CFD simulation of the laboratory-scale anaerobic digester to study the impacts of impeller geometric and operational parameters on its performance

Mohammad Esmaeel Kashfi, a Ramin Kouhikamali, a,b, Gholam Khayati, c Javad Mahmoudimehr a

a Faculty of Mechanical Engineering, University of Guilan, Rasht, Iran
b Department of Mechanical Engineering, Isfahan University of Technology, Isfahan, Iran
c Faculty of Chemical Engineering, University of Guilan, Rasht, Iran

Abstract

This research numerically surveys the effects of several main parameters of an agitated anaerobic digester on mixing rate and power input. The numerical simulation is handled applying the finite volume method (FVM), and it is validated with the available experimental data. The results indicate that doubling the blade length enhances the mixing rate by 39.9% and makes the power input enhance 13.5 times; increasing the number of blades (from 4 to 6) improves the mixing rate by 12.5% and makes the power input grow 1.4 times, and decreasing the blade tilt angle from 45° to 30° causes the mixing rate to drop by 14% and decreases the power input 1.8 times. Furthermore, the observations show that the mixing rate and power input are adversely influenced by the wastewater concentration. At last, the most effective impeller design, among 144 cases investigated, is found out and suggested.

Keywords: mechanical mixing; impeller design; CFD technique; wastewater; power input

* Corresponding author. Tel.: +98 (31)3 3915242; Mobile: +98 9124024253; Fax: +98 (31)3 3912628; Postal Code: 8415683111; E-mail address: r.kouhikamali@iut.ac.ir, kouhikamali@guilan.ac.ir.
Co-author e-mail address: mohammad_kashfi57@yahoo.com, khayati@guilan.ac.ir, mahmoudimehr@guilan.ac.ir
1. Introduction

The increase in awareness of global warming and the increase in energy prices has brought more attention to renewable energies in recent years. This kind of energy can be obtained from various types of renewable sources (such as waste and wastewater) and through the use of anaerobic digestion processes. Anaerobic digestion (AD) technique is widely used in Waste Water Treatment Plant (WWTP) to improve the quality of wastewater and to stabilize the sludge produced from wastewater treatment [1]. The overall performance of the AD systems depends significantly on the mixing [2]. A proper mixing leads to uniformity of materials and temperature in wastewater and causes the efficient material transfer from substrate to microorganisms in the large volume of the reactor [3]. According to investigations, methane gas production rate and the effluent quality using anaerobic digesters are significantly affected by the mixing [4, 5]. Furthermore, in the field of the mixing process, the quality of the mixture [6] as well as power consumption in agitated tanks [7, 8] has always been worrying for researchers. There are three major mixing techniques such as the impeller mixing [9, 10], slurry recirculation [11], and gas recirculation. Each of the methods mentioned has advantages and disadvantages, and impeller mixing is the most impressive technique among the modes of mixing [12, 13]. To obtain a good mixture, the continuously stirred tank reactor (CSTR) is typically used [14, 15], and computational fluid dynamics (CFD) techniques emerged as a helpful tool to predict and model the fluid flow [16, 17]. The numerical simulation can assist the researcher to detect the effects of several parameters, such as the geometry of the reactor, the number of blades, the tilt angle of the blade, and the rheological properties of the fluids on both flow pattern [18] and the mixing rate without the need to construct a commercial reactor.
Vesvikar and Al-Dahhan [19] numerically simulated the mixing process, which was provided by sparging gas, within a digester to determine flow patterns. The numerical results were consistent with the experimental data obtained from the computer automated radioactive particle tracking (CARPT). Buwa et al. [20] reported the numerical and experimental investigations for mixing in the tank agitated with different impellers. In this study, the mixing performance was evaluated, and the numerical results were in good agreement with experimental data. Wu [21] examined several turbulence models for non-Newtonian fluids in agitated anaerobic digesters to evaluate the power number and the flow number obtained from the CFD with those of laboratory data. Among the turbulence models studied, the realizable $K$-$\varepsilon$ and the standard $K$-$\omega$ models were found to be more suitable than the others. Magelli et al. [22] reported that the dimensions and agitator types have a strong effect on the mixing time (or homogenization) of the stirred tanks. Trad et al. [23] used CFD-based simulations to investigate the influences of the agitator geometry on the flow patterns inside a digester, and the numerical results were consistent with the experimental data. Lebranchu et al. [24] studied the impacts of impeller types and shear stress on biogas production using the CFD simulation method and the results obtained from the experimental data.

