RESEARCH ARTICLE OPEN ACCESS

Validation in the improved performance of Centrifugal pump using CFD

Mohammed Al-Shebani
The Public Authority for Applied Education and Training
State of Kuwait

ABSTRACT
It is the time when manufacturing the parts of the machine and testing it is difficult. The system is upgraded and now software such as Ansys can be used to test the machine performance. The improvement in the centrifugal pumps can be made using computational fluid dynamics. The changes and modifications in the geometry of impeller finest geometry are experimentally improved and testified.
Keywords: Impeller, Centrifugal pumps, Computational Fluid Dynamics (CFD)

Date of Submission: 25-01-2021 Date of Acceptance: 10-02-2021

I. INTRODUCTION

Centrifugal pump is a fast turning machine that is utilized in all the significant fields. It very well may be seen that the powers or the periodic forces inside the centrifugal pump for the turn of the impeller may make weakness the bearing, shaft, and diverse different segments (Patil, Joshi, & Naik, 2017). Centrifugal pump is widely utilized in many fields. The basic principle for the operation of the centrifugal pump is that it consists of a shaft that is mounted on a rotating impeller inside a volute which imparts energy to the fluid which is being moved. The pumps utilize centrifugal force to increase the fluid velocity as it passes the impeller and exits at the tip of the impeller. This action results in the conversion of mechanical energy into kinetic energy. Alteration in the centrifugal pump reduces the expected life of the parts of the pump.

A centrifugal pump acts as a transmitter for the transportation of fluids by the modification of one form of energy to another energy of the fluid progress (Al-Obaidi, 2019). The energies which undergo a modification are rotational kinetic energy and hydrodynamic energy and the energy that causes prompting of the flow of fluid comes from a particular source. The generated flow moves along the pump with specific directions and modulation. Anybody who knows the name of centrifugal pumps must be aware of the proper functioning schedule of the pump.

Centrifugal pumps consist of groups of machines that eventually incorporate to produce outcomes at various industrial levels. If you look upon the history of an industry, it must contain the technical theories of the usage of centrifugal pumps. Besides the working level of history, the experimenters place great efforts in enhancing the working efficiency of a centrifugal pump. The increased proficiency in working of the pump includes the frequency of flow rate within a centrifugal pump. The fluid flow passes through various parts of the pump like impellers. While passing through the impellers, the fluid gets accelerated velocity for operating the system (Deshmukh & Samad, 2019). All parts of the pump altogether add worth to the proper functioning of a centrifugal pump.

All parts of the pump like impellers, blade, inlet, outlet, and modification of the come altogether present the ideational work structure of the centrifugal pump. If you aim to study the working structure of a pump then you must have to analyze every single motion to depict the overall working procedure of the pumps. These pumps are the kind of the main and famous pumps that have been in use for many times in various niches of technique (Lim & Sohn, 2018). For making it easier, it is a twirling pump that adds aid in the flowing of the fluid with the help of centrifugal force that is why they are termed centrifugal pumps. In today's times to fulfill the necessities is quite problematic. So it is necessary to keep in notice all the parameters in thinking for the betterment of centrifugal pumps. These pumps are quite beneficial for transporting large quantities of fluid from one point to another with the conversion of energies. The transportation of fluid mainly depends upon the flow rates of the fluid in pumps. For a deep study of the centrifugal pumps, some researchers have planted an experimentation sample of centrifugal pumps in the laboratory. In the experimentation sample, they study the involvement of regulating parameters such as the flow rate, pressure at both monitoring points like inlet and outlet. With the advent of recent theories and numerical calculation of the pump, the study of every part of the pump has become easy to make an account.

Centrifugal pumps are commonly constructed for the process of fluid progression with specific rates. The concern for improving the performance of the pump has been an interest for a couple of years (Lim & Sohn, 2018). The performance of the centrifugal pump is been depending upon the geometrical ways of fluid progression along with the impellers and casing of the pump. Moreover, for analyzing the optimization of the impeller blades, assurance of Hydraulic layout of the metallic volume, and overall achievement of the centrifugal pumps, the CFD codes bring in to use. The working mechanism of centrifugal pumps generates fluid flow and also makes it rise from a lower to a higher rate. When the fluid passes through impellers it gains velocity as well as pressure. Centrifugal pump is the most widespread pump of the industrial regions. To increase the validation performance of the pump, the technical design mechanism is necessary. Additionally, design optimization is the key to enhancing the working of centrifugal pumps. The numerical analysis also plays an important role in the optimization of the induced impellers in pumps.

For the productive growth of several industries, the key factor always remains relevant to the pumping strategy of the industry. The centrifugal pumps hold the capacity of handling heavy flows that is why they act as basic equipment in the progress of the enterprise. Only the 4th part of the total pump production is unaware of the significant employment of the centrifugal pumps (Morrison, Yin, Agarwal, & Patil, 2018). In the current working of a centrifugal pump commercial computational fluid dynamics code, ANSYS, CFX is used to facilitate the flow of fluid in the 3D structure of the centrifugal pump. The piping design can also be improved for the proper functioning of a pump (Bao, Yang, Kim, & Kim, 2017). With the availability of many formats of pumps, only the exact design of a pump is definite for availing of the valuable facility. It is essential to evaluate the working of a pump under different operating circumstances to ensure the proficiency of energy and avoid the downfall of the equipment. In this concern, the operating tool CFD is best to check the capability of a centrifugal pump. It helps the engineer to simulate the functioning of a centrifugal pump accurately. Before bringing in practice any physical tool or machine expects to have the design process (Wang et al., 2017). The more you get to know about the formulation of the product, the more you can earn benefit from it.

