**TRIBHUVAN UNIVERSITY**

**INSTITUTE OF ENGINEERING**

**PULCHOWK CAMPUS**

A PROJECT REPORT ON

# CFD ANALYSIS OF CAVIATION IN CENTRIFUGAL PUMP


SUBMITTED BY:

BIPIN KHATI (077BME011)

BISHNU DEV MANDAL (077BME013)

KRISHNA GUPTA (077BME019)

SANJIV PAHARI (077BME038)


SUBMITTED TO:

DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

AUGUST, 2023

# ABSTRACT

Cavitation is a tricky problem in the way fluids flow, especially in machines called centrifugal pumps. It can cause harmful effects on how these pumps work and their physical parts. This study used computer simulations to understand cavitation better in these pumps. We looked closely at how cavitation forms inside the pump, and how it affects things like how the fluid moves, where pressures change, and how well the pump works. By studying how fluid moves and cavitation happens, we learned how it makes pumps less efficient and reliable. This helps us figure out ways to make pumps work better and last *longer*. *This research shows that computer simulations are really useful for understanding* and solving cavitation problems in pumps, which can lead to better technology for moving fluids.

***Keywords:*** *Cavitation, Centrifugal Pump, CFD*

# ACKNOWLEDGEMENT

We extend our heartfelt appreciation to the Department of Mechanical and Aerospace Engineering, IOE, Pulchowk Campus, Lalitpur, for affording us the invaluable opportunity to embark on a project aimed at enriching the insights gained during our Bachelor's journey in Mechanical Engineering.

Equally, our gratitude extends to our supervisor, Assistant Professor Mr. Neeraj Adhikari, whose guidance and support steered us through the project's intricacies. We would also like to extend our thanks to our friends and experts whose invaluable suggestions illuminated our path.

This endeavor has been a collaborative and enlightening experience, made possible by the collective efforts of these individuals and institutions.

Bipin Khati (077BME011)

Bishnu Dev Mandal (077BME013)

Krishna Gupta (077BME019)

Sanjiv Pahari (077BME035)

# TABLE OF CONTENTS


ABSTRACT ..... i

ACKNOWLEDGEMENT ..... ii

LIST OF FIGURES ..... v

LIST OF ABBREVIATIONS ..... vi

LIST OF SYMBOLS ..... vii

CHAPTER 1 : INTRODUCTION ..... 1

1.1. Background ..... 1

1.2 Problem Statement ..... 3

1.3 Objectives ..... 3

1.4 System Requirements ..... 3

CHAPTER 2 : LITERATURE REVIEW ..... 4

2.1 Cavitation Phenomenon and Centrifugal Pump Performance ..... 4

2.2 Typical cavitation occurrence situations ..... 5

2.3 Cavitation mechanism ..... 5

2.3 Interrelationship between Flow Rates, Pump Head, and Cavitation ..... 6

2.4 Impeller Geometry's Role in Cavitation Performance ..... 7

2.5 Theoretical Framework for Cavitation Prediction and Management ..... 7

2.6 Holistic Understanding and Future Research Implications ..... 8

CHAPTER 3 CHAPTER THREE: METHODOLOGY ..... 9

3.1 CAD Modelling ..... 10

3.2 Meshing ..... 10

3.3 Simulation of Centrifugal Pump ..... 10

3.3.1 Assumptions ..... 11

3.3.2 Boundary conditions ..... 11

3.3.3 Solution Parameters ........................................................................................................ 11
CHAPTER 4 : RESULT AND DISCUSSIONS ....................................................................................... 12
4.1 Contours of Velocity streamline, Velocity and Static Pressure ................................. 12
4.1 Variation of Volume fraction with rpm ........................................................................ 14
4.2 Variation of volume fraction with mass flow rate ....................................................... 19
4.3 Variation of volume fraction with suction pressure .................................................... 20
CHAPTER 5 : CONCLUSION AND FUTURE ENHANCEMENT ...................................................... 22
5.1 Conclusion ............................................................................................................................ 22
5.2 Scope of Future Enhancement ........................................................................................ 22
References ................................................................................................................................ 23

# LIST OF FIGURES

Figure 1.1: Centrifugal Pump ........................................................................................ 1
Figure 1.2: Open Impeller, semi enclosed Impeller and Enclosed Impeller....................... 2
Figure 2.1: Pump performance curve.................................................................................. 4
Figure 2.2: Thermodynamic diagram of phase change [2] ................................................ 5
Figure 2.3: Collapsing bubble shape [3] ............................................................................. 6
Figure 3.1: Block diagram of methodology ......................................................................... 9
Figure 4.1: Velocity streamline contour ........................................................................... 12
Figure 4.2: Velocity contour .............................................................................................. 13
Figure 4.3: Static Pressure Contour .................................................................................. 13
Figure 4.4: Volume fraction contour at 800 rpm .............................................................. 14
Figure 4.5: Volume fraction contour at 2000 rpm ............................................................ 15
Figure 4.6: Volume fraction contour at 3000 rpm ............................................................ 15
Figure 4.7: Volume fraction contour at 6000 rpm ............................................................ 16
Figure 4.8: XY plot of volume fraction vs. radial position at 800 rpm ............................ 17
Figure 4.9: XY plot of volume fraction vs. radial position at 2000 rpm .......................... 17
Figure 4.10: XY plot of volume fraction vs. radial position at 3000 rpm ........................ 18
Figure 4.11: XY plot of volume fraction vs. radial position at 6000 rpm ........................ 18
Figure 4.12: Volume fraction contour at 8 kg/s Figure 4.13: Volume fraction XY plot at 8 kg/s ............................................................................................................ 19
Figure 4.14: Volume fraction contour at 45 kg/s Figure 4.15: Volume fraction XY plot at 45 kg/s .......................................................................................................... 19
Figure 4.16: Contour of volume fraction at suction pressure zero ................................... 20
Figure 4.17: XY plot of vapor fraction vs. radial position................................................ 21

# LIST OF ABBREVIATIONS

| | |
|---|---|
| CFD | Computational Fluid Dynamics |
| RPM | Round per Minute |

# LIST OF SYMBOLS

| | |
|---|---|
| $\rho$ | Density of two- phase mixture |
| $v$ | Quality-averaged velocity vector of two-phase mixed flow |
| P | Pressure |
| g | Acceleration due to gravity |
| $P_v$ | Vapor Pressure |
| U | Tip velocity |
| $V_R$ | Radial Velocity |
| $P_B$ | Barometric pressure |
| $P_\infty$ | Free stream pressure |
| R | Radius |
| S | Surface Tension constant |
| $\sigma$ | Cavitation number |

# CHAPTER 1 : INTRODUCTION

## 1.1. Background

Centrifugal pump is a hydraulic machine which converts mechanical energy into hydraulic energy by the use of centrifugal force acting on the fluid. These are the most popular and commonly used type of pumps for the transfer of fluids from low level to high level. It's is used in places like agriculture, municipal (water and wastewater plants), industrial, power generation plants, petroleum, mining, chemical, pharmaceutical and many others.

When a certain mass of liquid is made to rotate by an external source, it is thrown away from the centrifugal axis of rotation and a head is impressed which enables it to rise to a higher level.

Centrifugal Pumps can be used for viscous and non-viscous liquids and has higher efficiency.