Meister et al. [25] reported that for concentrated wastewaters, the $K$-$\varepsilon$ model is more appropriate than the $K$-$\omega$ model with regard to the simulation performance. Mendoza et al. [26] performed a CFD simulation to assess the influences of impeller rotational speed and fluid viscosity on the flow fields in a stirred tank. Mao et al. [27] determined the indexes to increase the AD performance and to minimize the mixing time using CFD simulation. Müller et al. [28] attempted to optimize an externally mixed bio-gas plant through coupling of CFD simulations with an optimization software. Cui et al. [29] focused on the numerical simulations to reveal the
impacts of various impellers on the flow field and the mixing performance within digesters. Rave et al. [30] applied a scale-adaptive simulation (SAS) of single-phase flow to evaluate the flow pattern in a baffled stirred tank at a high Reynolds number and also illustrated the effect of the blade geometry.

According to the provided literature survey, there were several works that were trying to discover and improve the mixing pattern inside a digester; however, these authors perceived that the influences of several main geometric parameters (such as the blade size, the number of blades, and tilt angle of the blade) and operational parameters (such as the agitation speed and the wastewater concentration or total solid) have not been studied perfectly and in an integrated approach. This study employs UDF-assisted Fluent to numerically investigate the influences of the mentioned parameters on mixing rate and also determines power input (or operational cost) from the numerical solution reports in an agitated anaerobic digester. For this purpose, the total solids (TS) of 2.5%, 7.5%, 12.1%, the impeller diameter sizes (D) of \( d/3 \), \( d/2 \), \( 2d/3 \), (where \( d \) denotes the digester diameter), blade tilt angles of 30°, 45°, two different numbers of blades (i.e., 4 and 6 blades), and agitation speeds of 250, 500, 750 and 1500 rpm for the cylindrical digester are investigated. Finally, the results of the different settings for the design parameters are compared in terms of power input and mixing rate, which is related to power production, and the best choice is proposed.

2. Theoretical model and methods

In this study, the continuity, momentum, and turbulence equations are used to determine the hydrodynamics of flow in the agitated anaerobic digester. Due to the complexity of the AD process, the simplifying assumptions are considered as follows:
i. The behavior of the wastewater (or fluid) at a concentration of TS≥2.5% is non-
Newtonian and, the wastewater is incompressible and isothermal.

ii. The realizable $K$-$\varepsilon$ model is used to characterize the turbulent flow.

iii. The standard wall functions are used for near-wall treatment.

iv. The temperature of the mixing system, including the wall and the wastewater is fixed at
35°C.

v. The computations are done for a three-dimensional non-time-dependent turbulent flow.

2.1. Governing equations

2.1.1. Continuity equation

The equation of continuity is described by the following equation:

$$\frac{\partial \rho}{\partial t} + \text{div} (\rho \mathbf{V}) = 0$$

(1)

Herein, $\mathbf{V}$ (m/s) and $\rho$ (kg/m$^3$) denote the velocity vector and the density of the fluid.

2.1.2. Conservation of momentum

The momentum equation is represented by the following expression:

$$\rho \frac{DV}{Dt} = -\nabla p + \nabla \cdot \tau + \mathbf{F}$$

(2)

The variables $\rho$ (Pa), $\tau$ (N/m$^2$), and $\mathbf{F}$ (N/m$^3$) define the fluid’s static pressure, the viscous stress
tensor, and the body force vector per unit volume.

