



بسم الله الرحمن الرحيم

Sudan University of Science and Technology
College of Graduate Studies



A Study of Cavitation in Pumps and Flow Systems

دراسة التكهف في المضخات وانظمة التدفق

Thesis submitted in partial fulfillment of requirement for the degree
of M.Sc. in Mechanical Engineering (Power)

Prepared by:

Eng. Abdalla Gouda Mohammed Abdalla

Supervisor:

Dr. Mohyedin Ahmed Abdelghadir

May 2018

الآية

بسم الله الرحمن الرحيم

قال تعالى

(يرفع الله الذين آمنوا منكم والذين أوتوا العلم درجات)

صدق الله العظيم

Dedication

I want to dedicate this thesis to my parents who give me the greatest support in my life and study.

Acknowledgements

Firstly, I would like to express my gratitude to Allah for enabling the guidance to complete this work. I also express my gratitude to my parents, and family who made my life fruitful for their invaluable support during my work.

A further special thanks to Dr. Mohyedin Ahmed Abdelghadir for his constant guidance and support throughout this work. I would also like to thank all my committee members for their encouragement and support of my thesis work.

I wish, also, to thank all the staff of the mechanical engineering department of Sudan University of Science and Technology. Academic an administration, or technical staff, for their kind collaboration, help and kindness.

I would like to express my sincerest appreciation to my brothers and colleagues at Mechanical engineering.

ABSTRACT

Cavitation is the appearance of vapor cavities inside an initially homogeneous liquid medium, occurs in very different situations. According to the flow configuration and the physical properties of the liquid, cavitation is the most commonly required one in all the researches on the pumps and flow systems but they are too hard to manipulate with the primary methods. In this research was study the effects of rotational speed (rpm); flow rate (Q); fluid temperature (T) and pressure (P) on cavitation in pumps and flow system, to improve the pumps efficiency, and to choose a suitable method circuits and software programming. This research includes mechanism of cavitation formation, cavitation Reduction Techniques and ANSYS FLUENT Software. It was determined that a change in rotational speed, fluid flow rate and temperature are caused by changes in cavitation and find out that the cavitation in pumps increase with the rise of rotational speed and increase with increase in fluid flow rate and increase when the fluid flow rate is being so less and decrease with temperature rising and decrease with increase in exit pressure.

المستخلص

التكهف هو ظهور تجاويف بخار داخل وسط السائل متجانسة في البداية ويحدث التكهف في حالات مختلفة جدا. يعتبر التكهف هو الاكثر شيوعا في البحوث المتعلقة بالمضخات وانظمة التدفق ولكن من الصعب جد التعامل معها بالطرق الاساسية. في هذا البحث تمت دراسة تأثير سرعة الدوران , معدل التدفق , درجة الحرارة والضغط , تحسين كفاءة المضخات تم اختيار الدورة المناسبة و اختيار برنامج حاسوبي لمحاكاة النموذج. هذا البحث يحتوي على آلية تشكيل التكهف ، تقنيات الحد من حدوث التكهف. برنامج المحاكاة(انيسيس) وجد ان التغير في سرعة الدوران ومعدل تدفق المائع والتغير في درجة حرارة المائع ينتج عنها تغيير في حودث التكهف. وجد ان التكهف يزيد بزيادة سرعة الدوران وزيادة معدل تدفق المائع وايضا يزيد عندما يكون معدل تدفق المائع اقل من الحد المطلوب بكثير وايضا يقل التكهف بزيادة درجة حرارة المائع.

Table of contents

Title	Page
Dedication	II
Acknowledgments	III
Abstract	IV
المستخلص	V
Table of Contents	VI
List of Figures	VIII
List of Tables	X
List of Appreciation	XI
CHAPTER I	
2-1 Introduction	2
1-1-1 Main Parts of a Pump	2
1-1-2 Application	2
1-2 Cavitation	2
1-2-1 Why Cavitation is Important	4
1-2-2 Hydrodynamic Effects	5
1-2-3 Cavitation Damage	5
1-2-4 Extraneous Effect	5
1-3 Problem Statement	7
1-4 Research Objective	7
1-5 Methodology	7
CHAPTER II	
2-1 Literature Review	9
2-2 Forms of Cavitation	9
2-3 Cavitation Occurrence	9
2-4 Cavitation Characteristic	10
2-5 Cavitation Mechanism	10

