PAPER • **OPEN ACCESS**

CFD simulation of turbulent flow behaviour in a mixing reactor with Rushton impeller

To cite this article: V Sharan *et al* 2020 *J. Phys.: Conf. Ser.* **1716** 012025

View the article online for updates and enhancements.

240th ECS Meeting
Oct 10-14, 2021, Orlando, Florida
Register early and save up to 20% on registration costs
Early registration deadline Sep 13
REGISTER NOW

This content was downloaded from IP address 106.195.40.208 on 06/08/2021 at 06:21

CFD simulation of turbulent flow behaviour in a mixing reactor with Rushton impeller

V Sharan[1], K Rohit[1], M Ravishankar[1], D Bhuvaneshwar[1] and R Harish[1]

[1]School of Mechanical Engineering, Vellore Institute of Technology,
Chennai, Tamil Nadu- 600127, India
E-mail: harish.r@vit.ac.in

Abstract. As in the current industrial process, mixing is an essential process for various streams of application in engineering fields. The aim of this paper is to predict flow behaviour of the mixing process in mixing tank with the Rushton impeller and find out the best suitable design that creates required turbulence for producing a better degree of homogeneity in mixing. The rotation of the impeller is modelled using MRF in a single-phase medium treating the walls with a no-slip boundary. The mixing characteristics of single and multiple impellers are investigated by varying the impeller speeds and the results of velocity and pressure contours are compared to identify the best impeller with enhanced mixing.

1. Introduction

In many industries mixing is one of the major operations that is been take place for various wide range of applications. For this process an essential mixing tanks are widely used for executing all such mixing operation in industries like Pharmaceutical, oil, Agri-chemical preparations, paint manufacturing, chemical, food, water treatment and biochemical plants are used in wide variety of range with different types of tanks stirred tank, aeration tank and agitated tank based upon the applications. Blending substances and particles, dispersion of gas and liquid, solid suspension, heat transfer is also varying upon with the case conditions [1,2]. Maintain the balance quantity of the substance which is based upon the concentration level, their general operation is to mix the soluble substances such as solids, liquids and gases in which the impeller is used to make the interaction between the particles [3,4]. Some of the drawbacks where the accumulation of particles and stay as a stagnant due to improper mixing which may be depend upon the type of impeller used or the type of mixing tank design that is not suitable for the application that is used for [5,6]. Hence a careful choice of impeller must be selected for the achievement of better mixing quality in bringing the homogenous quality as an output, the characteristics of the impeller design plays a wide role such as the type of impeller, size of the impeller, no of blades used in the impeller ,speed at which the impeller is rotating plays a major role in the mixing performance[7,8]

Hence here the to study the flow physics inside the mixing tank where the impeller is set to a moving reference frame and understanding the turbulence visualization is done by using Computational fluid dynamics (CFD). Generally, CFD is used to analyzing the fluid flow system using numerical methods such that the simulation is run by the computer-controlled programs [9,10]. The possible source of failure can be predicted and analyzed using the software. There are various flows within the steady state and transient condition based upon the simulation that is executed depending on application [11]. Hence in order to achieve a promising result, CFD can be employed in gaining depth knowledge about the turbulence that operation that take place in the mixing tank due to the impeller motion of rotation about a particular axis and this method of using the CFD is way in a cost-free effective method [12]. Some of the impeller configuration dealing with hydrodynamics that relates the mixing performance in a mixing reactor is been tested out with computational fluid dynamics [13]. It is to be noted that the quality of the solid suspension mainly depends upon the fractal iteration as with

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

increasing the homogeneous degree of freedom [14,15]. The power consumption can be reduced drastically by the rigid flexible impeller which is to a punched rigid one [16]. The homogeneity quality is based upon the lumped parameter that insist the mixing time aspect ratio with the range of multiple impellers [17,18]. As building up a mixing tank is a major part of the designing in this application. Hence a mixing tank with an impeller is designed such a way that the mixing of components should take place very well [19,20]. As there are two cases which have been considered such as designing a “single impeller” and a “multiple impeller”. Both the single impeller and multiple impellers has different variation in performance drastically due to the nature of the design, as the selection of suitable impeller is mandatory of a mixing application to get a successful mixing rate.
The dimensions of the components are shown:

