows in mechanically stirred tanks, as demonstrated by the works of Dabiri *et al.*[22] and more recently by Reid *et al.*[23] It allows us to model agitator motion without the need for a moving mesh. The principle behind the MRF method is to divide the simulation domain into two distinct zones. A moving (rotating) zone corresponding to the region around the stirrer, where the Navier–Stokes equations are solved in a reference frame rotating at the same speed as the stirrer system and a fixed (static) zone representing the rest of the tank, where the equations are solved in a stationary reference frame. At the interfaces between these zones, coupling conditions ensure the continuity of physical quantities such as velocity and pressure. This approach is particularly well suited for steady-state and turbulent studies and provides a good compromise between accuracy and computational efficiency.

RESULTS AND DISCUSSION

In order to evaluate the simulation method used, we relied on the experimental work of Wu *et al.*[24] and Driss *et al.*[25] These studies served as a reference and it should be noted that the same geometric conditions, as well as the same fluid, were reproduced during the simulation, using a six-blade Rushton turbine with vertical baffles symmetrically positioned to enhance mixing performance. The numerical results obtained for tangential velocity (Fig. 2) and radial velocity (Fig. 3) were compared with experimental data from the literature and showed satisfactory agreement.

The dimensionless quantities used in these comparisons are defined as follows: *Vt** is the dimensionless tangential velocity, *Vx** is the dimensionless axial velocity, *Vr** is the dimensionless radial velocity and *Z** is the dimensionless liquid height in the tank.

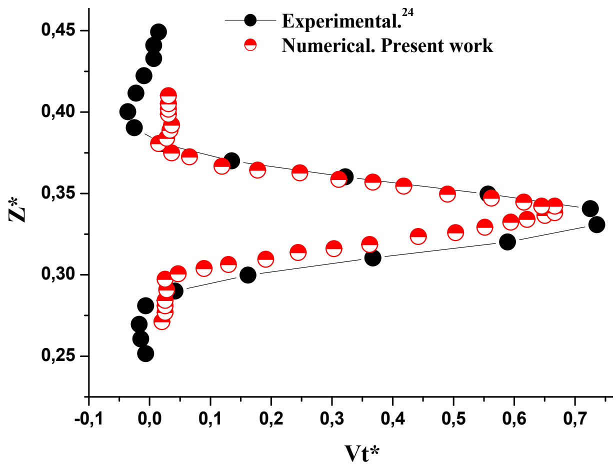


Fig. 2. Tangential velocities (Re = 40000).

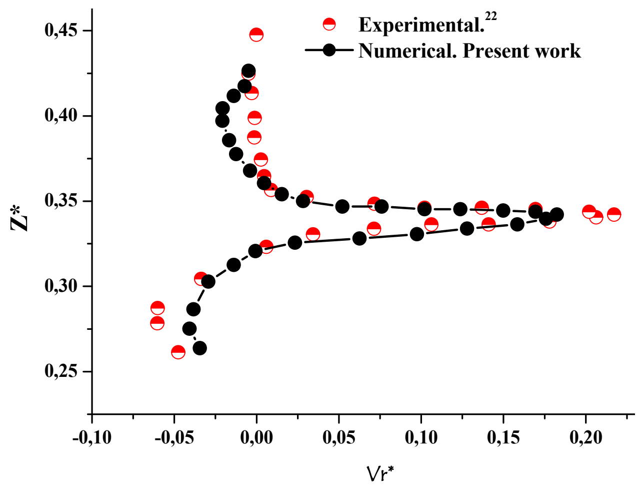


Fig. 3. Radial velocities (Re = 17000).

Effect of the number of blades on the flow generated

Figs. 4 and 5 show axial velocity profiles for different numbers of blades. It can be observed that the velocities are more intense when the agitator has a high number of blades (*e.g.*, 12 blades). In this case, the fluid motion is more intense, resulting in instability of the free fluid surface. On the other hand, for a 3-blade

Rushton turbine, the axial velocity in the free fluid surface is null, which explains why the number of blades has a positive influence on the stream flow. Negative velocity signs indicate the presence of vortices above and below the impeller.

