OPTIMISATION OF MIXING TANK USING MULTIPLE REFERENCE FRAME (MRF) APPROACH IN CFD SIMULATION

Duration of the project : 3 months

Project Mentors : Athul Akhileshan, Lalit Sharma, Nandish R H

4th year mentor : Azeem

Team Members : Aditya Kumar, Ninad Shetty, Gowtham R

Introduction

Project statement:

This project mainly aims at investigating the fluid dynamics inside a mixing tank, commonly used in various industries such as chemical, pharmaceutical, and wastewater treatment plants and optimize its working through CFD simulations. The objective is to understand the flow patterns and performance within these tanks to optimize their design and minimize operational costs.

Inspiration:

The optimization of mixing tanks through CFD simulations using the Multiple Reference Frame (MRF) approach offers cost-effective and efficient solutions. By harnessing computational fluid dynamics, this method eliminates the need for expensive experimental techniques like Laser Doppler Anemometry and Particle Image Velocimetry. The MRF approach, known for its computational efficiency and independence from experimental data, presents a promising avenue for improving mixing tank performance. This research aims to leverage the MRF technique to accurately model the hydrodynamics within mixing tanks, ultimately contributing to their optimal design and enhanced operational efficiency.

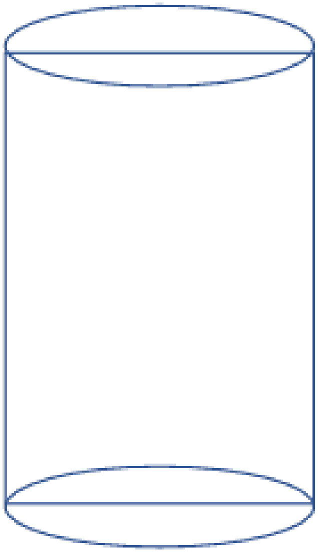
Background info:

The optimization of mixing processes in industrial applications plays a crucial role in enhancing efficiency and product quality. This report focuses on the application of Computational Fluid Dynamics (CFD) simulation techniques, specifically employing the Multiple Reference Frame (MRF) approach, to optimize mixing within a tank system. The MRF method allows for a comprehensive analysis of fluid flow and mixing behaviour, leading to informed design modifications for improved performance.

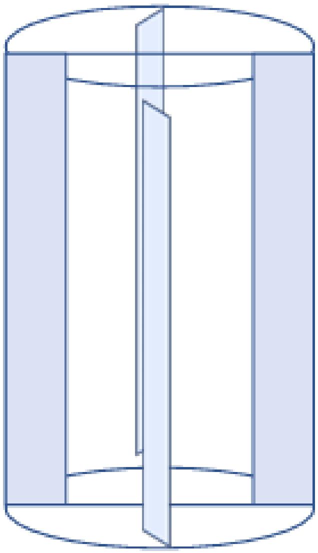
Methodology

The methodology employed in this project involves the modelling of a tank geometry using Fusion 360 software or SpaceClaim (ANSYS). Multiple Reference Frame (MRF) approach within ANSYS Fluent, a computational fluid dynamics (CFD) technique that divides the domain into stationary and rotating zones. This enables the simulation of impeller rotation within the mixing tank, crucial for understanding flow patterns, turbulence, and mixing efficiency. The process began with geometry setup in ANSYS DesignModeler, followed by mesh generation in ANSYS Meshing, ensuring adequate resolution near the impeller and tank walls. Boundary conditions were then defined, specifying impeller rotation speed, and wall conditions. The MRF model was configured in ANSYS Fluent, and simulations were run to solve for flow characteristics. Post-processing of results was conducted using ANSYS CFD-Post to analyse velocity profiles, turbulence intensity, and mixing effectiveness. This comprehensive approach, coupled with the capabilities of ANSYS software, facilitated the optimization of mixing tank reactors, leading to improved design and operational efficiency.

4 Different models are used in this project. The first one is a simple mixing tank, used for basic understanding of the project. Then we add in baffles and alter its size & shape. Three such models are created, which are explained in the image below.



