CFD ANALYSIS OF MODIFIED IMPELLER FOR PAINT MIXER

by

Muhammad Danial Bin Azmin

24759

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical)

JANUARY 2020

Universiti Teknologi PETRONAS

32610 Bandar Seri Iskandar

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

CFD ANALYSIS OF MODIFIED IMPELLER FOR PAINT MIXER

by

Muhammad Danial bin Azmin

24759

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

In partial fulfilment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

MECHANICAL

Approved by,

____

(Assoc. Prof. Ir. Dr. Mokhtar bin Awang)

UNIVERSITI TEKNOLOGI PETRONAS

Bandar Seri Iskandar, Perak

January 2020

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or person.

MUHAMMAD DANIAL BIN AZMIN

ABSTRACT

By using the Computational Fluid Dynamics (CFD) program, the impeller's output could be visualized without creating a prototype which would cost the time and money. For this research, simulation of computational fluid dynamics is performed on two forms of impeller design (Counterflow and Sawtooth) and modification of their design (4-Sided Impeller and Inclined Blade Sawtooth Impeller) for evaluation 2 technique of simulating was used to analyse the rotation of the impellers which are the Multiple Reference Frame method and the Moving Wall method. The simulations result shown by vector of velocity and wall shear of vectors. The efficiency of the impellers is then measured on the basis of the result obtained comparatively. The results of this project are important in determining an effective impeller to mix industrial paint in a homogeneous way.

ACKNOWLEDGEMENT

In the name of Allah, The Most Merciful and all praises to Him the Almighty that in His will I managed to complete this final year project within the allocated time.

I am using this opportunity to express my outmost gratitude and special thanks to my project supervisor, Assoc. Prof. Ir. Dr Mokhtar bin Awang for his special advices and guidance throughout the completion of this project in spite of his extraordinarily busy schedule. Without his assistance, it is likely that I would not be able to complete the project in time.

In addition, a special thanks to my family for always encouraging and loving me in this final project year. Not to mention, my fellow friends who are actively exchanging ideas when this project is complete. The relentless encouragement motivates me to explore information and enjoy the journey of this project.

Last but not least, I consider this gap in my career growth as a significant milestone for me.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATION

μ	-	Dynamic Viscosity Coefficient
D	-	Vessel Diameter
ρ	-	Density
N	-	Rotor Speed
Re_i	-	Impeller Reynolds Number
u	-	Velocity Components
e	-	Total Energy per Unit Volume
q	-	Heat Flux
P	-	Stress Tensor
$P(\rho,T)$	-	Scalar Pressure
I	-	Unit Diagonal Tensor
T	-	Temperature
k	-	Turbulence Kinetic Energy
ε	-	Turbulence Dissipation Rate
z	-	Height

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND STUDY

In many industrial applications mixing plays an important role, not limited to, petrochemical, food processing and paint manufacturing. The mechanically stirred vessel is one of the most common tools used for mixing and the most important component of this vessel is the impeller. The flow resulting from mechanical agitation is known to be turbulent.

careful analysis and selection of effective impellers will produce an efficient and homogeneous combination. Companies that rely on mixing operations need to invest in mixing equipment that can be used on a wide variety of production and application items, including fluid volumes and varying levels of viscosity. Homogeneity in industries is key, because resources are used.

The measuring efficiency of the mixing performance of an impeller may be factors such as impeller geometry, number of impeller blade, impeller size, working fluid, the impeller configuration often operating angular velocity. Mixing phase also involves three processes, called propagation, dispersion, and diffusion. Distribution known as the fluid's bulk diffusion which is called macromixing. Usually, the quantity of a mixing device is defined in terms of the dimensional Reynolds impeller number Rei, expressed in the fluid vessel as a function of the density of the fluid, the dynamic viscosity μ, the diameter of the vessel, D, and the speed of the rotor, N in rev / s.

An important role of Computational Fluid Dynamics (CFD) in fluid flow is to analysis use of numerical approach and simulation. By using this program, we can research complex and creative design and implementation in any setting easily, removing prototype requirement. CFD better in selecting the best design, also predict any possible outcome before any real fabrication.

1.2 Problem Statement

The problem with the current design of the impeller is that although they have the conditions needed to produce turbulent flow, the flow is either too localized or too inefficient to effectively transfer the kinetic energy momentum and turbulence. While these original designs should work well enough when used in less viscous liquid, but initial findings indicate that the performance of the impellers deteriorated at more viscosity liquid as the Counterflow impeller showing considerable loss in efficiency. This demands a redesign that can fix the initial design flaw.

1.3 Objective

The objectives of this project are:

I. To evaluate and analyse mixing performance of the original impeller designs and their modified counterpart by analyse its velocity vector and wall shear vector.

1.4 Scope of Study

Research scope for study and designing impeller for mixing paint project involves mainly the topic of mechanical engineering technical drawing design and fluid dynamic design. And the most important thing is Computer Assisted Engineering, which explains a great deal about Computational Fluid Dynamic, CFD software. There are also associated core subjects of mechanical engineering, such as mechanics, and engineering materials.

1.5 Scope of Work

The research was carried out to test and analyse the efficiency of mixing of two initial impeller designs, the Sawtooth and the Counterflow impeller, and their improved version, the Inclined Blade Sawtooth and the 4-side Impeller.

The designing of the impellers is using CATIA V5 software. The impeller and the vessel were modelled as laboratory scale equipment. Which the scale is varied depends on the application.

For this study the research is carried out using numerical rather than experimental method. By the use of Computational Fluid Dynamics (CFD) software, the impeller's fluid flow could be visualized and its parameters tabled and displayed for evaluation and analysis.

The CFD program that is used for this analysis will be ANSYS Fluent 18.2 to simulate impeller rotation and fluid flow. The simulation method used is a study of 3-Dimensions. The design of the impeller will be imported from CATIA into ANSYS Design Modeler in order to define the position of the impeller in relation to the mixing vessel on the X-Y-Z plane before being meshed, to define the boundary condition and to fluently imported where the necessary parameters are set. In the CFD Postprocessing the simulation results are evaluated graphically and statistical analysis.

1.6 Significance of Study

The simulation results graphical analysis such as the velocity streamline and the wall shear vector would provide insight on how the impeller's structure and design affects the flow as the impeller passes through the fluid. The statistical evaluation will show the degree of improvement of the new designs compared to their original counterpart and will decide which of the designs may produce better mixing for the case of paint mixing. This diagram will allow the impeller to understand and know the induced flow, which in turn will help to pick the best impeller for paint mixing application and the future of impeller design.

CHAPTER 2

LITERATURE REVIEW

2.1 Mixing

This diagram will allow the impeller to understand and know the induced flow, which in turn will help to pick the best impeller for paint mixing application and the future of impeller design [1].

In a turbulent flow, a large number of eddies of different sizes coexist, and the process of splitting the bulk flow into smaller eddies is called dispersion. Dispersion is referring to the combining of the intermediate length scale hence it is called mesomixing. The degree of homogeneity resulting from dispersion is restricted to the size of the smallest eddies that could occur in a specific liquids [3].

Diffusion is the smaller-scale mixing, which also known as micromixing, is comparatively slow but still more efficient process on a small length range. Mixing of low viscous liquid like water with 1 mPa viscosity with momentum transfer and turbulence occurs. Although highly viscosity liquid, causes frictional drag on the impeller, the high speed from the impeller to the region around the impeller [4][5].

2.2 Viscosity

A fluid's viscosity is a measure of its resistance to gradual deformation cause of strain [7]. In liquids it can be translated informally as the "thickness" of the liquid In other words, with higher viscosity fluid getting more friction between molecules, which makes it more difficult for fluids to deformed [8].

Viscosity is important because mixing of viscous material can require a variety of industrial processes. It is well known that viscosity have a significant effect on the fluid flow, mixing, and mass and energy transport. Therefore, an understanding of liquid viscosity impact on mixing is highly essential[9].

The fluid temperature also influences the viscosity of a liquid, and the relation between fluid viscosity and temperature is inversely proportional. Liquids have lower viscosity at higher temperatures due to the high-energy molecule which resists the cohesive force between them.

A fluid is known as a Newtonian fluid if it behaves in accordance with the Law of Newton, where the viscosity is independent of stress. Newtonian can be called gasses, water and other liquids such as industrial paint. Non-Newtonian liquids violate Newton's Law in which the viscosity is dependent of the stress applied [10]. For process of mixing, the higher viscos fluid would generally require more kinetic energy and more momentum from the impeller to overcome the viscous force[11]. Therefore, a careful selection of impeller design is important as an efficient impeller could produce higher turbulence at lower rotational speed. The table below shows the viscosity of different substance.

Table 2.1: Viscosity for Different Substances at 20˚C

Substance	**Viscosity (mPa.s)**
Waters	1.0016
Whole Milk	2.12
Honeys	4000
Glycerines	648
Paint for industrial	100

There is no common viscosity for industrial paint because the value will differ depending on the composition of the paint such as binder, thickener, solvent, any additive and extender.

2.3 Impeller

Impeller is a mechanical component used for transferring the kinetic energy and the momentum from the impeller's rotational motion to the surrounding fluid. In a mixing vessel, the impeller serves as an essential part in order to provide a homogeneous solution by reducing the fluid concentration gradients [11]. When an impeller travels through the fluid, the impeller's motion is transmitted to the surrounding fluid and the impeller's geometry and design causes fluid flow.

An impeller's mixing efficiency depends on factors such as the geometry of the impeller, the size of the impeller, the number of impeller blades, the fluid used, the impeller configuration and the angular velocity. Usually, the quantification of a mixing device is known as the dimensional Reynolds impeller number Rei, expressed in the fluid vessel as a function of the density of the fluid, the dynamic viscosity of μ, the diameter of the vessel, D, and the speed of the rotor, N in rev / s. As shown in Equation below [12]:

$$Re_i = \frac{\rho N D^2}{\mu}$$

For this analysis, an industrial paint density norm will be used for 880 kg / m3, with dynamic viscosity of 0.05, rotational impeller speed will be determined as 5 rev / s, and vessel diameter of 0.16 m, resulting in a Reynolds impeller number of Rei=14080. Because the flow at Rei > 10,000 is considered totally turbulent, this research can be viewed as a completely turbulent mixing review.

2.4 Turbulence

Turbulent flow can be characterized as a fluid movement pattern with dramatic changes in its pressure and velocity as compared to a laminar flow in which the fluid flows in parallel layers without any interruption in layer formation. [13]. Turbulence occurred by excess kinetic energy in sections of a fluid flow that surmounted the damping effect of the viscosity of the fluid.

Turbulence flows depended on the viscosity of the fluid, and it is easier to occur in a lower viscosity of fluid like water, also it is more difficult to produce on high viscous fluid. The turbulence flow could be predicting by using a number of Reynolds which is a ratio of a fluid kinetic energy to its viscous of damping [14].

Turbulence is important in a mixing phase and is accomplished from a constant supply of energy which is the impeller's rotational motion. The supply of energy needs to be both required and maintained as turbulence dissipates at a rapid rate as its kinetic energy is transformed by viscous shear stress into internal energy. It produces eddies of varying size in the vessel, allowing more eddies to be created before it reaches a small scale that is necessary for diffusion to happen. The rate on which diffusion happens is known as the Kolmogorov Length Scale [15].

2.5 Computational Fluid Dynamics

An important role of Computational Fluid Dynamic (CFD) played in fluid flow analysis using numerical approach and simulation. This method make engineers easier on research and implement both new and complex designs in a virtual world, removing the need for any prototype [16]. This software also helps in determine the best design, predict the possible outcome before proceed to any real fabrication.

Mixing parameters can be studied using CFD include the flow types, mixing time, power requirements and velocity patterns [17]. Impeller in architecture played crucial role in agitated mixing and showing result of flow pattern that later could be theoretically visualized using CFD.

A variety of methods were employed in mechanically agitated mixing for turbulent flow simulation. With regard to CFD, the typical approach is the Reynolds-averaged Navier-Stokes (RANS) equation. The RANS method averages the equations over a time interval or a series of similar fields and is commonly used in the estimation of stable-state solutions [18].

The Navier-Stokes equations represent the turbulence characteristic and are the basis used to define the flow phenomenon. These equations are based on the conservation laws: continuity, momentum, and energy conservation laws give below:

Continuity Equation,

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho u) = 0 \qquad \textbf{(2.4.1)}$$

Momentum Equation,

$$\frac{\partial \rho u}{\partial t} + \nabla.(\rho u u) = -\nabla.P \qquad \textbf{(2.4.2)}$$

Energy Conservation Equation,

$$\frac{\partial \rho e}{\partial t} + \nabla . eu = -\nabla(u.P) - \nabla . q \qquad \textbf{(2.4.3)}$$

Where u, ρ, e, q are the velocity components, density, total energy per unit volume and heat flux respectively. P is the stress tensor, and in a Newtonian fluid it is defined by:

$$P = p(\rho, T)I + \frac{2}{3}\mu(\nabla . u)I - \mu[(\nabla u)+(\nabla v)^T] \qquad \textbf{(2.4.4)}$$

Where $p(\rho,T)$ is the scalar pressure, I, is a unit diagonal tensor, T is the temperature and μ is the dynamic viscosity coefficient.