2-6	Effect of Rotational Speed on Cavitation	10
2-7	Effect of Blade Angle on Cavitation Phenomenon in Axial Pump	11
2-8	Effect of Fluid Temperature on Cavitation	14
2-9	Computational Fluid Dynamic	16
2-10	CFD – An Overview	16
2-11	A typical CFD simulation consists of several stages, described	17
2-12	Background and History	17
CHAPTER III		
3-1	Introduction	23
3-2	Calculation Models used in the ANSYS FLUENT	23
CHAPTER IV		
4-1	Effect of Rotational Speed (RPM) on Cavitation	35
4-2	Effect of Fluid Flow Rate (Q) on Cavitation	37
4-3	Effect of Temperature (T) on Cavitation	40
4-4	Effect of Pressure (P) on Cavitation	42
CHAPTER V		
5-1	Conclusion	46
5-2	Recommendations	47
5-3	References	48

List of Figures

Figures	Page
Figure 3-1 step 1 in program	23
Figure 3-2 step 2 in program	24
Figure 3-3 step 3 in program	25
Figure 3-4 step 4 in program	26
Figure 3-5 step 5 in program	26
Figure 3-6 step 6 in program	27
Figure 3-7 step 7 in program	28
Figure 3-8 step 8 in program	29
Figure 3-9 step 9 in program	30
Figure 3-10 step 10 in program	32
Figure 3-11 step 11 in program	32
Figure 4-1 volume fraction at 2600rpm	35
Figure 4-2 volume fraction at 2500rpm	36
Figure 4-3 volume fraction at 2160rpm	36
Figure 4-4 volume fraction at 1750rpm	36
Figure 4-5 volume fraction at 1450rpm	37
Figure 4-6 relation between rotational speed and volume fraction	37
Figure 4-7 volume fraction at $10.5\text{m}^3/\text{s}$	38
Figure 4-8 volume fraction at $5.046\text{m}^3/\text{s}$	38
Figure 4-9 volume fraction at $5.05\text{m}^3/\text{s}$	39
Figure 4-10 volume fraction at $4.05\text{m}^3/\text{s}$	39
Figure 4-11 relation between the fluid flow rate and volume fraction	39
Figure 4-12 volume fraction at 250K	40
Figure 4-13 volume fraction at 280K	40
Figure 4-14 volume fraction at 350K	41
Figure 4-15 volume fraction at 400K	41

Figure 4-16	relation between fluid temperature and volume fraction	41
Figure 4-17	volume fraction at 2.5MPa	42
Figure 4-18	volume fraction at 3MPa	42
Figure 4-19	volume fraction at 3.5MPa	43
Figure 4-20	volume fraction at 4.5MPa	43
Figure 4-21	volume fraction at 5MPa	43
Figure 4-22	relation between the pressure and volume fraction	44

List of tables

Table	Page
Table 3-1 Cell Zone Conditions	27
Table 3-2 Boundary Conditions Inlet	28
Table 3-3 Boundary Conditions Outlet	29
Table 3-4 Solver Discretization	32
Table 3-5 Solution Controls	33

List of Appreciation

BEP	Best Efficiency Point
NPSH	Net Positive Suction Head
NPSHa	Net Positive Suction Head Actual
NPSHr	Net Positive Suction Head Required
RPM	Revaluation Per Unit
CFD	Computational Fluid Dynamics

CAPTER I
INTRODUCTION

CHAPTER I

INTRODUCTION

1-1 Background:

A pump is a device used to move liquids by mechanical action. Pumps can be classified into three major groups according to the method they use to move the fluid: direct lift, displacement, and gravity pumps. A wide variety of pump types have been constructed and used in many different applications in industry. Pumps must have a mechanism which operates them, and consume energy to perform mechanical work by moving the fluid. The activating mechanism is often reciprocating or rotary. Pumps may be operated in many ways, including manual operation, electricity an engine of some type, or wind action.

1-1-1 Main Parts of a Pump:

- (a) Impeller blade
- (b) Electric motor
- (c) Stuffing box
- (d) Coupling
- (e) Head shaft

1-1-2 Application:

Pumps are found in many applications. Single stage pumps are used for drainage, sewage pumping, general industrial pumping and slurry pumping. They are also popular with aquarium filters. Multiple stage submersible pumps are typically lowered down a borehole and used for water abstraction, water wells and in oil wells.

