International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 10 Issue: 06 | Jun 2023 www.irjet.net p-ISSN: 2395-0072

Optimization and Cavitation Analysis of Centrifugal Pump with Variable Working Parameters Using Ansys CFX

Nishant Kundu[1], Preetham A[2]

[1]B.tech, Dept of Mechanical Engineering, NIT Kurukshetra
[1]VD Engineer, SAE B.tech, Dept of Mechanical Engineering, NIT Kurukshetra

---

Abstract –

Centrifugal pumps are widely used in a variety of applications. However, their performance can be affected by a number of factors, including rotation speed (RPM), mass flow rate, blade number, and blade geometry. This study used computational fluid dynamics (CFD) to investigate the impact of these factors on the internal flow behavior of a centrifugal pump. The results showed that increasing the RPM led to higher output pressure, power output, and head rise, but also decreased efficiency due to heightened turbulence and frictional losses. The mass flow rate was also found to be a critical factor, with excessive flow rates leading to overloading and power losses, while insufficient flow rates resulted in diminished performance and overall efficiency. The blade number and geometry were also found to have a significant impact on flow behavior. Initially, a higher blade number enhanced efficiency by facilitating more efficient power transfer to the fluid. However, beyond a threshold of eight blades, diminishing returns were observed due to complex flow patterns and increased losses. Notably, blade geometry played a pivotal role, with the outlet blade angle exhibiting substantial influence on parameters such as efficiency and output pressure, while the inlet blade angle had limited significance.

Overall, this study provides valuable insights into the internal flow characteristics of centrifugal pumps. The findings of this study have a number of implications for the design and operation of centrifugal pumps and could also be used to improve efficiency and reliability in practical applications. The results could be used to design pumps that are more resistant to cavitation and also to develop new methods for optimizing the performance of centrifugal pumps.

Key Words: Centrifugal Pump, Ansys CFX, Vista CPD, CFD Analysis, CFD Post Processor, Stimulation, Component Systems, Cavitation.

1. INTRODUCTION

Centrifugal pumps, as fascinating hydraulic machines, can convert mechanical energy into hydraulic energy by harnessing the power of centrifugal force. By leveraging this force upon a fluid, these pumps induce velocity, which is subsequently transformed into a flowing motion. The intricate workings of a centrifugal pump are made possible through an assembly of mechanical components, each playing a crucial role in its operation. Fig.1.1 provides a detailed diagram of a centrifugal pump, highlighting the key elements that contribute to its operation, including the eye, impeller, volute casing, suction pipe, shaft, and delivery pipe.

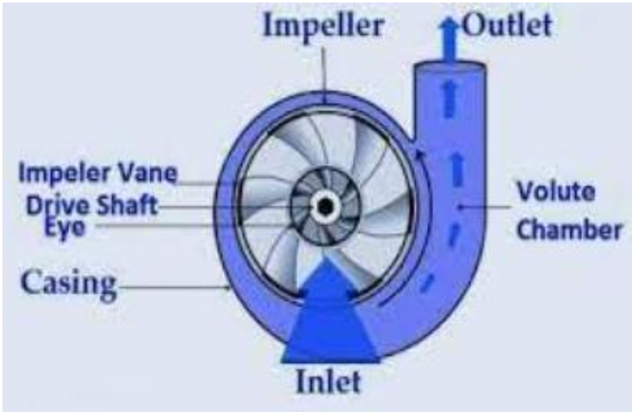


Fig.1.1 Various Components of Centrifugal Pump [7]

Several studies have focused on different aspects of centrifugal pumps, contributing to the understanding and improvement of their design and performance. Raut et al [1] explored the utilization of Ansys CFX for simulations, emphasizing geometry selection and the advantages of using the Ansys component system Vista CPD. Elida et al [2] provided insights into impeller geometry creation, meshing, and boundary condition implementation. Muttali et al. [3] analyzed the operating characteristics of centrifugal pumps, including the head-flow rate relationship and the impact of RPM on cavitation. Sun et al. [4] investigated the influence of rotational speed and flow rate variations on performance parameters. Bhupatni et al. [5] studied the effects of inlet and outlet blade angle variations on head and efficiency. Tabar et al. [8] developed a procedure for cavitation analysis in centrifugal pumps and validated it through impeller damage comparison. Jaiswal et al. [9] examined the relationship between Net Positive Suction Head (NPSH) and cavitation, proposing a cavitation model. These studies collectively contribute to the understanding, design, and performance optimization of centrifugal pumps, benefiting various industrial applications.

1.1 METHODOLOGY

In the study, several steps were followed to analyze a centrifugal pump using the CFX software. First, the Vista CPD component system was utilized to calculate the necessary parameters and determine the shape of the blades and volute. The design of the impeller blade and volute was created using the Bladegen feature and Create Volute feature. The data obtained from Bladegen was then transferred to Turbogrid to generate the mesh, which was subsequently transferred to the CFX Setup. Additionally, the mesh of the volute was also transferred to the CFX Setup. Figure 1.2 depicts the interconnection of various component systems of Ansys for the setup of our stimulation. Figure 1.3 shows the final geometry created and which is used for our simulation work. To conduct the simulation, the Shear Stress Transport K-ω model was employed.