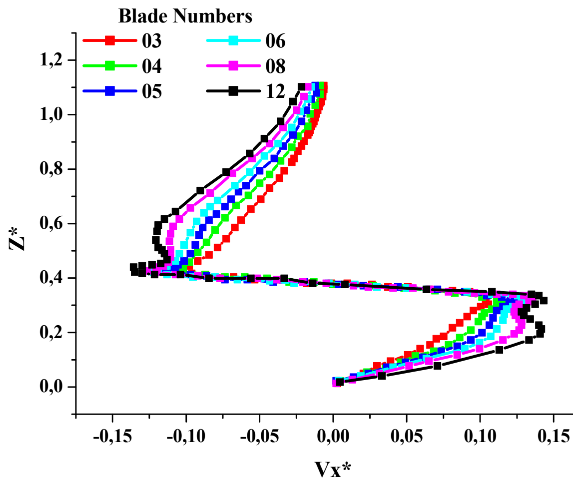


Fig. 4. Axial velocities for different numbers of blades (Re = 40000, X^* = 0.185).

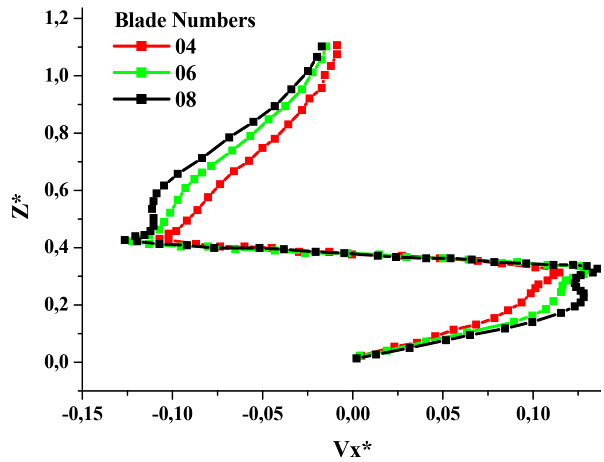


Fig. 5. Axial velocities for different numbers of blades (Re = 40000, X^* = 0.185).

Fig. 6 shows the tangential velocities for different numbers of blades. At the bottom of the tank, the fluid elements stagnate due to the wall effect at this location, which reduces the development of the flow. On the other hand, at the agitator, these velocities reach maximum values due to the increase in the number of blades, which leads to an increase in the discharge flow generated by the agitator blade. These phenomena are also illustrated in Fig. 7.

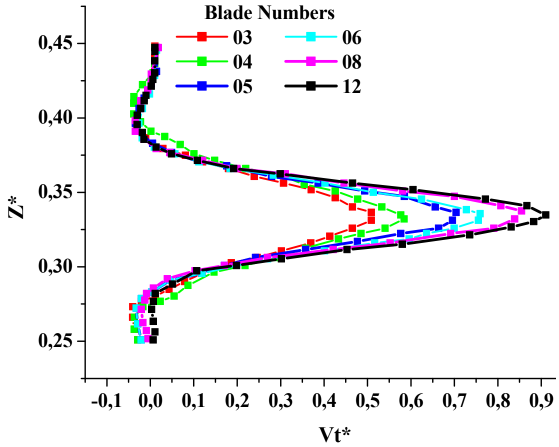
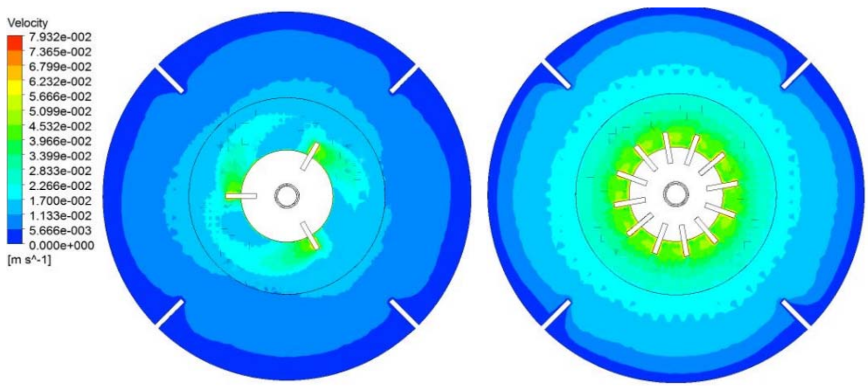


Fig. 6. Tangential velocities for different numbers of blades (Re = 40000, X^* = 0.185, baffled tank).