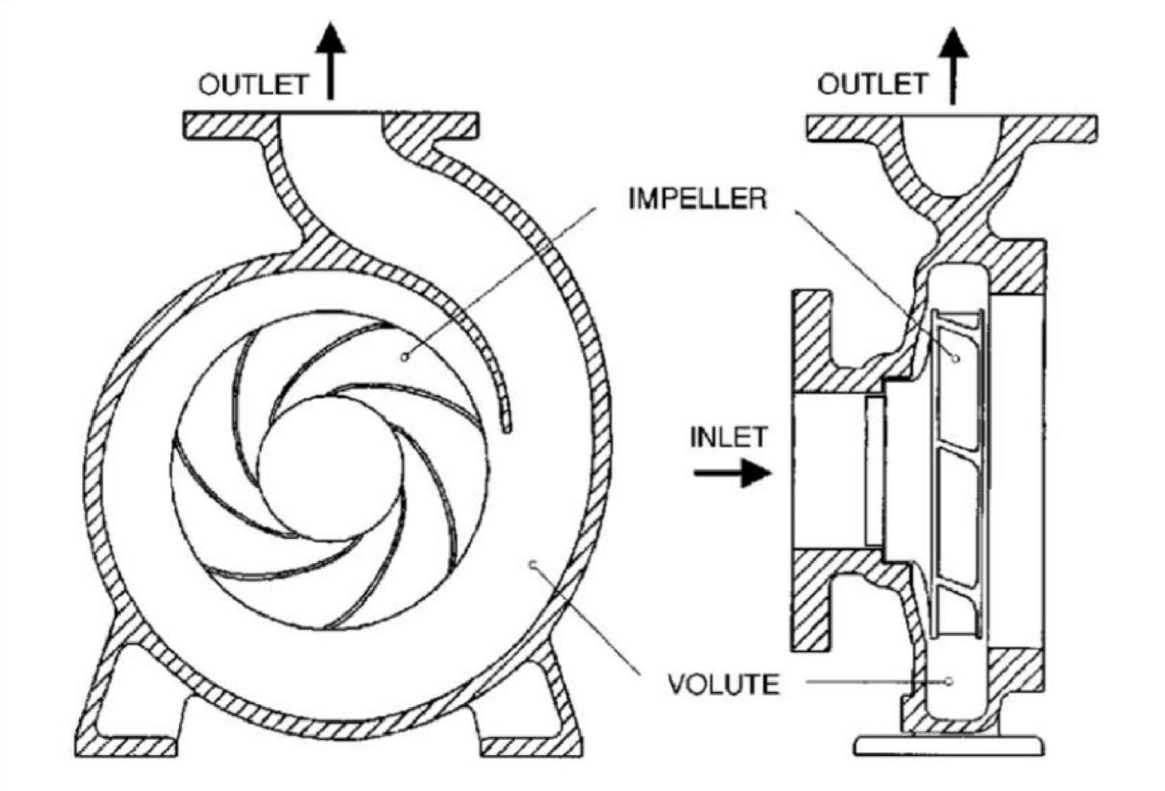


*Figure 1.1: Centrifugal Pump*

The main parts of centrifugal pump are:

**a.** Impeller

It is a wheel or rotor which is provided with a series of backward curved blades or vanes. It is mounded on the shaft which is coupled to an external source of energy which imparts the liquid energy to the impeller there by making it to rotate. Impellers are divided into three types: open Impeller, semi enclosed Impeller and enclosed Impeller.

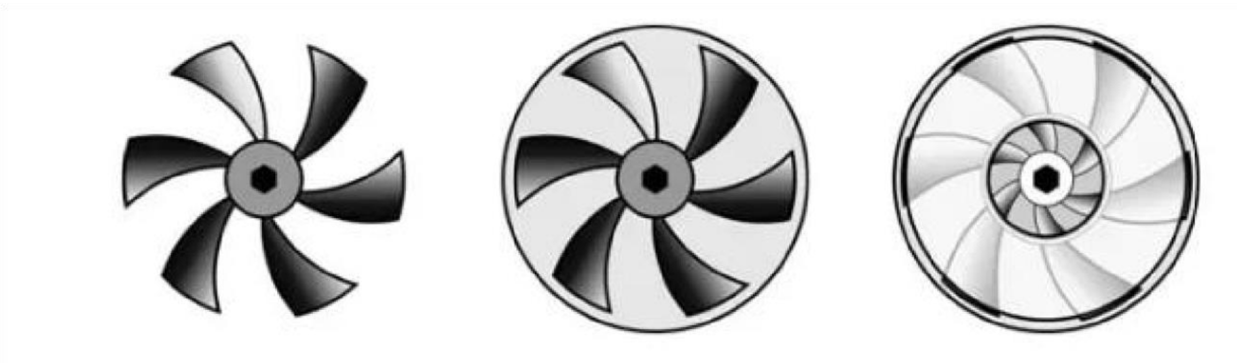

*Figure 1.2: Open Impeller, semi enclosed Impeller and Enclosed Impeller*

**b.** Delivery Pipe

It is a pipe which is connected at its lower end to the out let of the pump and it delivers the liquid to the required height. Near the outlet of the pump on the delivery pipe, a valve is provided which controls the flow from the pump into delivery pipe.

**c.** Casing

It is a pipe which is connected at the upper end to the inlet of the pump to the centre of impeller which is commonly known as eye. The double end reaction pump consists of two suction pipe connected to the eye from both sides. The lower end dips into liquid in to lift. The lower end is fitted in to foot valve and strainer.

Cavitation, an adverse phenomenon in centrifugal pumps, emerges when fluid pressure falls below vapor pressure, forming vapor bubbles that collapse upon entering high-pressure zones. In these pumps, cavitation typically occurs within the impeller, inducing erosion, pitting, noise, and vibration, consequently degrading hydraulic performance and mechanical integrity. The resultant shockwaves can even damage downstream components. Design optimization, material selection, operational adjustments, and predictive simulations are employed to mitigate cavitation's detrimental effects, ensuring prolonged pump efficiency and reliability.

## 1.2 Problem Statement

Cavitation poses a significant challenge for fluid-based rotating systems, a concern that cannot be avoided. Identifying areas prone to cavitation and assessing the extent of its impact on impellers is crucial for mitigating its effects. However, traditional experimental testing of cavitation is costly and lacks versatility. The occurrence of cavitation on impeller blades hinders the use of high RPM motors for applications requiring either high discharge head or low suction head. A comprehensive exploration of factors influencing cavitation is essential for the effective design of high-performance centrifugal pumps.

## 1.3 Objectives

- To conduct an in-depth examination of the phenomenon of cavitation in centrifugal pumps using Computational Fluid Dynamics (CFD) simulations.
- To investigate the effects of varying operational parameters, including RPM, mass flow rate, and suction pressure, on the formation, severity, and characteristics of cavitation within the pump.

## 1.4 System Requirements

### 1.4.1 Hardware Requirements

- Processor(s): Workstation class
- 4 GB RAM
- 25 GB hard drive space
- Computer must have a physical C:/” drive present
- Graphics card and driver: Professional workstation class 3-D
- OpenGL-capable

### 1.4.2 Software Requirements

- ANSYS 2023 R2 student version

# CHAPTER 2 : LITERATURE REVIEW

The second chapter of this study encompasses an exhaustive survey of relevant literature, focusing on the intricate phenomenon of cavitation and its implications for the operational performance of centrifugal pumps.

## 2.1 Cavitation Phenomenon and Centrifugal Pump Performance

Cavitation, a complex occurrence characterized by the creation and subsequent collapse of vapor-filled bubbles within a flowing fluid due to localized pressure drops below the vapor pressure threshold, emerges as a pivotal determinant in the functioning of centrifugal pumps. The efficiency ($\eta$) of a pump is given by the formula $\eta = gQH / bhp$, where $\eta$ is efficiency, g is gravity acceleration, Q is flow rate, H is head, and bhp is hydraulic energy. Pump efficiency is typically below unity due to practical factors. Cavitation has been found to one of the major factors that decreases pump efficiency. Cavitation decreases pump efficiency by introducing inefficiencies and disruptions in the fluid flow within the pump. The intricate interplay of hydraulic, mechanical, and thermodynamic factors has garnered considerable scholarly attention, particularly concerning the repercussions of pump rotational velocities, volumetric flow rates, and the intricacies of impeller configuration on the manifestation of cavitation [1]. This introduces formidable operational challenges, concurrently unveils avenues for optimizing the performance of centrifugal pump systems.

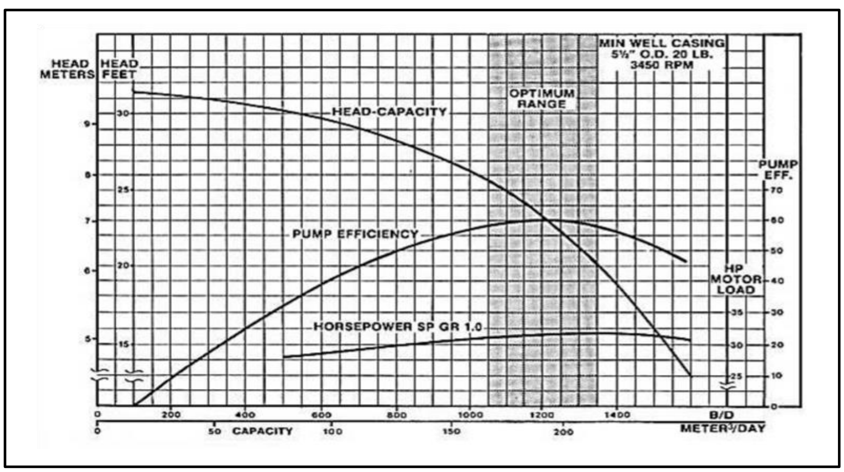


*Figure 2.1: Pump performance curve*

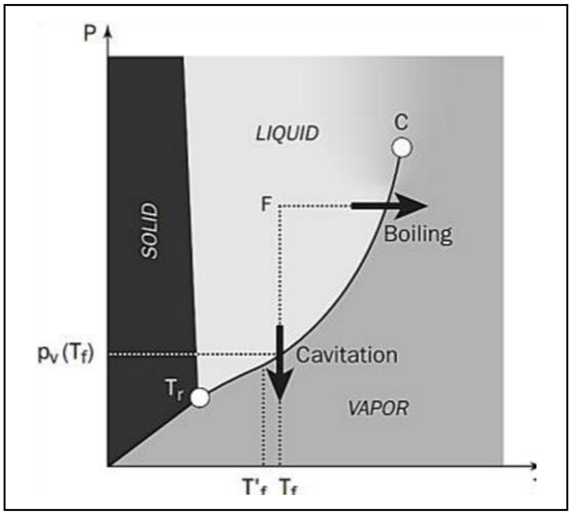


*Figure 2.2: Thermodynamic diagram of phase change [2]*

## 2.2 Typical cavitation occurrence situations

Cavitation is likely to occur in fluid flow under various circumstances:

A. In areas with tight geometries causing high velocities and pressure drop, seen in flow constriction and pump impellers.
B. When neighboring flows with disparate velocities generate turbulence and pressure fluctuations, as seen in jets and wakes.
C. In highly unsteady flows that accelerate fluid temporally, resulting in pressure drops.
D. Rough surfaces at fluid flow boundaries cause wakes and attached cavities.
E. Water in gaps between mechanical system parts can lead to cavitation when walls move.
F. Immersed solid bodies with sharp edges accelerate fluid, causing pressure drops and increasing cavitation likelihood.