In the CFX Setup, the Turbomode was utilized to analyze the centrifugal pump, with appropriate fluid and boundary conditions specified for both the rotating and stationary domains. The solver controls were set to 500 iterations, achieving the required convergence of 1e-04. The solver was initiated with double precision, and the results were subsequently analyzed in the CFD post-processor. Various parameters such as pressure, velocity, and efficiency were checked, along with the analysis of cavitation pressure pulsations, through the examination of their contours and plots. These steps provided valuable insights into the performance and behavior of the centrifugal pump under different operating conditions

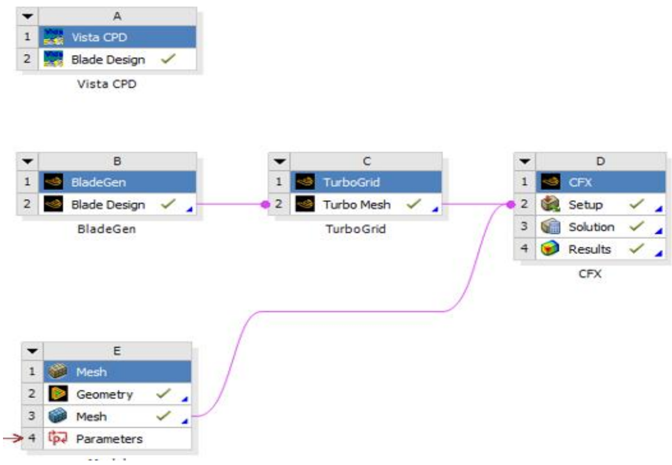


Fig.1.2 Setup for CFX Stimulation

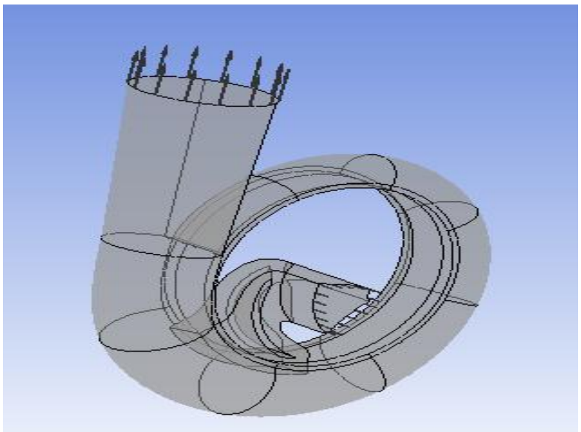


Fig.1.3 View of centrifugal pump geometry created by Vista CPD for CFD simulations

2. MODEL SETUP AND GOVERNING EQUATIONS

A. *Laminar and Turbulent Flow*

If flow streamlines are parallel to each other then the flow is called laminar one and if flow streamlines crosscut each other then the flow is called turbulent flow. Osborne Reynolds experimented and found a parameter whose value helps us to understand the flow nature. Equation 2.1 shows Reynolds number (Re)

$$Re = \frac{\rho * D * v}{\mu} \tag{2.1}$$

Where **ρ** is the density of the fluid, **μ** is the viscosity of the fluid, **D** is the diameter of the pipe and **v** is the velocity of the fluid. If Reynolds number is less than 2100 flow is laminar and if it is more than 4000 flow is turbulent.

B. *N-S Equations*

N-S stands for Navier Stokes Equations represented in equation 2.2. These equations account for Momentum conservation in viscous incompressible fluid flow. These Equations were established by French scientist Navier in 1821 and British physicist Stokes in 1845. Vector form is:

$$\rho \frac{D\vec{v}}{Dt} = \rho f - \nabla p + \mu \nabla^2 \vec{v} \tag{2.2}$$

Here ρ is density, f is an external force on fluid per unit volume, p is pressure, $\vec{v}$ is velocity vector and **μ** is dynamic viscosity.

C. *Continuity Equation*

Mass conservation equation for incompressible fluid flow. Where $\vec{v}$ is relative fluid velocity vector in equation 2.3.

$$\nabla \vec{v} = 0 \quad (2.3)$$

D. *Turbulence Model*

To describe turbulence, various models exist, such as the standard k−ε, renormalization group k−ε, standard k−ω, SST k−ω, and Reynolds stress model. In steady simulations, the SST k−ω turbulence model is used in combination with the incompressible continuity equation and the Reynolds time averaged Navier–Stokes equations. Additionally, the SAS-SST turbulence model is utilized in unsteady simulations to capture flow separations and vortices more accurately.

2.1 MESHING AND GRID INDEPENDENCE STUDY

The grid independence test plays a crucial role in validating the accuracy and reliability of a simulation model. It helps determine whether the results obtained from the simulation are influenced by the size or resolution of the grid used. To conduct a grid independence test, simulations are run at multiple grid sizes or resolutions, and the results are compared. The objective is to identify the minimum grid size or resolution needed to achieve accurate results. In this case, after performing the grid independence test, a mesh consisting of 540,046 elements was selected. Figure 2.1 depicts the results of the test, showing that a mesh with approximately 500,000 elements falls within the sufficiently converging zone, validating the correctness of its results. Our chosen mesh, consisting of 540,046 elements, is marked by a vertical line and falls within the safe zone, ensuring reliable simulations.