3 blades 12 blades

Fig. 7. Velocity champs (Re = 40000, baffled tank).

As presented in Table III, the numerically calculated power number for a six-bladed Rushton-type stirrer is 0.82 for a six-bladed Rushton-type stirrer with an unbaffled vessel and 6.72 with a baffled vessel. These results were compared with those found numerically and experimentally in the literature. The values are almost identical to those obtained in other experimental and numerical studies, making it possible to determine the effect of the number of blades on energy consumption.

Fig. 8 shows that the power number increases with increase of the number of blades, which can be explained by the increased contact surface between the stirrer and the fluid and also by the greater interaction between the radial flow generated by the stirrer and the side wall of the tank, resulting in a higher power requirement. In addition, the presence of baffles improves mixing quality, but also results in an 87 % increase in power consumption compared to an unbaffled tank (Fig. 8). This is because the baffles act as an obstacle to the flow and change its direction, which is mainly tangential in a tank without baffles.

TABLE III. Power number for Rushton turbine with and without baffles

Type	Present work	Exp.[24]	Num.[26]	Num.[27]	Exp.[28]
Baffle tank	6.72	6.07	5.40	–	6.0
Unbaffled tank	0.82	–	–	1.2	0.85

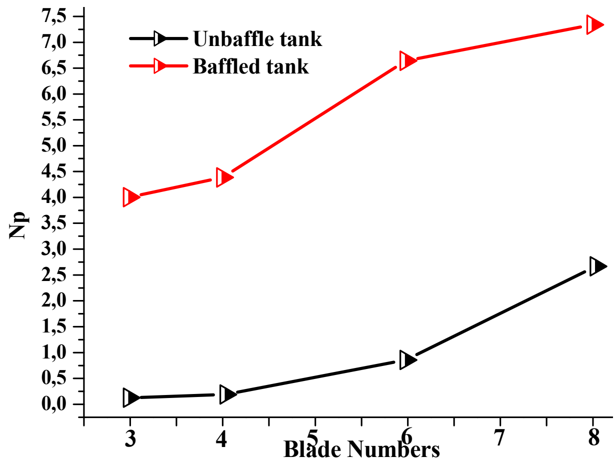


Fig. 8. Power number for different numbers of blades (Re = 40000).

Effect of the number of agitators on the flow rate

Figs. 9 and 10 illustrate the tangential and axial velocities of tanks with various stage numbers. As can be seen, the volume of fluid agitated is greater when there are three stages. Consequently, the intensity of the agitated fluid increases with the number of stirrers used. Figs. 9 and 10 illustrate the effect of the number of stirrers and Table IV provides the geometric dimensions used in the simulations. All configurations use the same type of stirrer, *i.e.*, a Rushton turbine with six blades mounted in a tank with vertical baffles. Table IV also indicates the distances between the stirrers for the multi-stage configurations. The standard geometry consists of a single Rushton turbine and was used in the validation phase. These results confirm that increasing the number of agitators enhances axial and tangential circulation, leading to better homogenization across the tank volume. Table IV.

Fig. 11 (a, b and c) shows the effect of the number of agitator stages on the induced flow in the tank. It can be seen that the tank equipped with three agitators has a larger volume of the agitated zone than the other configurations. However, this improvement is accompanied by an increase in energy consumption.