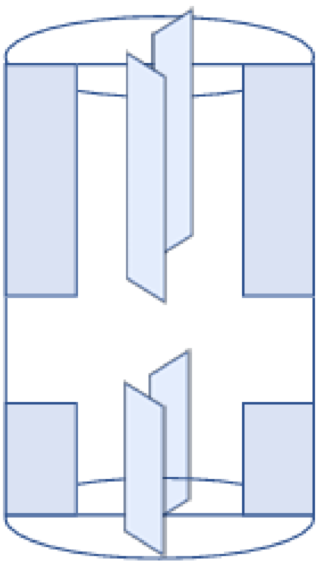
Simple Cylindrical Tank as discussed in Tutorial session



Addition of baffles

15 x 1.5 (cm)

4 equally spaced baffles separated by 90 degrees



4 equally spaced baffles separated by 90 degrees, split into two systems.

Width = 1.5 cm

Split it into two heights according to your decision. Once decided, do not change after this step.

(Keep in mind to leave the gap in between at least equal to the height of the impeller)

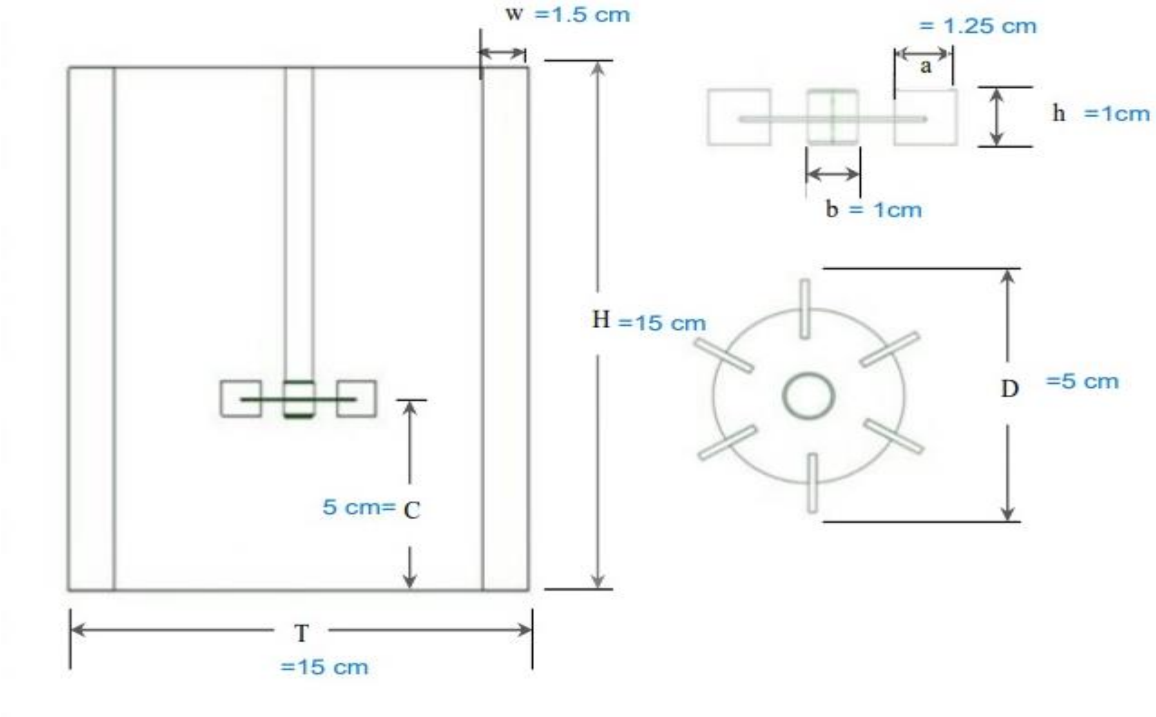
Create two models:

1. Both systems are aligned on each other
2. The systems are split 45 degrees apart

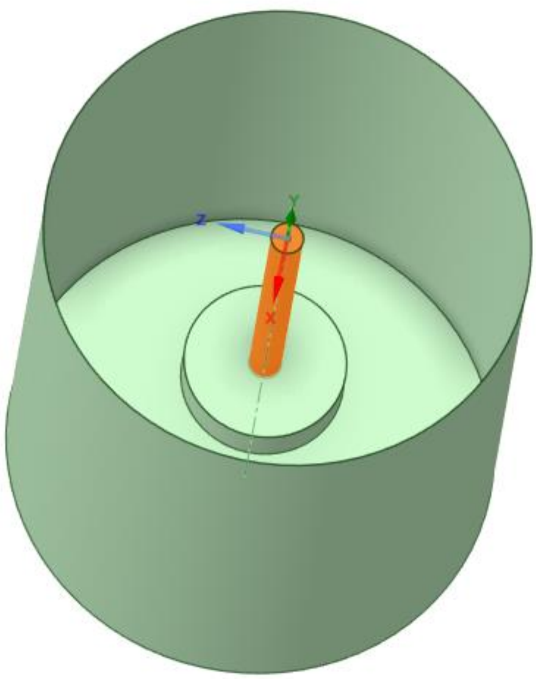
Results

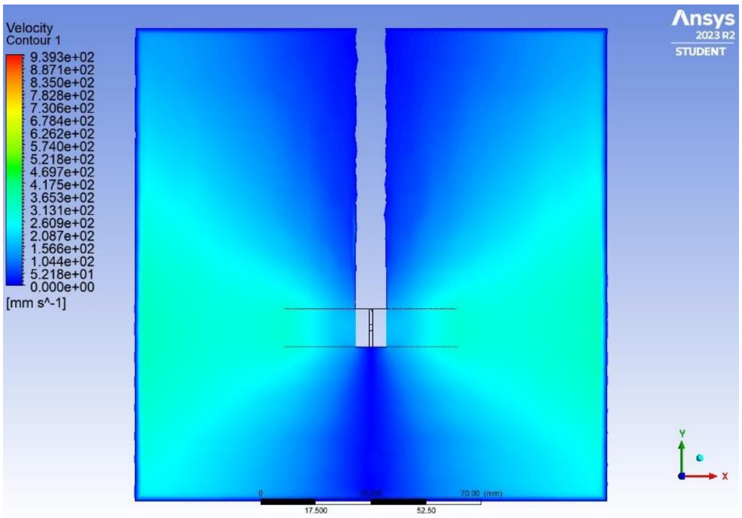
For the first set, a simulation of a simple mixing tank is performed to get a hang of the methodology and the post-processing of results.

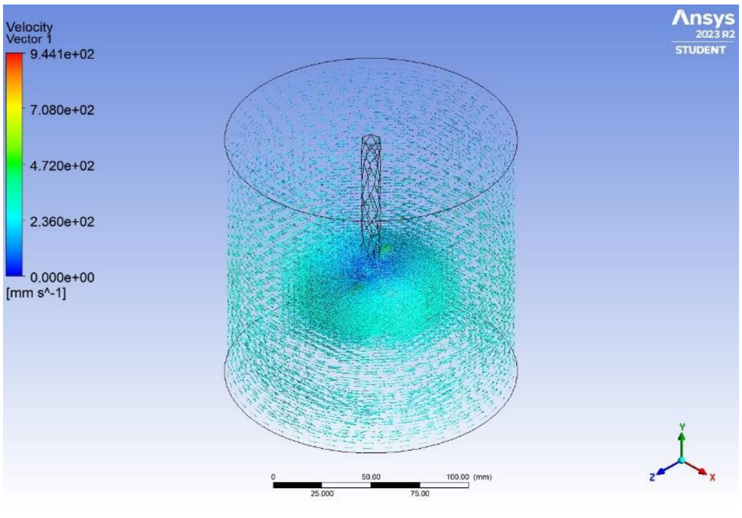
The Dimensions of the Base tank used for this purpose and all simulations thereafter with modifications are as follows:



Base model:

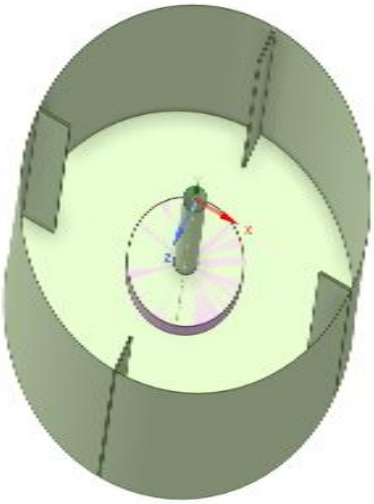




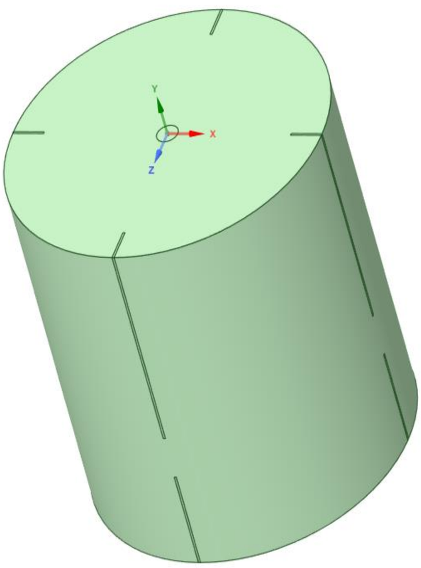


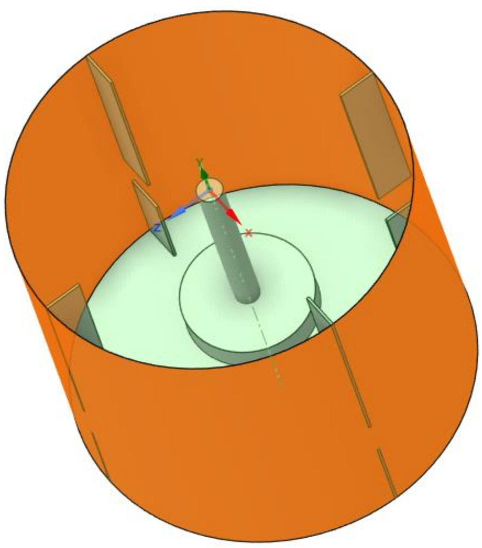
After this, a comparative study of three different models is done. The three different models differ in the baffle size, shape and their arrangement.