The k-ε model is a two-equation model that comes under RANS, where the Turbulent Kinetic Energy (TKE) and its dissipation rate (π) are used to characterize the unstable flow fields [16]. K-π model has good precision, time savings and is ideal for a wide range of studies on turbulence flow [18].

Generally speaking, the key drawback associated with the RANS models is that it fails to accurately predict the specific characteristics of complex flows, as the k-ε model assumes turbulence isotropy. Nevertheless, it is the most fitting CFD tool available in term of computational speed and reliability [19].

CHAPTER 3

METHODOLOGY

3.1 Methodology Overview

Analysis Methodology is types of methodology used to complete this project. Methodology by research is a way of systematically answering the research problem. This offers the best level for the researcher to schedule job progress with the ultimate aim of solving the job using the correct methodology in specifications.

The project begins by gather the needed information and having literature review of subject that involve in developing impeller. Detailed information is required to have proper design of the impeller as the best design for the impeller and to understand its fluid flow result before the design process starts. The initial configuration for the impeller is the Counterflow and Sawtooth impeller. The reason these two impellers are chosen is because they possess the specific flow feature, and this impeller has been commonly used in industrial paint mixing.

The initial prototypes of the impeller were first developed before being imported into the ANSYS where they would be pre-processed and simulated. The simulation result will then be evaluated and analysed in order to compare the flow characteristics and identify the weakness in the fluid flow induced.

Pre-processing and simulation process by graphically are then observed for the modified design then the result will be analysed and evaluating with the original impeller design to determine their efficiency and effectiveness. This project needs to design impeller and simulate the rotating impeller in a CFD software where the study's flow characteristics.

3.1.1 Computer Aided Design Process

The impellers design was developed by using software of CATIA V5. This software is chosen because of the software's familiarity and experience, and because it can provide a multitude of command tools that promote the impeller design process. The impellers were designed as laboratory equipment with a regular 80 mm diameter and a 20 mm diameter of the impeller.

CFD Pre-Processing

The impeller designs then imported from CATIA into ANSYS for pre-processing. It is then setting into meshing tool to mesh where the domains are distinct into small volumes of power. The mesh usually consists of three different parts, region of the enclosure and region of the revolving area, and region of the impeller.

CFD Setups

The simulations were prepared with fluent solver. By using different pressure based, absolute velocity, and steady state condition with gravity acting on the negative z-axis. The material chosen for the fluid is paint with a density, ρ, of 880 kg/m3 and constant viscosity, μ, of 0.04 kg/m.s as discussed on the literature review.

By the technique of Multiple Reference Frame (MRF), the cell zone for the rotating domain will be set to frame motion with 300 rpm at the z-axis direction. This system would conduct the fluid domain around the impeller, close to how the exhausted fan operates. As for the Moving Wall process, the moving domain wall, which reflects the shape of the impeller, is set to rotate in the z-axis direction at rotational velocity of 300 rpm.

CFD Post Processing

The post-processing was done using ANSYS CFD-Post. There are two sets of data taken from each simulation to be evaluated; The velocity streamline, velocity and wall shear vectors.

3.1.2 Design Process

Two of original impellers and their adjustment are designed using CAD software to set the dimensions of the impeller on a lab scale of 80 mm total impeller diameter and 20 mm impeller body diameter. The designs are designed to generate turbulence in the vessel and facilitate mixing with standard laboratory scale.

Counter flow impeller

The Counter flow impeller always impose a turbulent flow characteristic cause of the two blades at different orientation with a buffer plate in between. These two blades allow for two separate flow directions in one movement and thus help to generate turbulence and increase fluid mixing. As a typical laboratory scale, the impeller's blade will enforce a length of 20 mm with a width of 20 mm on the inside and 10 mm on the outside, the buffer plate will be 2 mm thick and the outer blade will be 8 mm long and 20 mm broad.

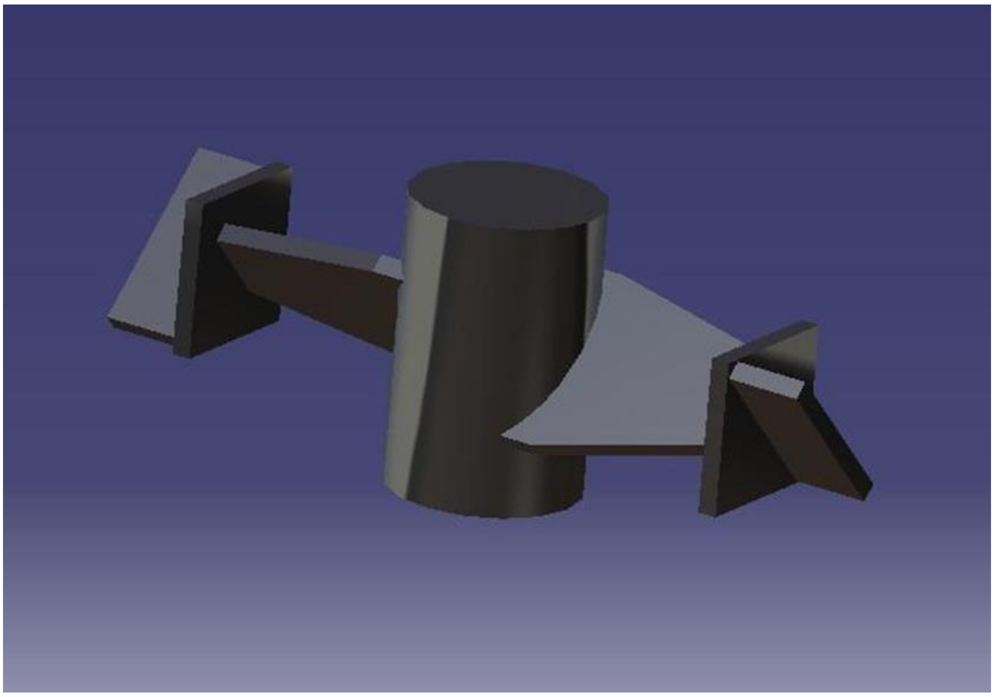


Figure 3.1.1: Counter Flow Impeller

Sawtooth Impeller

The Sawtooth impeller consists of a disk with twelve alternating protruded blades on its edge having a perpendicular height:7 mm from the base. This makes a slotted impact fluid flow as the fluid flows upward and downward the protruding edges as the impeller rotated such that turbulence flow can be generated.

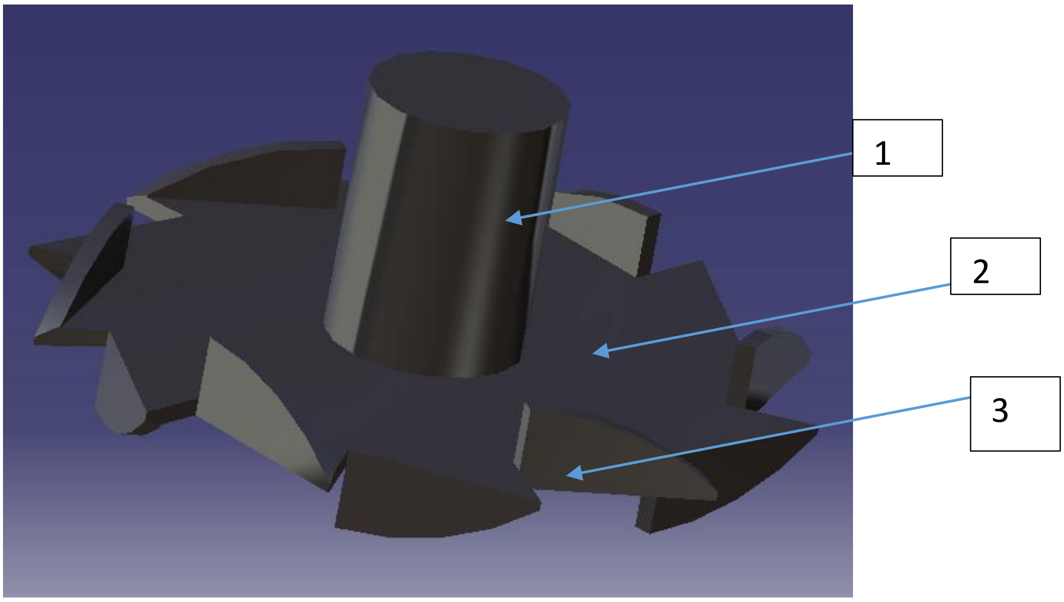


Figure 3.1.2: Sawtooth Impeller

Sawtooth impeller mainly would consist of three main parts which labelled in Figure 3.1.2 above:

1. *Impeller Body.*

   Purpose and dimension of the impeller body in the laboratory which same as discussed on the Counterflow impeller.

2. *Impeller disks plate*

   This part plays an important work when flowing the fluid flow which is to create the "slotted" fluid effect in competition with the protruded blades. The disk plate is 80 mm outer diameter and 2 mm thick.

3. *Protrude Blade*

This part is the most priority source of fluid flow as it will create slotted flow effect by the blades will directing the flow upwards and downwards. This design comprises 12 number of blades each with a height of 10 mm

The design of Sawtooth impeller in CATIA V5 need multiple steps because of its complex and detail design. The required steps include padding the impeller plate and 2-Dimensional sketching, designing arc-shaped 2-Dimensional sketching of its blades and creating circular patterns to create multiple blades.

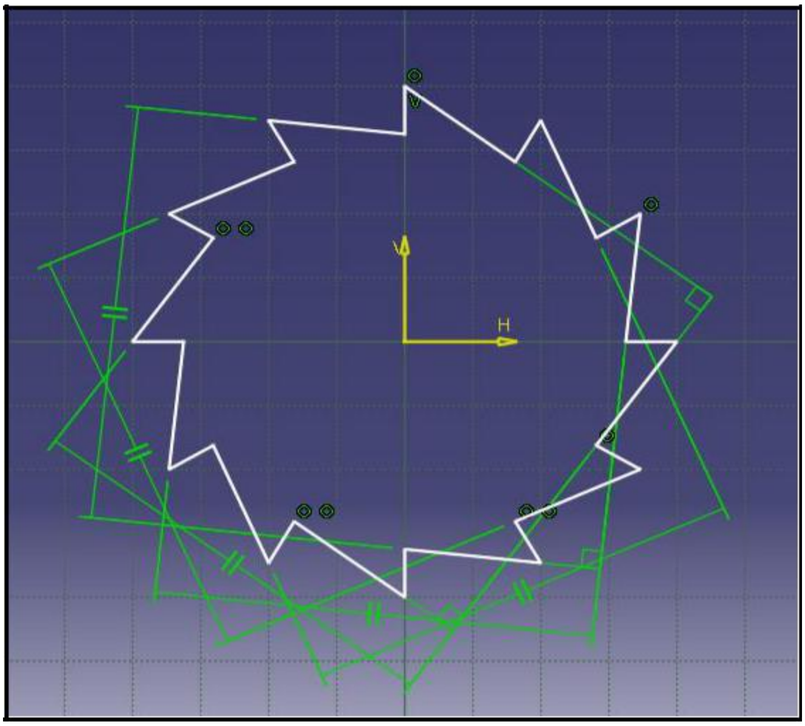


Figure 3.1.3: Dimensional of Sketching for Sawtooth Disk Plate

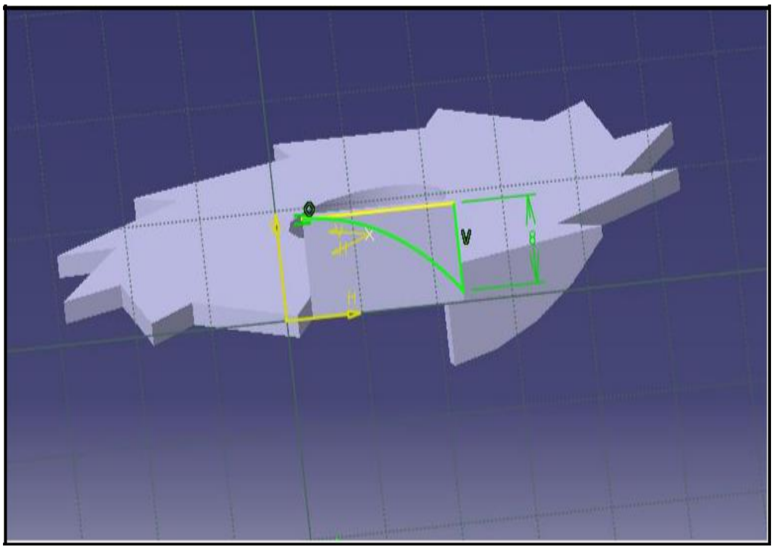


Figure 3.1.4: Dimensional of Sketching for Sawtooth Protruded Blade

The 4-Sided impeller

The 4-sided impeller are updated version for counter flow impeller with four blades at different orientations that designed to mimic the slotted fluid flow effect of the stitching sawtooth impeller while allowing the same path of fluid flow in different directions and thus retaining the design of the original criteria.