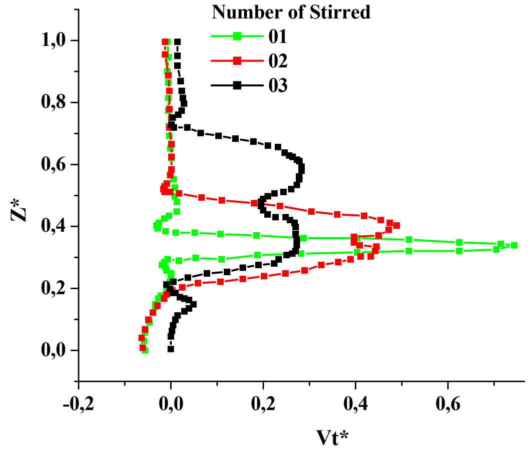


Fig. 9. Tangential velocity for different number of stirrers, Re = 40000.

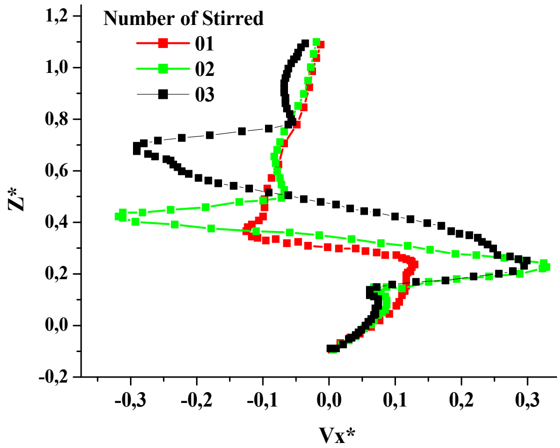
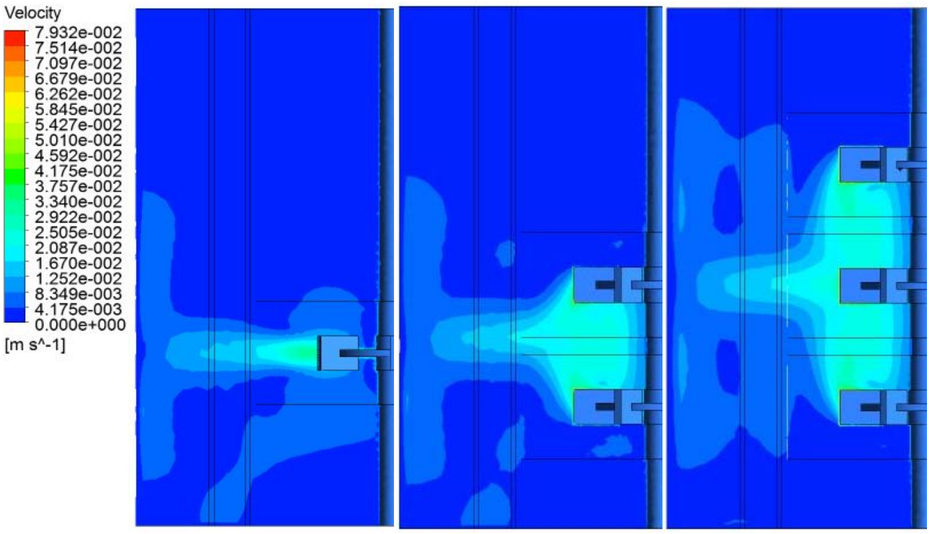


Fig. 10. Axial velocity for different number of stirrers, Re = 40000.

TABLE IV. Geometric parameters of multi-stage stirred tank configurations with Rushton turbines

Number of stirred	*C1*	*C2*	*C3*
2	0.80*D*	0.35*D*	0.35*D*
3	0.45*D*	0.35*D*	0.30*D*
Standard turbine	–	–	0.50*D*



a) standard turbine b) two stirrers c) three stirrers

Fig. 11. Velocity champs for different numbers of agitators.