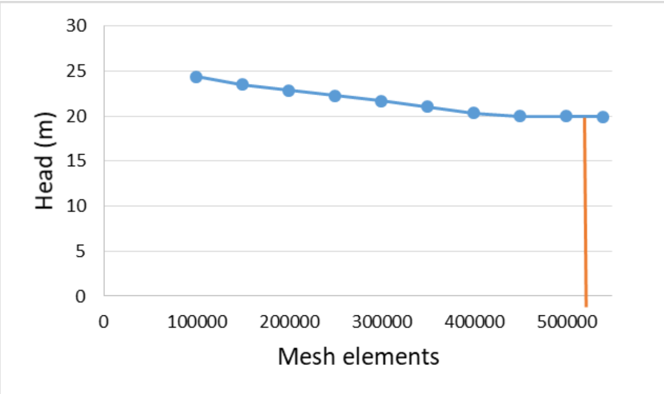


Fig.2.1 Variation of Head with Mesh Elements

2.2 BOUNDARY CONDITIONS

For designing pump geometry using Vista CPD few parameters are needed, we have taken rotational speed as 1450 rpm, volume flow rate as 280 m^3/h, density as 1000 kg/m^3, head rise as 20 m and the rest of other settings as default. Boundary conditions are vital in simulations across multiple disciplines, ensuring accurate and realistic results. These conditions encompass factors like initial conditions, behavior limits, and external influences. Precise definition and implementation of boundary conditions are crucial for accurate and effective simulations, making them indispensable in modeling and simulation endeavors. Static Pressure inlet and mass flow outlet is selected here for stimulation purpose. The static pressure within a system typically remains constant at atmospheric pressure (1 atm). However, the mass flow rate at the outlet of the system varies depending on specific conditions. For instance, if the volume flow rate differs, the corresponding mass flow rate will also differ. Here, the Volume flow rate is taken as 280 m3/h so for water mass flow rate comes out to be 77.78 kg/s. At once only one parameter is altered keeping all other constant and its effect is observed on other parameters.

2.3 VALIDATION STUDY

Validation studies are crucial in Computational Fluid Dynamics (CFD) simulations. They help establish the accuracy and reliability of simulation results by comparing them to experimental data or analytical solutions. To validate our model for a centrifugal pump, we compare operational characteristics like efficiency and head versus flow rate curves with results from another researcher, Table 2.1 shows the setup details used by Dr. Ramkrishna et, al [6] same inputs are used here too.

Flow rate (m^3/h)	3600
Head (m)	60
RPM	600
Efficiency	Maximum

Table 2.1 Details of the setup taken by Dr. Ramakrishna et. al [6]

Operating characteristics are plots of pump parameters such as head, efficiency, etc. concerning flow rate. These curves are prominent in understanding the fluid flow behavior in any fluid machinery. As visible in Fig.2.2 and 2.3 we get a good comparison between the results obtained by us and the results obtained by Dr. Ramakrishna et al. [8] We can see that success is achieved in getting both good accuracy as well as precision so we can go forward with our results.

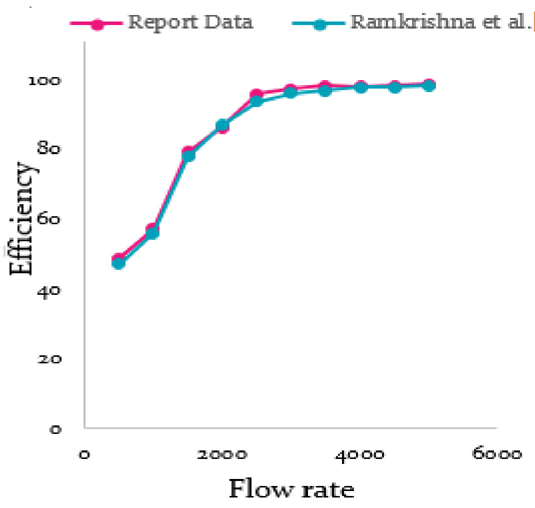


Fig.2.2 Efficiency vs Flow rate plot

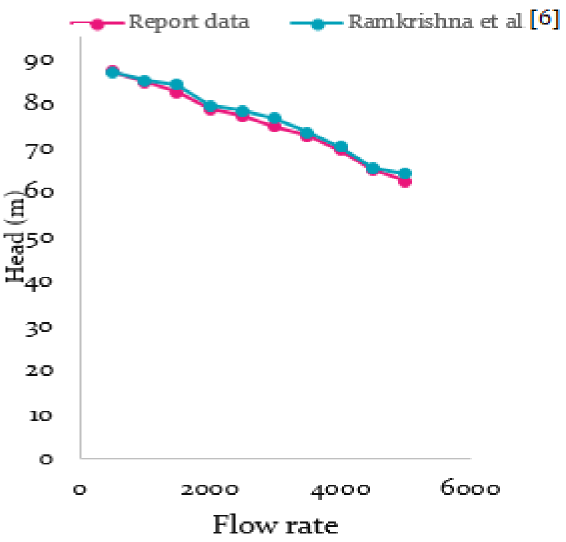


Fig.2.3 Head vs Flow rate plot

3. RESULTS AND DISCUSSIONS

Table 3.1 depicts a list of initial setup data for stimulations in Ansys CFX. This data is utilized for generating our pump geometry using Vista CPD and then according to the necessary alterations are made to it.

