VIVA-TECH IJRI
ANNUAL RESEARCH JOURNAL
VIVA-TECH INTERNATIONAL JOURNAL FOR RESEARCH AND INNOVATION
ANNUAL RESEARCH JOURNAL
ISSN(ONLINE): 2581-7280

Design and Optimization of Lily Impeller

Das Vishal[1], Kale Vishal[2], Kurhade Vipul[3], Gadag Pritesh[4]
[1](Department Of Mechanical Engineering, Viva institute of technology, India)
[2](Department Of Mechanical Engineering, Viva institute of technology, India)
[3](Department Of Mechanical Engineering, Viva institute of technology, India)
[4](Department Of Mechanical Engineering, Viva institute of technology, India)

Abstract : *An impeller, according to present invention which consist of hub and at least one blade supported by the hub . In this study, turbulent flow fields in a vessel stirred by a lily impeller will be investigated. The resultant turbulence will be numerically predicted using Computational Fluid Dynamics (CFD). Turbulence models will be developed using ANSYS solver. The Lily Impeller will be designed based on the original design of Jay Harman, by making changes on its parameters like length/ height of Impeller, angle of blades and also number of blades. We will simulate the impeller action in the vertical and horizontal planes of the stirred fluid volume. Velocity profiles generated from the simulations will be used to predict and compare the performance of the different lily impellers with different parameters and most optimum designs will be finalized. To validate the Computational Fluid Dynamics (CFD) model, the simulation results will be compared with experimental results from existing work and most satisfactory design will be choosed. The final impeller will be able to provide better turbulence characteristics that would improve the quality of mixing systems.*

Keywords - *Agitator, ANSYS, Computational Fluid Dynamics (CFD), Impeller, Lily Impeller, Mixing flows, Simulation, stirred mixers, Turbulence model.*

I. INTRODUCTION

The impeller or impellor is a rotor used to increase the pressure and flow of a fluid like Air or Water. It is the opposite of a turbine, which extracts energy from fluid, and reduces the pressure of a flowing fluid, Whereas Impeller uses Energy to push Fluid. An impeller is a rotating component of a pump that accelerates fluid outward from the center of rotation, thus fluid is being pumped. The velocity achieved by the impeller is transferred into pressure when the outward motion of the fluid is confined by the pump body. An impeller is usually a short cylinder with an open inlet (called an eye) in the center of rotation to accept incoming fluid from the fluid source, vanes to push the fluid radially outward and an outlet at circumference of the rotation. Lily Impeller is also a type of impeller, which was originally designed as a propeller of boats by Jay Harman but was never used as one. The device was designed based on the spiral shapes observed in nature. We can observe this shape everywhere and everyday in our life. Its shape is based on the logarithmic curve (known as the Fibonacci spiral) found in nature like nautilus shells and whirlpools, which accommodates a centripetal flow of liquid with little friction. As Harman Said the inventor of Lily Impeller, “there is no such thing as a straight line in nature”. All gases and fluids move in one spiral path because that's the path of least resistance. Based on this observation, Jay Harman, developer of Lily Impeller developed it.

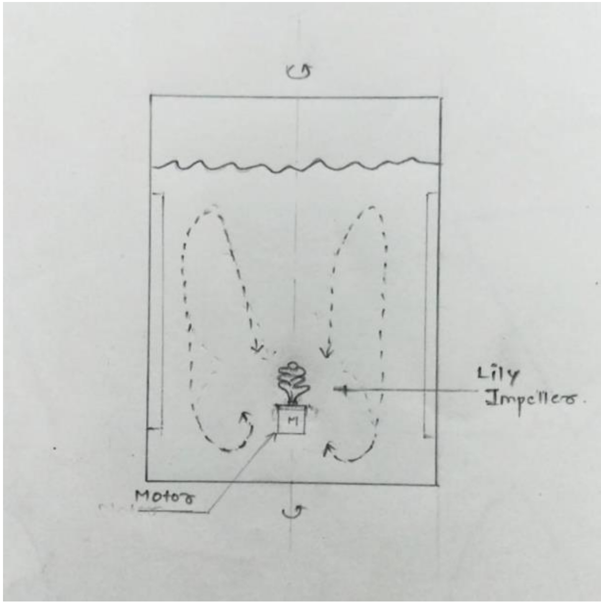

Fig.1 : Lily Impeller

II. LITERATURE REVIEW

Adnan Ghulam Mustafa et. al, 2020 [1] conducted the study to compare various mixing effects of different impellers (Agitator) and the most efficient one was chosen. The information about various impellers (Agitator) were studied from this paper.