Effect of the space between agitators

Figs. 12 and 13 clearly show that the effect of the distance between the mixers on the formation of recirculation loops is significant. When the distance between the two stirrers is equal to 0.35*D*, we observe that each stirrer develops only one recirculation loop (Case N°2 and Case N°3), suggesting that the two stirrers could be replaced by a single stirrer. On the other hand, when the difference exceeds 0.7*D*, each stirrer generates two recirculation loops (Case N°1). In addition, when

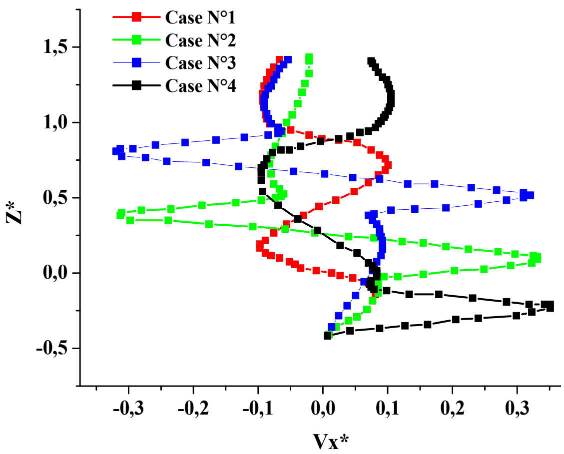
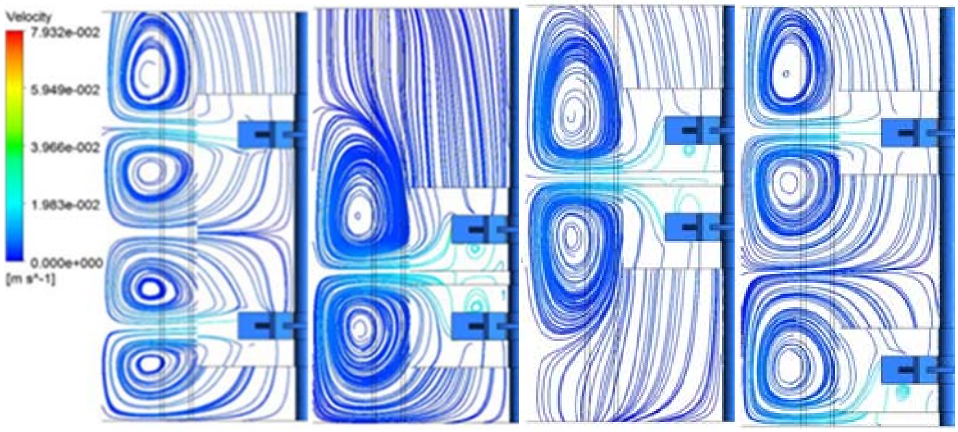


Fig. 12. Axial velocity for different positions of agitator (X^* = 0.185, Re = 40000).

the distance between the two stirrers increases further, as in Case N°4, three recirculation loops can be observed, as shown in Fig. 13. The flow intensity is higher in the lower part of the vessel compared to the upper part, which is caused by the large spacing between the two stirrers. This phenomenon is explained by the independent operation of each agitator, which has a positive effect on the efficacy of fluid mixing.



case N°1 case N°2 case N°3 case N°4

Fig. 13. Stream lines for different cases (Re = 40000).

CONCLUSION

The aim of this work is to study the influence of geometric parameters on the flow generated by a Rushton turbine, with and without baffles. The effect of the number of blades on energy consumption and mixing quality was clearly shown.

This study highlights the effect of the number of blades on the structure of the flow generated by the stirrer. The results show that increasing the number of blades has a significant effect on the volume of fluid swept, with a higher number of blades improving this coverage. In particular, an 8-blade Rushton turbine consumes 12 % more power than the standard 6-blade geometry.

The number of agitators in the tank also has a positive effect on flow intensity. In addition, the distance between the impellers plays a key role in the formation of recirculation zones. When the spacing is equal to 0.7*D*, recirculation loops are formed, which can disrupt the free surface of the fluid.

These findings provide useful insights for the practical design and optimization of stirred tanks, particularly in terms of improving mixing efficiency while managing power consumption. Parameters such as the number of blades and the presence of baffles significantly influence energy demand. The results also emphasize the importance of impeller spacing in controlling flow structures and surface stability.