Table 3.1 Initial Setup Data Stimulations

RPM	1450
Head	32 m
Mass Flow Rate	77.78 kg/s
Number of Blades	6
Inlet Blade Angle	13.98^0
Outlet Blade Angle	30^0

A. *Effect of Variation of RPM*

When the RPM of a pump is increased while keeping other factors constant, several things happen. The head, output pressure, power, and moment of the pump increase significantly. However, this increase in RPM also leads to higher turbulence and friction losses, resulting in a decrease in overall efficiency. Figure 3.1 illustrates the changes in the head, output pressure, efficiency, and moment as the RPM increases. Increasing the RPM of a centrifugal pump can enhance flow rates and pressures, but it can also cause cavitation, greater power consumption, and increased wear and tear on the pump components same as depicted in Figure 3.1. The ideal RPM for a centrifugal pump depends on the specific application and pump design. The following curves provide insights into how fluid flow is affected by the increase in RPM.

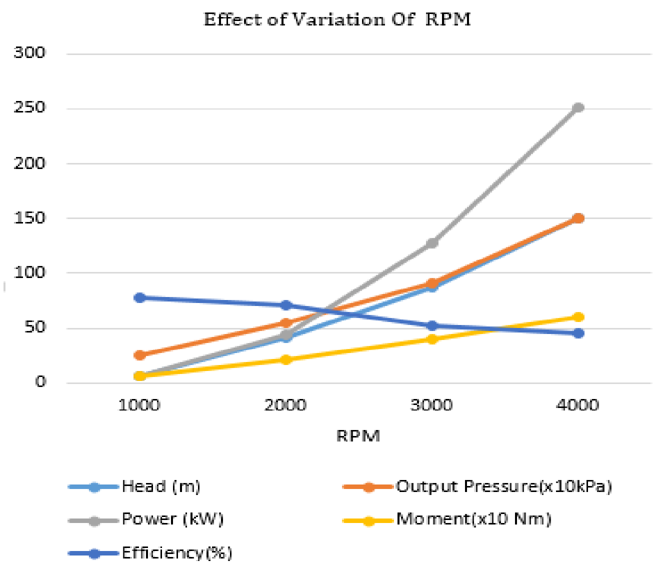


Fig.3.1 Variation of different parameters with RPM

B. *Effect of Variation of Blade Number*

An increase in the number of blades in a centrifugal pump can have a noticeable effect on its performance. Generally, as the blade number increases, the pump's efficiency tends to improve initially. This is because a greater number of blades can effectively capture and transfer fluid energy. However, there is a point where adding more blades can lead to diminishing returns. Beyond a certain blade number, the efficiency may start to decrease due to factors such as increased friction and flow disturbances Figure 3.2 shows a similar pattern. After reaching optimal blade number 8 factors like head rise, output pressure, and power output also showed a mild decrease. It is important to carefully consider the optimal blade number based on specific requirements and operating conditions to achieve the desired performance and maximize the pump's efficiency.