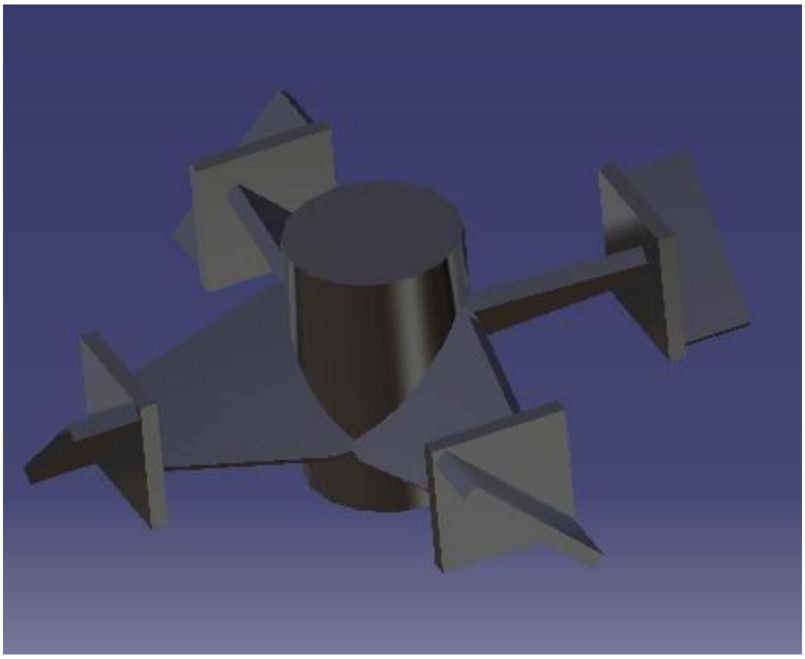


Figure 3.1.5: 4-Sided Impeller

Inclined Bladed Sawtooth Impeller

For this design is slight change to the Sawtooth impeller. In original impeller the protruded blades are oriented at 90˚, but on this impeller the blade oriented 45˚ thus the radial flow of impeller improved.

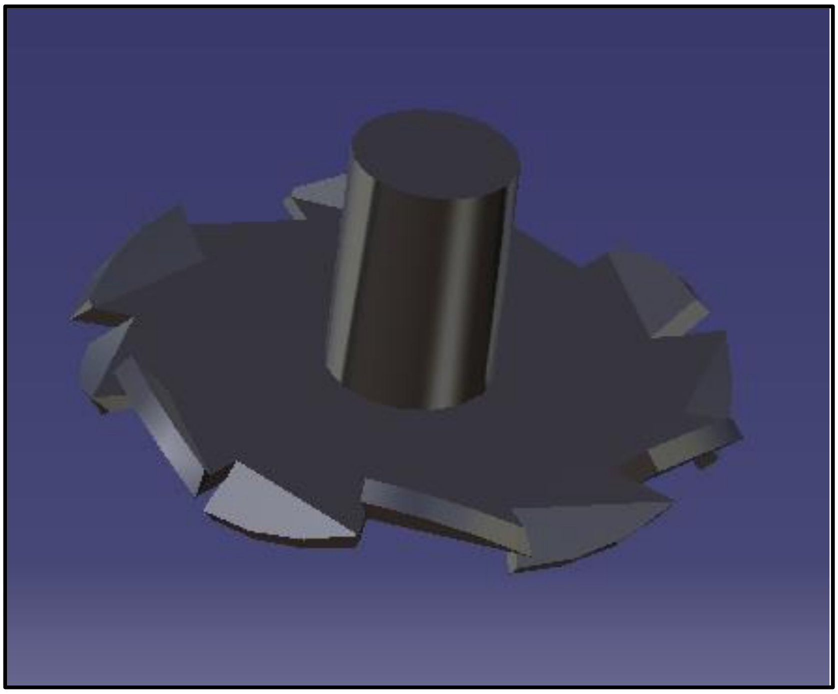


Figure 3.1.6: Inclined Blade of Sawtooth Impeller Design

Tilted protruded blades are positioned at 45 as shown in Figure 3.1.6 above with the intention of extending the draw region of fluid mixing by dispersing the concentrated flow at the middle region produced by the radial flow action. In other words, the flow would increase radially by providing wide area of the draw field, which would ultimately increase the vertical fluid flow.

3.2 General Simulation Structure

The modelling will be done using Ansys Fluent Workbench 18.2 and the process are as Figure 3.2.1 below:

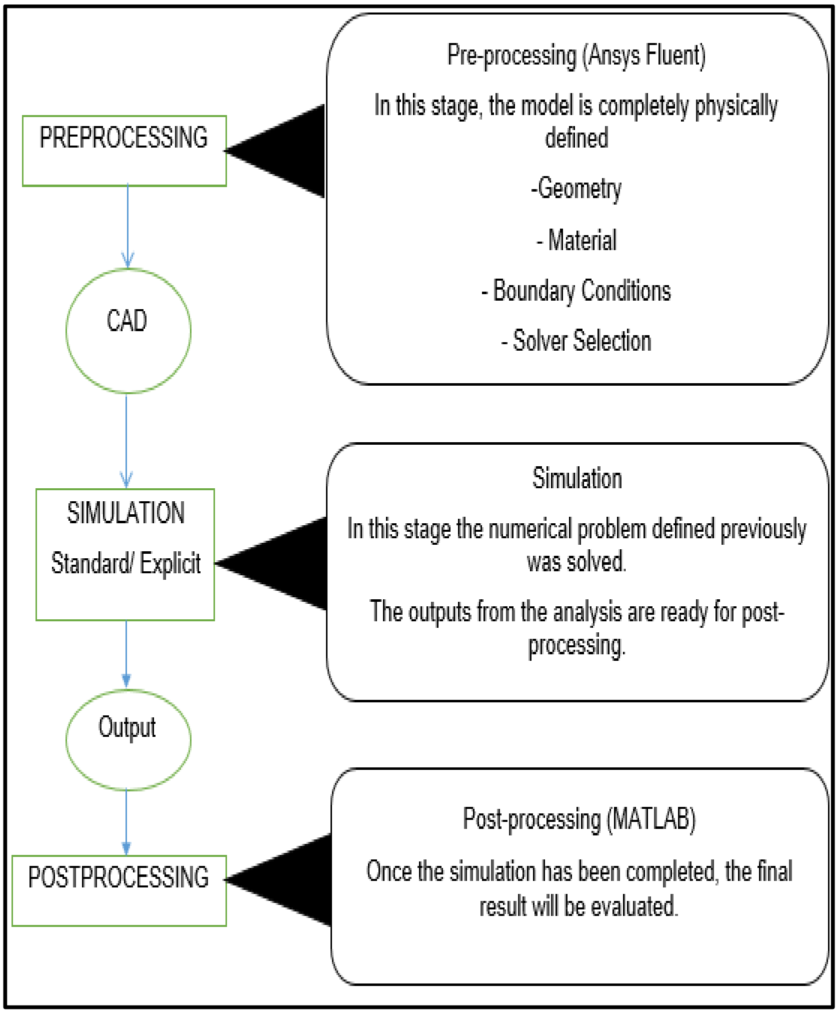


Figure 3.2.1: Ansys Fluent's General Procedure

In this study, the CAD is done before the pre-processing instead of during the pre-processing as the impeller design is done using CATIA before being imported to ANSYS as shown on Figure 3.2.1

3.3 CFD Pre-Processing

Pre-Processing is where the analysis domain, boundary state, and parameters are set before the simulation is performed. ANSYS Workbench 18.2 has three pre-processing phases which are Geometry, Mesh, and Setup. Geometry and Meshing will be covered in this section while Setup will be discussed in another section due to the elaborate steps required.

3.3.1 Geometry

In Geometry, the design of the impeller is could be created using ANSYS built-in CAD such as Project Modeler Space Argument. In this analysis, the impeller design was done using CATIA V5 and imported in Geometry Pre-Processing into ANSYS Design Modeler. As this study is about 3-Dimensional flow, the form of analysis must be selected as 3D before using the Geometry Pre-Processing method.

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Geometry
4	Directory Name	FFF
5	⊟ Notes	
6	Notes	
7	⊟ Used Licenses	
8	Last Update Used Licenses	
9	⊟ Geometry Source	
10	Geometry File Name	C:\Users\Owner\Desktop\FYP\Simulations\MW\MULTIFLOWmoving wall 2000\MULTIFLOW_files\dp0\FFF\DM \FFF.agdb
11	⊟ Advanced Geometry Options	
12	Analysis Type	3D
13	Compare Parts On Update	No

Figure 3.2.2: 3D Geometry Analysis Type Selection

Once the impeller designs were imported, the coordinate axes were defined with the impeller positioned to rotate along the *Z*-axis direction. The command used during Geometry Pre-Processing for this study is Enclosure and Boolean.

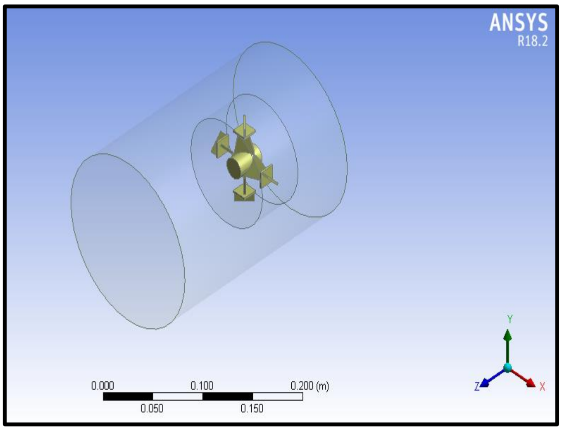


Figure 3.2.3: Positioning of the Impeller in Design Modeler

Enclosure

In Design Modeler two domains were generated using the Enclosure command. Which represented the region of the impeller and the mixing vessel called "rotating" and "closing" respectively.

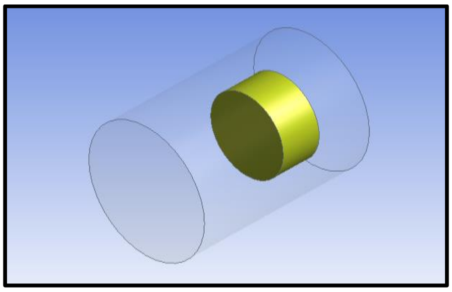


Figure 3.2.4: Rotating Domain

The rotating domain is a cylindrical fluid enclosure aligned in the *Z*-axis having a radius, positive *Z*-axis, and negative *Z*-axis cushion of 0.01m from the impeller.

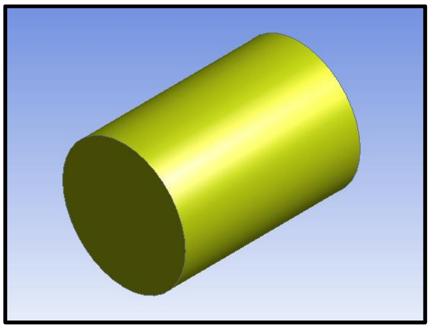


Figure 3.2.5: Enclosure Domain

The enclosure domain is a cylindrical fluid enclosure centered in the Z-axis with a 0.01 m x rotating domain radius, positive Z-axis and negative Z-axis cushion.

Boolean

Boolean is the command used to describe the two or more geometries that connect. Boolean is made up of facets of Join, Deduct, Overlap and Imprint. With the Boolean command in place, the research domain is divided into three different domains: Impeller, revolving and enclosure.

The Boolean command used for this analysis is the Deduct. Two sets of Booleans have been used, one is to remove the impeller from the rotating domain, leaving an impeller-shaped hole in the middle of the rotating domain empty. Another Boolean had been used to detach the revolving domain from the enclosure.