## 2.3 Cavitation mechanism

I. Fluid Nucleation: Nucleation involves state changes in substances around weak points called nuclei. This can happen due to conditions such as gaps forming between liquid molecules (homogeneous nucleation), gaps between liquid molecules and container walls (heterogeneous nucleation), presence of micro bubbles that grow with tension, and bubbles forming from external radiation.

II. Cavitation Inception: Cavitation inception relates to fluid pressure nearing vapor pressure, often quantified by the "cavitation number." When this number drops to specific values, bubbles form, and the cavitation number becomes the inception cavitation number ($\sigma_i$). Further reduction leads to more numerous and larger bubbles.

III. Bubble Growth: The growth of a single bubble to its maximum size follows the Rayleigh-Plesset equation, relating bubble radius change to fluid pressure. Assumptions include spherical symmetry and absence of thermal effects.

IV. Bubble Collapse: During bubble collapse, instability can lead to a re-entrant jet. If near a solid wall, the jet forms from the bubble's far side, hitting the wall with high speed, potentially causing material removal. Conversely, near a free surface, the jet is oriented away. (Figure 11 compares re-entrant jets measured by Lauterborn and Bolle [1975] and Plesset and Chapman [1971].)

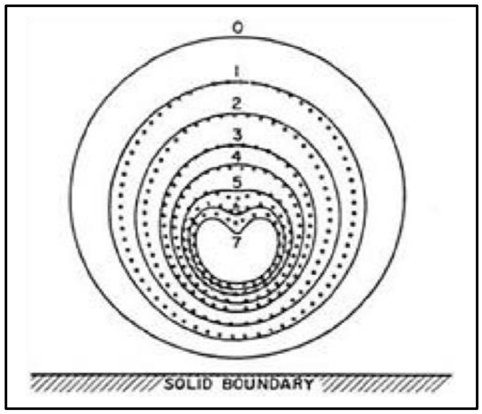


*Figure 2.3: Collapsing bubble shape [3]*

## 2.3 Interrelationship between Flow Rates, Pump Head, and Cavitation

Ahmed Ramadhan Al-Obaidi's seminal contribution in 2019 provides significant insights into the reciprocal interdependence between pump head, volumetric flow rates, and the propensity for cavitation initiation [1]. Al-Obaidi's work highlights that at elevated flow rates, cavitation phenomena are likely to transpire when the Net Positive Suction Head Available (NPSHa) descends beneath the prescribed Net Positive Suction Head Required (NPSHr) [1].

$$NPSH = \frac{P_1 - P_V}{\rho g}$$

$$\mathrm{NPSH_A} = \left(\frac{p_i}{\rho g} + \frac{V_i^2}{2g}\right) - \frac{p_v}{\rho g}$$

$$NPSHr = K_a \frac{V_{R1}^2}{2g} + K_b \frac{U_1^2}{2g}$$

This illustrates the interconnected relationship between hydraulic losses, volumetric flow dynamics, and the initial phases of cavitation.

## 2.4 Impeller Geometry's Role in Cavitation Performance

The inception of cavitation process mostly occurred at the eye of the impeller around or closed to the impeller blades leading edges [1]. The research conducted by Maxime Binama and colleagues in 2016 underscores the pivotal significance of impeller geometry in shaping the dynamics of cavitation [4]. Binama's study accentuates the profound influence of tailored impeller designs on the overarching performance attributes of centrifugal pumps, necessitating meticulous control through the amalgamation of experimental and numerical methodologies [4]. The intricate genesis of cavitation emerges from the convergence of tangential and axial velocities, precipitating the creation of vapor nuclei [4]. Binama's research elucidates the complex interplay intrinsic to the cavitation phenomenon, where vapor nuclei oscillate between implosive and explosive tendencies, engendering ramifications that extend beyond impeller passages to diffuser blades and volute channels.

## 2.5 Theoretical Framework for Cavitation Prediction and Management

The foundational work of W. A. Spraker in 1985 advances a theoretical framework that establishes a correlation between the Available Net Positive Suction Head (ANPSH) and the holistic domain of Total Cavitation Potential (TCP) [5]. This theoretical underpinning accentuates the profound impact of pump operational modalities on the emergence of cavitation phenomena [5]. The single bubble growth to its maximum radius (size) is governed by the Rayleigh-Plesset equation or its variations [5]. This equation gives out the relationship between the bubble radius variation and the surrounding fluid pressure.

$$\frac{P_B(t) - P_\infty(t)}{\rho_L} = R\frac{d^2R}{dt^2} + \frac{3}{2}\left(\frac{dR}{dt}\right)^2 + \frac{4\nu_L}{R}\frac{dR}{dt} + \frac{2S}{\rho_L R}$$

This equation is based on assumptions like spherical bubble symmetry and the absence of all thermal effects. The insights garnered from this theoretical construct furnish a valuable framework for prognosticating and effectively managing the diverse manifestations of cavitation across heterogeneous pump configurations.

## 2.6 Holistic Understanding and Future Research Implications

The combination of insights from Al-Obaidi's experiments, Binama's focus on impellers, and Spraker's theoretical framework emphasizes the importance of fully understanding the complex dynamics behind cavitation. Skillfully designed pump structures and thoughtful operational changes are crucial for reducing the harmful impacts of cavitation and ensuring reliable and energy-efficient centrifugal pump operation. As this field develops, ongoing research remains crucial for addressing new challenges and continually improving pump systems.