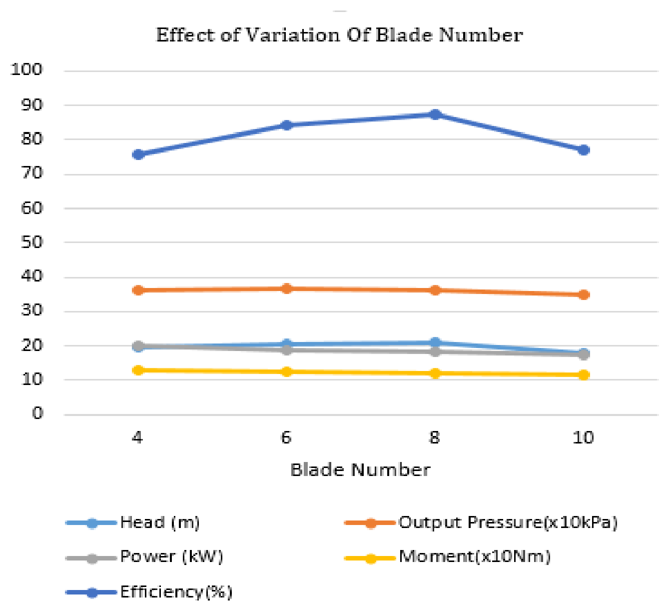


Fig.3.2 Variation of different parameters with Blade Number

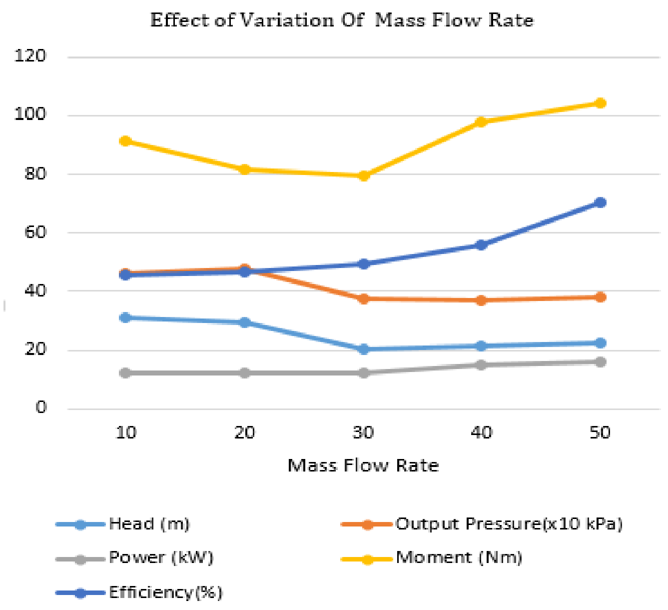


Fig.3.3 Variation of different parameters with Mass flow rate

C. *Effect of Variation of Mass flow rate*

The effect of varying mass flow rates in centrifugal pumps is significant and directly impacts their performance. As the mass flow rate increases, the pump experiences higher demand and needs to work harder to maintain the desired flow. This leads to an increase in the energy required to drive the pump and may result in reduced efficiency. Additionally, higher mass flow rates can cause increased turbulence and higher fluid velocities within the pump, which can potentially lead to issues such as cavitation and increased wear on the pump components. On the other hand, decreasing the mass flow rate below the pump's design point may lead to inefficient operation and reduced performance. Therefore, it is crucial to carefully consider and optimize the mass flow rate to ensure optimal pump performance, efficiency, and longevity. As visible in Fig.3.3, for our pump first on increasing mass flow rate a decrease is observed in the moment required to turn the blades, further the decrease in head rise and output pressure signifies reduced performance. As we increase the mass flow rate further, a simultaneous increase in efficiency is observed it signifies that the optimal mass flow rate for our pump design is around 50 lps (liters per second) and we are heading towards higher performance levels.

Changes in blade shape, such as curvature, twist, and blade angle, can influence crucial factors like pump efficiency, pressure rise, and overall operating characteristics. Careful consideration and analysis of blade geometry are essential to ensure optimal pump performance and maximize its effectiveness in various applications. A similar attempt is made in further two parts to find out how the performance of ourpump will be affected by changing its blade orientation. The blade Number is fixed at 6 for it and the value of blade angles is taken from the research work of Bhuptani et. al [5]. Initially, the Inlet blade angle is taken as 13.48 deg and the outlet blade angle is taken as 28 deg and then one is varied keeping the other constant.

D.*Effect of Variation of Inlet Blade Angle*

The angle at which the fluid enters the impeller blades determines how efficiently it is guided into the impeller. An optimal inlet blade angle ensures smooth and uniform flow, minimizing losses due to turbulence and recirculation. If the inlet blade angle is too steep, it can lead to excessive blockage and flow separation, resulting in reduced pump efficiency and increased energy consumption. On the other hand, if the angle is too shallow, it may cause insufficient guiding of the fluid, leading to inefficient utilization of the impeller blades. Here it can be seen in Figure 3.4 that as the inlet blade angle changed from 26 deg to 28 deg a decrease is observed in the moment required to turn the blades signifies ease in fluid flow. Other parameters are not affected significantly which indicates that for our centrifugal pump design inlet blade geometry is not a significant factor.

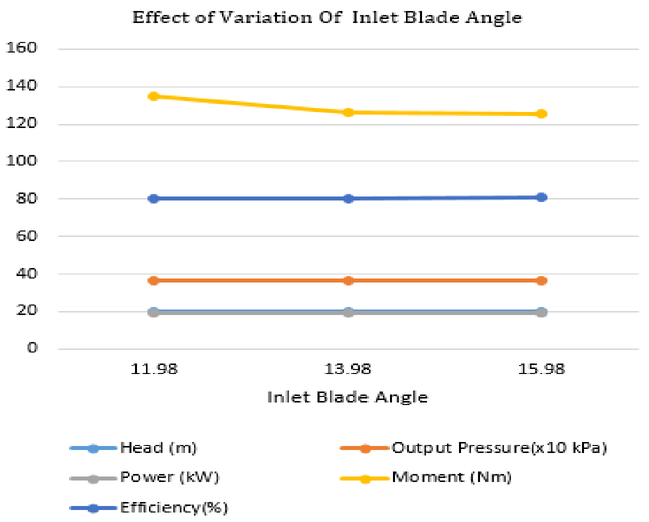


Fig.3.4 Variation of different parameters with Inlet Blade Angle

E. Effect of Variation of Outlet Blade Angle

An increase in the outlet blade angle typically leads to higher pressure output and improved pump efficiency. This is because a larger angle allows for better redirection and control of the fluid flow, minimizing energy losses. However, excessive angles can also cause problems such as flow separation and increased turbulence, leading to reduced p erformance. Conversely, a decrease in the outlet blade angle may result in lower pressure output and reduced efficiency. The fluid may experience more resistance and friction, leading to energy losses and decreased pump performance. Finding the optimal outlet blade angle that balances pressure output, efficiency, and overall pump performance is crucial. Figure 3.5 indicates a mild rise in all parameters as Outlet Blade Angle rises which is the same as expected and required.