For future work, optimizing the geometry and shape of baffles is identified as a promising direction. Since baffles play an important role in enhancing mixing but also increase power consumption, modifying their design may lead to better mixing performance with reduced energy costs. Moreover, the current numerical model does not account for multiphase flow or non-Newtonian behavior, which could be explored in future studies to enhance the model's applicability to real industrial processes.

ИЗВОД

CFD СИМУЛАЦИЈА ПЕРФОРМАНСИ РЕЗЕРВОАРА СА МЕШАЊЕМ

KAMLA YOUCEF[1], ZIED DRISS[2], MOHAMMED FOUKRACH[1], SERGIO ROSA[3] и TOUHAMI BAKI[4]

[1]Laboratory of Mechanics and Energy, Faculty of Technology, University Hassiba Ben Bouali of Chlef (UHBC), Algeria, [2]Laboratory of Electro-Mechanic Systems (LASEM), National School of Engineers of Sfax (ENIS), University of Sfax (US), Tunisia, [3]Polytechnic Institute of Bragança (IPB), Santa Apolonia Campus, Portugal и [4]Mechanical Faculty, Gaseous Fuels and Environment Laboratory, University of Sciences and Technology of Oran Mohamed Boudiaf (USTO-MB), BP 1505, El-M'Naouer, Oran 31000, Algeria

У овом раду извршена је нумеричка анализа тока флуида који генерише Раштонова турбина приликом свог рада. Топологија резултујућег тока флуида у великој мери зависи од неколико параметара, који укључују геометријску конфигурацију система и својства флуида који струји. У оквиру овог рада, једначине које описују систем засноване на k–ε моделу су решаване коришћењем методе коначних запремина. За неколико геометрија турбина, уз варирање броја лопатица од 6 до 12, анализирани су профили брзина, струјнице и димензије вртлога. Моделовање је укључило системе са једностепеним, двостепеним и тростепеним мешалицама, уз испитивање утицаја растојања између појединих мешалица на слику струјања. Коначно, нумеричке симулације извршене у оквиру овог рада валидиране су поређењем добијених резултата са експерименталним резултатима доступним у литератури, уз добро слагање експерименталних вредности са резултатима симулација.

(Примљено 15. априла, ревидирано 2. јуна, прихваћено 12. августа 2025)

REFERENCES

1. Z. Driss, in *Procedings of the International Conference on Advances in Mechanical Engineering and Mechanics, Hammamet, Tunisia*, 2006, pp. 1–6 (https://dx.doi.org/10.15866/irease.v7i3.3729)
2. L. Li, B. Xu, *Chem. Papers* **71** (2017) 1863 (https://dx.doi.org/10.1007/s11696-017-0180-1)
3. Y. Cui, H. Zhang, X. Li, M. Yang, Z. Guan, *Int. J. Agric. Biol. Eng.* **11** (2018) 111 (https://dx.doi.org/10.25165/j.ijabe.20181104.2729)
4. M. Jaszczur, A. Młynarczykowska, L. Demurtas *Energies* **13**(3) (2020) 640 (https://dx.doi.org/10.3390/en13030640)
5. F. L. Yang, S. J. Zhou, C. X. Zhang, *Korean J. Chem. Eng.* **32** (2015) 816 (https://doi.org/10.1007/s11814-014-0255-4)
6. L. Liangchao, C. Ning, X. Kefeng, X. Beiping, *Int. J. Chem. React. Eng.* **17** (2019) 20180215 (https://doi.org/10.1515/ijcre-2018-0215)
7. M. Kordas, G. Story, M. Konopacki, R. Rakoczy, *Ind. Eng. Chem. Res.* **52** (2013) 13818 (https://doi.org/10.1021/ie303086r)