Table 1. Dimension of the tank

Parameter	Symbol	Value(mm)
Tank Diameter	T	400
Tank Height	H	500
Impeller Blade Height	H	12
Impeller Blade Diameter	D=T/2	200
Impeller Clearance	C=D/3	66.66

The Rushton impeller is used here which the shaft holds concentric to centre of the mixing tank. This Rushton impeller consists of 6 blades which is well suited for creating required turbulence effect avoiding any accumulation of particles as a stagnant at any place in the tank. So as of Rushton impeller is well suited in multiphase medium such gas-liquid phase applications.

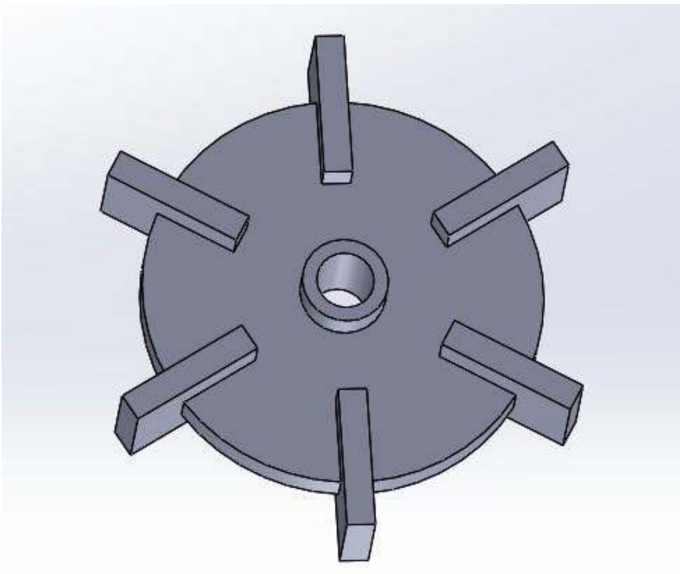

Figure 1 Schematic diagram of impeller

The figure 1 represent the schematic diagram of the Rushton impeller which have been used for executing the simulation

2. Mathematical modelling and numerical method

To solve the numerical simulation the finite volume solver is used where the turbulence in induced where this model is set to be as buoyancy. The Reynolds averaged Naiver-stokes (RANS) for getting the temperature field and the energy equation.

Here the Large eddy simulation turbulence model is executed for obtaining the turbulence effect. The energy that involves the surface and surface radiation and as to solve the governing equation:

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \nu\frac{\partial^2 u}{\partial y^2}, \quad (1)$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (2)$$

Here any of simplification is possible with the energy equation where the non linear problem plays a major role in balancing viscous problem where the Froude number which is dimensionless number plays a greater role in flow regimes. It is said that when the Froude number is lesser than one, it is tranquil and when the Froude number is greater than one, it is shooting flow

$$Fr < 1, \quad (3)$$

$$Fr > 1, \quad (4)$$

The variation in the continuity equation can be written in even two main components where the stream function plays a major role calculated by the empirical model of the power number which act a significant effect

$$u = + \frac{\partial \psi}{\partial y} \quad (5)$$

$$v = + \frac{\partial \psi}{\partial x} \quad (6)$$

Here the Reynolds number that is used to predict whether the flow is laminar or turbulent where the Re<2100 is said to be laminar and Re>4000 is said to turbulent. The values of the diameter of impeller, velocity ,density of the fluid used and viscosity is mandatory to find Reynolds number

$$Re = \frac{\rho v D}{\mu} \quad (7)$$

The density here is represented by „ρ‟ and the velocity is represented by „v‟ and the diameter of the impeller region is represented by „D‟

The difference of temperature between the source of heat and the ambient temperature where the „L‟ is the length of enclosure „v‟ is the kinematic viscosity. The Froude number in which the buoyancy forces the dimensionless number which relates the inertia and the mathematical representation of Froude number.

The fluids in the mixing reactor can consist of either Newtonian fluid or non-Newtonian fluid where it is known that the Newtonian fluid is of less viscous such as H2O fluid and non-Newtonian fluid has high viscous in nature such as oil. In detail the Newtonian fluid experience shear stress which is correlated with the strain rate where non-Newtonian varies based on applied stress.