2.1.3. Turbulence equations

In this study, due to the complexity of the governing flow, the realizable $K$-$\varepsilon$ model is used to
describe the turbulent flow characteristics in which $K$ (m$^2$/s$^2$) and $\varepsilon$ (m$^2$/s$^3$) denote the turbulence
kinetic energy and the turbulence dissipation rate; these parameters are defined by the following
transport equations:
\[
\rho \frac{DK}{Dt} = \nabla \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla K \right] + G_k - \rho \varepsilon \tag{3}
\]

\[
\rho \frac{D\varepsilon}{Dt} = \nabla \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \rho C_1 S \varepsilon - \rho C_2 \varepsilon^2 \left( \frac{K}{\varepsilon} \right) \frac{2}{\sqrt{\varepsilon}} \tag{4}
\]

Where \( G_k \) (J/m\(^3\)) and \( \mu_t \) (Pas) denote the generation of turbulence kinetic energy due to the mean velocity gradients and the eddy viscosity; the model constant values (dimensionless) \( C_2 \), \( \sigma_k \), and \( \sigma_\varepsilon \) are equal to 1.9, 1, and 1.2, respectively. Also, \( C_1 \) is defined as follows:

\[
C_1 = \max \left[ 0.43, \frac{\delta}{\delta + 5} \right] \tag{5}
\]

Herein, \( \delta = S \frac{K}{\varepsilon} \), and \( S \) (s\(^{-1}\)) denotes the modulus of the mean strain rate [31].

2.1.4. Non-Newtonian fluid model

As mentioned, the behavior of the wastewater with TS ≥ 2.5% can be considered as non-Newtonian. In the non-Newtonian fluid model used in the system, the rheological characteristics of wastewater depend on the TS values; therefore, these values can be obtained by the following expressions [32]:

\[
\eta = k \cdot \dot{\gamma}^{n-1} \tag{6}
\]

\[
\rho = 0.0367TS^3 - 2.38TS^2 + 14.6TS + 1000 \tag{7}
\]

The variables \( \eta \) (Pas) and \( \dot{\gamma} \) (s\(^{-1}\)) denote the non-Newtonian viscosity and the shear rate. Also, the constants \( k \) (Pa.s\(^n\)), \( n \) (dimensionless), and TS (%) define the consistency coefficient, the power-law index, and the concentration of the wastewater, respectively.

The details of rheological characteristics and densities of the wastewater with different TSs are reported in Table S1 (see supplementary material).

2.2. Description of major dimensionless parameters
The flow features within the agitated digester are affected by the impeller, and therefore the Reynolds number, \( Re \), is determined by the following equation [33]:

\[
Re = \frac{D^2 N^{2-n} \rho}{k} \left[ \frac{n}{8 \left( \frac{n}{6n+2} \right)^n} \right]
\]  

(8)

Herein, \( N \) (rps) and \( D \) (m) denote the agitation speed and diameter of the impeller, noting that the values of the constants \( n \) and \( k \) are summarized in Table S1.

In this study, the dimensionless parameter of \( N_p \) in which denotes the power number is defined by the following equation [34]:

\[
N_p = \frac{P_{in}}{\rho N^3 D^5}
\]  

(9)

Also, the power input of the system, \( P_{in} \) (W), can be obtained from the impeller torque, \( T \) (Nm), which is determined by the CFD simulation results:

\[
P_{in} = 2\pi NT
\]  

(10)

The mixing rate (%), represented by \( \alpha \), is an important dimensionless parameter in which is described as the ratio of the stream zone volume to the total volume of the digester. Furthermore, the stream zone is specified as the area with the velocity magnitude amount of greater than 5% of the maximum of the velocity of the entire computational domain [19]. After calculating the mixing rate using UFD-assisted Fluent, the methane production (power production) is determined from solving the governing equations numerically [35].