1-2 Cavitation:

Cavitation is the formation of vapor bubbles in a moving fluid where the pressure of the fluid falls beneath its vapor pressure. In essence,

cavitation results from a reduction in suction pressure, an increase in suction temperature, or an increase in the flow rate above that for which the pump has been designed [1]. Designers of pumps attempt to take into consideration the fact that pumps do not always run at peak efficiency, and try to take into account the operating range of the system. A centrifugal pump is usually operated comfortably within the range of 85% to 110% of its best efficiency point (BEP). However, many pumps are forced to operate outside of this range [2]. As a result, designers go to great lengths to ensure that cavitation bubbles do not collapse in the pump, but rather in the main piping system, far away from the impeller vanes.

There are several causes for cavitation in a pump and piping system, such as: high volumetric flow; a large decrease in the amount of fluid in the system which results in an abnormal increase in the temperature of the fluid; decrease in suction pressure due to changed conditions on the suction side of the pump; heating the fluid in the suction system, which leads to a higher fluid vapor pressure at the pump inlet; flow instability within the pump, which normally occurs at flow rates well below the pump's best efficiency point (BEP) flow rate; flow close to zero which results in rapid fluid heating in the pump casing, and quickly results in vapor-locking in the system; poor distribution of the fluid in pumps operating in parallel; oversized pumps operating at high capacity; pumping warm water with high vapor pressure; it is also hypothesized that cavities formed between the fluid and the vibrating parts of the pump that are in contact with the fluid; and a high percentage of leakage flows may lead to an increase in the temperature at the eye of the impeller, which would then possibly cause localized flashing [4-9].

High flow cavitation occurs more frequently when the Net Positive Suction Head (NPSH) margin (actual NPSH – required NPSH) is small,

possibly due to design constraints, and there is a reduction in the normal pump suction pressure as a result of an increase in the amount of suction piping pressure losses, or when the pump is operated well above its normal or rated flow. The Actual Net Positive Suction Head (NPSHA) is calculated using characteristics found at the pump's suction nozzle and is thus independent of the pump or the pump's characteristics. The Required Net Positive Suction Head (NPSHR) is the amount of net positive suction head that is required to avoid cavitation. It is independent of the system characteristics, and remains the same regardless of which system the pump is installed in. Excessive cavitation may lead to "vapor locking", a phenomenon where the fluid within the pump becomes mostly vapor, due to extreme vaporization of the fluid, or the pump being run for a long period of time at zero or near zero flow rate. Recovery from this phenomenon requires stopping the pump and allowing liquid to re-enter the pump [7].

1-2-1 Why cavitation is important

Cavitation is important as a consequence of its effects. These may be classified into three general categories. Effects that modified the hydrodynamics of the flow of the liquid, effects that produce damage on the solid-boundary surfaces of flow and extraneous effects that may not be accompanied by significant hydrodynamic flow modifications or damage to solid boundaries. Unfortunately for the field of applied hydrodynamics the effects of cavitation with very few exemptions are undesirable cavitation, [10] can produce serious and even catastrophic results. The necessity of avoiding or controlling cavitation imposes serious the design of many types of hydraulic equipment. In the field of hydraulic machinery, it has been found that all types of turbines, from a low specific speed (Francis) to the high specific speed (Kaplan) are

susceptible to cavitation. Centrifugal and axial pumps suffer from its effects and even the various types of positive-displacement pumps may be troubled by it. Although cavitation may be aggravated by poor design, it may occur in even the best-design equipment when the latter is operated under unfavorable conditions.

1-2-2 Hydrodynamic Effects

The various hydrodynamic effects of cavitation have their source in the interruption of the continuity of the liquid phase as cavities appear. As cavity volume displaces liquid, the flow pattern is modified and the dynamic interaction between the liquid and its boundaries is affected. The presence of cavitation increases the overall resistance to the flow. Usually the effect of cavitation on the guidance of the liquid by the boundary surface is to limit or lessen the force that can be applied to the liquid by the surface. Thus the two hydrodynamic results, of overall resistance to flow and reduced turning effect, combine to lower the performance of the equipment involved. In hydraulic machinery there is a drop in both the head and efficiency. Decrease in power and head are indications of cavitation causing a decrease in guidance and hence effective momentum transfer between liquid and impeller. The decrease in efficiency is a measure of the increased losses.

1-2-3 Cavitation Damage.

Cavitation damage is the most spectacular and the most widely recognized effect of cavitation. Cavitation damage is so closely related to the cavitation phenomenon that such damage is often simply referred to as cavitation damages solid flow boundaries by removing material from the surface. It has been found that cavitation can damage all types of metallic solids. Thus all metals, hard or soft, brittle or ductile, chemically active or chemically inert, can be damaged by cavitation.