# CHAPTER 3 CHAPTER THREE: METHODOLOGY

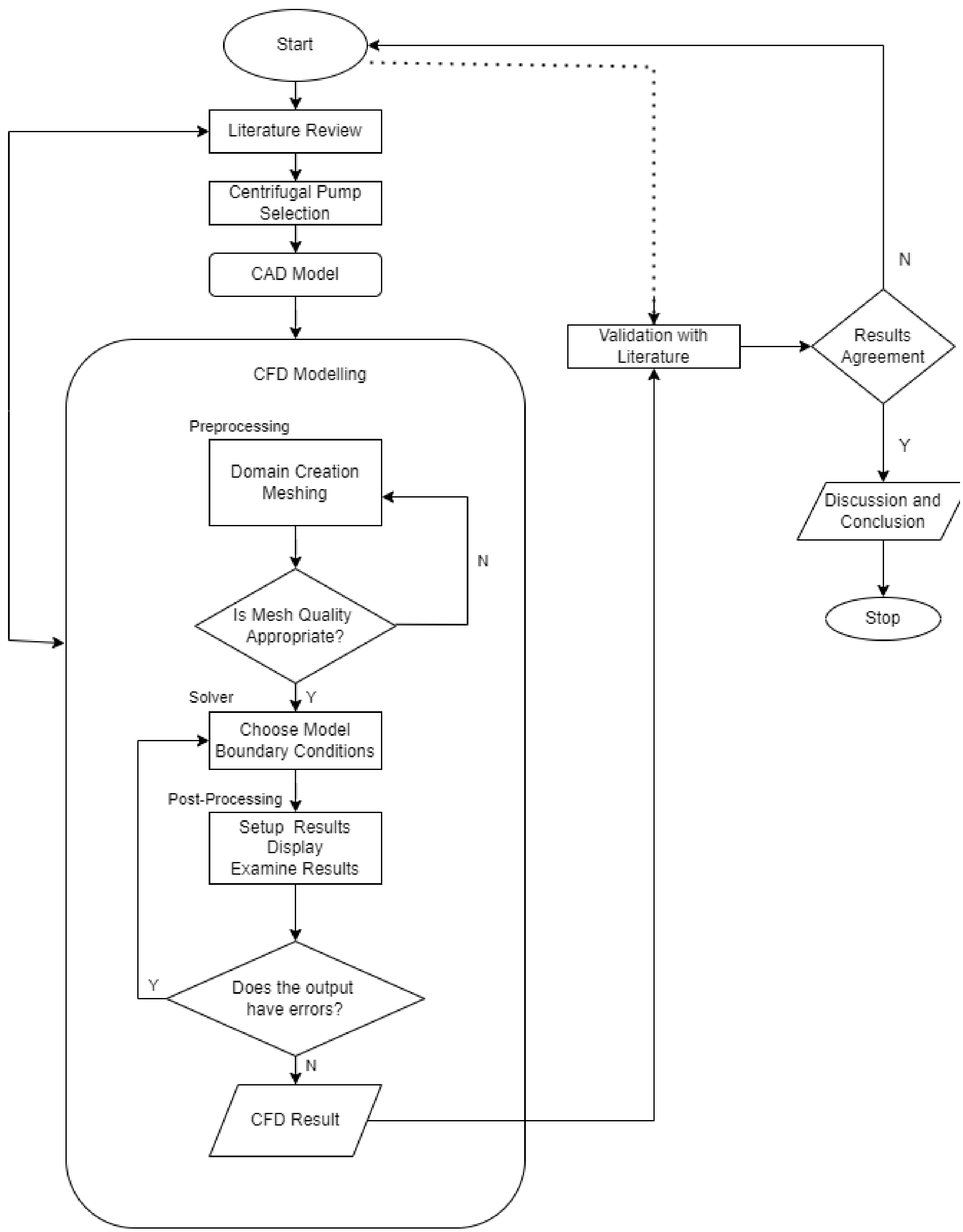


*Figure 3.1: Block diagram of methodology*

### 3.1 CAD Modelling

The centrifugal pump for the simulation was designed by CAD software SolidWorks which has eight blades. The CAD model of centrifugal pump is shown below.

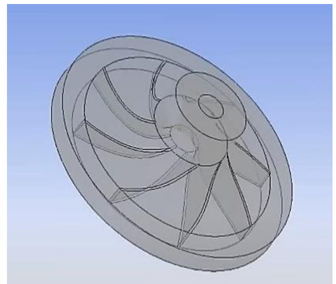

*Figure 3.2 Centrifugal pump CAD geometry*

### 3.2 Meshing

In this project, the geometry and the mesh of a eight blade pump impeller domain were generated with Ansys Workbench.The meshes with tetrehedral cells are used for the domain of impeller as shown in Figure 3.2.

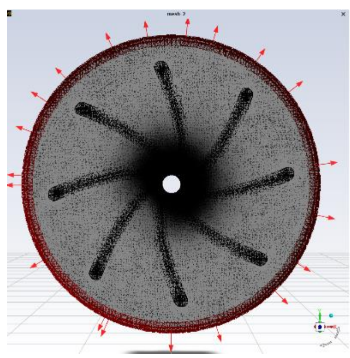

*Figure 3.3 Pump impeller with fine mesh*

| Mesh | Fine mesh |
|---|---|
| No of Nodes | 54375 |
| No of Elements | 266745 |

### 3.3 Simulation of Centrifugal Pump

After meshing of the model pump assembly CFD is used for simulation of the pump performance.The boundary conditions are applied.The performance results are obtained at different operating speed ,mass flow rate and suction head.The numerical simulation is carried out to detect the cavitation in centrfugal pump in terms of volume fraction.

### 3.3.1 Assumptions

For flow simulation inside the centrifugal pump,the following basic assumptions are considered:

- Incompressible fluid flow
- Steady state condition
- The walls were assumed to be smooth
- Constant fluid properties.

### 3.3.2 Boundary conditions

These are the set of conditions specified for the behavior of the solution to a set of differential equations at the boundary of its domain.Mathematical solutions are determined with the help of boundary conditions to many physical problems.These conditions specify the flow and thermal variables on the boundaries of a physical model.

The centrifugal pump has various components like impeller,inlet,outlet,hub and shroud.At inlet of pump,total pressure boundary condition was defined and mass flow rate boundary condition at outlet of pump was defined.And other surfaces were given as wall boundary conditions.And no slip wall condition is applied to rotating faces of impeller.The operating temperature is specified at inlet of pump.There must be no slip at fluid –wall interface.

### 3.3.3 Solution Parameters

The solution parameter is very important for solving any CFD problem. The standard k-Ɛ model is used for turbulence modelling with standard wall function. The standard k-Ɛ model is a widely used turbulence model in computational fluid dynamics (CFD) for simulating turbulent flow. It's a two-equation model that provides information about the turbulent kinetic energy (k) and the turbulent dissipation rate (Ɛ). The k-ε model aims to predict turbulence effects in a variety of flow situations, including those involving boundary layers and more.

In CFD simulation water and air vapor (mixture) was used as working fluid. Then, cavitation was studied in terms of volume fraction. The volume fraction equals one represents only presence of vapor in that region where cavitation occurs. Number of iteration used for the simulation of centrifugal pump analysis was 100.

# CHAPTER 4 : RESULT AND DISCUSSIONS

Cavitation causes loss of performance and degradation of life in the centrifugal pumps. Hence the analysis of cavitation is a very important aspect of any centrifugal pump. With the use of CFD tools it is possible to have a forecast about the cavitation places looking for the pressure field, since the cavitation has a direct relation with the vapor pressure at the flow fluid temperature, becoming possible to add improvements in the project of the equipment in order to prevent or to minimize the phenomenon, without the use of experimental methods that in the most cases showing high cost.

Comparing the cavitation parameters from literature and computed results, it is found that variation in cavitation with variation in rpm, mass flow rate and suction pressure followed the same nature of variation as followed in the literature.