The Coefficient Of Performance, COP (dimensionless), is determined by the following equation:

\[
COP = \frac{P_{out}}{P_{in}}
\]  

(11)
Hint: $\alpha$ and $P_{in} = F(D, \theta, n, N, TS)$, and also $P_{out} = G(\alpha)$

Where $P_{out}$ (W) is the power production. The parameters of the objective function (COP) depend on the impeller diameter ($D$), the blade tilt angle ($\theta$), the number of the blade ($n$), the agitation speed ($N$), the total solids (TS), and the mixing rate ($\alpha$). The COP is applied to figure out the optimal system.

### 2.3. Model’s geometry and boundary conditions

In this section, the illustrative view of the geometry of the computational domain, meshing, and boundary conditions is depicted in Figure S1 (see supplementary material). To create a suitable meshing in terms of quality, due to the complex geometry of the impeller, the unstructured grids using a mixture of elements (tetrahedral and triangular) are generated in ANSYS workbench. A steady-state modeling approach called the multiple reference frame (MRF) is performed to describe the rotational flow created by the impeller, noting that in this model, the flow field calculations in the bulk area (including all the domain in the computational zone except impeller) is determined by using the agitation speed, which is equal to the speed of the impeller. The geometric data of the laboratory-scale cylindrical digester, impeller, and the MRF size are summarized in Table 1.

### 2.4. Numerical solution technique

To discretize and solve the governing equations in the mixing model, the finite volume method (FVM) is performed by applying ANSYS Fluent commercial software (version 17.1) [36]. The velocity-pressure coupled differential equations are solved by using the SIMPLE algorithm. In this regard, initially, for solving the governing equations such as momentum and the turbulence equations, the first-order upwind scheme of about 100 iterations is done. At last, the equations mentioned are solved with the second-order upwind scheme. To achieve the
convergence criteria, the residual values of below $10^{-4}$ are employed for continuity, $V_x$, $V_y$, $V_z$, $K$, and $\varepsilon$ in Fluent.

### 2.5. Mesh study

In order to investigate the grid independence for the numerical solution of the mixing model in the current study, the CFD simulation of the model (for TS=7.5% and an agitation speed of 1500 rpm) is done for different grid numbers (585455, 953551, 1759752 and 2247207 cells), and the effects of the grid numbers on the velocity magnitude at different points around the impeller zone as well as the mixing rate are checked. By comparing the obtained results from CFD simulation of the mixing model in terms of the velocity magnitude amounts, which were previously reported by Kashfi et al. [35], and the mixing rate (see Table 2), among the four grids studied, it can be concluded that the model with the grid number of 1759752 cells is figured out to be adequately fine; in this grid conditions, the grid independence can be guaranteed by checking with higher resolution grids (2247207 cells).

### 3. Validation of CFD simulation

To confirm the numerical simulation in terms of the numerical method accuracy, the results obtained from the CFD simulation are compared with the available experimental data published by Hoffmann et al. [37]. The reasons for using Hoffmann’s model are its geometry similarities (such as impeller diameter, the geometry of the reactor, and the impeller type) with the present work geometry. To this end, the quantities of the radial velocity are evaluated at the impeller zone (or $z=50$ mm) for two agitation speeds of impeller (e.g. 250 and 500 rpm) and also obtained the points with the average of the difference percentage of 5% and 4%, respectively (more details in Figure S2 in the supplementary material). It is found that the CFD simulation results are
consistent with the experimental data. Hence, the CFD technique applied in this research can well predict the flow patterns.

4. Results and discussion

This section surveys the effects of various main geometric and non-geometric parameters of mechanically agitated anaerobic digester on the mixing rate and power input as the two major objective functions. The main geometric parameters, including impeller diameter size \((d/3, d/2, 2d/3)\), number of blades \((4 \text{ and } 6)\), and blade tilt angle \((30^\circ, 45^\circ)\) and also agitation speed \((250, 500, 750, 1500 \text{ rpm})\) and wastewater concentration \((\text{TS}=2.5\%, \text{ 7.5\%, 12.1\%})\) are the non-geometric parameters. Since the power input is related to the power number (see Equation 9); therefore, in this section, the reports of power number are presented.