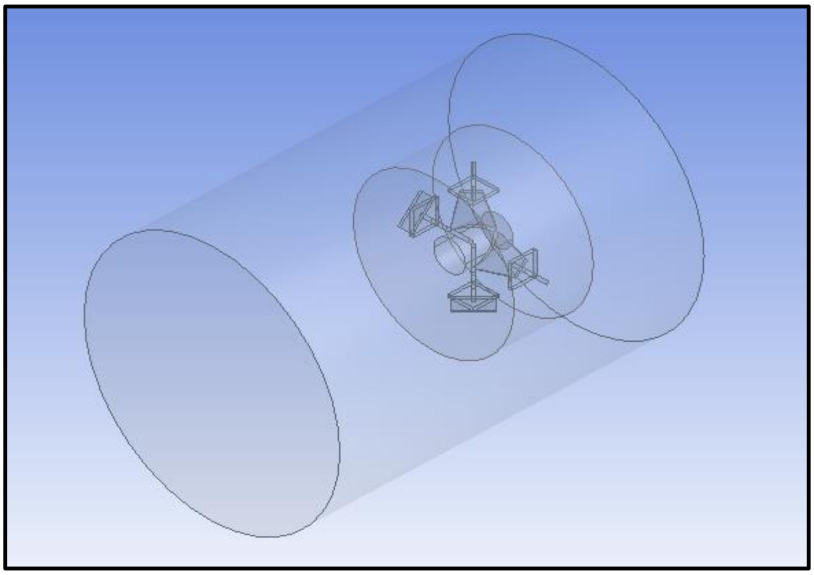


Figure 3.2.6: Finalized 4-Sided Impeller Geometry

3.3.2 Meshing

The finalized geometry was imported to the meshing tool to define and discretize the geometry into small control volumes for simulation and calculation. There are several ways to create a mesh, the mesh could be either manually or automatically generated and to decide whether to use a form of fine, medium or coarse structured mesh. Generally speaking, the finer the mesh, the more accurate the result is, but more computational time and power is required.

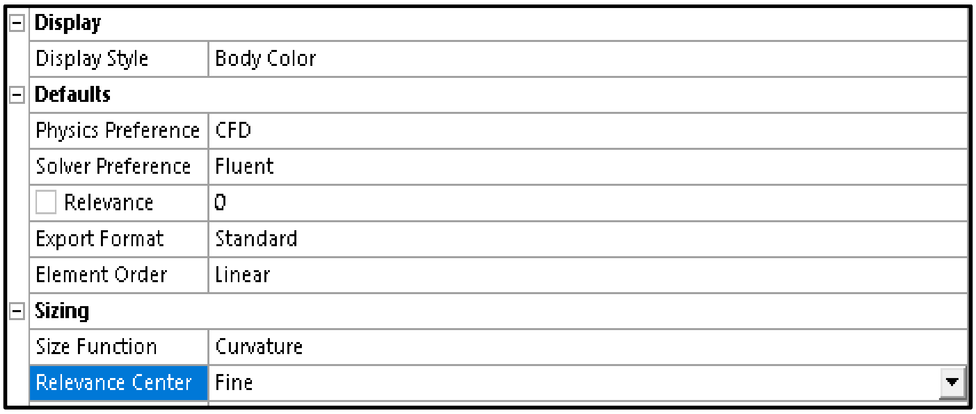


Figure 3.3.1: Fine Mesh Sizing Selection

The meshing quality required depends on the type of study, as certain study needed a higher quality mesh to obtain accurate specific parameters such as velocity, pressure and so on while comparative studies such as this one, meshing quality is not critical so long as all the analysis uses the same method of meshing. In the present study, a fine structured mesh is selected and automatic mesh generation was used. During meshing the inlets, outlets, etc. are usually defined for setting up boundary conditions, but since there is no inlet or outlet in this study, specific naming and defining of the domains are not required.

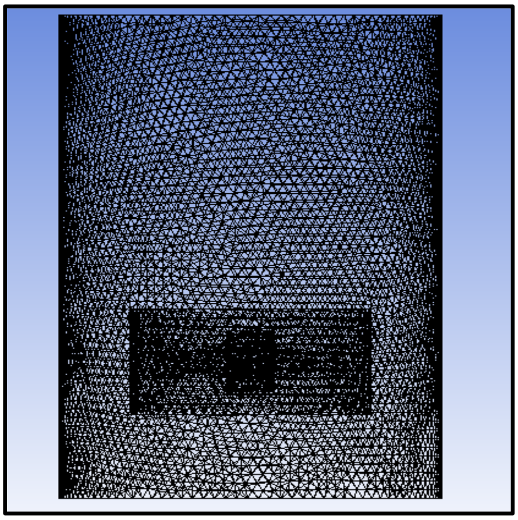


Figure 3.3.2: Automatic Generation Fine Meshed Region

3.4 CFD Setup

If the meshing is finished and imported into ANSYS Fluent, parameters such as the solver, fluid properties, cell zones, and boundary conditions, etc. need to be set for simulation. Fluent is set and runs for 3-Dimensional analysis.

3.4.1 General Solver

General solver is where the solver type for the simulation is selected, for this study, the solver that is used is steady, pressure-based, absolute velocity formulation. The distinction between a steady and a transient solver is that in a steady state the simulation lacks the terms that deal with time while in transient, these terms are taken into account and thus the complexity of the solution is increased and the computational time is increased. Stable state solver has proven adequate for turbulent mixing [16].

Density-based solver is usually used in simulation of high-speed compressible flow while pressure-based solver is used in low speed incompressible flow. For velocity formulation, relative velocity is usually used in the case where the majority of the fluid is rapidly rotating. Since in this study only the impeller is rotating, the absolute velocity formulation is used. Gravity plays a role in case of turbulent mixing as it affects the vertical fluid flow hence gravity is set to act on the negative *Z*-axis direction.

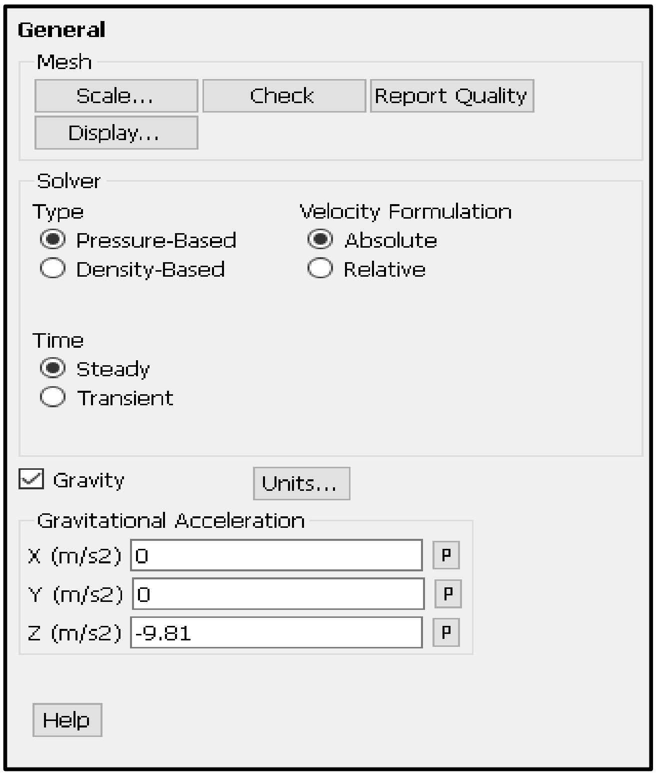


Figure 3.4.1: General Solver Used

3.4.2 Simulation Models

For standard wall function, this study uses viscous realizable k-π model. The realizable model was used instead of the standard model because it uses an additional eddy viscosity state and a dissipation transport equation that is derived from the variance in vorticity, resulting in a more precise solution thus increasing the computational time and is ideal for turbulent mixing studies. The basic wall function is the most widely used wall function and is ideal for a wide variety of simulation, the layout of wall functions only needs to be modified when highly sophisticated mesh is used.

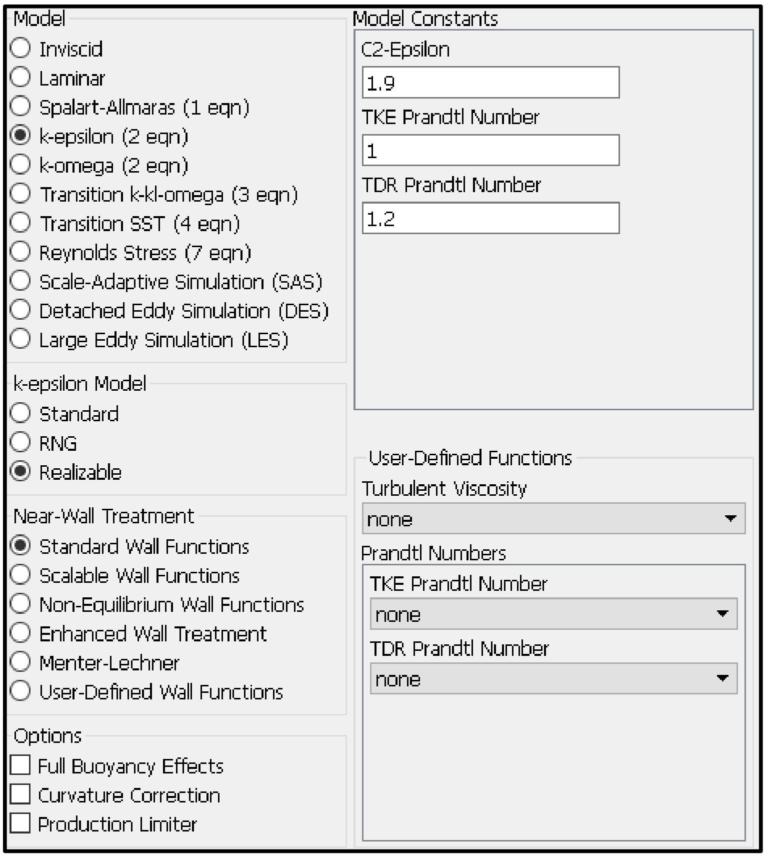


Figure 3.4.2: Simulation Models Used

3.4.3 Materials

The fluid properties and material of the impeller had been described in the fluent material tab. The fluid used for the analysis is the user-defined industrial paint with a density of 880 kg / m3 and a viscosity of 0.05 kg / m3 while the content of the impeller was set to 7700 kg / m3 of user-defined stainless steel.

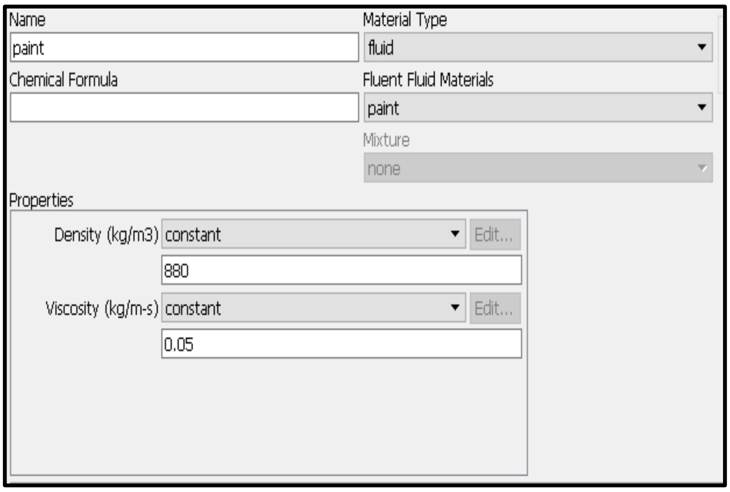


Figure 3.4.3: Fluid Properties

3.4.4 Cell Zone Conditions

In cell zone conditions, the behaviour of the domains was defined. In this study, two different methods are used to simulate the rotation of the impellers which are the Multiple Reference Frame (MRF) and the Moving Wall method. The multiple reference frame method utilizes the rotation of the domain to represent the impeller's rotation, so the conditions of the cell zone must be set in the MRF system. The rotating domain is set with frame motion rotating in the negative direction of the Z-axis at a rotational velocity of 300rpm while the rotating domain is set to stationary in the Moving Wall system.