2.1 Meshing and preprocessing

The Solid works 2019 software is used is used for modelling the mixing tank and its components before being imported to ANSYS Fluent for Pre-processing and meshing. Here two domains are created laterally which is one inner domain and outer domain. Such that the Boolean pattern of the impeller is created in the inner domain and only the inner domain is subjected to mesh motion. Here the tetrahedron mesh is executed to discretize even the uneven parts.

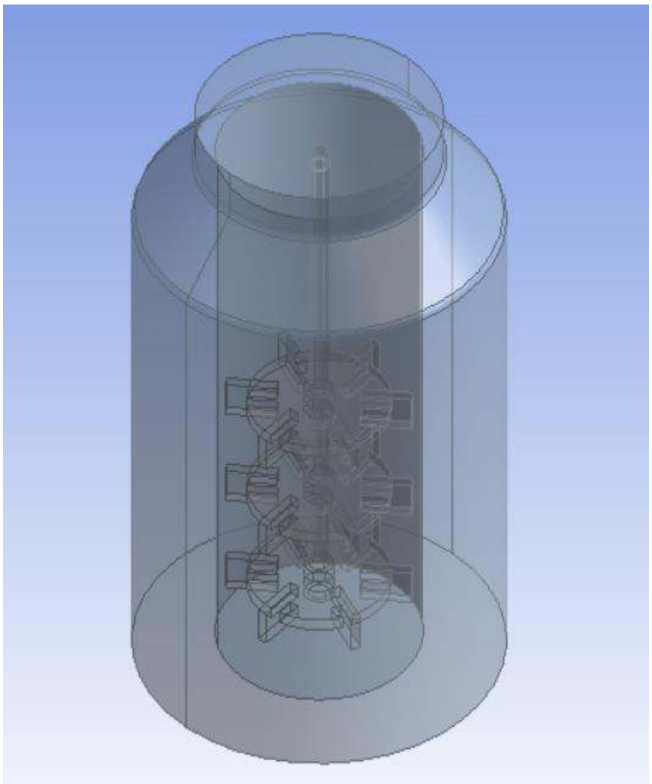

Figure 2 Geometry of the mixing tank

The above schematic diagram Figure2 represents the isometric view of mixing tank which has shaft attached with three Rushton impellers connected in a series as the shaft rotates, the impeller also rotates mutually at certain radian per second. Hence there are two cases where the tank with „single impeller‟ and „multiple impeller‟.

The case with single impeller has only one Rushton impeller attached with the shaft. So that a comparison is made finally with both single and multiple impellers with varying rotating speed of the shaft in radian per second. As it is to be known that definitely the simulation results differ due to the change in the geometry variations and finally the results are verified.

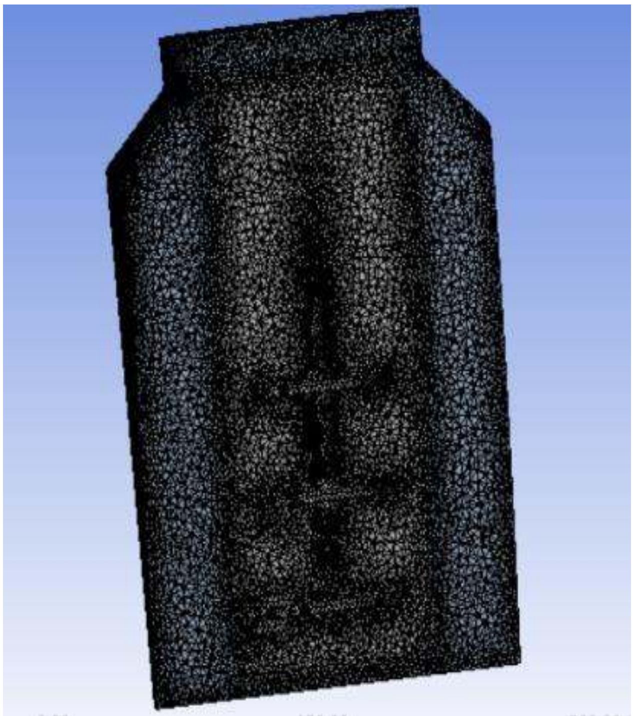

Figure 3 Insight view of Mesh

The figure 3 represent the cross-sectional view of mesh, hence an insight view if the impeller region can be viewed clearly where a finite mesh that had been done.