## 4.1 Contours of Velocity streamline, Velocity and Static Pressure

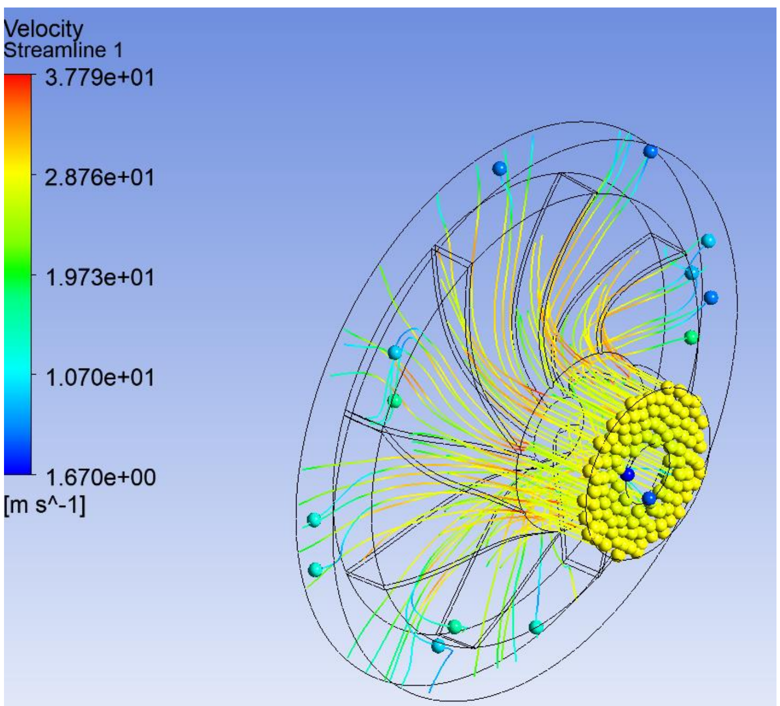


*Figure 4.1: Velocity streamline contour*

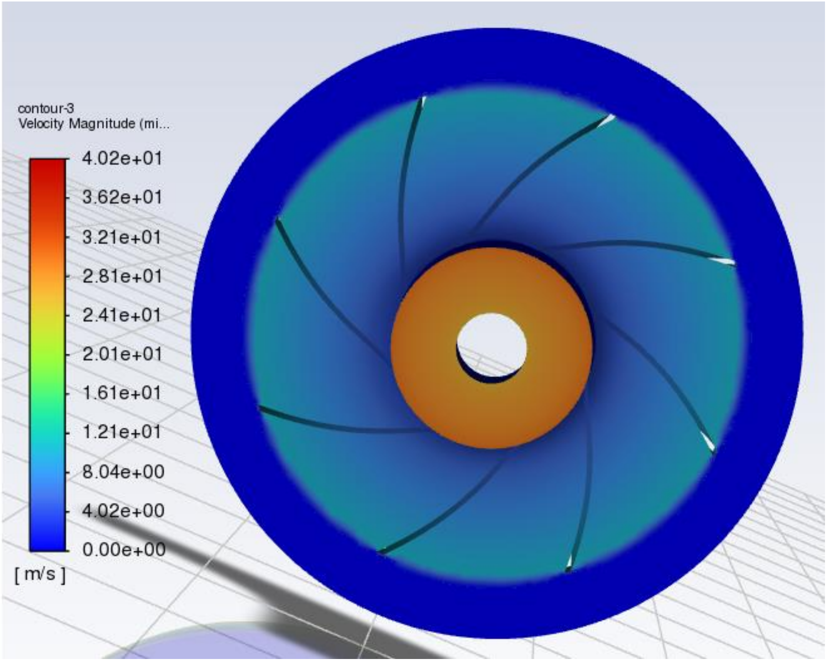


*Figure 4.2: Velocity contour*

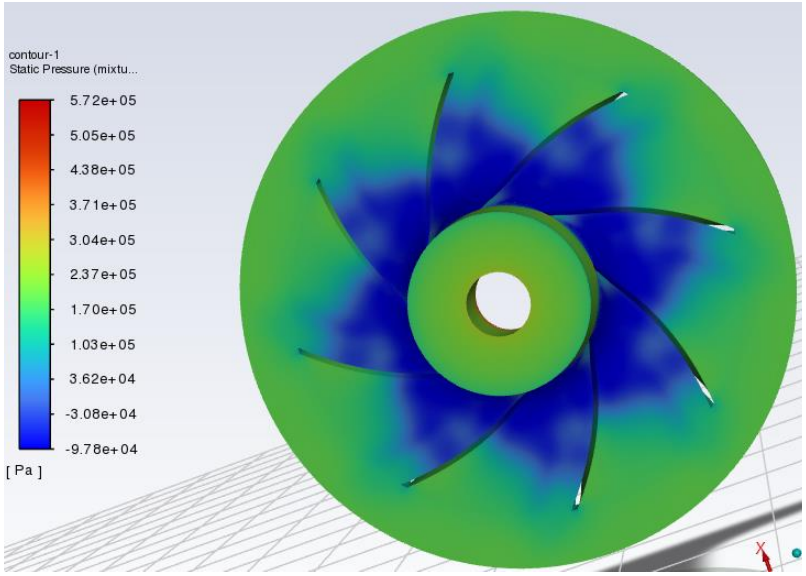


*Figure 4.3: Static Pressure Contour*

### 4.1 Variation of Volume fraction with rpm

Firstly, the value of the inlet and outlet pressure is kept constant at 29000 Pa and 200000 Pa respectively and the value of angular velocity is varied from 800 rpm to 6000 rpm. The volume fraction of vapor present in the fluid region is obtained from simulation at different rpm.