4.1. The effect of impeller geometry on the mixing rate and power number

The geometric characteristics of the impeller are of the most considerable factors affecting the mixing rate and the power number. Herein, the influences of the blade size, the number of blades, and the blade tilt angle are studied. The effect of the impeller geometric features on the mixing rate and power number is shown in Figure 1.

4.1.1. The effect of the size of the blade

Figure 1a shows the effect of the length of the blade on the mixing rate of the system, while blade tilt angle, number of blades, and TS are set at \(45^\circ, 4, \text{ and } 7.5\%\), respectively. This figure indicates that increasing the length of the blade, which increases the hydraulic diameter (or the impeller diameter), has led to a larger volume of wastewater to be swept and greater momentum to be penetrated into the lateral and upper layers of the digester. This observation was in another way reported by Vesvikar and Al-Dahhan [19]. They observed that a greater size of the blade resulted in a smaller dead zone.
The mixing rate index is not the only important engineering objective, and the power number plays an important role in the performance of the mechanical mixing system [38]. Table 3 is presented to study the effect of blade length on the system power input from the turbulence intensity (TI) point of view. This table shows that TI (maximum intensity as a representative) increases with increasing blade length.

The dimensionless parameter of TI is determined by the following equation:

\[
\text{TI} = \sqrt{\frac{2}{3}} \frac{K}{u}
\]  

(12)

Where \( K (\text{m}^2/\text{s}^2) \) is the turbulence kinetic energy, and \( u (\text{m/s}) \) is the mean velocity.

A greater TI is associated with higher viscous dissipation and, hence, a higher energy dissipation or greater power input. Figure 1b shows the power number calculated for various Re numbers. The predicted results showed a lower power number was observed for a higher Re number. This indicates that the more power input took place in a larger blade because the kinetic energy in the impeller is proportional to the fifth-order of impeller diameter (see Equation 9).

4.1.2. The effect of the number of blades

Figure 1c depicts the effect of the number of blades on the mixing rate within the agitated digester. As shown, for a fixed speed, the higher number of blades has led to a greater mixing rate. This is because for the higher number of blades, more surface area is in contact with the wastewater, and therefore a larger volume of wastewater is affected by the impellers. In addition, Table 4 reports the variations of the power number versus the number of blades. This table indicates the direct dependence of power number on the number of blades which has been previously shown by Nagata et al. [39]. Furthermore, it can be concluded that there is a direct
relationship between power number and the power input. This is because, for the same speed and
blade size, the kinetic energy in the impeller is constant.

4.1.3. The effect of the tilt angle of the blade

The influence of blade tilt angle on mixing rate is shown in Figure 1d. Herein, impeller
diameter size, number of blades, and TS are set at d/3, 4, and 7.5%, respectively. Figure 1d
shows that for the same rotational speed, the tilt angle of 30° resulted in a lower mixing rate as
compared to the tilt angle of 45°. This occurrence can be attributed to the fact that a lower tilt
angle which is equivalent to a higher deviation of the blades from the flow direction, causes a
smaller volume of the wastewater-swept region to rotate with the impeller. Therefore, it is
expected that the penetration of momentum into the digester and consequently the mixing rate of
the system to decrease. The power number at different blade tilt angles is listed in Table 5. As
observed, at the same rotational speed, by reducing the tilt angle of the blade, the power number
is decreased. Because of a higher deviation of the blade from the flow direction (by changing the
tilt angle from 45° to 30°), it is expected that a lower resistance to be occurred in the opposite
direction of rotation. Therefore, this indicates that decreasing the blade tilt angle from 45° to 30°
can reduce the power input. This observation has been previously confirmed by Major-
Godlew ska and Karcz [40].