Mostly the design of the impeller was held up to improve the performance of the centrifugal pump. Everybody knows that every time it is not possible to carry out the testing of parts of the machine. For testing the improved performance of a centrifugal pump, code methods like CFD are necessary to employ. After setting up several modifications to the operating system the performance of the pump can only be improved. Centrifugal pumps work as a group of machines to serve several different industries for various purposes (Ge, He, Huang, Zuo, & Luo, 2020). As the need for these pumps goes on increasing, the demand for adding improved performance is also getting high, that's why the researchers keep on finding the latest and safest ways of improving the performance of a centrifugal pump.

To state the efficiency of a centrifugal pump, the inside structure of impeller blades is essential to keep in the notice. The working of this pump as per distinct conditions depends upon the ability of impeller blades. These impeller blades work on the principle of rotation to cause the flow of fluid that eventually raises a velocity. The velocity of the fluid tends to generate vibration and the vibration frequency getting increase or decrease as per the rate of the flow in the pump.

A centrifugal pump with higher proficiency is more in demand for industrial uses. The CFD analysis estimates the impeller diameter to check the working ability of the pump.

It calculates the outcome caused by the movement of the impeller blades. The pump in good working condition enables an excellent amount of velocity for the discharge of the fluid. In this regard, the performance of the centrifugal pump increases with the ultimate increase in velocity. So to operate a centrifugal pump in different set conditions many factors play their role individually (Ding, Han, Xiang, Ge, & Zhang, 2018). Impeller blades are responsible for the productive rotation, the inlet vane angle, the outlet vane angle; the rate of flow will pave the path for the amount of velocity or speed on that you can easily estimate the performance of a centrifugal pump. In this way, the efficiency of a centrifugal pump gets improved.

The efficiency and working of a centrifugal pump can be enhanced using computational fluid dynamics (Dönmez & Yemenici, 2019). Interest in impeller design has considerably increased the efficiency of the design, pump performance, modification, and its working. Through computational fluid dynamics (CFD), the challenges that lower and decline the efficiency of the centrifugal pumps can be enhanced. The paper will provide an overview of validation in the improved performance of a centrifugal pump using computational fluid dynamics (CFD).

Aim

The improvement in the centrifugal pumps can be made using computational fluid dynamics. The changes and modifications in the geometry of impeller finest geometry are experimentally improved and testified.

II. LITERATURE REVIEW

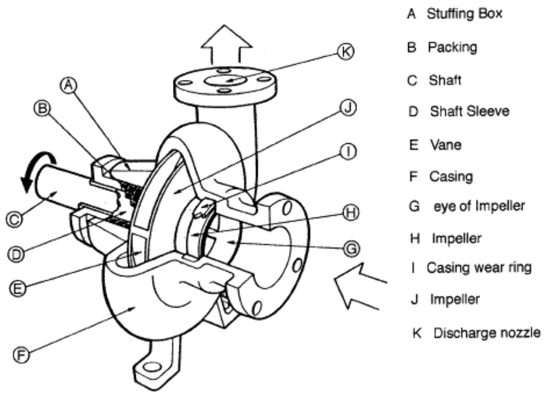


Figure 1: Main parts of the Centrifugal pump (Nuclear Power, 2017)

The centrifugal force is utilized by the pump right from the initial entry of fluid to the extraction of the fluid through the periphery on the rotating impeller. The whole phenomenon helps to convert the mechanical energy into kinetic energy (resulting from position and motion) that helps to shift the fluid to higher pressure and velocity areas. In the whole process, there is a likelihood that vibrations may occur. Naveen Varma et al., (2017) help to provide a description of the behavior and characteristics of the centrifugal pump.

The researchers state that the frequency mode of the centrifugal pump shapes and develops the corresponding frequency along with that, the shock, behavior (harmonic and random) are also shaped by the frequency mode of the pump (Benturki, Dizene, & Ghenaiet, 2018). The shaft that is already mounted on the rotating impeller that is placed inside of housing allows imparting/generate energy and power to move the fluid.

A survived closer scale pump has improved by V.S. Kadam etc., through extended the measurement from 770 millimeters to 820 millimeters to affixes the high-ranking efficiency by the requirements of discharge and head (Kadam, Gawade, Mohite, & Chapkhane, 2011). The pump, along with greater efficiency and a large steady operating region, is built. This analyzed CFD pump through an improved impeller is designed to check out the pump's efficiency and performance. It is based on the former research Jie Jin at al foundation (Jin, Fan, Han, & Hu, 2012). The extremist low fixed-speed centrifugal pump of the GSB20-320 hydraulic model has been developed and built by him. In terms of higher accomplishment on the extremist low and set speed, a performance test and numerical simulation have been adapted to analyze these hydraulic properties centrifugal pump.