2.2 Simulation set up

The Ansys fluent is used to run the simulation of the mixing process to take place. Here transient state condition is used with activating the turbulence condition of Large eddy simulation on selecting the sub-grid scale model as Smagorinsky-Lilly which will be helpful to find out the eddy-viscosity. The electrolytic medium is just taken as liquid-water H2O where the fluid properties are imported from the fluid database as the general density of water 998.2kg/m^3 and viscosity constant of 0.001003 kg/m-s. Here the inner domain is considered as the fluid medium having a no slip boundary condition to the walls. To subject the impeller to a rotating zone, the MRF model is used for executing the impeller to rotate at varying speed according to case conditions such as 25 rad/sec, 35 rad/sec, 45 rad/sec.

3. Result and discussion

Here two case conditions are executed having a simulation of mixing tank with „single impeller‟ and „multiple impeller‟ and a comparison of the velocity and pressure is made to conclude which has better performance.

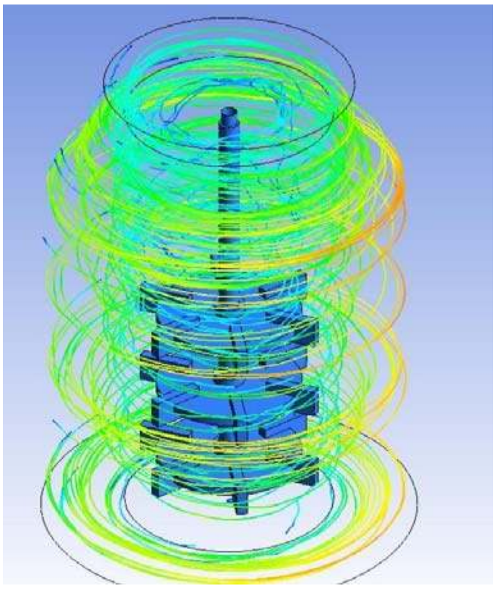

Figure 4. Streamline of mixing process

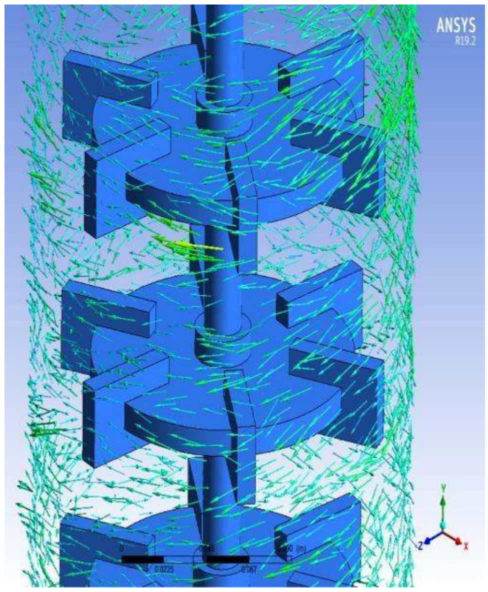

Figure 5. Insight view of vector pattern

The Figure 5 represents the velocity vector pattern that clearly indicates the impeller is rotating mutually creating a turbulence effect among the three impellers.

3.1. Case 1

In this the effect of velocity and the effect of pressure is considered with the „single impeller". This is simulated with varying rotational speed of the impeller such as 25rad/sec, 35rad/sec, 45rad/sec. Here the turbulence that is induced due to the swirl motion of the impeller, the pressure is way developed by the fluid medium to the friction that is created between the blades. Hence finally all the rotating speed plays a major role where each of it is finally used to calculate the Reynolds number, so that which one of the case has more turbulent effect can be predicted depending upon the obtained value of the Reynolds number. So, it is much mandatory to calculate each of the rotating speed case conditions.