4.2. The effect of agitation speed on the mixing rate and power number

The axial flow impeller is used in this research. This kind of impellers can create flows
parallel to the axis of the impeller stator and are suitable for creating vertical flows [41]. Figures
2a-d show the contours of velocity magnitude within the computational domain for the different
agitation speeds, while impeller diameter size, tilt angle, number of blades and TS are considered
to be fixed at d/3, 45°, 4 and 7.5%, respectively. The figures indicate that with increasing
agitation speed, the momentum diffusion to the lateral and upper layers of the wastewater increased. Therefore, the wastewater mixing rate is observed to be positively affected by increasing the agitation speed. This trend also occurs for other geometric conditions and rheological properties of the wastewater, which is not repeated to be shown for the sake of conciseness. As an illustration, increasing the agitation speed from 250 to 1500 rpm has led the mixing rate to increase from 44.8% to 97.5%. By using nonlinear regression in this case, the $\alpha - N$ function can be obtained:

$$\alpha = 30.142 \cdot \ln(N) - 117.91$$  \hspace{1cm} (13)

Herein, $\alpha$ (%) and $N$ (rpm) denote the mixing rate and the rotational speed of the impeller, and the quantity of the coefficient of determination, $R^2$, is sufficient ($R^2 = 0.9422$).

In the conditions studied, if the mixing rate of the system is the same as the value obtained at 1500 rpm of the impeller (the value is equal to 97.5%), according to Equation 13, the agitation speed of the impeller is determined 1270 rpm. Therefore, it can be deduced that from 1270 to 1500 rpm, there is no change in the mixing rate of the system, and increasing the agitation speed increases only the power input of the system. On the other hand, to calculate the minimum agitation speed, a mixing rate of 5% is allocated in the above equation. The impeller speed for the mixing rate of 5% is attained 59 rpm. It should be noted that the systems with agitation speeds less than this amount have negative values of the mixing rate, which is conceptually meaningless. In other words, mixing does not occur in the system (or the unmixed mode). Thus, based on the numerical results, it is suggested that the agitation speeds in the range of 59 to 1270 rpm are applied to analyze the performance of this mixing system.

Figure 2e illustrates the power number versus Re number. As observed, for entirely turbulent flows at large Re, the system power number does not show notable changes and tends to a
constant value. This prediction is in good agreement with the experimental results [34], and also this trend indicates that the higher of the Re number, the higher of the power input due to higher impeller torque. In this case, an exponential regression between power number and Reynolds number can be observed:

\[ N_p = 2 \text{Re}^{-0.08022} \]  

(14)

Notably, the determination coefficient, \( R^2 \), is satisfactory found to be larger than 0.97.

Based on the obtained results in this study, the power number predicted for impeller A-310 is equal to 0.315 under turbulent conditions, while at the same conditions, the amount of 0.3 was reported in the published data by Hemrajani and Tatterson [34].

4.3. The effect of TS on the mixing rate and power number

The TS of wastewater in the system is of the important parameters affecting the mixing efficiency. The TS influences the fluid dynamics through affecting the wastewater rheological behavior [33]. Figure 3 reveals the influence of TS on the mixing rate and the power number. Figures 3a-c show the contours of velocity magnitude within the computational domain for the different levels of TS, while the impeller diameter size, the tilt angle, the number of blades, and the agitation speed are set at \( d/3 \), 45°, 4, and 500 rpm, respectively.

As observed, for a higher TS, the momentum diffusion into the lateral and upper layers of the wastewater is reduced; therefore, the system mixing rate is inversely related to the TS. Furthermore, as shown in Figures 3a-c, for a higher TS, the highest velocity gradient is observed in the areas around the blades. The effect of TS on the power number is depicted in Figure 3d. As can be seen, the power number in a digester with higher TS is greater than those in a digester with lower TS. It can be concluded that a higher TS value in this condition leads to greater power input due to the same kinetic energy in both systems. For instance, at \( \text{Re}=2000 \), the predicted
power input in this research is 0.32 W in a system with TS of 2.5%, and 15.7 W in a system with TS of 7.5%. This observation may be attributed to the fact that the higher viscosity leads to a higher viscous dissipation.