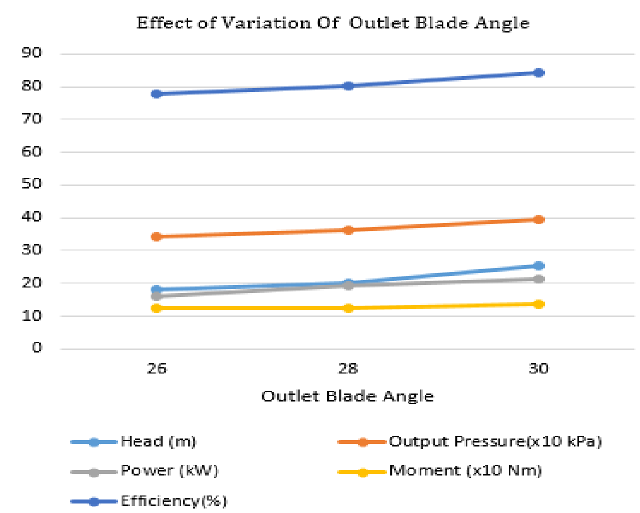


Fig.3.5 Variation of different parameters with Outlet Blade Angle

3.1 CAVITATION ANALYSIS

Cavitation Model

Cavitation occurs when the pressure falls below the vapor pressure which causes the formation of an air bubble in the pump impeller that bursts and causes erosion on the blade surface. Investigating cavitation behavior aids in improving pump reliability and lifespan, ensuring smooth operation and reducing maintenance costs. Understanding the factors that contribute to cavitation occurrence helps in identifying and mitigating potential risks of failure and damage. The degree of cavitation is estimated in terms of Net Positive Suction Head (NPSH) [9] & [10] as given in equation 3.1:

$$NPSH = \frac{Pi}{\rho g} + \frac{{v_i}^2}{2g} - \frac{P_v}{\rho g} \qquad (3.1)$$

Where P_i=Absolute Pressure at the inlet, v_i = Absolute velocity at the inlet, P_v =Vapor pressure of fluid (3170 Pa, Vapor pressure of water at 25^{0}C)

Apart from it Rayleigh-Plesset equation as shown in equation 3.2 is used for calculating time-varying cavitation bubbles of radius size 10-6 m (ANSYS CFX-Solver Theory Guide, 2016)

$$R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2}\left(\frac{dR_B}{dx}\right)^2 + \frac{2\sigma}{\rho R_B} = \frac{P_v - P_i}{\rho} \qquad (3.2)$$

Where R_B is radius of cavitation bubble, σ is the surface tension coefficient between liquid and vapor, ρ is density of water. The volume fraction of 100% and 0% are used for water and water vapor respectively.

The simulation analysis incorporates boundary conditions that specify the mass flow rate at the inlet (35 kg/s, initially) and the static pressure at the outlet (250000 Pa, Initially). In this analysis, the effects of varying the mass flow rate and static pressure are investigated, by varying them one by one keeping other variables constant, and the corresponding results are thoroughly examined and discussed. By systematically altering these boundary conditions, a comprehensive understanding of their influence on the simulation results is obtained.

Figure 3.6 shows the cavitation phenomenon on our impeller blades where volume with Vapour Volume Fraction is more than 0.8 is shown in red and the blades are shown in Grey. Figure 3.7 is of an actual impeller damaged due to cavitation and we can see that, the areas affected in the actual impeller are the same as we got as a result of our stimulation.

Fig.3.6 Stimulation Result of Cavitation Analysis showing cavitation-affected regions in red.

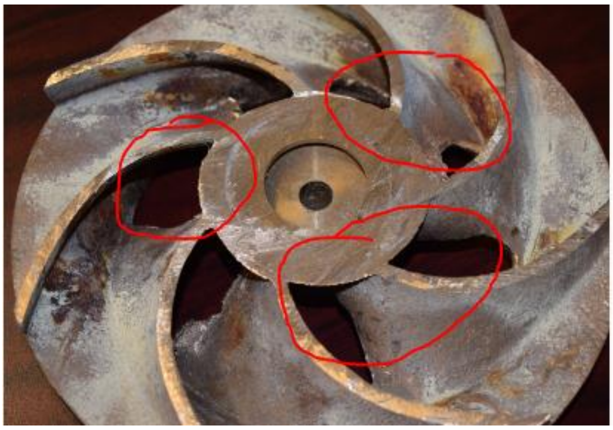
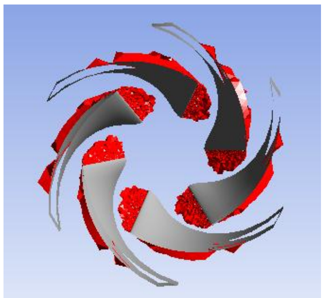


Fig.3.7 Actual Impeller damaged due to cavitation [11]

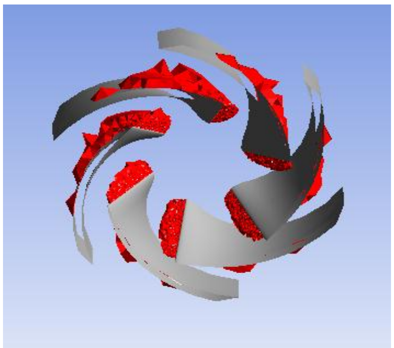
Variation of cavitation with mass flow rate

Higher mass flow rates ensure a greater supply of fluid to the pump, reducing the chances of localized pressure drops below the vapor pressure of the fluid. This, in turn, mitigates the risk of cavitation formation. On the other hand, lower mass flow rates can lead to reduced fluid availability, creating conditions conducive to cavitation. In such scenarios, the pump may experience significant damage and performance degradation due to cavitation effects. As visible from figures 3.8 and 3.9 with an increase in mass flow rate, cavitation volume where vapour volume fraction is more than 0.8 decreases but then begins increasing again.