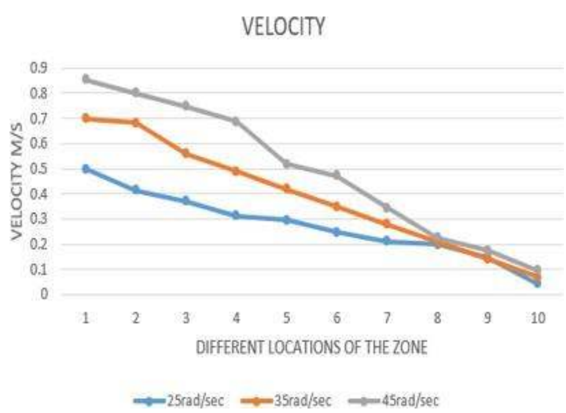

Figure 6. Graphical representation of velocity

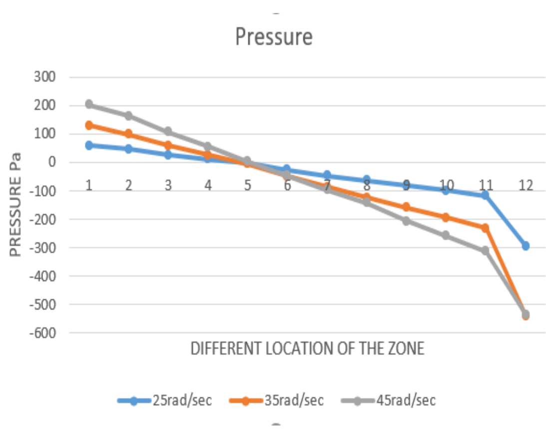

Figure 7.Graphical representation of pressure

The figure 6 represent the velocity pattern of the „single impeller", here the velocity pattern of three different impeller speeds 25rad/sec, 35rad/sec, 45rad/sec has been compared relatively.

The figure7 represent the pressure pattern of the „single impeller" where the pattern of pressure is compared relatively with three different rotational speed of the impeller 25rad/sec, 35rad/sec, 45rad/sec.

3.2. Case 2

In here the effect of velocity and pressure is considered with the „multiple impeller". This is simulated with varying rotational speed of the impeller such as 25rad/sec, 35rad/sec, 45rad/sec.

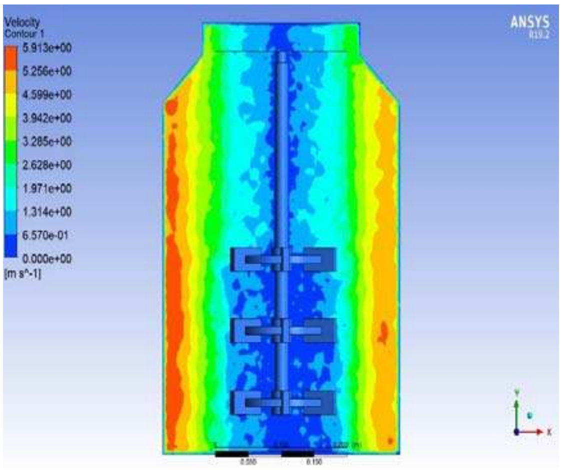

Figure 8. Velocity contour of multiple impeller

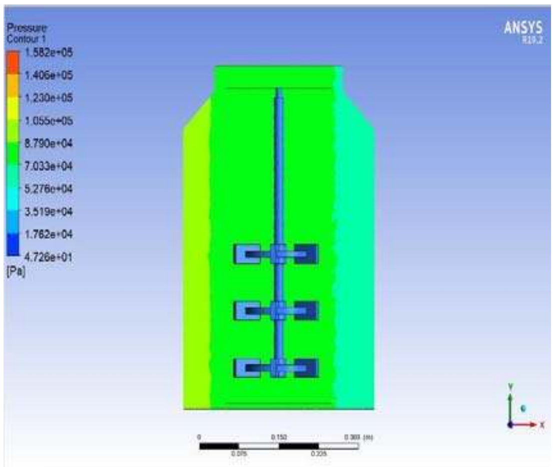

Figure 9. Pressure contour of multiple impeller

The figure 8 represent the velocity contour of the mixing tank with a multiple impeller that is attached to the shaft. Here the velocity is imparted due to the turbulence that is created by the rotational speed of the impeller pertaining to an axis. The figure 9 represent the pressure contour of the mixing tank with a „multiple impeller" attached to the shaft. Here due to the rotational speed of the impeller, turbulence is created effectively which causes pressure that is to be induced in the tank region