1) *Single Baffle System:*

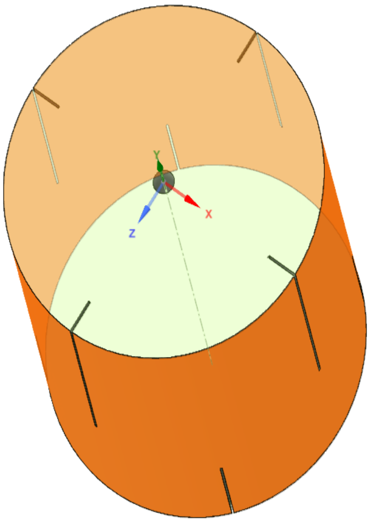


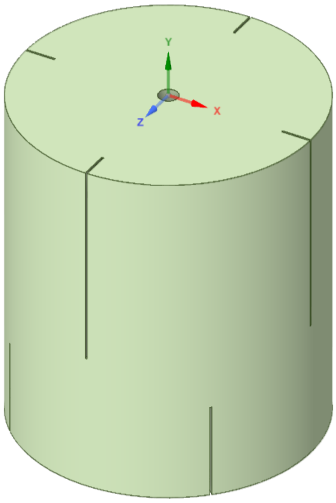
2) *Double Baffle System(parallel)*



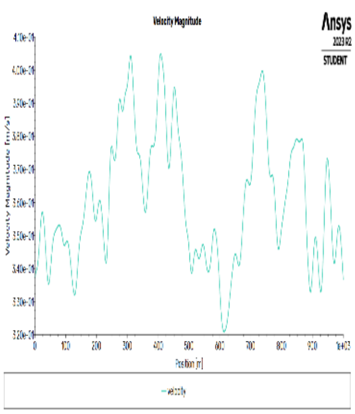
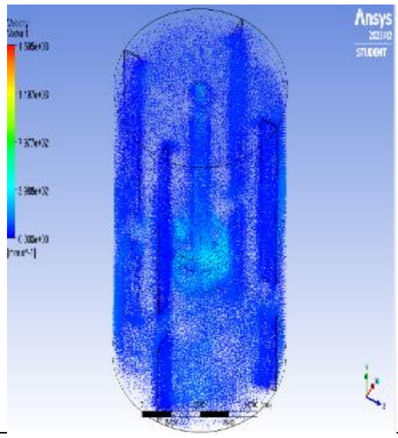
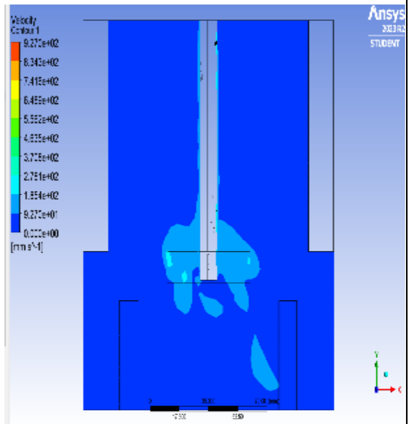
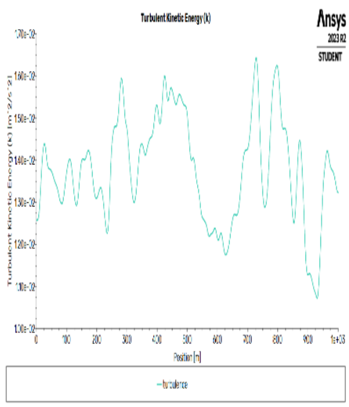
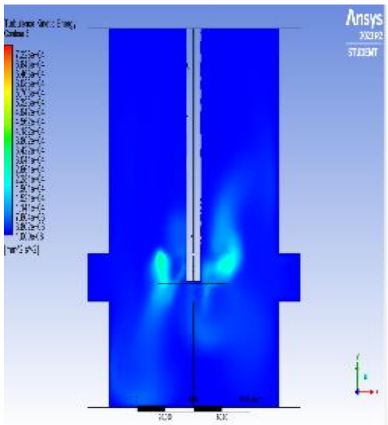
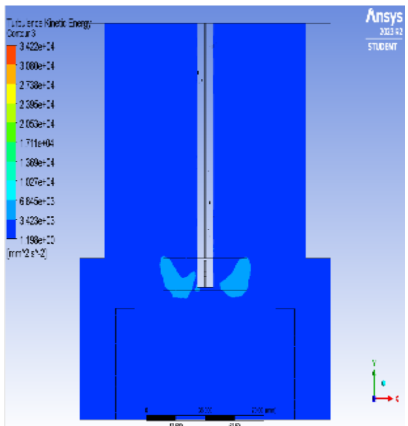


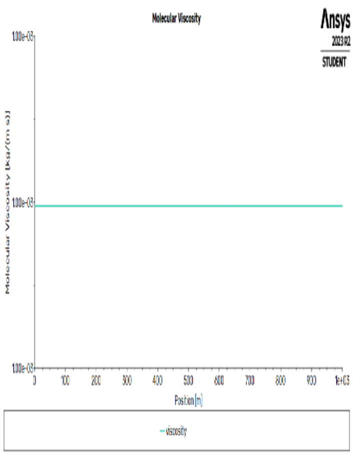
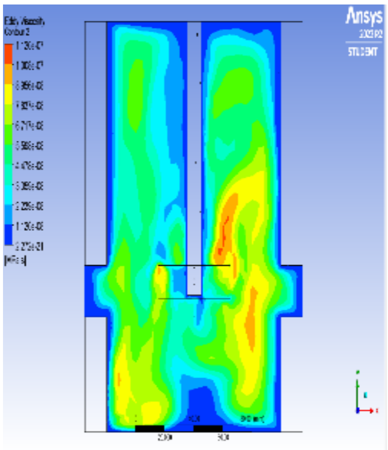
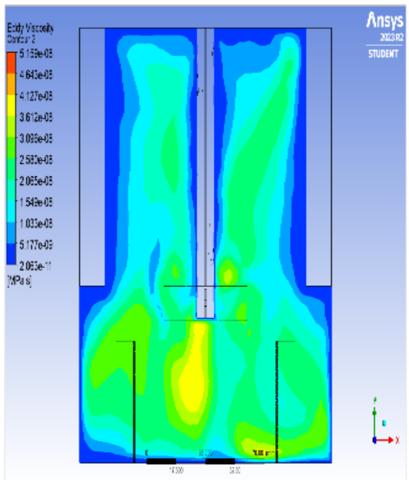
3) *Double Baffle System(Separated by 45 degrees)*