An Anjani Kumar Sinha et al have done a CFD examination of this centrifugal pump to polish up and discharge through distinct blade geometry. In this project, he also discussed the effects of the alternated design of these pumps. In Ansys software, a CFD module is a tool that is utilized to get out the development in the discharge of a water pump. After doing changes in this pump design, they realized that there occurs improved velocity from 1.2 m/ s to 1.7 m/ s. Overall results of discharge are enhanced by up to 42 %. Hence efficiency is increased by 25 per, and it is improved up to 60 per to 85 per. However, changing the design of this water pump also improves the peak pressure, distribution, velocity of water, and discharge. It simultaneously extends the efficiency of the pump when making an increase in the velocity. They determined the water pump's performance by using the thickness of the blade and subsisting outlet vane angles and inlet vane angles.

Krishna Kumar Yadav et al. have been analyzed the performance of this centrifugal pump by using the thickness of the blade then the efficiency was 84 %, and the results for outlet angle was 16.28 degrees (Yadav, Mendiratta, & Gahlot, 2016). For the inlet, the angle was 21.08 degrees, along the thickness of the blade is 10mm. The water pump efficiency was increased up to 89.19 % for blade thickness and optimized angles. After that, Mohankumar M analyzed the impacts of the pertinent design parameters by containing the inlet blade angle, the blade number, the impeller diameter, and the trimmed impeller profile that is on the three-dimensional water pump with the constant sate liquid flow (Mohan Kumar, Raj, & Vratharaj, 2014). Meshing CFD of Geometric modeling analysis the velocity distribution and pressure of the liquid. Patil et al. (2017) have analyzed the impacts of the ratio of the blade's thickness and the geometry of the blade over the outlet of the impeller onto this centrifugal pumps' low fixed speed performance (P. M. Patil, Gawas, Pawaskar, & Todkar, 2015).

In CFD., this project then imitated for handling conditions of the extremist low fixed speed of the water pump. Its efficiency was thoroughly studied unassisted, its effects, and the first, the parameter ratio of blade thickness was analyzed. The change in the pump's efficiency was observed, and the geometry outlet blade was also varied. The top-notch blade thickness ratio for the stated requisite was considered (Pei, Yin, Yuan, Wang, & Wang, 2017). The centrifugal pump's overall efficiency and an optimization parameter and effects of both parameters were analyzed. The standard design of the impeller's gotten results shows an increase in the efficiency raised about 4.743 %.

The pumps' designers are challenged continuously by providing vehicles, the machine that works to be more reliable, quietly, and efficiently also at a reasonable price. Several investigators have appealed for CFD as per numeric disguise tool to get out distinct investigation on a centrifugal pump. This part tells that the research work that is held through many investigators by CFD approach on the centrifugal pumps. Conventional procedures are rare considerably used that are greatly accurate and usually tend to utilize in few software packages of commercial. Nevertheless, it is highly slow in the case of computation period that results in totally impossible to sort out huge problems within a reasonable time to utilize it online. In such a paper, we thoroughly check on both popular and available procedures that may survey moderate acceleration through the conventional method. This method is then further grouped into two main categories like hardware techniques and enhanced Numerical methods.

Generally, the hardware techniques' acceleration is used combined with enhanced numerical and conventional procedures. The advanced numerical procedure was classified based on the Mesh, hybrid, and Meshfree method. This literature was observed on the popular paper on the advanced numerical methods. After the scope of this small paper, it will be after going into the detailing of each critical analysis and procedure for each method available on the literature. In such a paper, we rather provide the available methods that the conventional method might be utilized to get CFD real-time there consider over their application and list some basic and some unique interesting literature for heat transfer, free-surface, and multi-phase surface. It also discussed the two distinct types of the acceleration process, hardware techniques, and enhanced numerical methods.

Hardware techniques process is really common for appealing for the acceleration amongst several available processes. With the CFD community, the grants in the multi-core architecture of the computer during the recent most decades proposed very great success. Parallel programming utilizing the multi-core central processing unit CPU and multi-core general purpose graphic processing unit GPGPU architecture create it possible to control large problems. However, using the accessible computational power makes it tough to get real-time CDF by appealing for the hardware techniques process to the conventional process. It wants to achieve real-time CFD to use the CFD tools online for industrial operations that motivated investigators in such a community to build enhanced alternative processes. Using mathematical methods, we can get two-way acceleration by both hardware techniques and mathematical methods.

The standard process of imitating fluid flow is the Lagrangian and the Eulerian approach. If we talk about the Eulerian view, the coordinates are firmed we seek how the liquid passes by the fixed points and also measured the change of rate properties, for example, temperature, and velocity, etc. In the Lagrangian view, a fluid may be constituted a huge scale of particles where we hold on the track of each particle as per moves by time and space. The particles of fluid bring properties such as temperature, and velocity, etc. The process that partially depended on the Eulerian method is usually known as the mesh-based process. Others that are fully defending the Lagrangian method are typically known as a mesh-free process. Both ways have their distinct disadvantages and advantages.

However, there are such kinds of methods built based on both Lagrangian and the Eulerian approaches to get advantages by both grouped hybrid methods and frames. ROM reduced-order modeling replaces the real model with a completely shorter order model that still tells the significant phenomenon of the methods through satisfactory correctness. The major idea beyond this ROM model is finding a decreasing basis that has an important scale of degrees of liberty compared to the original kind of model. The finest and popular method we prefer to see is a paradigmatic basis for simulation for heat transfer and fluid for POD proper orthogonal decomposition.