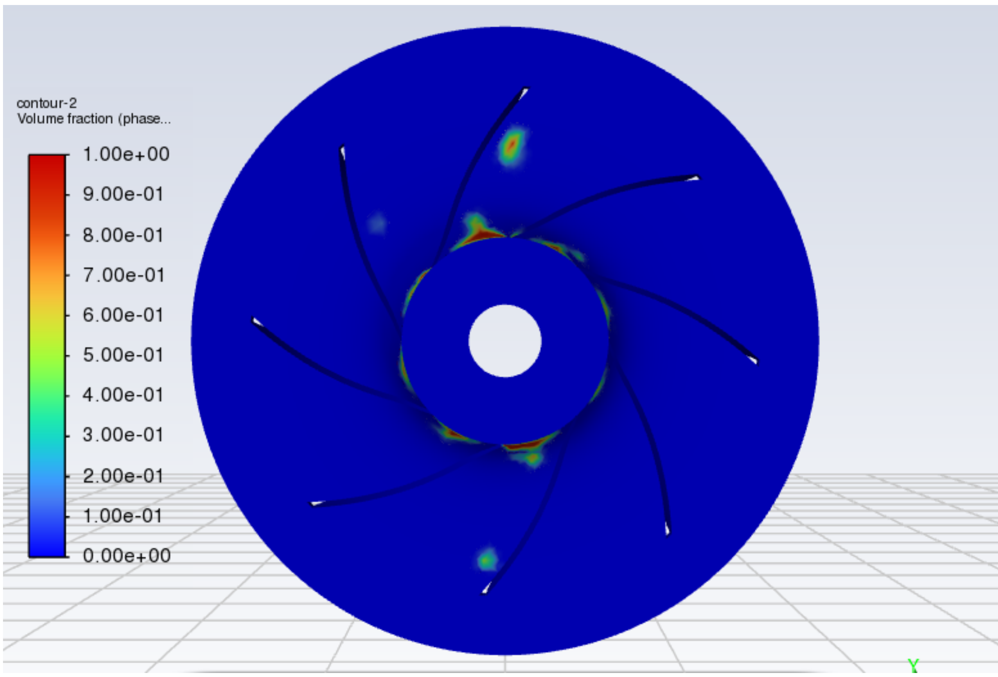


*Figure 4.4: Volume fraction contour at 800 rpm*

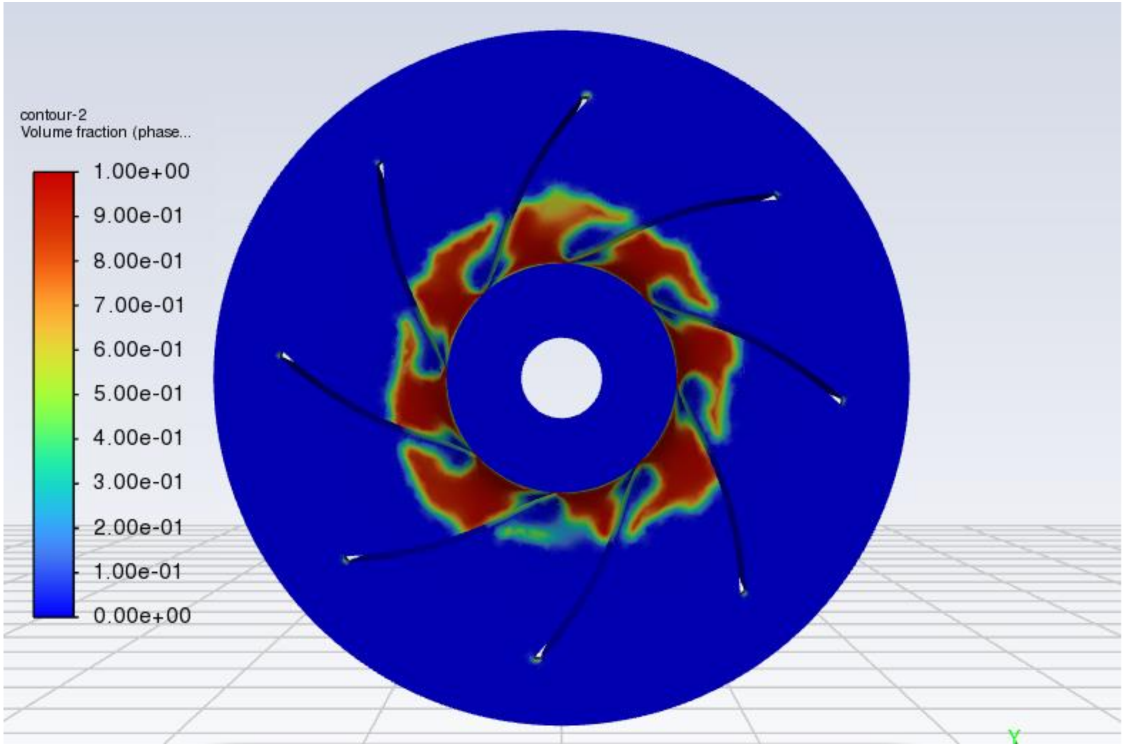


*Figure 4.5: Volume fraction contour at 2000 rpm*

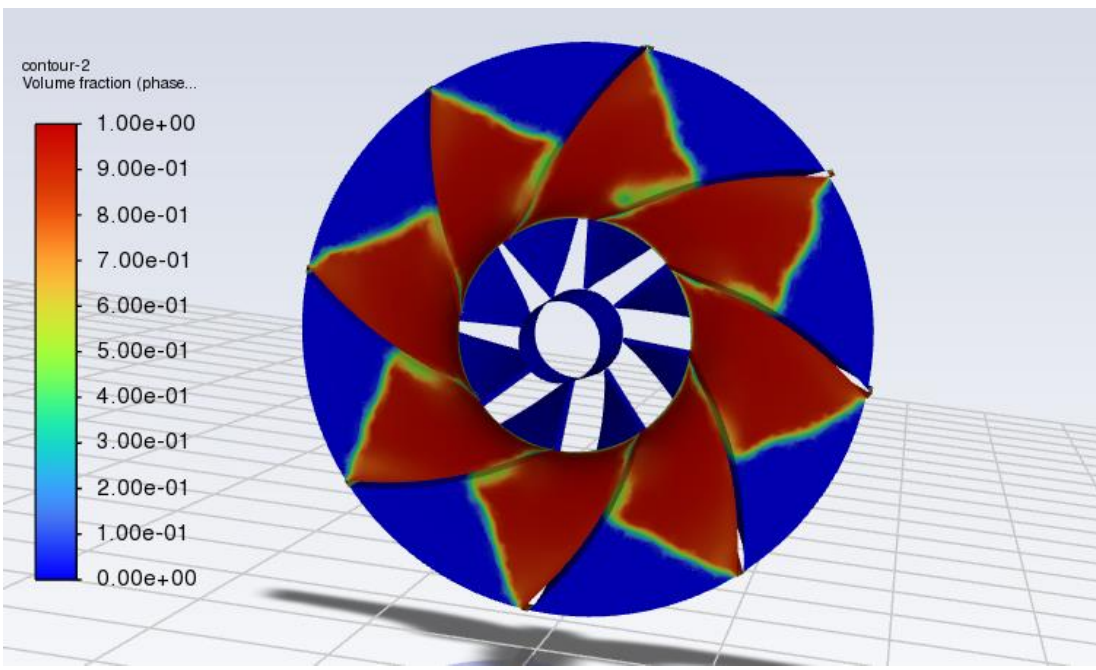


*Figure 4.6: Volume fraction contour at 3000 rpm*

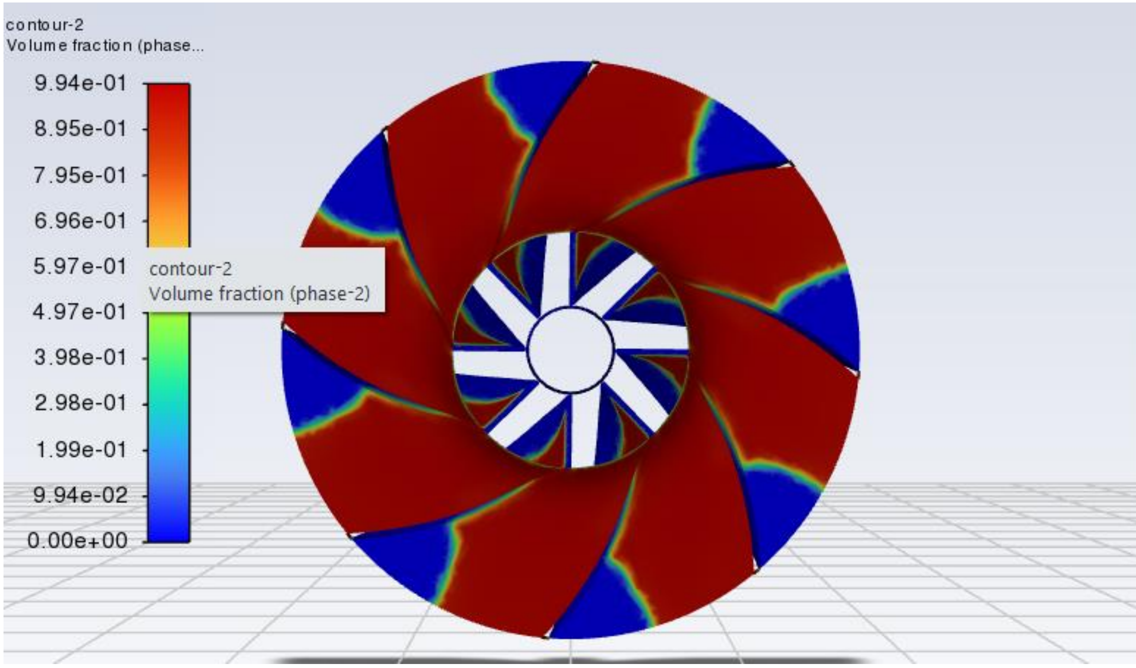


*Figure 4.7: Volume fraction contour at 6000 rpm*

The volume fraction contour shows the amount of water vapor bubble present in the fluid region of centrifugal pump. The hot colors shows the increasing intensity of vapor bubbles. As the rpm of the pump is increased the vapor formation also increases linearly. Up to 3000 rpm the outlet volume fraction of vapor is zero i.e. pure liquid flows out. But above 3000 rpm the outlet volume fraction increases meaning it consists of both phases of water. This implies that pressure at the exit is not strong enough for cavitation on impeller. Since cavitation is the measure of metallic erosion caused by the bursting of the vapor bubbles, the cavitation on impeller is maximum at around 3000 rpm. Above 3000 rpm the cavitation increases in the volute casing of the centrifugal pump.

The simulation was plotted in graph showing volume fraction (y-axis) vs. radial position of the impeller (x-axis) for different rpm as shown below:

Here in the figure below, the components of centrifugal pump are named as:

- solid_1_1 means impeller blade
- sup means the impeller casing

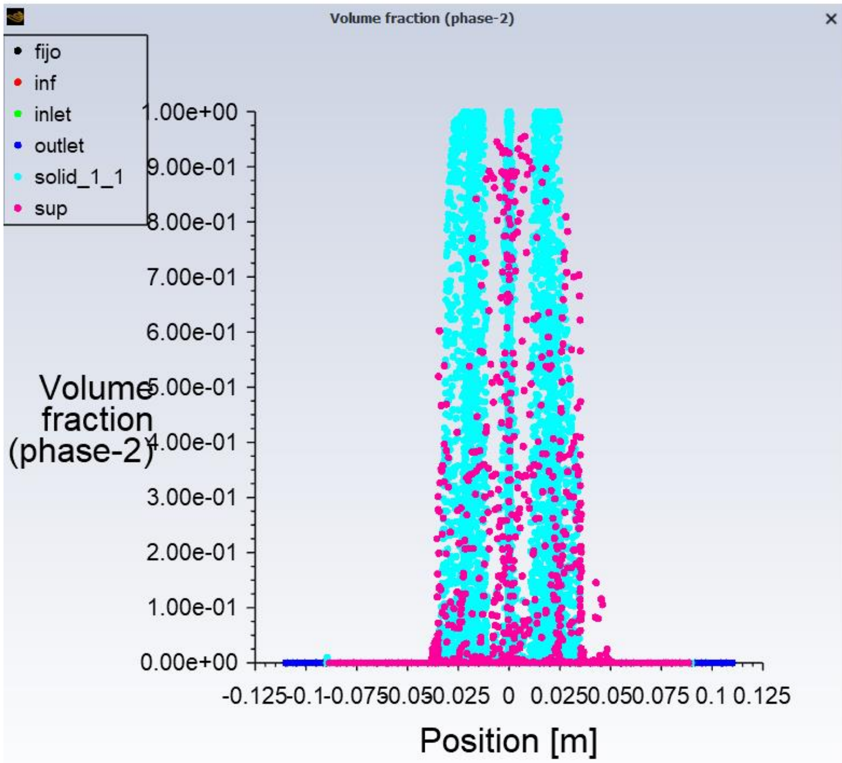


*Figure 4.8: XY plot of volume fraction vs. radial position at 800 rpm*

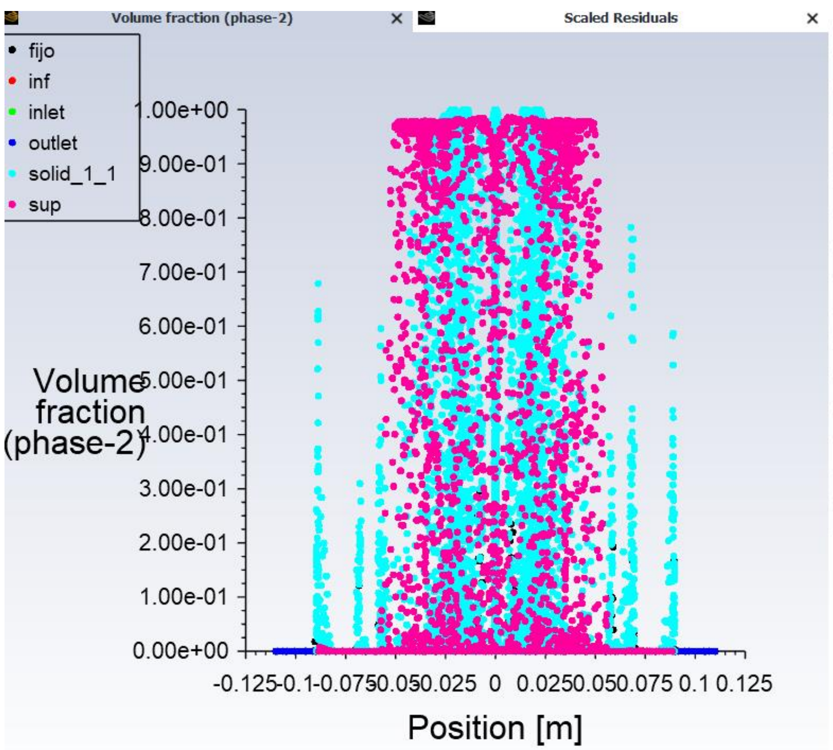


*Figure 4.9: XY plot of volume fraction vs. radial position at 2000 rpm*

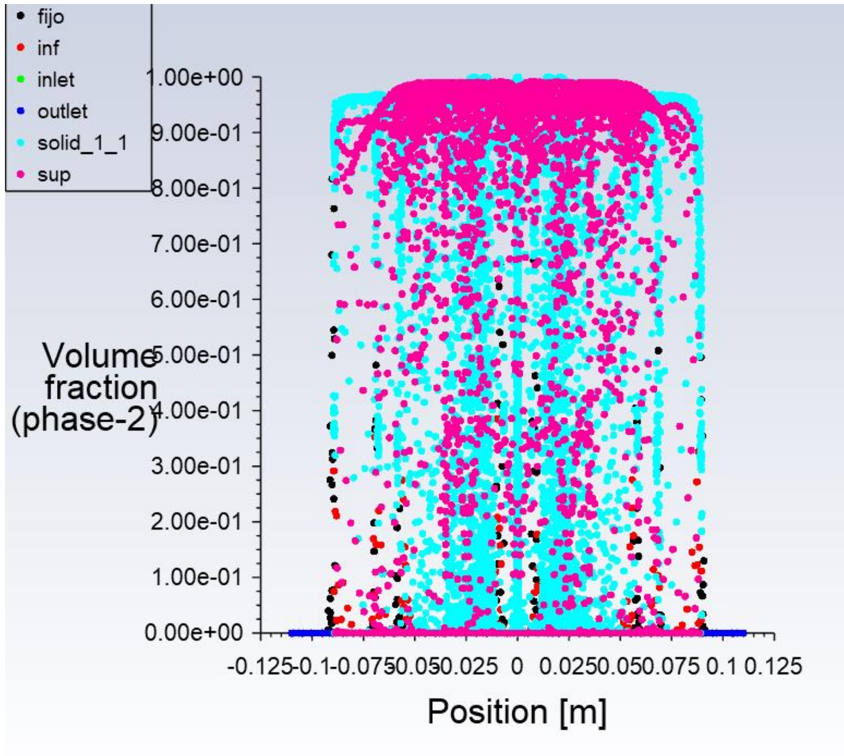


*Figure 4.10: XY plot of volume fraction vs. radial position at 3000 rpm*

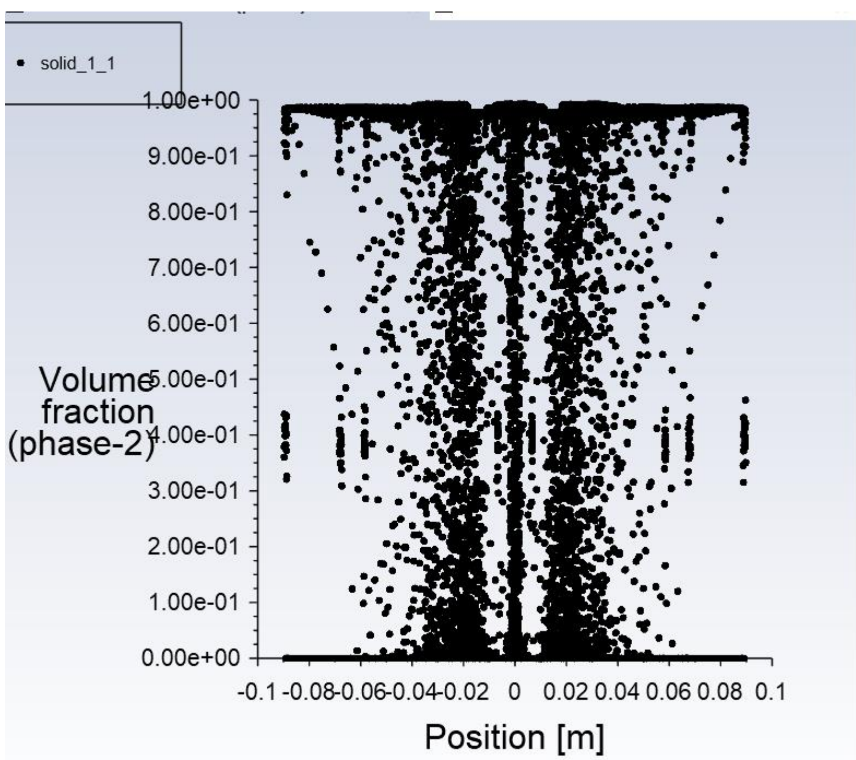


*Figure 4.11: XY plot of volume fraction vs. radial position at 6000 rpm*

Here the area of the covered by the dots represent the measure of cavitation. Due to the width of impeller, there is variation of vapor fraction on the same radial point along the axial direction. This gives a range of vapor fraction on same radial position as shown in the above graphs. With increase in the rpm there is increase in the width of the graph from the center.

### 4.2 Variation of volume fraction with mass flow rate

Now for a constant rpm of 3000, the volume fraction was observed for varying mass flow rate. With 120 iterations, simulation was performed and following results were obtained:

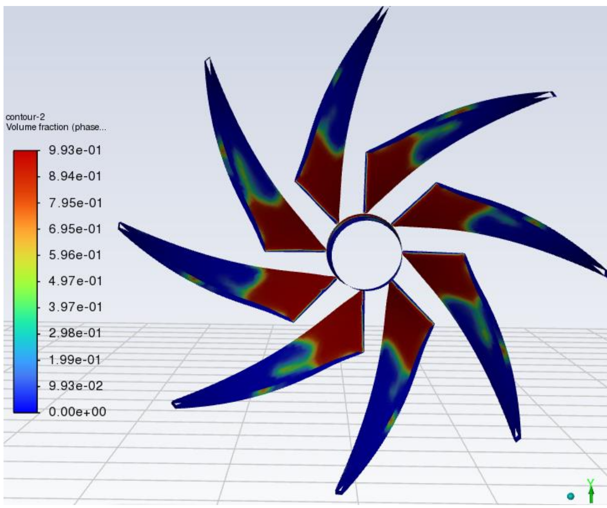


*Figure 4.12: Volume fraction contour at 8 kg/s*

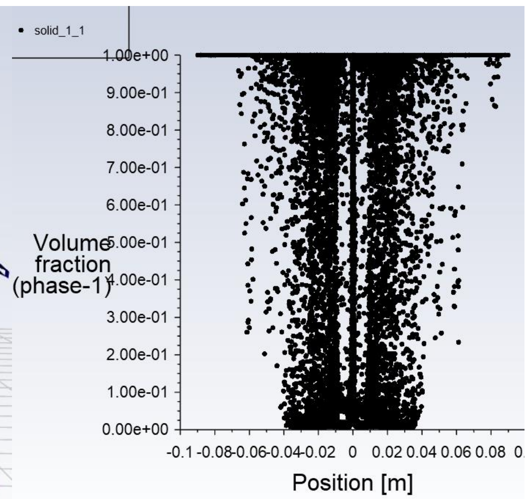


*Figure 4.13: Volume fraction XY plot at 8 kg/s*

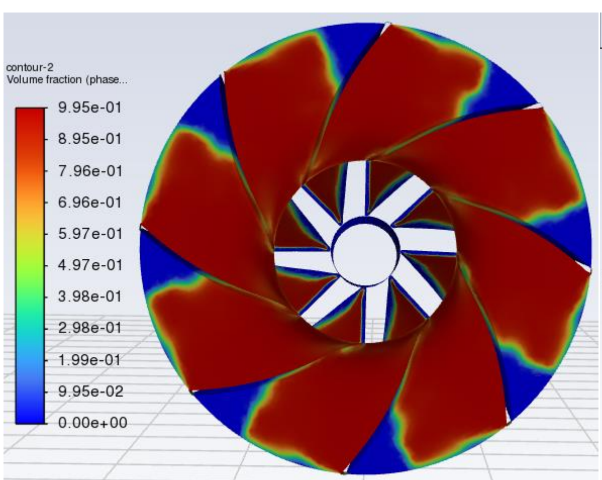


*Figure 4.14: Volume fraction contour at 45 kg/s*

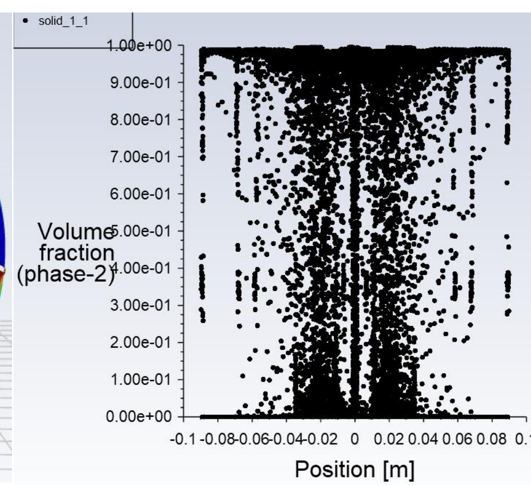


*Figure 4.15: Volume fraction XY plot at 45 kg/s*

This implies that cavitation increases with increase in mass flow rate.

## 4.3 Variation of volume fraction with suction pressure

When the suction pressure was dropped to zero keeping all other parameters constant, following results were obtained from simulation.