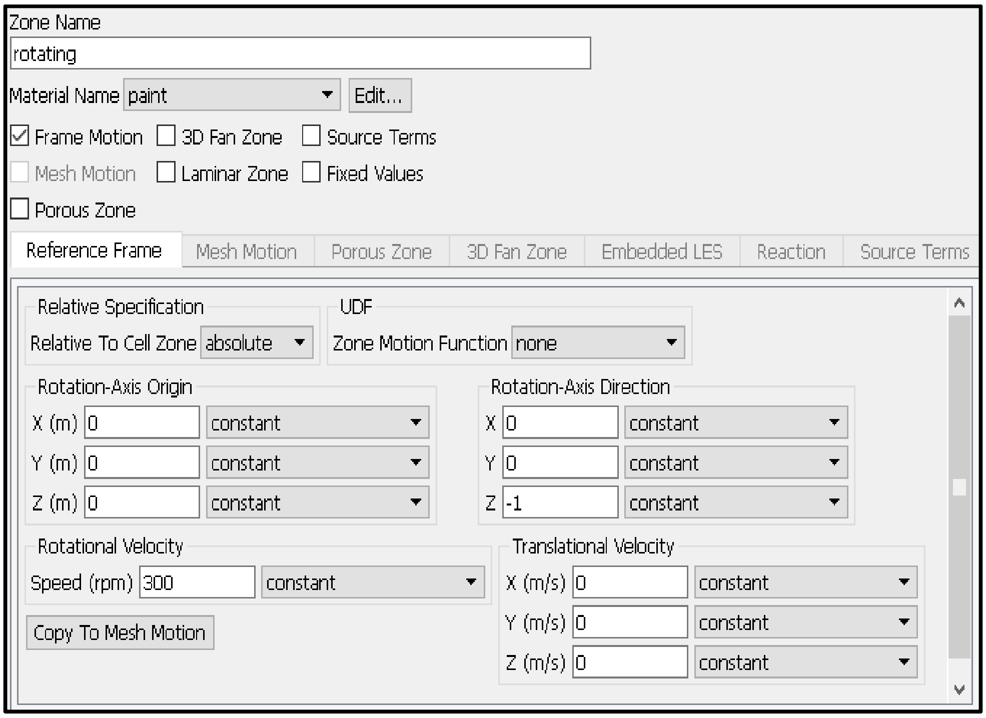


Figure 3.4.4: Multiple Reference Frame Method Rotating Domain Cell Zone Condition Setup

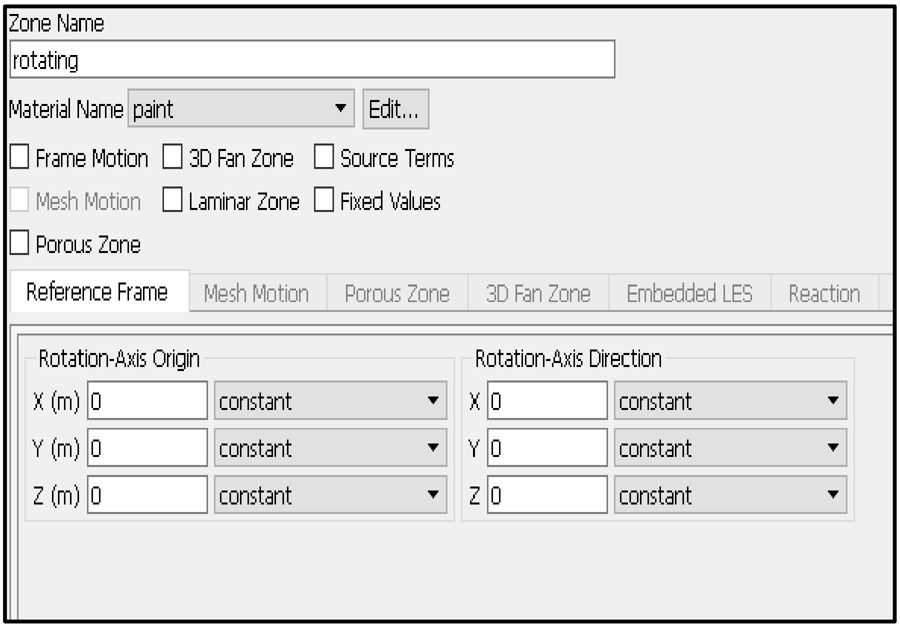


Figure 3.4.5: Moving Wall Method Rotating Domain Cell Zone Condition Setup

3.4.5 Boundary Conditions

In the setup of boundary conditions, the domain walls are described as being either stationary, rotating, or translated. In the MRF method, the rotating domain wall is set to be stationary while the moving domain wall is set to rotate at an absolute rotational velocity of 300 rpm in the z-axis direction in the Moving Wall method. The enclosure wall is set to stationary for both the system.

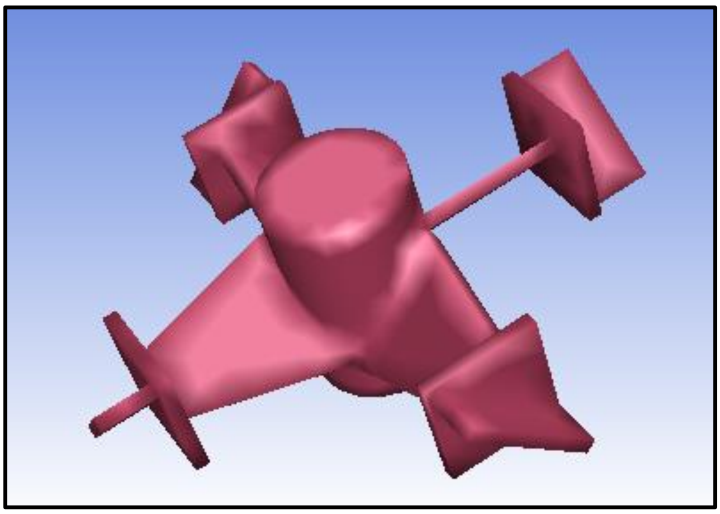


Figure 3.4.6: Rotating Domain Wall

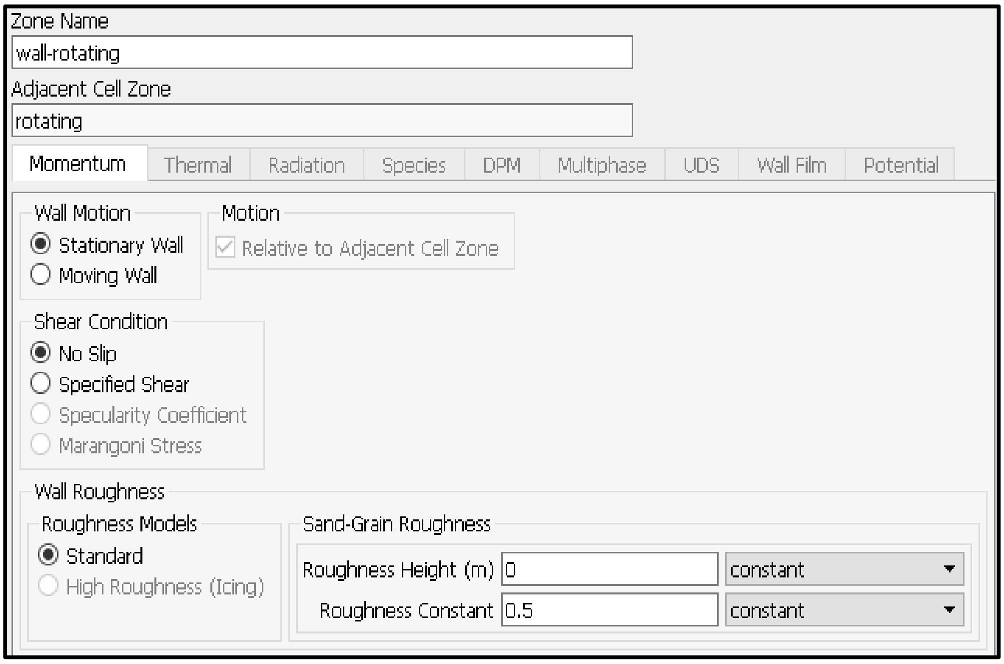


Figure 3.4.7: Multiple Reference Frame Method Rotating Domain Wall Boundary Conditions Setup

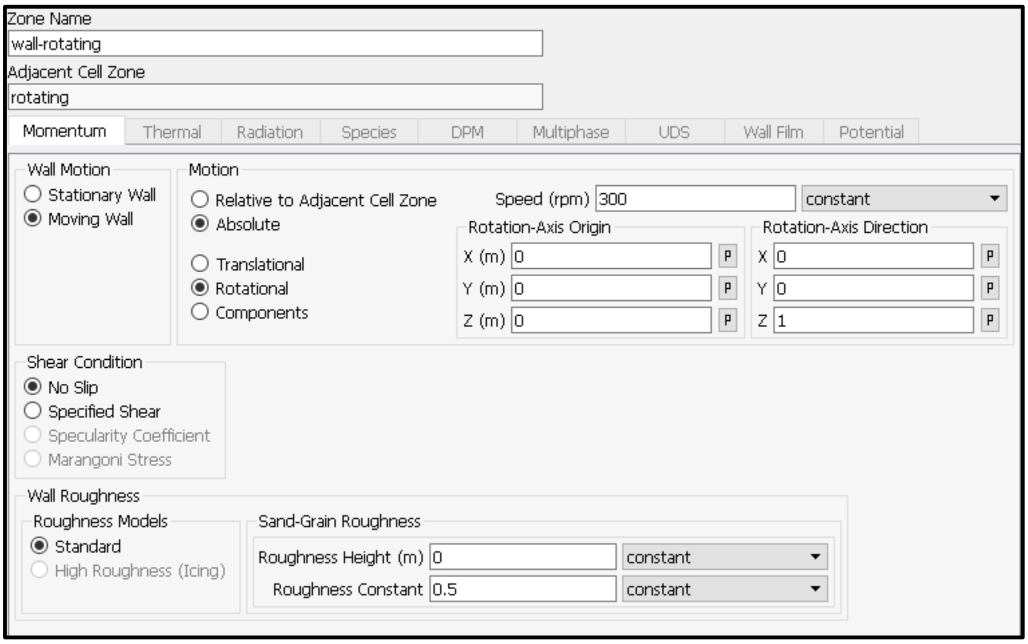


Figure 3.4.8: Moving Wall Method Rotating Domain Wall Boundary Condition Setup

3.4.6 Solution Methods

The solution methods define the method used to compute the solution of the simulation. The methods used for this analysis are SIMPLE Velocity-Pressure Coupling Scheme with Least Squares Cell Based Gradient Spatial Discretization, Second Order Pressure, Second Order Upwind Momentum, Second Order Upwind Turbulence Kinetic Energy, Second Order Upwind Turbulent Dissipation Rate. The methods were left at default except the Turbulent Kinetic Energy and Turbulent Dissipation Rate which were set to First Order Upwind by default.

According to ANSYS Fluent user guide, the SIMPLE scheme is used as default as it is suitable for a wide range of application and mesh skewness. The same could be said for the Least Squares Cell Dependent Gradient, though it is the least accurate of all choices, it is made default because it balances time and accuracy of the computation. The transition from First Order Upwind to Turbulent Kinetic Energy and Turbulent Dissipation Rate is designed to increase computational accuracy. The First Order Upwind is the simplest Upwind scheme possible and Second Order Upwind Scheme improve the scheme by adding an additional data point which offers more accuracy on approximation of spatial derivative [20].

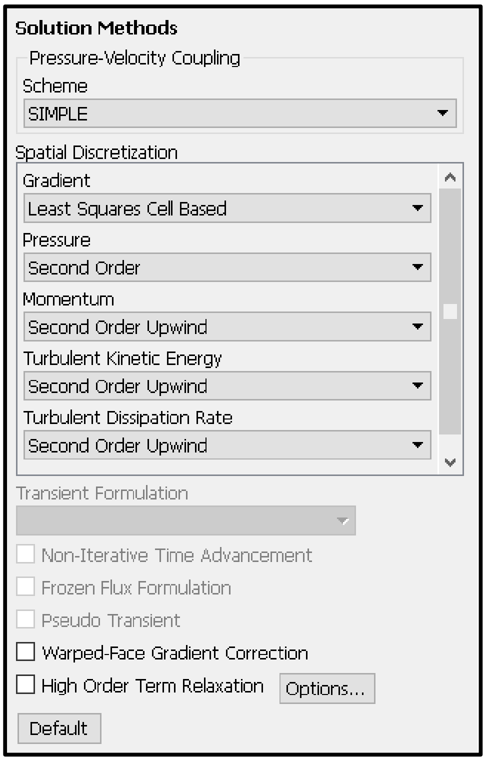


Figure 3.4.9: Solution Methods Setup

3.4.7 Initialization

Until starting the calculation, initialization is needed to provide the initial calculation values, while at the same time checking for errors during calculation setup. ANSYS Fluent provides two forms of initialization which are hybrid and normal. The meaning of both the initialization is as follows, according to Fluent's user guide:

Hybrid Initialization: Hybrid initialization uses a combination of different interpolation method and also using Laplace equation to solve the velocity and the pressure. Other variables such as temperature, turbulence and specific fractions will be fixed automatically based on interpolation and domain averaged value.

Standard Initialization: Standard initialization in ANSYS Fluent requires manually assigning the variable and the value. It is also can be used to observe small changes and assign difference in pressure, velocity and turbulent kinetic energy.

Generally speaking, in most case studies, hybrid initialization is more flexible and realistic to use because the software agrees on the best initial values for the case study, whereas Standard initialization is typically used when the initial calculation values are essential to the solution and the user needs to specify them. The Hybrid initialization approach was employed for this analysis.