Rubber, plastic, glass, quartz, concrete and other nonmetallic solids are likewise susceptible to cavitation damage. There are four basic principles, for cavitation damage [10,11]. The smaller the molecular size of the liquid is and the lower the viscosity, the easier it is for the liquid to penetrate into the surface pores of the metal, i.e. the penetration of water into the surface of the metal is deeper than in the case of oils. The greater the pressure, the deeper and quicker the liquid penetrates into the pores of the material. The smaller the area of the pore the greater the pressure produced when the vapor bubbles collapse. The higher the frequency of vibration, the more intensive is the destruction of the surface layer of the metal. The phenomenon of cavitation should be distinguished from the corrosion and erosion of metals.

(a) Corrosion of metals is caused solely by chemical and electro-chemical processes, the similarity of this process to cavitation occurs in corrosion by a liquid which is subjected to high frequency pressure oscillations.

(b) Erosion of metals consists in the abrasion of the surface of the metal walls by solids carried- by liquid or the washing away of the walls by the flowing particles of clean liquid moving at a high velocity e.g. in high.

1-2-4 Extraneous Effect

Two of the most common effects of cavitation that may not involve major modification in the liquid flow or damage to the solid surfaces are cavitation noise and vibration induced by cavitation. It has been found experimentally, [12,13], that considerable noise is produced by the collapse of cavities. It is possible that noise is evolved during the entire process but, if so, the intensity is much lower than that produced during the collapse so that little effort has been made to isolate and identify it. The importance of cavitation noise depends largely on the individual installation. For example, in a power-house of a factory in which the noise level from other sources is already high, the addition of cavitation

noise may be hardly recognized and may offer no problem. On the other hand, the universal opinion in naval circles, [14,15], is that the most serious wartime effect of cavitation on surface or subsurface craft is the production of noise. Its presence makes it impossible to preserve secrecy of movement and of fears considerable assistance to the enemy in determining the exact location of the vessel on which it originates. The cavitation process is inherently an unsteady one and may involve large fluctuating forces. If one of the frequency components of these fluctuations matches a natural frequency of a portion of the equipment, vibration may result. The vibration is usually of fairly high frequency ranging from several thousand cycles per second.

1-4 Problem Statement:

Cavitation is the rapid process of formation and collapse of vapor bubbles in a liquid. can lead to results reduced performance. today most researcher in the world focus on how to use technology to reduce cavitation and to solve this problem. and to search to the technique to use it, one of the methods used software simulation. Computational fluid dynamics (CFD).

1-3 Objectives of Research:

1- To reduce the cavitation in pumps under variable (pressure (P), temperature (T), rotational speed (RPM), fluid flow rate (Q)).

Methodology:

- **Study for literature review**

In general, there are some parameter in take to reduce cavitation in pumps and flow systems like rotational speed, number of blade, flow rate, fluid temperature, blade angel and pressure.

- Study the effect of rotational speed, flow rate, fluid temperature and exit pressure to reduce cavitation in pumps and flow systems using ANSYS Software.

CHAPTER II
LITERATURE REVIEW

CHAPTER II

LITERATURE REVIEW

2-1 Introduction:

A review of recent of cavitation in pump and flow system activities is can be divided in to aspects experimental studies and numerical studies both of them focus on performance of system.

A method that predicts the cavitation performance for pumps and inducers is presented.

2-2 Forms of Cavitation:

Braun [16] indicates that the recognized forms of liquid cavitation are:

- (a) Gaseous cavitation generally contains one or multiple species of dissolved in the fluid and occurs as the pressure falls below the saturation pressure of the particular gas component.
- (b) Pseudo cavitation is a form of gaseous cavitation during which the gas bubble expands on account of depressurization without further gas mass diffusion from the liquid to the gas phase.
- (c) Vaporous cavitation is the result of a thermodynamic non equilibrium event when the pressure falls below the vapor pressure of the liquid at the prevalent temperature.

2-3 Cavitation Occurrence:

Chang [17] stated that cavitation occurs when pumps are:

- (a) Operating at flow rates either much greater or much less than their BEP, in areas of low pressures (pump inlet at high flows) or high local velocities (caused by recirculation).
- (b) Operating the system at either higher than design temperatures or lower than design suction pressure can be a cause of cavitation.
- (c) Not selecting the right pump.
- (d) The pump suction is starved because of the formation of air pockets or fouling of pipes.