Brenner T. A. et al. have sort out the problem of multi-phase heat transfer by utilizing ROM reduced-order modeling where they get a very great agreement along with the complete order models. They have talked about several practical problems of proper orthogonal decomposition POD in their paper. Habashi T et al. and Lieu T et al., and Lappo V et al. have real-time CFD simulation by utilizing ROM reduced-order modeling. Lieu T et al. have built a complete air-craft layout, and the results were similar to the method of Lattice Boltzmann. It is a distinct type of process that sorted Lattice Boltzmann's equation instead of the equation of the Navier Stokes. In this model, a fluid flow through a particle mesh where these are met along the lattice mesh discrete nodes.

In engineering, for more information about the LBM Lattice Boltzmann model and its application, we prefer the development of the lattice Boltzmann model. Extension to this model for a thermal multiphase flow and heat transfer has already been described. The convicting heat transfer issue has been discussed by Rosdzimin A. R. M. et al., where they get advantages for the LBM model by directly sorting the equation of Navier Stokes. This lattice Boltzmann model process will be utilized to imitate the Knudsen number low transfer heat many multiplexes issues, for example, the droplet of a thermal behavior type on a rigid surface and improved heat transfer on nanofluids. By using this model, it enables the investigators to sort out the multiplex heat transfer and multi-phase problems in around a few past years by the real-time CFD such as Gevelera M. et al. has executed LBM lattice Boltzmann model to imitate the several multiplex fluid flows and also the real-time CFD simulation.

The centrifugal pumps during their utilization in engineering applications show the effect caused by its main factor that is vibration; while operating the centrifugal pump, it is necessary to keep a record of the intensity of vibration. The reason for keeping such a record is that it concerns the safety purpose related to the pump; the irregular of more vibration may lead to a risk of safe functioning of the pump. For making the performance of centrifugal pump qualitative, the aspect of vibration plays a vital role. The calculation of vibration from every corner of the pump evaluates the best functioning. The more you reduced the intensity of the vibration, the more you are close to utilizing the pump safely.

A centrifugal pump is a machine which more likely to operate within an intermediate frequency range. The speed of velocity indicates a certain level of vibration in the pump. Every direction in a particular centrifugal pump evaluates it is performed through the status of vibration. Sometimes with the increase in flow rate, there appears a gradual reduction in vibration of the pump, while at some points the flow rate increases along with the gain of the vibration. Additionally, the consistent change in flow among x, y, z directions of the pump show a definite difference in the vibration structure, its consistency depends upon the consistent difference along with the directions.

The design of frequency in all these three directions relies on the difference of amplitude. The monitoring points of the pump exhibit more excitation regularities than other points. Besides this, as we have discussed above that the flow rate exists side by side with the vibration regularity in a centrifugal pump, the flow rate incorporates with mechanical force to produce high frequency in the pipelines of the pump. This duo of mechanical force and flow rate regulates the functioning of the pump with increased frequency in vibrations. Furthermore, the mechanical force creates less vibration at the foot side of the pump as compared to other monitoring points like the inlet and outlet section of the centrifugal pump. The characteristic of vibration is more effective in dealing when you get successful in finding the source of it. The mechanism of vibration frequency is a numeric method of calculating the flow rates and mechanical force.

All parts of the pump either it is inlet or outlet, the foot point or any other, all perfectly engage to give outcome in industry. The flow rate in the pump along with variable vibration frequency at different parts collectively carries out the function of a centrifugal pump. The vibration pattern of the pump mainly relies on the flow of incited pulses and mechanical force that work on a centrifugal pump. Sometimes it may happen the data collection of impellers varies though they are the parts of the same pump their working may generate a gap between them. Also, the gap depends upon the dents and the number of dents. The centrifugal pumps with vane diffusers are more likely less in speed than the standard speed of 190. The basic source of vibration in such pumps are mechanical force and flow rate. Their duo perfectly helps in generating the vibration pulses. The efficiency of the pump increase or decreases with the amount of flow throughout the pump.

The adjustment of the valve opening pipe involves operating the progression condition of the pump while controlling and changing the flow. There are mainly three conditions on which the flow of the pump can be monitored; 0Qdec, 1.0 Qdec, and overfilling flow status like 1.5 Qdec flow rates create vibration spectra that would assess the oscillation state of the pump at that moment. All these things are needed to be in knowledge while using a centrifugal pump at the industry level. The pattern or design of the flow rate makes utilize these centrifugal pumps into many functions. For studying these different flow rates, inlet pipe and inlet closet are being used in estimating the palpitation of the cycle.

This is necessary to keep in notice because the performance of a centrifugal pump can vary with the amplification of vibration with different velocity and as result; it will make you observe important effects on the number of vibrations that were carried out. Moreover, overflow may cause the velocity of vibration to be as low as 0.59mm/s. The centrifugal pump's velocity depends upon the pulsation regularity. The flow rate in a centrifugal pump symbolizes the number of vibrations as per its velocity. As much as the flow will increase that eventually cause the reduction in amplitude of the vibration.

While functioning, the centrifugal pump shows the dominant vibration range at the inlet pipe and inlet cavity. As the flow is used to make its way through the inlet pipe and inlet cavity, it may show the fluctuation of vibration due to the operating system of a centrifugal pump. At the operating procedure of the centrifugal pump, the amplification of vibration may vary with the irregular pressure at different points. The condition in which you are going to operate the pump will affect all aspects like the vibration of the flow rate. This characteristic at both points like inlet and outlet generates a frequency difference in the flow.