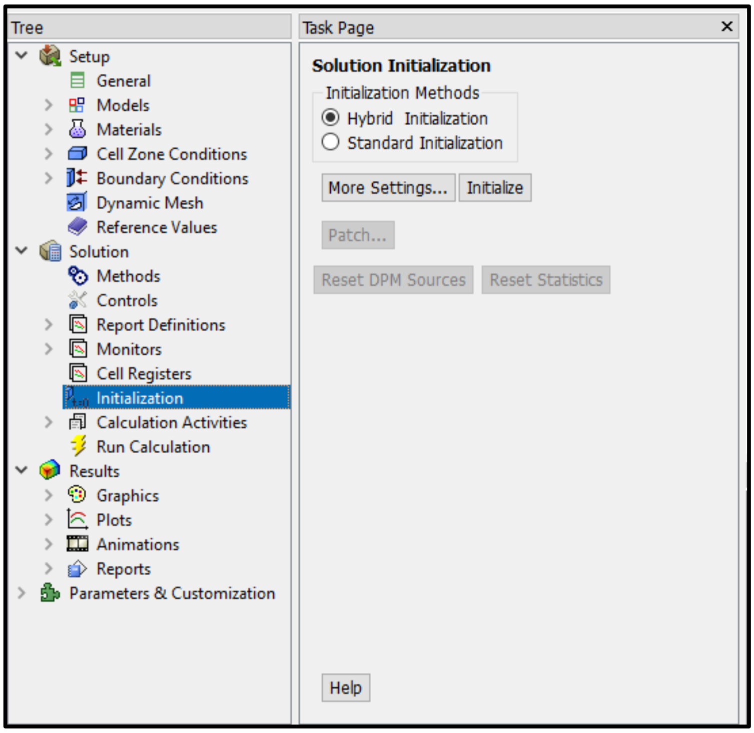


Figure 3.4.10: Solution Initialization Setup

3.4.8 Run Calculation

Once all the required setting up is done, the final step is to set number of iterations, since this is a steady state analysis which means that this study is time independent, the time step and step size is not required. The software is designed for this study to run every 1 iteration for 2000 iterations, with reporting and profile update intervals- The calculation was completed for 2000 iterations in 2-3 hours for the computational period of time.

3.5 CFD Post-Processing

Once the calculation is complete, the results are imported from Fluent into CFD- Post for processing before they could be presented in a graphical and statistical manner.

3.5.1 Wall Shear Vector Setup

The Wall Shear Vector is a graphical representation that is used to visualize the direction of fluid flow on the impeller surface as the impeller rotates. The Wall Shear Vector is created on the “wall rotating” location which is the impeller, using Vertex Sampling, Reduction Factor with a Factor of 1. The symbol used is the Line Arrow with a value of 2 for size and a value of 1. Reduction Factor option is used as the number of Seed Points are not as critical in Wall Shear Vector as they are in Velocity Streamlines.

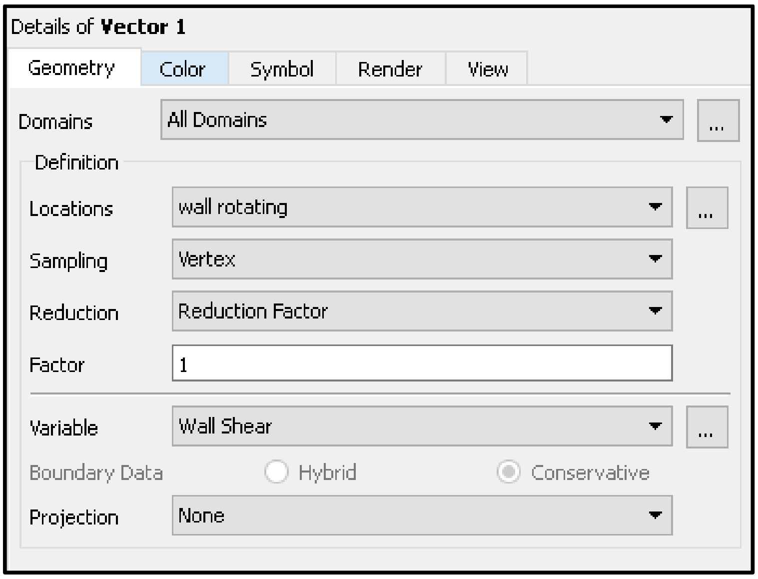


Figure 3.5.1: Wall Shear Vector Setup

3.5.2 Velocity Vector Setup

The Velocity Vectors, as opposed to the direction of fluid flow on the impeller surface, is a graphical representation of the direction of fluid flow in the mixing vessel. The Velocity Vector is generated using Vertex Sampling, Reduction Factor with a Factor of 3, at the "enclosure" position which is the mixing vessel. The symbol used is the 10-size Line Arrow. A high value for scale is needed to give a clearer view of the flow direction.

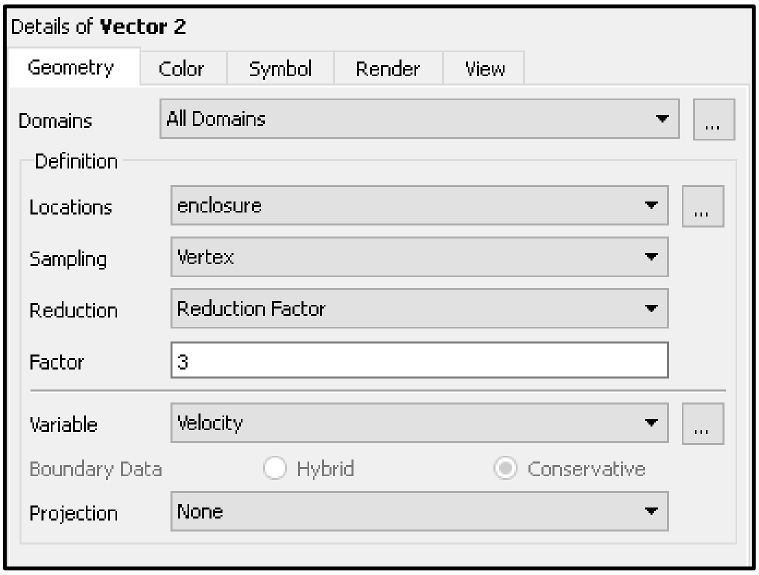


Figure 3.5.2: Velocity Vector Setup

3.5.3 Statistical Graphs Setup

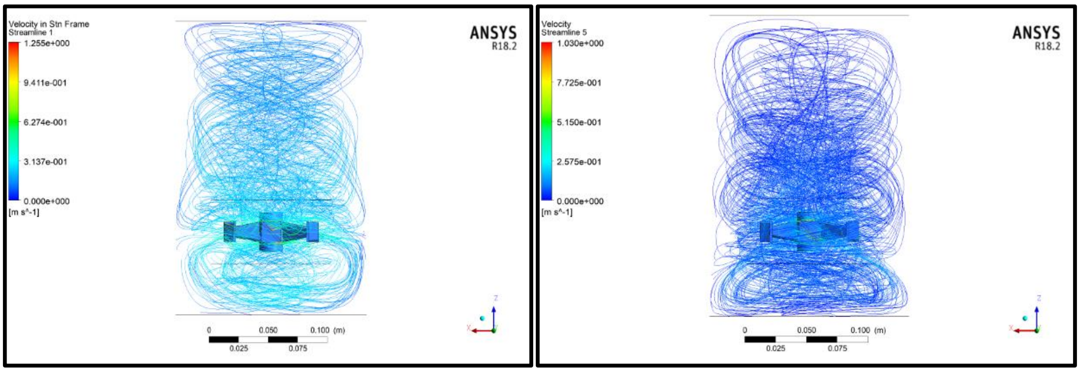
The statistical graphs are created to provide visualization on the numerical data of each impeller designs. The graphs are used by analysis of the Velocity produced by each impeller design to determine the mixing efficiency of the impellers. A lines are created at different height in the mixing vessel to represent the mixing parameters of the impellers at different regions in the mixing vessel. The chart created is an XY chart type with the *X*-axis representing the radial distance and *Y*-axis representing the studied parameters. As the graphs produced in Fluent reflect the performance of a given impeller at different heights, Fluent data had to be exported to Microsoft Excel to produce a graph representing the performance of all the impellers at each individual height.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Moving Wall Vs. Multiple Reference Frame Method

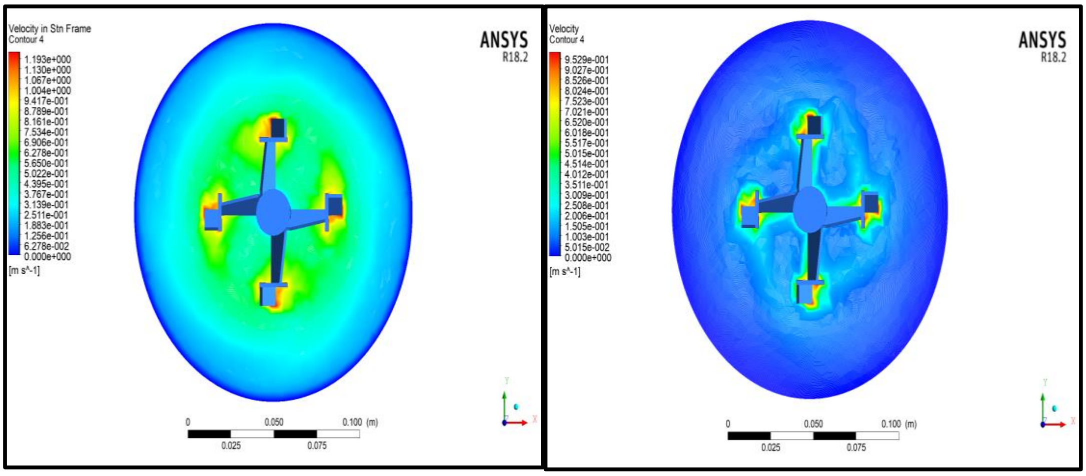
Difference between these 2 methods, as discussed in the section, is that the Multiple Reference Frame (MRF) method uses the rotation of the entire Rotating domain (with the impeller shaped cavity inside), Whereas the Moving Wall (MW) method uses the Rotating Wall rotation, reflecting only the geometry of the impeller to simulate the impeller movement. MRF is one of the most common methods used in analyzes involving rotary elements and in section, The MRF system will be tested and compared with the MW system to decide the best approach to use in the case study.



(a) **(b)**

Figure 4.1.1: Velocity Streamline Generated by (a) MRF; (b) MW Method

The velocity streamlines in Figure 4.1.1 above show that both methods have an almost equal fluid flow although the MRF created streamlines have a slightly higher velocity because of the additional fluid momentum in the rotating domain.

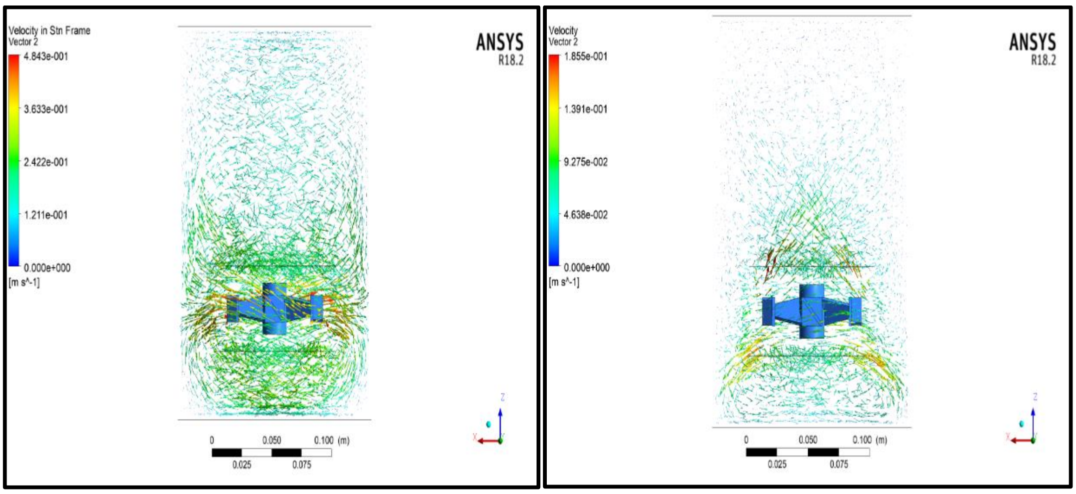


(a) **(b)**

Figure 4.1.2**:** Horizontal Velocity Contour Generated by (a) MRF; (b) MW Method

Figure 4.1.2 shows that the MW method provided a more accurate representation of the flow induced by the impeller compared to the horizontal velocity contour produced by the MRF method.

A circular shape of high velocity region is observed in the contour generate by the multiple reference frame method as fluid in the circular shaped domain rotates and transferring its momentum to around fluid area, leading to inaccuracy of the contour generated. The MRF approach can therefore capture an exact contour around the surface of the impeller.



(a) **(b)**

Figure 4.1.3: Velocity of Vector Generated from (a) MRF; (b) MW Method

The velocity vectors in Figure 4.1.3 show that the velocity contours created by the multiple reference frame technique show fluid flow in all directions outward from the rotating domain as the cylindrical shaped fluid rotates, overshadowing the velocity vectors produced by the cavity within the domain and making it impossible to determine the actual flow provided by the cavity itself. The MW method will produce a more discernible flow vector, as the fluid is drawn into the impeller region and projected upwards.

Based on the observation from Figure 4.1.1, Figure 4.1.2, and Figure 4.1.3 above, it could be stated that the MRF method is not suitable for this case study. As turbulent mixing case study depends on the momentum and kinetic energy transfer, the additional energy and momentum carried by the fluid within the rotating domain influences the simulation outcome, creating difficulties in analysing the flow and direction fields. Whereas, the MW approach can provide consistent results and is observed in the figures above for this case study, accurate representation of flow fields and direction of flow. The following parts will be addressed using the results obtained by the MW method because of the accuracy and reliability given by the MW method.