Senan Thabet, Thabit H. Thabit, 2018 [2] conducted the study at Computational Fluid Dynamics (CFD) in order to display various pros and cons of CFD. The various applications and various options for methods of analysis of various Fluids were studied.

Ian Torotwa, Changying Ji, 2018 [3] presented a case study on Design and operation of mixing systems using agitated vessels, which is a difficult task due to the challenge of obtaining accurate information about impeller-induced turbulence. They have used Computational Fluid Dynamics (CFD) to provide detailed understanding of such systems. In this study, experimental tests and Computational Fluid Dynamics(CFD) simulations were performed to examine the flow characteristics of fluid in four impeller designs (anchor, saw-tooth, counter-flow and Rushton turbine), in achieving the solution in homogeneity.

Harshal Patil et. al, 2018 [4] presented a case study on CFD simulation to conduct an investigation on flow behaviour of (Fluid) water in a fully baffled stirred tank with Rushton turbine as impeller. The goal of the study was to develop a CFD model of fluid flow in the tank and optimize the dimensions of the inner rotating fluid zone for the MRF model. The best inner rotating fluid zone was found when the simulation results for mean radial velocity, mean tangential velocity and mean axial velocity was reasonably matched with literature data from Wu and Patterson (1989) research paper.

Ian Torotwa, Changying Ji, 2018 [5] presented a case study using Counter Axial Flow Impeller design and their characteristics that improves the turbulence characteristics using CFD. The different designs for conventional impeller types were studied with a multistage combination of impeller and their efficiency were studied.

Romil Gadiya et. al, 2017 [6] conducted the study on mixing tanks used for mixing of two or more chemicals in the chemical industry. The required Modification of the existing design of the mixing tank by changing the mixing system like industrial agitators was studied in this paper.

Akash J Patil et. al, 2017 [7] presented a study on the mechanical design of agitator for mixing polyelectrolyte having a viscosity of 1.5 cp, by considering the fluid forces that are imposed on the impeller surface by the fluid.

The analysis shows that the forces are a resultant of turbulent flow of fluid and static fluid forces. The study shows that the loads are active and are transmitted from the impeller blades to the agitator shaft and then to the gearbox.

M. Sanju Edwards, V. G. Ganesan, 2016 [8] conducted a study at mixing in stirred tanks which are the most common unit in the world in operations of process industries. The base model of the rushton agitator was taken to study the mixing performance using CFD and was compared with reference literature by plotting performance parameters.

Saeed Asiri, 2012 [9] conducted a study to design and implement a new kind of agitators called differential agitator. The Differential Agitator is an electro-mechanic set consisting of two shafts. The first shaft is the bearing axis while the second shaft is called an agitating group consisting of the axis of the quartet upper bearing group and the triple lower group. A numerical analysis, manufacturing and laboratory experiments were carried out to design and implement the differential agitator in their experiment. Knowing the material prosperities and the loading conditions of Agitator, the FEM using ANSYS 11 was conducted to get the optimum design of the geometrical parameters of the differential agitator elements, while the experimental test was performed to validate the advantages of the differential agitators to give a high agitation of lime in the water as an example. The study ended with results to maximize agitator performance and optimize the geometrical parameters to be used for manufacturing of differential agitator.

Yeng-Yung Tsui, Yu-Chang Hu, 2011 [10] conducted a study to investigate the mixing of flow generated by helical ribbon blade impellers and to show that with the help of CFD the performance of the mixing system can be significantly improved by optimizing the geometric configurations of the impeller. To fulfill this objective, a numerical model was developed to solve the Navier-Stokes equations for the flow field. However, difficulties arose due to the rotation of the impeller in the vessel. In order to ease the problem, the velocity field was assumed to be in a quasi-steady state in their study and the multi frame of reference was adopted to tackle the rotation of the impeller.

III. PROBLEM DEFINITION

The various Instruments like agitator which are used for different mixing processes produce low turbulence flow which slows the rate of mixing and are also costlier and bulky in size. The Lily impeller on the other hand is cheap and compact in size and will take less time for mixing for the same volume of liquid, and which may help to produce efficient turbulence flow for mixing compared to other instruments.

IV. METHODOLOGY

The main objective of our project is to mix various fluids together more efficiently compared to the other industrial mixing instruments like agitator, using the turbulence flow characteristics of fluid. To achieve the above stated objectives various calculations, Solid Modeling and Analysis (CFD) have to be done.