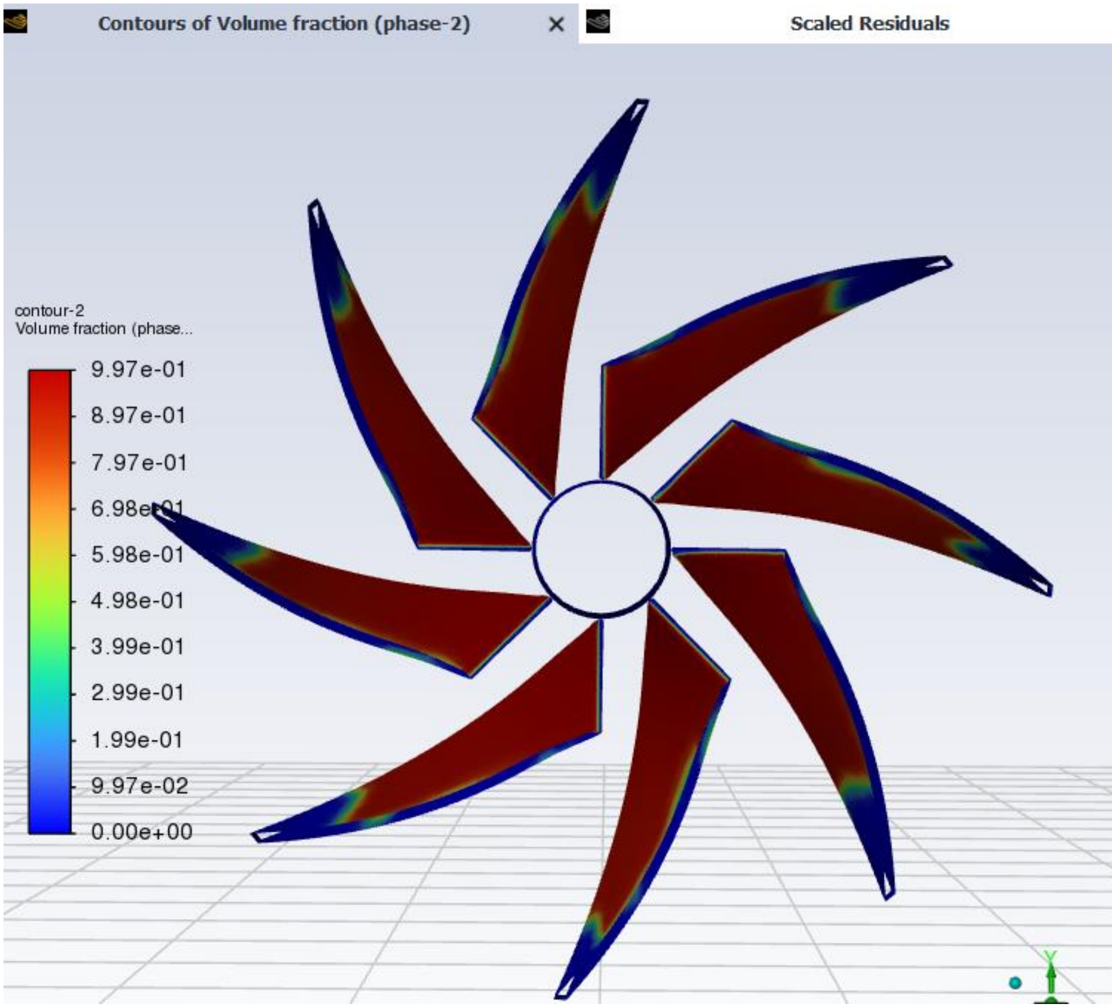


*Figure 4.16: Contour of volume fraction at suction pressure zero*

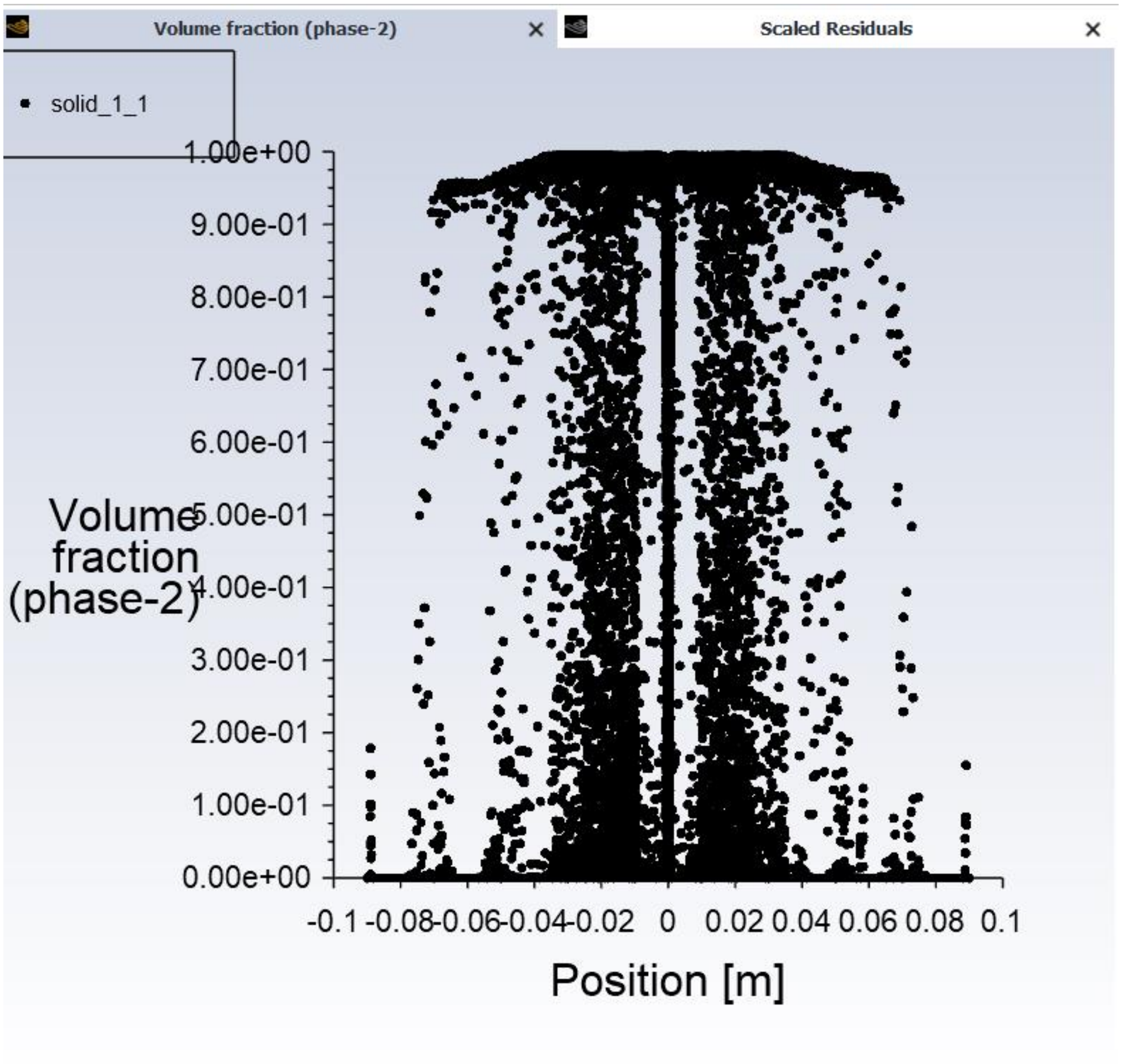


*Figure 4.17: XY plot of vapor fraction vs. radial position*

This implies that cavitation increases with decrease in suction pressure keeping the outlet pressure constant.

# CHAPTER 5 : CONCLUSION AND FUTURE ENHANCEMENT

## 5.1 Conclusion

In conclusion, our project focused on the comprehensive Computational Fluid Dynamics (CFD) analysis of cavitation within a centrifugal pump, examining its behavior under varying conditions of RPM, mass flow rate, and suction pressure. Through meticulous simulations, we revealed a consistent trend: cavitation increases with an increase in RPM, mass flow rate, and suction pressure.

The interplay of these parameters with cavitation was clearly evident in our results. As RPM, mass flow rate, and suction pressure increased, the likelihood and severity of cavitation also escalated. This observation underscores the delicate balance between operational settings and the potential for cavitation-related challenges in centrifugal pumps.

## 5.2 Scope of Future Enhancement

There are a few investigations that can be done for better understanding of cavitation in centrifugal pump. For now, below are the recommendation for future researchers:

1. Relation between geometry of blade and cavitation: Investigation can be done on effect of varying guide beta angle on cavitation.
2. Effect of roughness of blade on cavitation

# References

[1] A. Al-Obaidi, "Investigation of Cavitation in Centrifugal Pumps," *Journal of Fluid Dynamics,* pp. 112-126, 2019.

[2] M. S. P. A. R. B. CHAPMAN, "Collapse of an initially spherical vapour cavity in the," *Journal of Fluid Mecanics,* vol. 47, no. 13, pp. 283-290, 1970.

[3] M. Binama, "Cavitation Effects in Centrifugal Pump-A Review," *Int. Journal of Engineering Research and Applications,* pp. 52-63, 2016.

[4] M. Binama, "Impeller Geometry Effects on Cavitation Performance in Centrifugal Pumps," *Journal of Fluid Mechanics,* pp. 432-448, 2016.

[5] W. Spraker, "Total Cavitation Potential and its Relation to Pump Operation," *Transactions of Fluid Mechanics,* pp. 567-582, 1985.