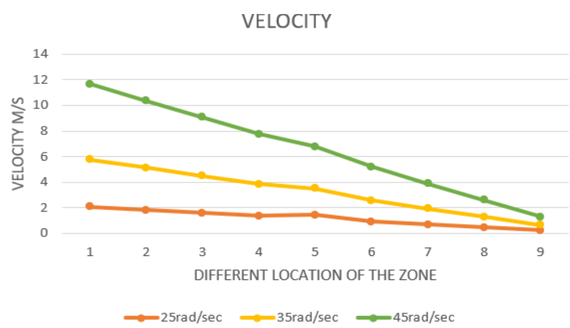

Figure 10. Graphical representation of velocity

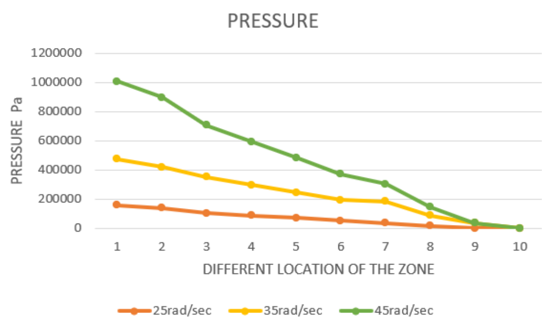

Figure 11.Graphical representation of pressure

The figure 10 represent the velocity pattern of the „multiple impeller", here the velocity pattern of three different impeller speeds 25rad/sec, 35rad/sec, 45rad/sec has been compared relatively. The figure11 represent the pressure pattern of the „multiple impeller" where the pattern of pressure is compared relatively with three different rotational speed of the impeller 25rad/sec, 35rad/sec, 45rad/sec

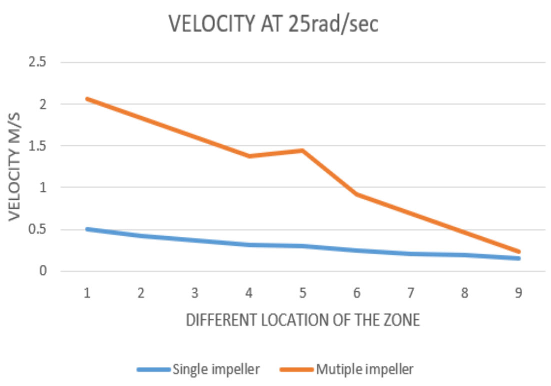

Figure 12. Graphical representation of velocity

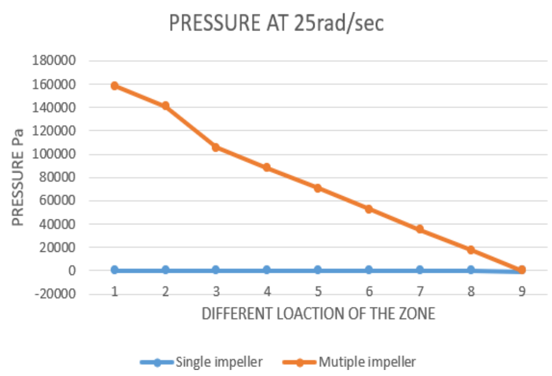

Figure 13. Graphical representation of pressure

4. Conclusion

Hence the flow physics of the mixing process can be easily studied using CFD. It has been come to know that the impeller flow characteristics differ varyingly according to the increase in the impeller rotational speed 25rad/sec, 35rad/sec, 45rad/sec. With all the graphs of velocity and pressure pattern of both „single impeller" and „multiple impeller" it is to be known that increase in the speed of impeller increases the velocity and pressure gradually. So, has to select a best impeller which creates more turbulent effect a comparison is made between both single and multiple impellers. Here a single case condition of impeller rotational speed 25rad/sec is taken. The figure 12 and figure 13 represent the comparison of both single and multiple impeller which the velocity and pressure that is induced. As it is predicted that the performance of the „multiple impeller" is comparatively better than „single impeller" in producing more turbulent effect this is essential for mixing. Hence the selection of impeller design is mandatory which gives good mixing quality. This concludes that using CFD, it is easy to find the best suitable design of the mixing tanks that will guarantee the best homogenous mixing quality as the output reducing the time and cost