Also, the impact of TS on the TI dimensionless parameter for concentrations 2.5% and 12.1% in the agitated anaerobic digester are studied. The results of this investigation are shown in Figure 4, while impeller diameter size, tilt angle, number of blades and agitation speed of impeller are considered to be set at $d/3$, 45°, 4 and 500 rpm, respectively.

As observed, the system with a higher TS leads to a higher TI. As a result, the dissipation of the viscosity and also energy consumption are increased that caused the power input of the system to increase. Furthermore, as can be revealed in Figure 4, the highest TI gradient is seen in the areas around the blades, and the movement of the fluid is done slowly toward the walls and upper layers of the digester with higher TS due to the effects of the fluid viscosity. These reasons also make the mixing rate parameter decrease in the system with a higher TS. At last, it can be deduced that the results obtained from the TI point of view are in complete agreement with the above presented results.

### 4.4. Practical application of this study

By considering the mixing rate, which is directly related to methane production and power production, and the power input as the main objectives of the present work; the impeller whose number of blades, tilt angle, size and agitation speed are equal to 4, 30°, 62 mm and 250 rpm, respectively, is found out as the best design among the 144 models examined (see supplementary material, Table S2). Also, for large-scale of the most efficient impeller design, it is important to simultaneously evaluate the influence of mixing on both power input and methane production, noting that it is necessary to apply the criteria of the similarity such as geometric and mixing
energy level, which were presented in some studies (see, e.g. [42, 43]). After getting the detailed
geometric information of the anaerobic digester from the mentioned criteria, the CFD simulation
can be used to simulate the flow field in real anaerobic digesters. These results can assist to sense
to real mixing efficiency in the digesters.

5. Conclusions

This research numerically studied the effects of some main parameters (especially impeller
gometry related ones) on mechanical performance of an anaerobic digester. The results showed
that doubling the blade size, increasing the blade tilt angle from 30° to 45°, and increasing the
number of agitating blades from 4 to 6 improved the mixing rate by 39.9%, 14%, and 12.5%,
however, caused the power input to increase 13.5 times, 1.8 times, and 1.4 times, respectively.
The four-blade impeller with a tilt angle of 30° and with a diameter of one-third of the tank
diameter was the most efficient design.

Supplementary data is available at:

http://scientiairanica.sharif.edu/jufile?ar_sfile=173731

References

1. Tchobanoglous, G., Burton, F.L. and Stensel, H.D. “Wastewater Engineering; Treatment and
Reuse”, 4th Edn., McGraw-Hill Companies, Inc. (Metcalf and Eddy, Inc.), New York, USA
(2003).


List of Tables:

Table 1. Dimensions of laboratory-scale cylindrical digester, impeller, and MRF.

Table 2. Grid independence analysis regarding mixing rate (α).

Table 3. Maximum turbulent intensity (TI) for different impeller diameter sizes.
Table 4. Power number for different number of blades.

Table 5. Power number for different blade tilt angles.

List of Figures:

Figure 1. Effect of (a) impeller diameter size on mixing rate (b) impeller diameter size on power number (c) blade number on mixing rate (d) blade tilt angle on mixing rate.
Figure 2. Contours of velocity magnitude within the computational domain for the agitation speed of (a) 250 (b) 500 (c) 750 (d) 1500 (rpm); (e) Effect of agitation speed on power number for the impeller diameter size of \(d/3\).

Figure 3. Contours of velocity magnitude within the computational domain for the TS of (a) 2.5 (b) 7.5 (c) 12.1 (%); (d) Effect of TS on power number for the impeller diameter size of \(d/3\).

Figure 4. Contours of turbulence intensity (TI) within the computational domain for the TS of (a) 2.5 (b) 12.1 (%).