Working of the centrifugal pump at different flow rates showcases the intensity of the vibration at the inlet and outlet sections. If the pump carries out the flow at a lower rate like 0Qdec or the overloading rate, these different conditions will allow the pump to produce different frequencies of vibration. With the change of flow, the pump works with different conditions. The amplitude of the number of vibrations mainly depends upon two peak points like 0Qdec and 1.5 Qc.

The working mechanism of the centrifugal pump counts the vibration frequency at various respected parts of the pump. The vibration is the outcome of mechanical force and the flow rate within the pump. As well as the condition of the monitoring system influence the vibrating state of the pump. The current supply remains the same for almost every condition except for a minor change due to the flow rate. The well functioned pump can accelerate the vibration for beneficial outcomes. The monitoring points of the pump exhibits more excitation regularities than other points.

The vibration in a centrifugal pump varies with the faulty and healthy impeller, faulty impeller proceed inflow with a high head as compared to a healthy impeller. Moreover, the vibration acceleration at some rates of the flow varies proportionally with the respective rate of the flow. The same is the case with higher and lower head; a faulty impeller at some rates of the flow offers a higher head while operating the system of the pump. In moving with the procedure the acceleration of the vibration also act linearly proportional to the flow rates and get increase after reaching specific rates. High-pressure centrifugal pumps cause pressure fluctuations because of different faulty impellers at different flow rates of the centrifugal pump.

Every single detail about the centrifugal pump is essential to know for operating it in the right ways. The main function of the pump works on the flow rate that collaborates with the mechanical force to produce valuable findings. Also, the thorough knowledge about the working of a pump prevents it from damages and breakdowns, while operating it, it is necessary to check and balance the flow rate and vibration, and frequency for a smooth process. Each part of the pump regulates of rather gets instructions by the schedule of vibrations those to produce through the inlet and outlet section of the pump.

If the system gets in touch with some error or break down the technology for dealing with centrifugal pump helps sort out the problem. A centrifugal pump consists of various parts like impellers; each part integrates well to generate a valuable effect. If you want to experience the working of a centrifugal pump, ahead towards the industries, their working lot can show the effects of centrifugal pump productively. Though every part in the pump particulate in the well-functioning of a pump but the impellers' role is vital in producing the high energy with the acceleration of vibration. The frequency of the vibrations in the pump varies directly with the consistency of the flow if the flow rate goes on increase the vibration increase and if the rate of flow drops, the vibration amount automatically drops.

Wang et al, studied that the vibration and resonance qualities of the five-stage marine pump are changed when and vibration inspection in the audible spectrum. Research identifies the difference in the vibratory characteristics of the centrifugal pump when cavitation occurred in that machine. Chen at al formulate a hybrid numerical method to initiate vibration and resonance imposed by a change in the flow of the marine centrifugal pump. The results of the numerical showed that the fluctuation in pressure amplitude depends on the grid size near the wall and the BPF was the vibration spectrum of the volute.

All the studies concluded that the vibration and resonance reduction in the centrifugal pump was based on the implementation of a vibration-induced mechanism. So, after installing a FRIS in the centrifugal pump the result analyzed showed that the maximum vibration intensity of the pump had been reduced by 88.0 % at M1-M4. And the vibration intensity of the inlet flange is slightly higher at the base pump and largely higher at the bracket pump. However, the vibration intensity at the outlet flange is slightly higher at the bracket pump and largely higher at the body pump whereas the vibration intensity is lowest at the connecting plate. Another positive result that has been calculated by installing FRIS in the centrifugal pump is that the maximum vibration velocity is reduced by 83.3% of the APF at M1-M4. As the rate of flow increases, the vibration velocity level of the pump decreases at M1-M4 and M9.

Another way to improve the performance of the centrifugal pump through Computational Fluid Dynamics is by the use of an auxiliary mass neutralizer which is commonly known as a Vibration Absorber. Vibration Absorber is also beneficial to reduce the load response affecting the whole mechanism. Domestic centrifugal pumps need a high application to carry water. Because of erosion related to the impeller, it is bearing, and mainly due to the presence of unbalanced residual forces, in the long term of using a centrifugal pump produces unpleasant sounds along with high vibration. The Tuned Vibrator Absorber (TVA) includes the mass damping and stiffening producing elements within it. Fraham introduced the solution of TVA in 1909 for deducting the mechanical vibrations imposed by the monotonic harmonic forces. Turned Vibrator Absorber solutions have been largely used around the world for vibration control in mechanical and engineering systems.

III. METHODOLOGY

The recommended project is designed to bring out the various modifications in the architect of the impeller, its development processing, and scrutinizing of the centrifugal pump which will further produce the result of improved efficiency. The basic steps used in the methodology are listed below.

I. The current design of the impeller has been studied to examine the recent performance and the various faults and issues that have been arising due to it.

II. The modification has been made in the design of the impeller based on the requirement of the head, discharge through the impeller will be gratified to meet the demands of Pump Testing Standard (IS 6595 Part-1).

Modification in outlet angles of the existing impeller
↓
Geometrical modeling of the impeller model
↓
Static Analysis of the impeller model
↓
CFD Analysis of the impeller Models-Base model, Model no.1, 2, 3 & 4
↓
The casting of the best impeller model from CFD Analysis
↓
Performance testing & Validation of the improved results

Figure 2: Methodology Flow Chart

3.1 Alteration in outlet angles of the impeller

The modification made in the outlet angles of the impeller is done based on the objectives listed below:

a) To evaluate the performance of CFD analysis related to the contemporary design of the centrifugal pump is focused on lowering down the pressure produced in the impeller outlet along with the low amount of the velocity generated in the process.

b) Another modification that has been carried was based on the optimization of the design related to the impeller geometry to increase the amount of the velocity produced in the outlet of the impeller.