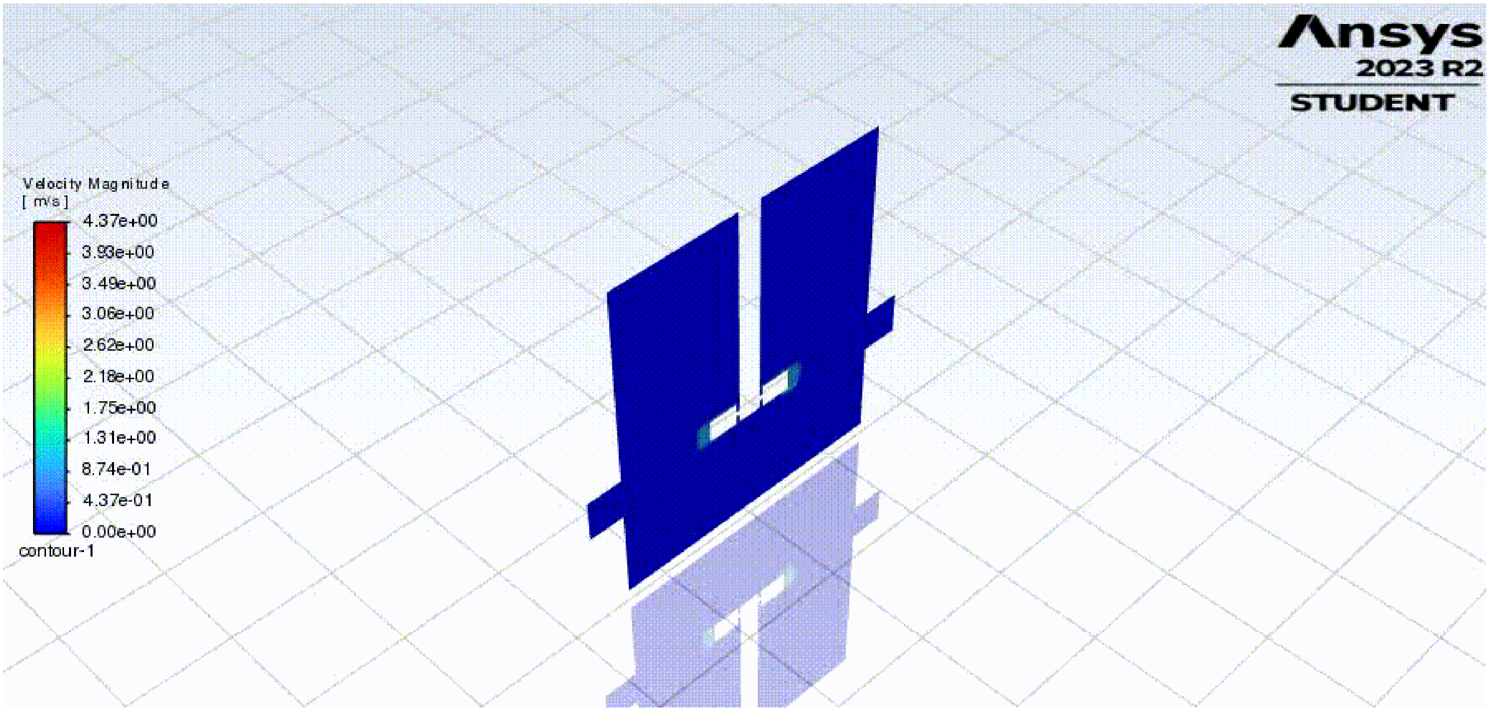




Comparative Study

Model Name	*Single Baffle System*	*Double baffle system concentric*	*Double baffle system separated by 45-degree*
VELOCITY			
TURBULENCE			

<table>
<tr>
<td>VISCOSITY</td>
<td>
</td>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>REPORT DEFINITION</td>
<td>
<table>
<tr><td>strain-rate</td><td>146.0241</td><td>s^-1</td></tr>
<tr><td>viscosity</td><td>0.001003</td><td>kg/(m s)</td></tr>
<tr><td>velocity</td><td>0.3397195</td><td>m/s</td></tr>
<tr><td>pressure</td><td>-25.88875</td><td>Pa</td></tr>
<tr><td>turbulence</td><td>0.01277119</td><td>m^2/s^2</td></tr>
</table>
</td>
<td>
<table>
<tr><td>strain-rate</td><td>148.3464</td><td>s^-1</td></tr>
<tr><td>viscosity</td><td>0.001003</td><td>kg/(m s)</td></tr>
<tr><td>velocity</td><td>0.3670429</td><td>m/s</td></tr>
<tr><td>pressure</td><td>-35.9255</td><td>Pa</td></tr>
<tr><td>turbulence</td><td>0.01519142</td><td>m^2/s^2</td></tr>
</table>
</td>
<td>
<table>
<tr><td>strain-rate</td><td>83.89521</td><td>s^-1</td></tr>
<tr><td>viscosity</td><td>0.001003</td><td>kg/(m s)</td></tr>
<tr><td>pressure</td><td>-8.912692</td><td>Pa</td></tr>
<tr><td>velocity</td><td>0.2072995</td><td>m/s</td></tr>
<tr><td>turbulence</td><td>0.005623488</td><td>m^2/s^2</td></tr>
</table>
</td>
</tr>
</table>