References

[1] Deyin G and Chao C 2019 Numerical simulation of solid-liquid mixing characteristics in a stirred tank with fractal impellers *Advanced Powder Technology* **30** pp 2126-38
[2] Cees H and Ruben V 2018 Inter-compartment interaction in multi-impeller mixing experiments and multiple reference frame CFD *Chemical Engineering Research and Design* **136 pp** 870-885
[3] Arturo A and Victor R 2015 Optimal location of axial impellers in a stirred tank applying evolutionary programing and CFD *Chemical Engineering Research and Design* **100** pp 203-211
[4] Weipeng H and Zhoulong X 2019 Effect of impeller clearance on floc growth behaviors in a baffled square stirred-tank reactor *Separation and Purification Technology* **2121** pp 233-244
[5] Manjunath B and Sanja M 2019 CFD simulation of single-phase flow in flotation cells: Effect of impeller blade shape, clearance, and Reynolds number *International Journal of Mining Science and Technology* **29** pp 657-669
[6] Hougliang Z and Ting Z 2014 Experimental and CFD studies of solid–liquid slurry tank stirred with an improved Intermig impeller *Transactions of Nonferrous Metals Society of China* **24** pp 2650-59
[7] Masoud R and Arsalan P 2007 CFD study on mixing by coupled jet-impeller mixers in a large crude oil storage tank *Computers & Chemical Engineering* **31** pp 737-744
[8] Johannes W and Bjorn W 2020 Computational fluid dynamics (CFD) as a tool for industrial UF/DF tank optimization *Biochemical Engineering Journal* **160** pp 107617

[9] Arjun K and Raviraju V 2019 Improved mixing of solid suspensions in stirred tanks with interface baffles: CFD simulation and experimental validation *Chemical Engineering Journal* **358** pp 621-633
[10] Quanhong Z and Hang X 2019 CFD study on double- to single-loop flow pattern transition and its influence on macro mixing efficiency in fully baffled tank stirred by a Rushton turbine *Chinese Journal of Chemical Engineering* **27** pp 993-1000
[11] Andrej B and Iztok Z 2006 Individual impeller flooding in aerated vessel stirred by multiple - Rushton impellers *Chemical engineering journal* **116** pp 85-95.
[12] Vrabel P and Van R 1999 Mixing in large scale aerated reactors with multiple impellers *Chemical Engineering Research and Design* **77** pp 291-302
[13] Joelle A and Catherine X 2006 Design of multiple impeller stirred tanks for the mixing of highly viscous fluids using CFD **61** pp 2913-20
[14] Yuyun B and Zhigang H 2006 Gas Dispersion and Solid Suspension in a Three-Phase Stirred Tank with Multiple Impellers *Chemical engineering journal* **193** pp 801-825.
[15] Naoki D and Yukimasa M 2001 Suspension of solid particles in multi-impeller three-phase stirred tank reactors *The canadian journal of chemical engineering* **79** pp 712-8054
[16] Driss Z and Bouzgarrou G 2010 Computational studies of the pitched blade turbines design effect on the stirred tank flow characteristics *European journal of mechanics* pp 236-245
[17] Franco M and Giuseppina M 2013 Mixing time in high aspect ratio vessels stirred with multiple impellers *Chemical engineering science* pp 712-720.
[18] Ranade V and Van H 1994 A computational snapshot of gas liquid flow in baffled stirred reactors *Chemical engineering science* pp 5175-92
[19] Gunyol O and Mudde R 2006 Computational Study of Hydrodynamics of a Standard Stirred Tank Reactor and a Large-Scale Multi-Impeller Fermenter *International Journal for Multiscale Computational Engineering* **7** pp 559-576
[20] Zied D and Sarhan K 2001 CFD simulation of the laminar flow in stirred tanks generated by double helical ribbons *Central European Journal of Engineering* **4** pp 413-422