Description	Base Model	Model No 1	Model No 2	Model No 3	Model No 4
Blade Angle	20^0	19.36^0	25.11^0	26.88^0	20.01^0
Mass flow rate	11 lit/sec				

Table 1: Alteration in the outlet angles of Impeller

3.2 Geometrical displaying and interlocking of altered impeller model

To perform the static analysis of the centrifugal pump or the computational fluid dynamic analysis of the centrifugal pump. It is necessary to convert the 2D model of the pump into its version of the solid model which has been done by implementing the process of CATIA.

The abbreviation of the CATIA is a computer-aided three-dimensional interactive application. It is a software suite formed to multi-platform works such as computer-aided manufacturing (CAM), computer-aided engineering (CAE), and computer-aided design (CAD) along with the PLM and 3D model production. CATIA was developed by the Dassault System which is a French company.

As CATIA assists multiple stages of product manufacture from design, conceptualization, and engineering to its production. It is a CAx-software referred to as 3D Product Lifecycle Management. It has been used as facilitating tool which collaborates with engineering by its integrated cloud service and is used in the disciplines of shape and design, electrical, fluid surfacing along with mechanical engineering and electronic system design. CATIA has also been used in a broad range of industries from defense to aerospace and packaging design along with the usage in the field of architecture.

The geometric modeling of the impeller has been presented in figure 3. While meshing of the impeller model has been shown in figure 4. Table .2. is depicting the details of meshing where the remodeling of model 2 has been done. There are 68803 nodes in it along with the 321001 numbers of the element. The quality of model 2 is rated as 0.85.

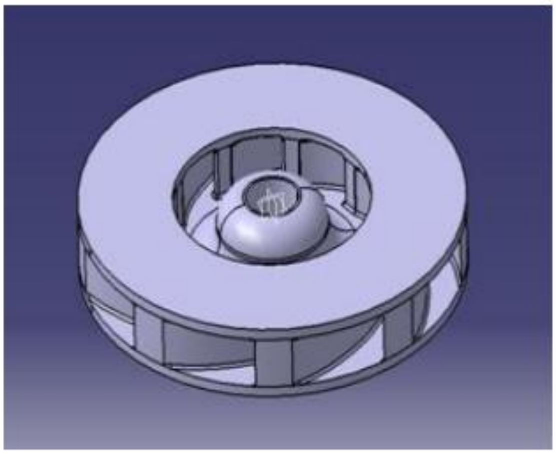


Figure 3: Geometric displaying of the impeller

Figure 4: Interlocking of impeller Model

Description	No. of Nodes	No. of Elements	Quality (Skewness)
Model No 2	68803	321001	0.85

Table 2: Interlocking details

3.3 Static Analysis

The workbench of the ANSYS static structural module is been used to bring out the comparative study based on the numerical static analysis related to the impeller of the centrifugal pump. ANSYS is an American company that manufactures and markets simulation software regarding Multiphysics engineering's such as product evaluation, product design, and its operation along with that the company provide its services and products to the customer present worldwide. The ANSYS was founded back in 1970 by John Swanson. ANSYS also went public on the forum of NASDAQ back in the 2000s. The American made various acquisitions of other companies related to engineering design, admitting additional technology in the field of fluid dynamics, electronic analysis, and design along with their other physics evaluation and checking. On 23 December 2019, ANSYS also became a part of the NASDAQ-100 index.

In Table 2. the comparative analysis has been done between the basic model of the impeller and the modified impeller of the centrifugal pump. The comparative analysis has been made based on vector displacement, tangential displacement, radial displacement, and equivalent von mises stress respectively. The values for the base design are 0.0023 mm, 0.0023416 mm, 0.000482 mm, and 30.84 MPa respectively. Whereas the values for the impeller of the modified design are 0.0035 mm, 0.003364 mm, 0.0011165 mm, and 8.98 MPa respectively.

Quantity	Base Design	Modified Design (Rev_02)
Vector Displacement	0.0023 mm	0.0035 mm
Tangential Displacement	0.0023416 mm	0.003364 mm
Radial Displacement	0.000482 mm	0.0011165 mm
Equivalent Von Mises Stress.	30.84 MPa	8.98 MPa

Table 3: Assessment of static analysis

3.4 CFD Analysis

The CFD analysis of the evaluated design of the centrifugal water pump is been carried out by using the analysis software of ANSYS. As this project is related to the flow of fluid and variables such as pressure and velocity are added in the project. So, the ANSYS software has been selected in evaluating the study related to the CFD module of the centrifugal pump.

The geometries of the design have been modified then been incorporated into the software of ANSYS. The analysis of the CFD has been performed to find the considerable improvement shown in the department of head, velocity, and pressure, etc.

The results after CFD analysis are discussed in figure 5 and figure 6. In Table 3:

related to impeller velocities, the average velocity of the impeller outlet and pump outlet of the base model has been compared with the impeller outlet and pump outlet of Model No.1, Model No.2, Model No.3, and Model No.4 respectively. The base model averages velocities are 12.15 and 3.85 respectively. While the average velocities of impeller outlet of Model No.1, 2, 3, and 4 are 11.75, 12.97, 12.72, 12.23 respectively. However, the averages velocities of pump outlets of Model No.1, 2, 3, and 4 are 3.79, 3.64, 3.79, and 3.86 respectively.