(a) 10kg/s



(b) 20kg/s



(c) 30kg/s

Fig.3.8 Variation of Volume under cavitation with Mass flow rate

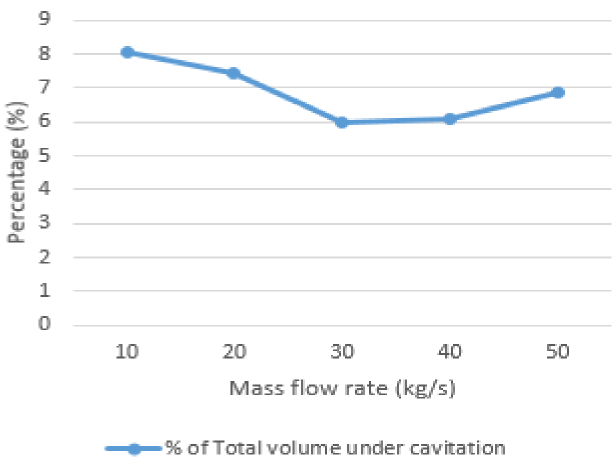
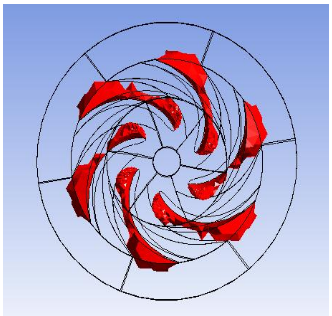


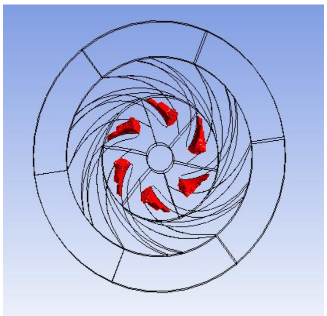
Fig.3.9 Variation of Volume under cavitation with Mass flow rate

Variation of Cavitation with Outlet Pressure

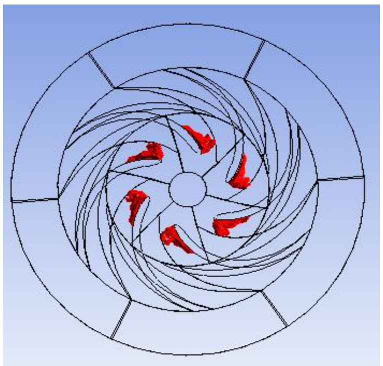
Cavitation can be initiated when the local pressure drops below the fluid's vapor pressure, leading to vapor bubbles forming. Figure 3.11 depicts how a sharp decrease is observed in cavitation volume with an increase in pressure from 100000Pa to 500000 Pa



(a) 1,00,000 Pa



(b) 3,00,000 Pa



(c) 5,00,000 Pa

Fig.3.10 Variation of Volume under Cavitation with Outlet Pressure

Figure 3.11 depicts a sharp decrease in the percentage of volume under cavitation as outlet pressure increases which is the reason for the increase in pressure at a local level that prevented bubble formation and ultimately cavitation.

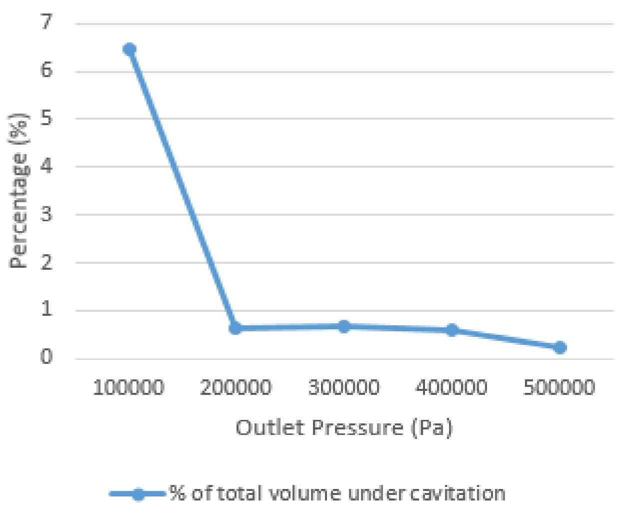


Fig.3.11 Variation of Volume under Cavitation with Outlet Pressure

4. CONCLUSIONS

In this research study, we conducted numerical simulations to provide a comprehensive understanding of the internal flow behavior of a centrifugal pump. Leveraging the powerful computational fluid dynamics (CFD) simulation software, Ansys CFX, we explored the impact of varying RPM, mass flow rate, blade number, and blade geometry on pump performance and got following results.