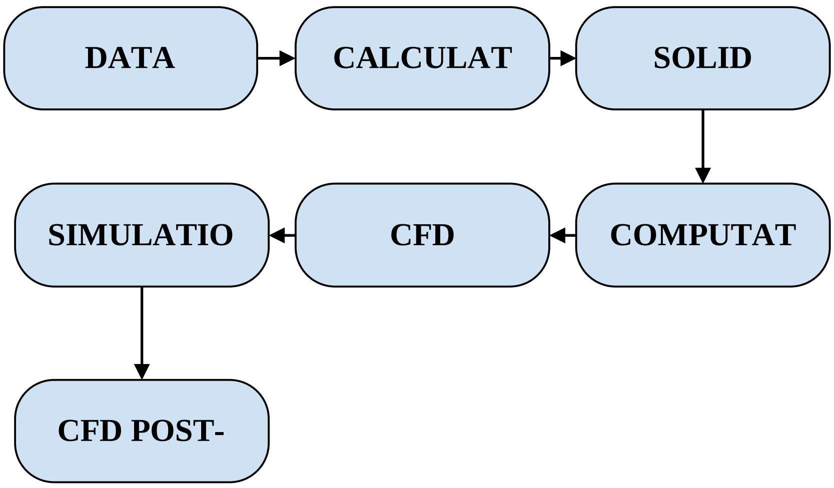

Fig. 2 : Blog Diagram of Methodology

4.1 Data Collection

The data collection can be defined as the process of collecting and measuring information of targeted variables in an established system, which then enables one to answer relevant questions and evaluate results. The aim of data collection is to capture quality evidence that allows analysis to lead to the formulation of agreeable answers to the questions that have been posed. For our project various data has to be collected like data about the fluid that will be used for mixing (for e.g. viscosity, density, Reynolds number, etc of that fluid), dimensions of the vessel that will be used for mixing of the fluids. These dimensions will be used in CFD analysis for defining the boundary conditions in the analysis. Based on the calculations motor for the impeller will be selected.

4.2 Calculations

Calculations based on the data collected in the previous step will be used for the calculations like calculations of the length/height of impeller, angle of blades and number of blades of impeller. Based on the calculations of impeller, vessel/ container in which the process of mixing will be carried out will be selected. On the basis of the same the Motor for the Impeller will be selected.

4.3 Solid Modelling

Modeling a physical process is an activity to analyze, evaluate, simulate and define the common knowledge in a referencing manner that can be easily understood by the people. Its main aim is to provide crystal clear information about the object through simulations and convey the conceptual understanding regarding physical models. For Modelling of Lily Impeller we are going to use SolidWorks Software using the values from the previous step of Calculation.

4.4 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a method that uses techniques from physics, applied mathematics and computer science to model, predict and visualize how fluids, that is, gas or liquid, flows. A qualitative and quantitative prediction can be put together with the help of CFD which uses mathematical modeling tools, numerical computation and software tools to frame, understand, construct and therefore, predict the required scenarios.

4.4.1 CFD Meshing and Pre-Processing

The impeller configurations, fluid volume and the container will be modelled as separate regions in solidworks software, before being imported into ANSYS for pre-processing and meshing. Elaborate interfaces between the contacting fluid regions and boundary conditions will be created in ANSYS workbench design modeler. A mesh will be generated to discretize the domains into small control volumes, where the conservation equation is to be approximated by computer numerical calculations. The mesh for the mixing simulation set-up contains two main zones, tank-fluid region and impeller region, modelled as separate interacting fluid domains. An increased mesh density will be used near the impeller and the tank walls in order to outline the boundary layer flow details.

4.4.2 Simulation Process and Computation

The simulations will be prepared in Ansys solver, using the pressure-based steady state and absolute velocity conditions with gravity acting in the negative y-axis direction. The created fluid regions will be set to viscous type in the standard model with standard wall functions. The material we are going to choose for our analysis will be water-liquid with a density(ρ) of 998.2 kg/m3 and constant viscosity(μ) of 0.001 kg/m-s. Cell zone conditions will entail the impeller-fluid interface, which consists of the impeller surface and the fluid regions around the impeller. Mesh interfaces and contact regions will be confirmed to be the exact points where interactions occurred.

4.4.3 CFD Post-Process

Analysis This stage covers an important step which views/show the result analysis for the different shape and configuration of the impeller. Mixing of the fluid can be checked from the result of the volume fraction of Fluids. The pressure drop should be low as possible for the good mixing and for a high-efficiency impeller. Contour plot is the function applicable in CFD-Post whereby it displays the graphical results according to the data generated in Ansys. This function is significant because it allows the user to view the trend of the process happening in the domain and be able to identify the exact location of the reaction happening during the simulation. In this study, the variables of contour plot that will be applied is velocity, pressure and volume fraction.