In Table 4. related to the computational fluid design of the impeller model, the angle, pressure, and the impeller outlet are being compared between the base design and the Model No.1, 2, and 3. The value of angle and pressure of base design are 20^0 and 165587.4 Pa respectively. Whereas the angle of the Model No.1, 2, 3, and 4 are 19.36^0, 25.11^0, 26.88^0 and 20.01^0 respectively. However, the pressure of the Model No.1, 2, 3, and 4 are 158907.9 Pa, 204563.3 Pa, 168234.1 Pa, and 185132.3 Pa.

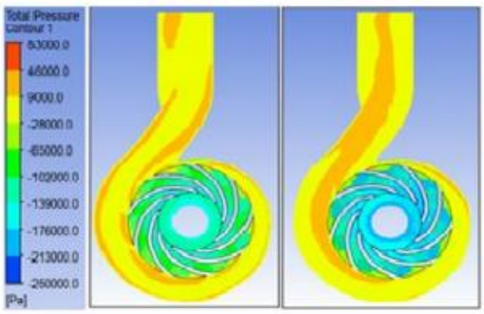
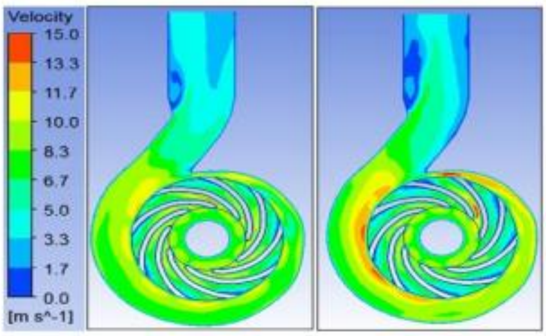


Figure5: Pressure contours of impeller models



Base Model Model No2

Figure 6: Velocity contours of impeller Model in CFD

Average Velocity (m/s)		
Cases	Impeller Outlet (m/s)	Pump Outlet (m/s)
Base	12.15	3.85
Model No 1	11.75	3.79
Model No 2	12.97	3.64
Model No 3	12.72	3.79
Model No 4	12.23	3.86

Table 4: Impeller velocities

Description	Angle	Pressure in (Pa)	Impeller Outlet (m/s)
Base	20^0	165587.4	12.15
Model no 1	19.36^0	158907.9	11.75
Model no 2	25.11^0	204563.3	12.97
Model no 3	26.88^0	168234.1	12.72
Model no 4	20.01^0	185132.3	12.23

Table 5: CFD analysis of impeller models

3.5 Performance Testing (Readings)

After performing the analysis of computational fluid dynamics (CFD), the values of Model No.1, Model No.2, Model No.3, and Model No.4 have been compared among them. Then the casting of the modified impeller of the centrifugal pump of Model No.2 has been selected based on its angle, inlet impeller, and outlet impeller velocities and pressure and is termed as Impeller No.2 now. Then the next step has to fix the impeller. Impeller No.2 is installed in the casting and then evaluated for further testing.

The results after performance testing are given in Table 5. And Figure 7: Table 5: Related to the Experimentation results of the Base Impeller Model and Impeller No.2 have been compared. The comparison has been made upon the Discharge, Pump Input, Pump Output, and Efficiency in percentages of the Base Impeller Model and Impeller No.2 respectively. The values of Discharge, Pump Input, Pump Output, and Efficiency in percentages of the Base Impeller Model are 11.20 lit/sec, 3.181 kW, 2.098 kW, 66.0% respectively. Whereas the values of Discharge, Pump Input, Pump Output and Efficiency in percentages of the Modified Impeller (Impeller No.2) are 12.02 lit/sec, 2.349 kW, 3.272 kW, and 71.79 % respectively. However, the results of efficiency in percentages and discharge in lit/sec of Base Model of Impeller and Modified Model of Impeller are respectively given in the Graphical Representation of Figure 7. It is then concluded from the results that the Modified Model of Impeller is the improved version of the Base Model of Impeller.

Case	Base Impeller Model	Modified Impeller (Model No 2)
Discharge (lit/sec)	11.20	12.02
Pump Input (kw)	3.181	2.349
Pump Output (kw)	2.098	3.272
Efficiency in %	66.0	71.79

Table 6: Research results

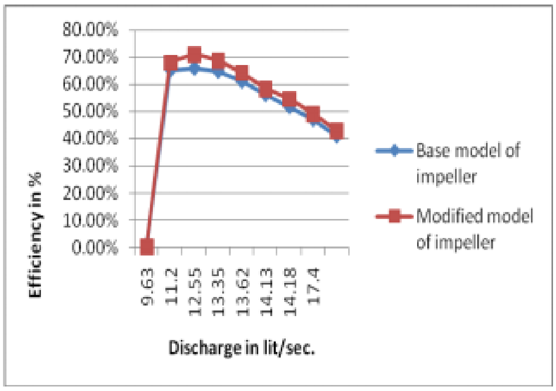


Figure7: Graphical illustration of results

The performance of the centrifugal pump can be improved by using the methods by the Computational Fluid Dynamics (CFD) and one of the methods includes the usage of the vibratory isolator to ensure that the transmission regarding the excitations can be decreased from the different part of the centrifugal pump. A floating raft isolation system is also known as FRIS is a type of vibratory isolator and its influences the vibratory characteristics of the marine centrifugal pump in a better way.