2-4 Cavitation Characteristic:

- (a) **High pressure:** The pressure peak can reach several hundred *MPa* or even climb to several *GPa*. This pressure is higher than the elastic limit of most engineering materials.
- (b) **Small dimension:** Dimensions of the micro-jet are very small (from a few micrometers up to several hundred micrometers). Thus each impact on the solid surface concerns only a very small area
- (c) **Short time:** The duration of the impact is about several microseconds.
- (d) **High temperature:** Because of the localized dissipation of energy during collapse, local temperature can be very high (several thousand °C) [20].

2-5 Cavitation Mechanism:

According to Whiteside's the failure mechanism depends on the ratio of the intensity to the material resistance. However, the mechanism for failure changes over time because of damage is dependent on liquid viscosity: An increase in liquid viscosity causes a reduction in the number and size of cavitation bubbles [19,20].

2-6 Effect of Rotational Speed (rpm) on Cavitation:

For a pump rotates at any running speed, the liquid entering the impeller eye turns and is split into streams by the leading edges of the impeller blades, this splitting locally drops the fluid pressure below that in the suction pipe. This situation becomes more dangerous when the incoming liquid is at a pressure with small margin above the vapor pressure corresponding to the liquid temperature. Cavities or bubbles of vapor create along the impeller blades just behind the inlet edges, and this is well known as cavitation in pumps. Cavitation has undesirable effects; erode the blade surface, shorten seal and bearing life as a result of

increased noise and vibration, choke the flow passages in the impeller, worsen the pump performance. The pump manufacturer provides the pressure in the pump suction exceeds the vapor pressure to prevent cavitation. This value is known as the required net positive suction head NPSHR which is a characteristic of the pump design and manufacturing. NPSHR depends on the pump rotational speed as follows; $2 \text{ NPSHR} \propto N$. It should be noted however that at very low speeds there is a minimum NPSHR plateau, NPSHR does not tend to zero at zero speed. It is therefore essential to carefully consider NPSHR in variable speed pumps. On the other hand, the pressure exceeds the vapor pressure in the pumping system is expressed as the available net positive suction head NPSHA and it is a characteristic of the system. Manufacturers and industry BPMA, define the onset of cavitation as the value of NPSHR when there is a head drop of 3% compared with the head for cavitation free performance. Al-Arabi et al. [25] visually observed cavitation develops at the state where the pump head drops by 3%, but the onset of cavitation appears long before the pump performance is affected. At 3% drop of total head severe cavitation is highly likely and design on the basis of that could cause an erroneous prediction estimate of pump performance. The present work considers six different values of rotational speed of 1476 rpm, 1644 rpm, 1932 rpm, 2190 rpm, 2466 rpm, and 2682 rpm, the rotational speed is precisely controlled with the electronic inverter. The pump discharge is modulated and kept in the range of 780 L/h \pm 5 L/h for all the values of rotational speeds, the adjustment of the pump discharge by a pre-calibrated electromagnetic flow meter is achieved by trials and errors. Since the available net positive suction head NPSHA is based on the vapor pressure at the corresponding fluid temperature, suction pressure which is modulated by the control valve upstream the pump inlet and measured by pre-calibrated dial gauges and

set to be 650 mmHg “vacuum”, and the velocity of flow through the suction line. Then the value of the available net positive suction head is calculated and it is found to be 0.754 m for all values of the rotational speed. To assign the cavitation status for each rotational speed, the affinity laws are used to predict the required net positive suction head at the rotational speeds, while the manufacturer’s data at 2850 rpm is considered as the baseline data.

The conditions at rotational speeds of 1476 rpm, 1644 rpm, and 1932 rpm are normal pump operation with no cavitation, while for rotational speeds of 2190 rpm, 2466 rpm, and 2682 rpm the pump is subjected to cavitation. Therefore, the cavitation strength (%) is calculated for each cavitation condition. The cavitation strength values are 10.24%, 58.10%, and 40.63% for the rotational speeds of 2190 rpm, 2466 rpm, and 2682 rpm respectively. Therefore, the studied conditions is divided into two groups, group 1 represents normal operation (no cavitation) of the pump at 1476 rpm, 1644 rpm, and 1932 rpm. While group 2 represents pump operates at cavitation at 2190 rpm, 2466 rpm, and 2682 rpm (cavitation).