Obstacles faced

Creating the geometry with different baffle systems as well as getting used to Design-Modeller, Space Claim etc

Meshing 3D objects of different curvatures with sizing named selection

Methods of meshing

Application glitch

Conclusion

In a mixing tank, a low strain rate typically indicates slow or gentle mixing. This means that the contents within the tank are being mixed at a gradual pace, which can have several implications:

1. *Homogeneous Mixing: A low strain rate allows for thorough mixing without causing excessive shear forces. This promotes the uniform distribution of components throughout the mixture, leading to a homogeneous blend.*
2. *Reduced Shear Stress: Slow mixing reduces the shear stress exerted on the components within the tank. This can be beneficial for delicate materials or processes where excessive shear could lead to degradation or alteration of the mixture properties.*
3. *Controlled Process: Low strain rate mixing enables better control over the mixing process. It allows operators to adjust parameters such as mixing speed and duration more precisely to achieve the desired outcome.*
4. *Time Efficiency: While low-strain rate mixing may take longer compared to high-strain rate mixing, it can be more efficient for certain applications where gentle blending is required. It allows sufficient time for components to interact and react without the risk of overmixing.*

Overall, a low strain rate in a mixing tank signifies a controlled and gentle mixing process, which can be advantageous for achieving uniform blends and preserving the integrity of sensitive materials.
HENCE IN OUR MODELS,
The double baffle system separated by 45 degrees stands out with a strain rate of 83.89521s-1 compared to other models.

Low velocity can indicate that the flow within the tank is relatively stable and uniform. This may be desirable in some applications where turbulent flow could cause undesirable effects, such as air entrainment or excessive energy consumption. Here the fluid of our interest is water, hence low velocity but relatively near to other models would be better.
HENCE IN OUR MODELS,
The double baffle system separated by 45 degrees stands out with a velocity of 0.2073m/s compared to other models whose velocities are near but more (0.3393&0.367).

Normal Operating Conditions: Pressure close to atmospheric pressure, typically around 0 to -34,500 Pa. This indicates stable and smooth operation without significant suction or vacuum effects. Potential for gentler mixing: With less negative pressure, there may be less risk of excessive suction or cavitation, potentially resulting in gentler mixing conditions.
HENCE IN OUR MODELS,
The double baffle system separated by 45 degrees stands out with a pressure of –8.9Pa compared to other models.

Reduced turbulence implies a smoother and more predictable flow pattern, which typically leads to more uniform mixing. Components within the tank are less likely to experience localized agitation or shearing forces, resulting in a more consistent blend of materials. Lower turbulence generally corresponds to less energy being expended to maintain fluid motion. This can lead to lower operating costs and reduced wear and tear on mixing equipment. Excessive turbulence can result in the entrainment

of air bubbles into the fluid, which may interfere with the mixing process or cause quality issues in the final product. Smaller turbulence reduces this risk, resulting in a cleaner and more efficient mixing operation.
HENCE IN OUR MODELS,
The double baffle system separated by 45 degrees stands out with the lowest turbulence compared to other models.

Hence, after considering all the factors Double baffle system separated by 45 degrees is the best tank for the highest optimisation of the mixing tank.

Future work

We need to add more parameters in the simulation like impeller blades or changes in the size of the rotating domain.

References

- *https://doi.org/10.30780/specialissue-ICACCG2020/026*

- *https://doi.org/10.1080/00986445.2019.1694919*

- *https://doi.org/10.1080/00986445.2021.1975681*

- *https://youtu.be/oa-xcE0_0UY*

- *Ansys Fluent Theory Guide*

- *MRF and SM ANSYS workshop document*

- *COMPUTATIONAL FLOW MODELING FOR CHEMICAL REACTOR ENGINEERING, V.V.Ranade.*