8. H. Ameur, Y. Kamla, *Instal* **6** (2020) 42 (https://doi.org/10.36119/15.2020.6.5)
9. Y. Kamla, H. Ameur, M. I. Arab, B. Azeddine, *Data Brief* **39** (2021) 107669 (https://doi.org/10.1016/j.dib.2021.107669)
10. X. Dang, J. Guo, L.Yang, S. Xue, B. Ai, X. Li, L. Chen, W. Li, H. Qin, J. Zhang, *J. Taiwan Inst. Chem. Eng.* **131** (2022) 104 (https://doi.org/10.1016/j.jtice.2021.11.016)
11. S. Youcefi, A. Mokhefi, M. Bouzit, A. Youcefi, *Int. J. Heat Tech.* **41** (2023) 1489 (https://doi.org/10.18280/ijht.410611)
12. S. Bonnot, F. Cabaret, L. Fradette, P. A. Tanguy, *Chem. Eng. Res. Des.* **85** (2007) 1129 (https://doi.org/10.1205/cherd06215)
13. N. Hashemi, F. Ein-Mozaffari, S. R. Upreti, D. K. Hwang, *Chem. Eng. Sci.* **151** (2016) 25 (https://doi.org/10.1016/j.ces.2016.05.003)
14. C. Yang, J. Yao, X. Chen, M. Xie, G. Zhou, Z. Xu, B. Liu, *Chem. Eng. Res. Des.* **202** (2024) 377 (https://doi.org/10.1016/j.cherd.2023.12.040
15. H. H. Mortensen, F. Innings, A. Håkansson, *Chem. Eng. Res. Des.* **121** (2017) 245 (https://doi.org/10.1016/j.cherd.2017.03.016)
16. S. M. Mahmoudi, M. Yianneskis, in *Fluid Mechanics of Mixing. Modelling, Operations and Experimental Techniques*, R. King, Ed., Springer, Dordrecht, 1992, pp. 11–18 (https://doi.org/10.1007/978-94-015-7973-5_2)
17. K. Rutherford, K. C. Lee, S. M. M. Mahmoudi, M. Yianneskis, *AIChE* **42** (1996) 332 (https://doi.org/10.1002/aic.690420204)
18. A. W. Nienow, *Chem. Eng. Sci.* **52** (1997) 2557 (https://doi.org/10.1016/S0009-2509(97)00072-9)
19. P. M. Armenante, Y. T. Huang, T. Li, *Chem. Eng. Sci.* **47** (1992) 2865 (https://doi.org/10.1016/0009-2509(92)87143-E)
20. G. Montante, A. Brucato, K. C. Lee, M. Yianneskis, *Canadian J. Chem. Eng.* **77** (2009) 649 (https://doi.org/10.1002/cjce.5450770405)
21. A. Hadjeb, M. Bouzit, Y. Kamla, H. Ameur, *Green Sci.* **19** (2017) 83 (https://doi.org/10.1515/pjct-2017-0053)
22. S. Dabiri, A. Noorpoor, M. Arfaee, P. Kumar, W. Rauch, *Water (Switzerland)* **14** (2021) 105 (https://doi.org/10.3390/w13121629)
23. A. Reid, R. Rossi, C. Cottini, A. Benassi, *Meccanica* **15** (2024) 1 (https://doi.org/10.1007/s11012-024-01824-z)
24. H. Wu, G. K. Patterson, *Chem. Eng. Sci.* **44** (1989) 2207 (https://doi.org/10.1016/0009-2509(89)85155-3)
25. Z. Driss, A. Kaffel, B. B. Amira, G. Bouzgarrou, M. S. Abid, *Am. J. Energy Res.* **2** (2014) 67 (https://doi.org/10.12691/ajer-2-3-4)
26. D. A. Deglon, C. J. Meyer, *Minerals Eng.* **19** (2006) 1059 (https://doi.org/10.1016/j.mineng.2006.04.001)
27. C. D. Tacă, M. Păunescu, *Chem. Eng. Sci.* **56** (2001) 4445 (https://doi.org/10.1016/S0009-2509(01)00096-3)
28. J. Karcz, M. Major, *Chem. Eng. Process. Proc. Intens.* **37** (1998) 249 (https://doi.org/10.1016/S0255-2701(98)00033-6).