2-7 Effect of Blade Angle on Cavitation Phenomenon in Axial Pump:

The contours of computed vapor volume fraction on the rotor at blade angles 80°, 70°, 60°. It is at the leading edge of the blades at blade angle 60°. The static pressure distribution on the hydrofoil at the tip section at blade angles 80°, 70° and 60°. The static pressure decreases sharply below the vapor pressure at the leading edge of the blades at blade angle 60°, while the static pressure at blade angles 70° and 80° is above the vapor pressure. The vapor volume distribution on the hydrofoil at tip section at blade angles 80°, 70° and 60°. The vapor volume fraction increases sharply at the leading edge of the blades at blade angle 60°, while at blade angles 70° and 80° is equal to zero. The total pressure distribution on the hydrofoil at tip section at blade angles 80°, 70° and 60°.

The total pressure decreases sharply below the vapor pressure at the leading edge of the blades at blade angle 60° , while at blade angles 70° and 80° is above the vapor pressure. Obviously that cavitation does not appear at blade angles 80° and 70° while it is present at blade angle 60° at the leading edge of the suction side due to the decrease in static pressure below the vapor pressure. The computed density contours on the rotor at blade angles 80° , 70° , 60° are presented. It is obvious that there is no density at blade angles 80° and 70° , while the density decreases sharply at blade angles 60° at the leading edge of the blades due to the decrease in static pressure below the vapor pressure.

Suction and Discharge Recirculation usually occurs during reduced flows, and the flow of some fluid around the impeller to the suction side. If this is found in the inlet of the impeller, then it is known as suction recirculation. If this is found at the outlet of the impeller, then it is known as discharge recirculation [22]. Recirculation is inevitable in every impeller design. The discharge recirculation can be reduced in design, but this would result in a reduction in the rated efficiency of the pump. The suction recirculation can likewise be reduced, which would result in an increase in NPSHR. In order to avoid recirculation, the recommendation is not to exceed certain suction specific speeds (a dimensionless ratio describing operating conditions in a pump). Although useful, this advice cannot be applied blindly to all cases. During recirculation, heat is added to the fluid being pumped due to the pump losses. As a result, if the pump operates in this mode for an extensive period of time, temperatures may increase leading to vaporization and potentially an explosive and dangerous condition may exist [21]. During suction recirculation, a loud crackling noise is produced around the suction of the pump, for discharge recirculation, at the discharge volute or diffuser. Noise produced by recirculation has a greater intensity than that produced by cavitation, and

is normally characterized by a random, knocking sound. Suction or discharge recirculation can be determined by monitoring the pressure pulsations found in the suction and discharge of the pump. Piezoelectric transducers are normally placed close to the impeller on either the suction or discharge side of the pump. Data obtained may be analyzed using a spectrum analyzer to generate a plot of the pressure pulsations versus the frequency of selected flows. On this plot, a sudden increase in the magnitude of the pressure pulsations would represent the beginnings of recirculation. Pitot tubes installed at the eye of the impeller can also help determine the onset of suction recirculation. With the pitot tube directed into the impeller eye, suction recirculation will occur when the flow reversal from the eye impinges on the pitot tube with a rapid rise in the gauge reading [22].

2-8 Effect of Fluid Temperature on Cavitation:

Cavitation is a major concern in many pumping applications because it can result in reduced pump performance and/or impeller blade damage. Cavitation performance dependent upon the fluid and its operating temperature. This dependency can result in a performance change and a lower required net positive suction head to the pump and is referred to as a thermodynamic effect of cavitation. The magnitude of the decrease is dependent upon the physical and thermodynamic properties of the fluid, as well as the vapor distribution, pump design, rotative speed, and flow coefficient. In reference [23] the required net positive suction head for a cavitating inducer was considerably less in 37° R (20.5 K) hydrogen than that for a similar inducer in room temperature water. presents an empirical method of determining the cavitation performance of small low suction specific speed centrifugal pumps. The method is based on

cavitation tests using deaerated water to 300° F (421 K), several hydrocarbons, Freon, and other fluids [2].