4.2 Graphical Analysis

In this section, the graphical analysis of the fluid flow induced by the impellers will be discussed. Graphical analysis provides a visualization on the flow developed as a result of the impellers' geometry as they rotate in the mixing vessel.

4.2.1 Wall Shear Vector

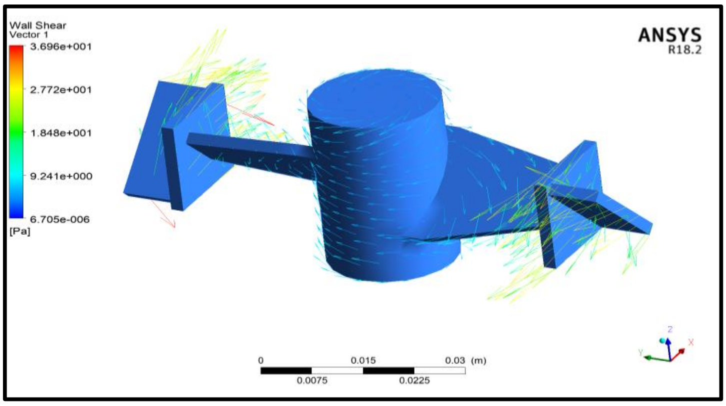


Figure 4.2.1: Wall Shear Vector of Counterflow Impeller

Figure 4.2.1 earlier shows that as the Counterflow impeller rotates counter-clockwise and moves through the fluid, the flow on the inside of the blade is directed downward. The fluid flow is noted to be axially guided through the plate and observation on the outward of blade shows that the fluid is flow guided upward as it slides on the blade surface, whereas at the bottom of the outer blade the flow looks downwards instead of coming in contact with the tip of the blade.

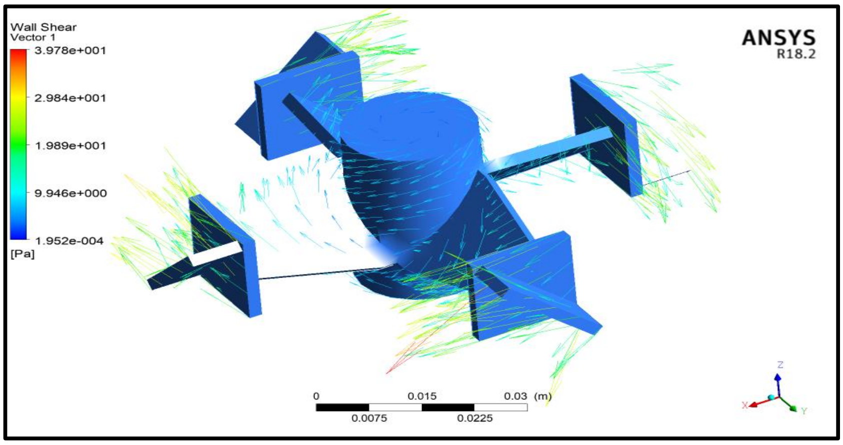


Figure 4.2.2: Wall Shear Vector of 4-Sided Flow Impeller

Figure 4.2.2 above indicates that the induced flow on the inner blade is alternately directed upward and downward. When the fluid enters the buffer layer, the flow is directed axially, and an alternating up and down movement on the outer blade may be observed. The flow alternating up and down is close to that of the Sawtooth impeller, and may lead to the "slotted" fluid flow effect.

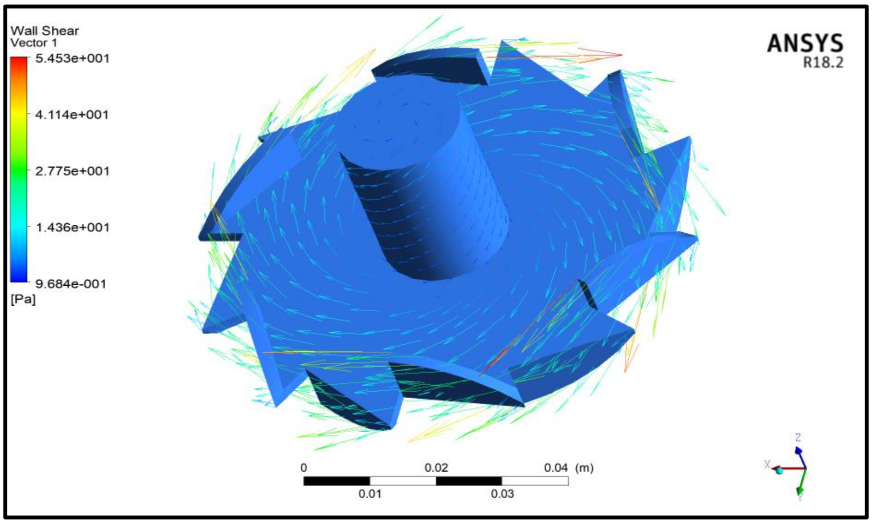


Figure 4.2.3: Wall Shear of Vector for Sawtooth Impeller

From the wall of shear vector in Figure 4.2.3 above, it could be observed that the protruded blades of the Sawtooth impeller directed the flow radially in all directions while simultaneously directing the flow alternately up and down through the disk plate, producing the "slotted" effect of fluid flow.

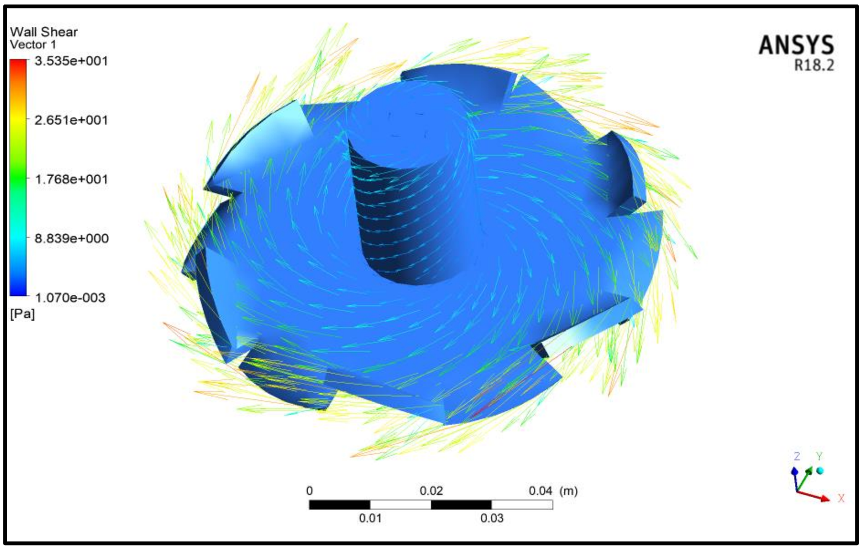


Figure 4.2.4: Wall Shear of Vector for Incline Blade Impeller

Observation made on Figure 4.2.4 shows that the inclined blade of this impeller design caused the fluid to flow more radially outwards compared to the original design.

4.2.2 Velocity Vector

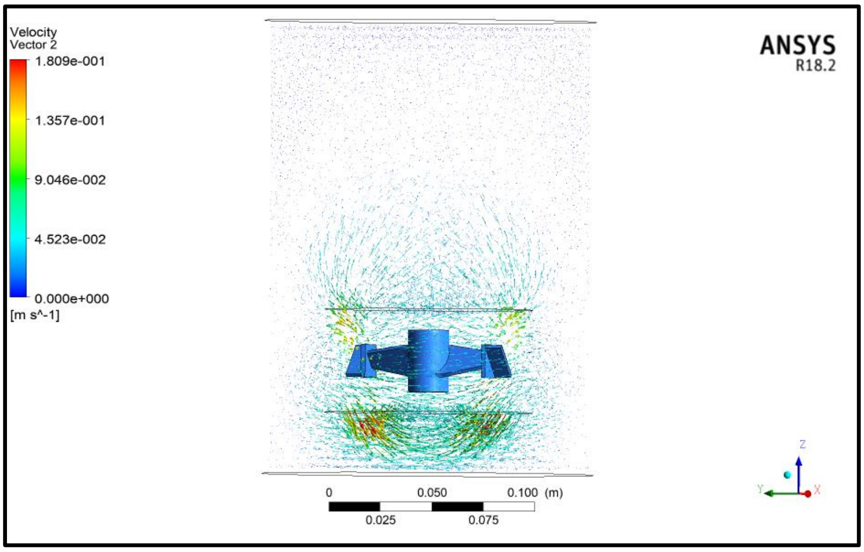


Figure 4.2.5: Velocity of Vector for Counterflow Impeller

In Figure 4.2.5 indicates that the fluid flows radially in all directions while spiralling flow can be observed in the center region, high flow across the area of the impeller is observed with low flow at the top of the vessel.

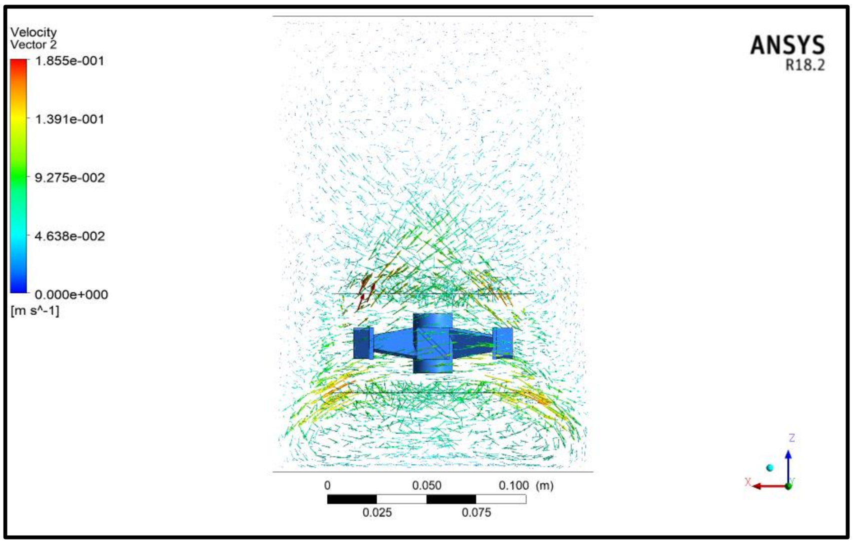


Figure 4.2.6: Velocity Vector of 4-Sided Impeller

From Figure 4.2.6 above it can be seen that the fluid at the impeller's bottom region is drawn into the area of the impeller before being projected in several directions upwards. Fast flow in the underside of the vessel and around the impeller area could be observed.

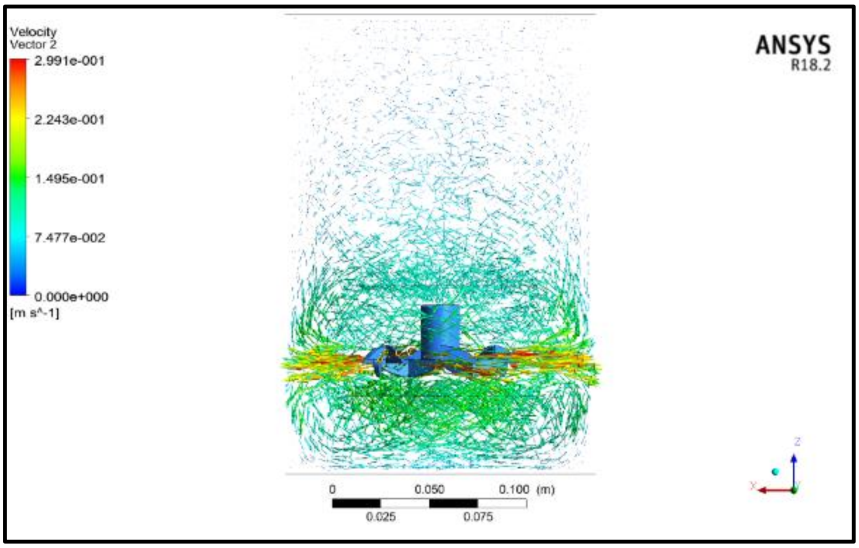


Figure 4.2.7: Velocity of Vector for Sawtooth Impeller

In Figure 4.2.7 above shows that the fluid is sucked into the impeller region at the bottom of the vessel and projected upward. Around the same time, downward vectors observed at the vessel's top region indicate that the fluid from the vessel's upper region is also drawn downward, forming a loop. Stark outward radial flow.