(1) Raising the RPM produced a higher output pressure, energy output, and head increase. However, this was accompanied by a drop in efficiency caused by heightened turbulence and friction-related losses.

(2) Too much mass flow rate caused an excessive amount of load on blades, which lead to power losses and decrease in efficiency. Conversely, inadequate flow led to poor performance and decreased productivity.

(3) We observed that the number and geometry of blades had a major impact on the flow behavior. Initially, increasing the number of blades allowed for better power transfer to the fluid, leading to enhanced efficiency. However, once the number of blades surpassed eight, the efficiency of the system decreased due to the complex flow patterns that caused additional losses.

(4) Blade geometry played a pivotal role, with the outlet blade angle exhibiting substantial influence on parameters such as efficiency and output pressure, while the inlet blade angle had limited significance.

(5) Our research examined the effects of varying mass flow rate and outlet pressure on cavitation within the centrifugal pump. Increasing the mass flow rate initially decreased the volume under cavitation, but it subsequently began to rise again.

(6) As for outlet pressure, a sharp decline in cavitation occurred once the pressure exceeded 100,000 Pa, which can be attributed to localized pressure increases that impeded bubble formation.

Overall, this study provides valuable insights into the internal flow characteristics of centrifugal pumps, elucidating the role of RPM, mass flow rate, blade number, and blade geometry in their performance. By enhancing our understanding of pump design and operation, these findings contribute to improved efficiency and reliability in practical applications.

5. ACKNOWLEDGEMENT

I would like to express my heartfelt gratitude to Dr. Rajesh Kumar, my mentor, for his invaluable guidance and support throughout this research. His expertise, encouragement and insightful inputs have been instrumental in shaping this study and expanding my knowledge in the field of centrifugal pumps. I am truly grateful for his patience, dedication, and mentorship. I would also like to sincerely thank my colleague, Preetham A., for his collaboration and assistance during this research endeavor. His contributions, discussions, and shared experiences have been immensely beneficial in enhancing the quality and depth of this work.

Furthermore, we would like to acknowledge the online resources that have played a significant role in this research like ChatGPT, YouTube, Google Scholar, Edx etc.

Lastly, we would like to express our gratitude to my college, NIT Kurukshetra, for providing us with the necessary resources and infrastructure to carry out this research. The academic environment and opportunities for learning and growth have been instrumental in our development as a researcher.

6. REFERENCES

[1] Prathamesh Raut , Rohit Rathod , Rohit Tidke , Niraj Rathod , Sanchitee Rokade and Prof. Nishant Kulkarni "Design and CFD Analysis of Centrifugal Pump" International Journal for Research in Applied Science & Engineering Technology, Volume 10 Issue XII Dec 2022

[2] Farah Elida Selamat and Wan Hariz Iskandar Wan Izhan "Design and Analysis of Centrifugal Pump Impeller for Performance Enhancement" Journal of Mechanical Engineering Vol SI 5(2), 36-53, 2018

[3] Raghavendra S. Muttalli, Shweta Agrawal and Harshla Warudkar "CFD Simulation of Centrifugal Pump Impeller Using ANSYS-CFX" International Journal of Innovative Research in Science, Engineering, and Technology, Vol. 3, Issue 8, August 2014

[4] Tianyu Sun, Renkun Wang and Zihao Zeng "Investigations of Rotational Speed and Flow Rate on Centrifugal Pump Performance Using CFD" IEEE 13th International Conference on Mechanical and Intelligent Manufacturing Technologies 2022

[5] Amit h. Bhuptani, Prof. Ravi K. Patel and K.M. Bhuptani "Design and Analysis of Centrifugal Pump"

[6] R. Ramakrishna, S. Hemalatha and D. Srinivasa Rao "Analysis and performance of centrifugal pump impeller" Materials Today: Proceedings 50 (2022) 2467–2473

[7] https://mechanicalnotes.com/wp-content/uploads/2020/03/Centrifugal_Pump-1.jpg

[8] Navid Shervani-Tabar, Reza Sedaaghi, Reza Mohajerin, Mohammad T. Shervani-Tabar, and Raed I. Bourisli "Experimental and Computational Investigation on the Cavitation Phenomenon in a Centrifugal Pump" Proceedings of the Eighth International Symposium on Cavitation (CAV 2012)

[9] A. K. Jaiswal, A. U. Rehman, A. R. Paul, and A. Jain "Detection of Cavitation through Acoustic Generation in Centrifugal Pump Impeller" Journal of Applied Fluid Mechanics, Vol. 12, No. 4, pp. 1103-1113, 2019

[10] Ahmed Ramadhan AL-OBAIDI "Experimental Comparative Investigations to Evaluate Cavitation Conditions within a Centrifugals Pump Based on Vibration and Acoustic Analyses Techniques" Archives of Acoustics – Volume 45, Number 3, 2020

[11] https://blogs.heattransfersales.com/blog/pump-cavitation

7. BIOGRAPHIES

Nishant Kundu
Mechanical Engineer
Research enthusiast

Preetham A
Mechanical Engineer
VD & VI engineer @ Team Nitrox
CEO & Founder @ DAMN
Coding enthusiast