During the cavitation process, vapor forms in a flowing liquid when the local static pressure falls below the vapor pressure of the local fluid. Heat required to form the vapor cavity must come from the surrounding liquid. During the vaporization process, the liquid in a thin layer adjacent to the cavity is cooled. The extent of this cooling depends on the properties of the liquid and the flow conditions. result in lower required net positive suction heads at increased liquid temperatures. In the Venturi studies of similar cavities the change in the depression of the cavity pressure below fluid inlet vapor pressure was equal to the change in the inlet pressure requirements. Thus the decrease in required inlet pressure is a measure of the thermodynamic effect of cavitation. The thermodynamic effect of cavitation for a pump in any fluid is defined herein as the difference between the required net positive suction head for a given liquid and temperature and that obtained in room temperature water under similar flow conditions. This infers constant rotative speed, constant flow coefficient, and the same head-rise coefficient ratio. It is assumed that the same vapor volume exists at the same head-rise coefficient ratio. This provides the same vapor blockage in the flow passage and maintains a constant ratio of fluid to blade velocity. This assumption can be affected by the type of cavitation in a pump.

Pump cavitation performance is independent of fluid temperature to 175° F (353 K). A reduction in the required net positive suction head of 2.0 to 6.0 feet (0.6 to 1.8 m) was measured in 250° F (394 K) water under that in room temperature water at similar operating conditions. This improvement in cavitation performance is attributed to the thermodynamic effects of cavitation, and is increased with increasing flow coefficient and decreasing head-rise coefficient ratio. At a head

coefficient ratio of 0.90 and at the design flow coefficient of 0.147, the measured reduction in required net positive suction head is 2.9 feet (0.9 m) between 80' and 250' F (300 and 394 K) [24].

2-9 Computational Fluid Dynamic:

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full scale testing, e.g. flight tests.

2-10 CFD – An Overview:

Computational Fluid Dynamics (CFD) is a computer based mathematical modelling tool that can be considered the amalgamation of theory and experimentation in the field of fluid flow and heat transfer. It is now widely used and is acceptable as a valid engineering tool in industry. CFD calculations are based upon the fundamental governing equations of fluid dynamics: the conservation of mass, momentum and energy. These equations combine to form the Navier-Stokes equations, which are a set of partial differential equations that cannot be solved analytically except in a limited number of cases. However, an approximate solution can be obtained using a discretization method that approximates the partial differential equations by a set of algebraic equations. There are a variety of techniques that may be used to perform

this discretization; the most often used are the finite volume method, the finite element method and the finite difference method. The resulting algebraic equations relate to small sub-volumes within the flow, at a finite number of discrete locations.

2-11 CFD steps) A typical CFD simulation consists of several stages described below:

- (a) Approximation of the geometry.
- (b) Creation of the numerical grid within the geometrical model.
- (c) Selection of models and modeling parameters
- (d) Calculation of the variable values.
- (e) Determination of a sufficiently converged solution.
- (f) Post Processing.
- (g) Solution Verification and Validation

2-12 History of The Program:

The fundamental basis of almost all CFD problems is the Navier–Stokes equations, which define many single-phase (gas or liquid, but not both)

fluid flows. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations

Historically, methods were first developed to solve the linearized potential equations. Two-dimensional (2D) methods, using conformal

transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s.[26]

One of the earliest type of calculations resembling modern CFD are those by Lewis Fry Richardson, in the sense that these calculations used finite differences and divided the physical space in cells. Although they failed dramatically, these calculations, together with Richardson's book "Weather prediction by numerical process",[27] set the basis for modern CFD and numerical meteorology. In fact, early CFD calculations during the 1940s using ENIAC used methods close to those in Richardson's 1922 book. The computer power available paced development of three-dimensional methods. Probably the first work using computers to model fluid flow, as governed by the Navier-Stokes equations, was performed at Los Alamos National Lab, in the T3 group. This group was led by Francis H. Harlow, who is widely considered as one of the pioneers of CFD. From 1957 to late 1960s, this group developed a variety of numerical methods to simulate transient two-dimensional fluid flows, such as Particle-in-cell method (Harlow, 1957),[28] Fluid-in-cell method (Gentry, Martin and Daly 1966),[29] Vorticity stream function method (Jake Fromm, 1963), and Marker-and-cell method (Harlow and Welch, 1965). Fromm's vorticity-stream-function method for 2D, transient, incompressible flow was the first treatment of strongly contorting incompressible flows in the world. The first paper with three-dimensional model was published by John Hess and A.M.O. Smith of Douglas Aircraft in 1967. This method discretized the surface of the geometry with panels, giving rise to this class of programs being called Panel Methods. Their method itself was simplified, in that it did not include lifting flows and hence was mainly applied to ship hulls and aircraft fuselages. The first lifting Panel Code (A230) was described in a paper written by Paul