Installing the FRIS in the centrifugal pump, the vibration intensity of the centrifugal pump at inlet flange is slightly higher than as of pump bracket and larger than as of pump body and the vibration intensity is lowest at the connecting plate. Centrifugal marine pumps are mainly used in the cooling water system, drainage system, and water circulating system besides protecting the ship from a fire emergency.

Marine centrifugal pumps generate a high amount of vibration and noise during the operational hours causing air pollution which directly affects the crew member's health along with the reduced shelf life of the equipment and exposing the ship's position. It is thereby reducing the resonance and vibration of centrifugal pumps becomes one of the leading problems in operating marine pumps. In the recent past, so many studies have been carried out on the issues related to resonance and vibration of the marine centrifugal pump and some of these studies showed favorable results.

IV. CONCLUSION

The impeller model 2 shows that there is a considerable enhancement of performance if the pressure drops.

The consumption of power in the modified impeller is less when compared to the base model.

Experimental analysis has shown that geometry after modification is improved. The efficiency is calculated to be enhanced from 66% to 71.79%. Therefore, validation of improvement in the centrifugal pump using computational fluid dynamics is effective.

REFERENCES

[1]. Al-Obaidi, A. R. (2019). Monitoring the performance of centrifugal pump under single-phase and cavitation condition: A CFD analysis of the number of impeller blades. Journal of Applied Fluid Mechanics, 12(2), 445–459.

[2]. Bao, N. T., Yang, C., Kim, B., & Kim, J. (2017). Internal Flow Analysis of Seawater Cooling Pump using CFD. Journal of The Korean Society of Marine Environment & Safety, 23(1), 104–111.

[3]. Benturki, M., Dizene, R., & Ghenaiet, A. (2018). Multi-objective optimization of the two-stage centrifugal pump using NSGA-II algorithm. Journal of Applied Fluid Mechanics, 11(4), 929–942.

[4]. Deshmukh, D., & Samad, A. (2019). CFD-based analysis for finding critical wall roughness on the centrifugal pump at design and off-design conditions. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 41(1), 58.

[5]. Ding, D., Han, Q.-L., Xiang, Y., Ge, X., & Zhang, X.-M. (2018). A survey on security control and attack detection for industrial cyber-physical systems. Neurocomputing, 275, 1674–1683.

[6]. Dönmez, M., & Yemenici, O. (2019). A Numerical Study on Centrifugal Pump Performance with the Influence of Non-Newtonian Fluids. International Journal of Sciences, 8(04), 39–45.

[7]. Ge, Z., He, D., Huang, R., Zuo, J., & Luo, X. (2020). Application of CFD-PBM coupling model for the analysis of gas-liquid distribution characteristics in a centrifugal pump. Journal of Petroleum Science and Engineering, 194, 107518.

[8]. Jin, J., Fan, Y., Han, W., & Hu, J. (2012). Design and analysis of a hydraulic model of

the ultra-low specific-speed centrifugal pump. Procedia Engineering, 31, 110–114.

[9]. Kadam, V. S., Gawade, S. S., Mohite, H. H., & Chapkhane, N. K. (2011). Design and development of split case pump using computational fluid dynamics. International Conference On Current Trends In Technology,' NUiCONE–2011.

[10]. Lim, S. E., & Sohn, C. H. (2018). CFD analysis of performance change by the inner surface roughness of a double-entry centrifugal pump. Journal of Mechanical Science and Technology, 32(2), 697–702.

[11]. Mohan Kumar, M., Raj, H. E. D., & Vratharaj, M. (2014). Analysis of the effect of Impeller Parameters on Performance of Centrifugal Pump using CFD. International Journal of Research in Mechanical Engineering, 1(03).

[12]. Morrison, G., Yin, W., Agarwal, R., & Patil, A. (2018). Development of Modified Affinity Law for Centrifugal Pump to predict the effect of Viscosity. Journal of Energy Resources Technology, 140(9).

[13]. Patil, M. N., Joshi, G. S., & Naik, P. V. R. (2017). Validation in the improved performance of Centrifugal pump using CFD. Int. Res. J. Eng. Technol, 4(6), 1674–1678.

[14]. Patil, P. M., Gawas, S. B., Pawaskar, P. P., & Todkar, D. R. (2015). Effect of geometrical changes of the impeller on centrifugal pump performance. International Research Journal of Engineering and Technology (IRJET), 2(2), 220–224.

[15]. Pei, J., Yin, T., Yuan, S., Wang, W., & Wang, J. (2017). Cavitation optimization for a centrifugal pump impeller by using the orthogonal design of the experiment. Chinese Journal of Mechanical Engineering, 30(1), 103–109.

[16]. Wang, C., Shi, W., Wang, X., Jiang, X., Yang, Y., Li, W., & Zhou, L. (2017). Optimal design of multistage centrifugal pump based on the combined energy loss model and computational fluid dynamics. Applied Energy, 187, 10–26.

[17]. Yadav, K. K., Mendiratta, K., & Gahlot, V. K. (2016). Optimization Of Design Of Mixed Flow Centrifugal Pump Impeller Using CFD. International Journal of Research in Engineering and Technology, ISSN, 2319–1163.

Appendices

Appendix 1 - Graphical illustration of results