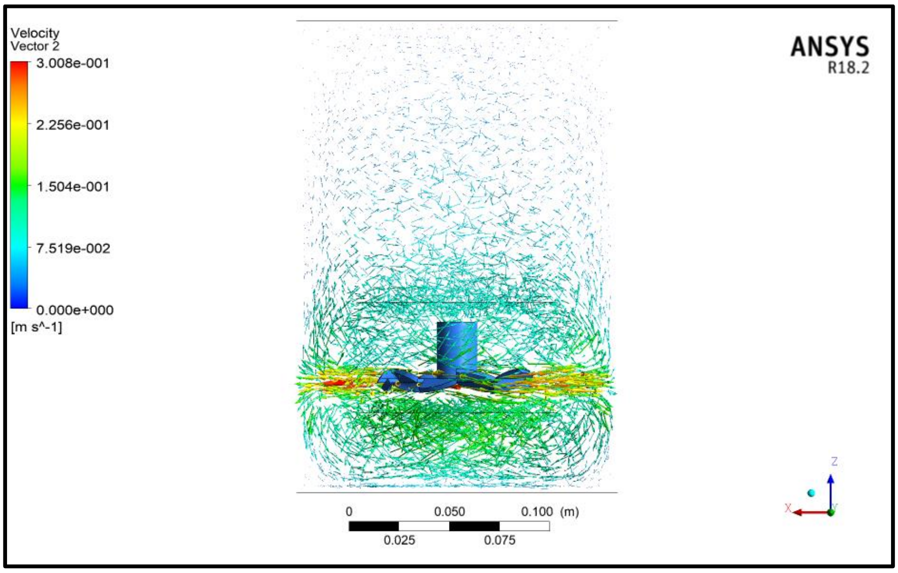


Figure 4.2.8: Velocity of Vector for Inclined Blade Impeller

Velocity vector from figure 4.2.8 indicates that the flow of fluid caused by Incline Blade Sawtooth impeller is identical to the Sawtooth impeller, whereas the flow concentration of the Sawtooth impeller around the impeller region seems to be scattered and the flow diverted to the other part of the vessel.

4.3 Statistical Analysis

The graphical analysis provides a visual representation of the impeller-induced flux. Although it provides an insight into how the fluid forms, it is still too arbitrary. Therefore, a statistical analysis is needed to reinforce the graphical analysis and provide numerical data for a credible analysis. Fort this study, the mixing performance will be quantified as the Flow Velocity. Velocity is chosen as it represents the presence of movement in a fluid, and it could be predicted that if a fluid velocity is present at a certain point in the vessel, mixing will occur at that point.

4.3.1 Mixing Performance at Bottom Region of Mixing Vessel

Table 4.1: Flow Velocity in mm/s at z=20mm From Bottom of Vessel

Impeller / Radial Distance (mm)	Counterflow	4-Sided	Sawtooth	Angled-Blade Sawtooth
0	29	60.2	123	127
8.89	30.7	71.1	139	143
17.8	36.8	79.8	146	152
26.7	41	79.4	144	151
35.6	41.9	74.4	136	144
44.4	42.4	67.1	128	136
53.3	43.3	57.6	115	125
62.2	37.9	43.5	103	112
71.1	22.3	24	66.4	75.1
80	1.6	1.9	0	0
Mean	**32.7**	**55.9**	**110**	**116**

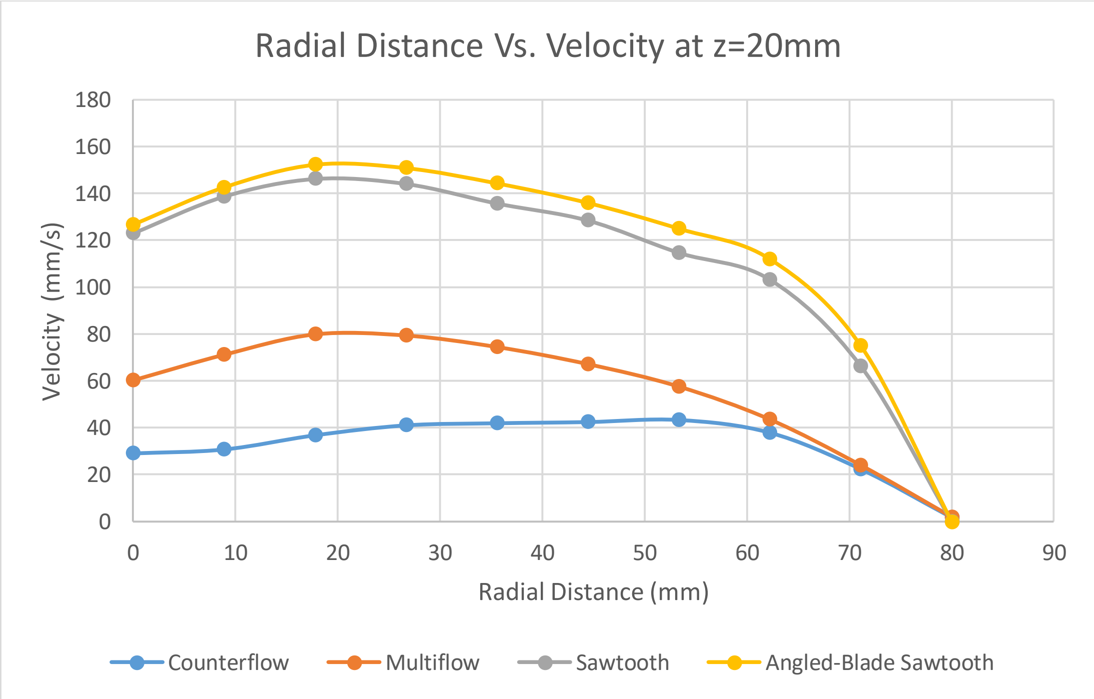


Figure 4.3.1: Graph of Radial Distance Vs. Velocity at Height of 20mm from Bottom of Vessel

For this study, the vessel is 2L beaker that has a height of 0.22 m and its diameter are 160 mm, At a height of 20mm from the bottom of the vessel, this graph will indicate the fluid flow at the bottom of the vessel, from the observation the counterflow impeller has the lowest fluid velocity of all the designs of the impeller, with a highest flow velocity 43.43 mm / s at a radial distance of 53.33 mm, meanwhile the Inclined Blade impeller has the maximun fluid flow velocity at highest speed of 154 mm/s at 17.85 mm its radial distance.

4-sided impeller demonstrated a major increase in speed relative to its counterpart, for its highest flow rate of 78.9 mm / s at a radial distance of 17.7 mm, nearly doubling the counterflow impeller's maximum velocity. Nevertheless, the 4-sided impeller seems too incapable of radially boosting the flow, with the outer area still having poor flow speed. Compared to the original version, the Inclined Blade impeller improvise significantly, providing a higher flow rate to all its radial distances. Another thing to notice is that the Sawtooth and Inclined Blade impellers have a recorded flow speed of stagnation which is 0.0 mm / s at a radial distance of 80 mm, while the Counterflow and it counterpart have a recorded flow velocity of 1.59 mm/s and 1.95 mm/s respectively at 80mm of radial distance. Though the flow rate low, that means both impellers would perform better to provide radial fluid flow than the Sawtooth impeller and its counterpart.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

Different types of simulation approaches available mean that the subject matter needs to be researched more carefully to choose the best approach for the specific case, as certain methods are ideally suited for certain purposes.

In this analysis, instead of the Multiple Reference Frame method, the Moving Wall method was chosen, as it represents a more accurate impeller simulation in turbulent mixing than the MRF method. MRF method is unsuitable for use due to the additional fluid momentum it possesses and affecting the end result since in turbulence mixing simulation, fluid momentum plays a major role. In case studies such as ceiling fans in a space or exhaust fans, MRF may be used safely because in those situations, fluid momentum isn't as crucial. RANS method that saves computing time was chosen. Furthermore, it was evidenced that steady state k-ε RANS is adequate in representing the fluid flow in turbulent mixing.

From observation the 4-sided and the Inclined blade impeller were found to be more powerful than their original counterpart in providing higher turbulence fluid flow. The 4-sided impeller, which could have potentially improved the flow of fluid due to the additional flow direction, which resulting in better radial flow across the vessel at all height coverage rates. It did increase the flow intensity in the vessel, providing strong flow from the bottom to the top of the vessel, while maintaining good radial flow as the original design.

The Inclined blade sawtooth impeller has been improve to provide greater radial coverage by means of its angled bladed construction, which in turn improves coverage of vertical area potentially than its counterpart. Although it is observed to be effective in increasing the coverage of radial area, slightly. The coverage for vertical region was also marginally improved but this was expected at the expense of strong fluid in the centre area, as the impeller distributed the concentration of flow in the center area and projected it upwards and outwards.

This redesign has proven a success for the industrial paint mixing case based on the results obtained. However, this does not necessarily mean that the modified designs are better as the complex turbulent mixing nature dictated that it all comes down to the application of the impeller as different of vessel configuration, working fluid and working angular speed could change the result drastically.

5.2 Recommendation

More studies are required to build an innovative impeller properly. Related criteria such as the density of fluid flow and its operating viscosity, the position of impellers and the working speed of rotation should be taken into consideration when designing an impeller, since they would have a direct effect on the output.

For future studies, by considering study the effect for the sawtooth impeller of inner-to-outer blade ratio, the weight of impeller, blade width, Counterflow impeller and disk radius, number of blades, blade height, and blade angle. Research of the effectiveness of the 4-side and inclined blade impeller in different configurations for its vessels, fluid properties, and operating speed should also be done in order to further confirm the effective flow performance of these impellers.

REFERENCES

[1] T. Kumaresan and J. B. Joshi, “Effect of Impeller Design on the Flow Pattern and Mixing in Stirred Tanks”, Chem. Eng. J., vol. 115, no. 3, pp. 173–193, 2006.

[2] L.-F. Ge, C.-Y.; Wang, J.-J.; Gu, X.-P.; Feng, “CFD Simulation and PIV Measurement of the Flow Field Generated by Modified Pitched Blade Turbine Impellers”, Chem. Eng. Res. Des., no. 92, pp. 1027–1036, 2014.

[3] J. Thompson et al., “An Experimental and CFD Investigation Into the Mixing in a Closed System Stirred Vessel”, Sustain. Des. Manuf., pp. 720–731, 2014.

[4] J. Guhu, D., Dudukovic, M.P., Ramachandran, A., Mehta, S., Alvare, “CFD Based Compartmental Modeling of Single Phase Stirred-Tank Reactors”, J AIChE, vol. 52, pp. 1836–1846, 2006.

[5] University Teknologi Mara Faculty of Chemical Engineering, “Chapter 6 Mixing”, Bioprocess, pp. 1–66, 2018.

[6] F. S. Holland, F.A., Chapman, "Liquid Mixing and Processing in Stirred Tanks", Chapman and Hall Ltd., 1966.

[7] Y. A. C. J. M. Çengel, Fluid Mechanics: Fundamental and Applications, Third Edition in SI Units, McGraw Hill Education, 2014.

[8] A. W. Fitch, H. Jian, and X. Ni, “An Investigation of the Effect of Viscosity on Mixing in an Oscillatory Baffled Column Using Digital Particle Image Velocimetry and Computational Fluid Dynamics Simulation”, Chem. Eng. J., vol. 112, no. 1–3, pp. 197–210, 2005.

[9] T. K. Reid, Robert C.; Sherwood, The Properties of Gases and Liquids, McGraw-Hill Book Company, Inc, 1958.

[10] John R. Rumble, CRC Handbook of Chemistry and Physics (99th ed.). Boca Raton, FL: CRC Press, 2018.

[11] P. J. Fellows, Food Processing Technology: Principles and Practice (3rd ed.),. Woodhead Publishing, 2009.

[12] S. Yanniotis, S.; Skaltsi, S.; Karaburnioti, "Effect of Moisture Content on the Viscosity Of Honey At Different Temperatures", J. Food Eng., pp. 372–377, 2006.

[13] H. E. Segur, J. B.; Oberstar, "Viscosity of Glycerol and Its Aqueous Solution", Ind. Eng. Chem., vol. 43, pp. 2117–2120, 1951.

[14] "AIChE Equipment Testing Procedure in Mixing Equipment (Impeller Type) 3rd Edition", AIChE, New York, 2001.

[15] R. R. Hemrajani and G. B. Tatterson, Mechanically Stirred Vessels. 2004.

[16] Ian Torotwa and Changying Ji, "A Study of the Mixing Performance of Different Impeller Designs in Stirred Vessels Using Computational Fluid Dynamics", Designs, vol. 2, no. 1, pp. 1–16, 2018.

[17] P. Pakzad, L.; Ein-Mozaffari, F.; Chan, "Using Computational Fluid Dynamics Modeling to Study the Mixing of Pseudoplastic Fluids With a Scaba 6SRGT Impeller", Chem. Eng. Process., no. 47, pp. 2218–2227, 2008.

[18] C. Barrue, H.; Bertrand, J.; Cristol, B.; Xuereb, "Eulerian Simulation of Dense Solid-Liquid Suspension in Multi-Stage Stirred Vessel", J. Chem. Eng. Jpn., vol. 34, pp. 85–594, 2001.