Table 1. Dimensions of laboratory-scale cylindrical digester, impeller, and MRF.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Digester diameter</td>
<td>17.4</td>
</tr>
<tr>
<td>Digester height</td>
<td>26</td>
</tr>
</tbody>
</table>
Impeller diameter size \( 6.2 \)
blade height \( 1.5 \)
blade thickness \( 0.3 \)
Dist. from bottom \( 8.6 \)
\( D_{\text{MRF}} \) \( 9 \)
\( H_{\text{impeller}} \) \( 2.2 \)

Table 2. Grid independence analysis regarding mixing rate \( (\alpha) \).

<table>
<thead>
<tr>
<th>No.</th>
<th>Number of cells</th>
<th>( \alpha ) (%)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>585,455</td>
<td>99.76</td>
<td>--</td>
</tr>
<tr>
<td>2</td>
<td>953,551</td>
<td>98.8</td>
<td>1</td>
</tr>
<tr>
<td>Impeller diameter size</td>
<td>Max. TI (dimensionless)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>------------------------</td>
<td>-------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$d/3$</td>
<td>0.4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$d/2$</td>
<td>0.61</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3. Maximum turbulent intensity (TI) for different impeller diameter sizes $^a$. 
\[
\frac{2d}{3} \quad 0.76 \\
^a (4\text{-blade, 500 rpm, } 45^\circ, \text{ and TS}=7.5\%) 
\]

<table>
<thead>
<tr>
<th>Number of blade</th>
<th>(N\text{ (rpm)})</th>
<th>(N_p)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>250</td>
<td>1.36</td>
</tr>
<tr>
<td>6</td>
<td>250</td>
<td>1.8</td>
</tr>
</tbody>
</table>

**Table 4.** Power number for different number of blades \(^a\).
Table 5. Power number for different blade tilt angles $^a$.

<table>
<thead>
<tr>
<th>Tilt angle (°)</th>
<th>$N$ (rpm)</th>
<th>$N_p$</th>
</tr>
</thead>
<tbody>
<tr>
<td>45</td>
<td>250</td>
<td>1.36</td>
</tr>
<tr>
<td>30</td>
<td>250</td>
<td>0.84</td>
</tr>
</tbody>
</table>

$^a (D = d/3, 45^\circ, \text{ and TS}=7.5\%)$
\[
\begin{array}{ccc}
45 & 500 & 1.22 \\
30 & 500 & 0.68 \\
\end{array}
\]

\(^a(D=d/3, \text{4-blade, and TS=7.5\%})\)
Figure 1
Figure 2
Figure 3
Figure 4
**Biographies**

**Mohammad Esmaeel Kashfi** received his MSc and Ph.D. degrees in Mechanical Engineering-Energy conversion from the University of Guilan, Iran, in 2015 and 2022, respectively. His main research interests include the design and CFD simulation of mixing systems, renewable energy, and non-Newtonian fluid flows.

**Ramin Kouhikamali** received his Ph.D. degree in Mechanical Engineering-Energy conversion from Amirkabir University of Technology, Iran, in 2008. He is currently a Professor at the Department of Thermo-Fluids, Isfahan University of Technology. His main research interests include the experimental and numerical study of two-phase flows, desalination units, water treatment, and renewable energy. He is the author and co-author of many papers in these fields.

**Gholam Khayati** is currently an Associate Professor of Chemical Engineering at the University of Guilan. His main research interests are bioseparation and wastewater treatment. He has presented many research articles in various journals in these fields.

**Javad Mahmoudimehr** received his BSc degree in Mechanical Engineering from the University of Guilan, Iran, in 2004. He earned his MSc and Ph.D. degrees at Iran University of Science & Technology, Iran, in 2007 and 2012 respectively. He is currently an Associate Professor in the Mechanical Engineering Faculty at the University of Guilan. His research interest fields include energy, optimization, and renewable energy. He published many papers in different journals in these fields.