V. CONCLUSION

In this project we are trying to replace Agitator by Lily Impeller which consumes less energy compared to Agitator which is used for mixing various liquids all over the world in various mixing facilities. In this project we are planning to achieve it by changing the parameters of the impeller like angle of blade, number of blades and height/length of blades. After carrying out the analysis we will be able to choose the most optimum parameters for impeller to use it in place of Agitator for mixing, which will also help to study the flow characteristics of lily impeller that would improve the turbulence characteristics which helps the better mixing process.

Acknowledgements

It's a great pleasure for us to undertake this project. We would like to express our special thanks of gratitude to our project Faculty Advisor Prof. Henisha Raut ma'am and project guide Prof. Niyati Raut ma'am for their valuable inputs, able guidance, encouragement, whole-hearted cooperation and constructive criticism throughout the duration of our project. We deeply express our sincere thanks to our Head of Department Prof. Niyati Raut ma'am and principal Mr. Arun Kumar for encouraging and allowing us to present the project on the topic "Design and Optimization of Lily Impeller" at our department premises for the partial fulfillment of the requirements leading to the award of B-Tech degree. We thank our group members for their contribution and cooperation for making this project. This project was made from the support and contribution of our group members. So we will thank each one of us. During the project, we came to know about so many new things and we are really very thankful to them. Secondly we would also like to thank our parents and friends who helped us a lot in finalizing this project within a limited time frame. It helped us to increase our knowledge and skills. We are making this project not only for marks but to also increase our knowledge and skills. Thanks again to all who helped us.

REFERENCES

Journal Papers:

[1] Adnan Ghulam Mustafa, Mohd Fadhil Majnis, Nor Azyati Abdul Muttalib, "CFD Study on Impeller Effect on Mixing in Miniature Stirred Tank Reactor", *CFD Letters 12, Issue 10* (2020) 15-26.

[2] Senan Thabet, Thabit H. Thabit, "Computational Fluid Dynamics: Science of the Future", *IJRE, Vol. 5 No. 6,* 2018, pp. 430-433.

[3] Ian Torotwa, Changying Ji, "A Study of the Mixing Performance of Different Impeller Designs in Stirred Vessels Using Computational Fluid Dynamics", *MDPI*, 2018.

[4] Harshal Patila, Ajey Kumar Patelb, Harish J. Pantc, A. Venu Vinoda, "CFD simulation model for mixing tanks using multiple reference frame (MRF) impeller rotation", *ISH Journal Of Hydraulic Engineering,* 2018.

[5] Ian Torotwa, Changying Ji, "Mixing Performance of Counter-Axial Flow Impeller using Computational Fluid Dynamics", *IJCET, Vol. 8 No. 2,* 2018.

[6] Romil Gadiya, Ankush Patel, Rajdip Raulji, Ghanshyam Gandhi, B P Patel, "Design Modification and Analysis of Mixing Tank-A Review", *SJIF*, 2017.

[7] Akash J Patil, Sachin B Rajude, Mayur S Thok, Ajit R Pawar, "Review On Design Of Agitator", *IJRTI, Vol. 2, Issue 4,* 2017

[8] M. Sanju Edwards, V. G. Ganesan, "Performance Study Of Mixing Agitator Using Computational Fluid Dynamics", *IJMRME, vol 2, issue 1,* 2016.

[9] Saeed Asiri, "Design and Implementation of Differential Agitators to Maximize Agitating Performance", *International Journal of Mechanics and Applications,* 2012, 2(6): 98-112.

[10] Yeng-Yung Tsui, Yu-Chang Hu, "Flow Characteristics In Mixers Agitated By Helical Ribbon Blade Impeller", *Engineering Applications of Computational Fluid Mechanics, Vol. 5, No. 3,* 2011, pp. 416–429.

Books:

[11] R. K. Rajput, M. E., M.I.E, C.E, *Power Plant Engineering* (New Delhi: Laxmi Publications (P) LTD, 2001).

[12] Dr. R. K. Bansal, Ph.D, M.I.E, *Fluid Mechanics and Hydraulics Machines* (New Delhi: Laxmi Publications (P) LTD, 2000).

[13] Hassan AREF, James B. KADTKE, Ireneusz ZAWADZKI, *Fluid Dynamics Research* (North-Holland, vol. 3, no. 1-4, 1998, pp. 63 - 74).