Rubbert and Gary Saaris of Boeing Aircraft in 1968.[30] In time, more advanced three-dimensional Panel Codes were developed at Boeing (PANAIR, A502),[27] Lockheed (Quadpan),[31] Douglas (HESS), McDonnell Aircraft (MACAERO),[15] NASA (PMARC)[32] and Analytical Methods (WBAERO,[17] USAERO[33] and VSAERO[34][35]). Some (PANAIR, HESS and MACAERO) were higher order codes, using higher order distributions of surface singularities, while others (Quadpan, PMARC, USAERO and VSAERO) used single singularities on each surface panel. The advantage of the lower order codes was that they ran much faster on the computers of the time. Today, VSAERO has grown to be a multi-order code and is the most widely used program of this class. It has been used in the development of many submarines, surface ships, automobiles, helicopters, aircraft, and more recently wind turbines. Its sister code, USAERO is an unsteady panel method that has also been used for modeling such things as high speed trains and racing yachts. The NASA PMARC code from an early version of VSAERO and a derivative of PMARC, named CMARC,[26] is also commercially available. In the two-dimensional realm, a number of Panel Codes have been developed for airfoil analysis and design. The codes typically have a boundary layer analysis included, so that viscous effects can be modeled. Professor Richard Eppler of the University of Stuttgart developed the PROFILE code, partly with NASA funding, which became available in the early 1980s.[27] This was soon followed by MIT Professor Mark Drela's XFOIL code.[28] Both PROFILE and XFOIL incorporate two-dimensional panel codes, with coupled boundary layer codes for airfoil analysis work. PROFILE uses a conformal transformation method for inverse airfoil design, while XFOIL has both a conformal transformation and an inverse panel method for airfoil design.

An intermediate step between Panel Codes and Full Potential codes were codes that used the Transonic Small Disturbance equations. In particular, the three-dimensional WIBCO code,[24] developed by Charlie Boppe of Grumman Aircraft in the early 1980s has seen heavy use.

Developers turned to Full Potential codes, as panel methods could not calculate the non-linear flow present at transonic speeds. The first description of a means of using the Full Potential equations was published by Earll Murman and Julian Cole of Boeing in 1970.[29] Frances Bauer, Paul Garabedian and David Korn of the Courant Institute at New York University (NYU) wrote a series of two-dimensional Full Potential airfoil codes that were widely used, the most important being named Program H.[30] A further growth of Program H was developed by Bob Melnik and his group at Grumman Aerospace as Grumfoil.[31] Antony Jameson, originally at Grumman Aircraft and the Courant Institute of NYU, worked with David Caughey to develop the important three-dimensional Full Potential code FLO22[32] in 1975. Many Full Potential codes emerged after this, culminating in Boeing's Tranair (A633) code,[33] which still sees heavy use.

The next step was the Euler equations, which promised to provide more accurate solutions of transonic flows. The methodology used by Jameson in his three-dimensional FLO57 code[30] (1981) was used by others to produce such programs as Lockheed's TEAM program[34] and IAI/Analytical Methods' MGAERO program.[32] MGAERO is unique in being a structured cartesian mesh code, while most other such codes use structured body-fitted grids (with the exception of NASA's highly successful CART3D code,[33] Lockheed's SPLITFLOW code[34] and Georgia Tech's NASCART-GT).[35] Antony Jameson also developed the

three-dimensional AIRPLANE code[36] which made use of unstructured tetrahedral grids.

In the two-dimensional realm, Mark Drela and Michael Giles, then graduate students at MIT, developed the ISES Euler program[35] (actually a suite of programs) for airfoil design and analysis. This code first became available in 1986 and has been further developed to design, analyze and optimize single or multi-element airfoils, as the MSES program.[38] MSES sees wide use throughout the world. A derivative of MSES, for the design and analysis of airfoils in a cascade, is MISES.

CHAPTER III
DESIGN AND METHODOLOGY

CHAPTER III

DESIGN AND METHODOLOGY

3-1 Introduction:

This chapter is interested in how to reduce the cavitation in pumps and flow systems and computational fluid dynamics (ANSYS fluent flow) steps and used pump. Water inlet pump under operations conditions.

3-2 Program ANSYS (fluent flow) steps:

When the program is opened there are many steps flowed in order to compute the cavitation in pumps and the steps below expresses that.

Step 1 Impeller

Start the 3D version of ANSYS FLUENT. Read the mesh file, centrif-pump.msh . Check and display the grid.

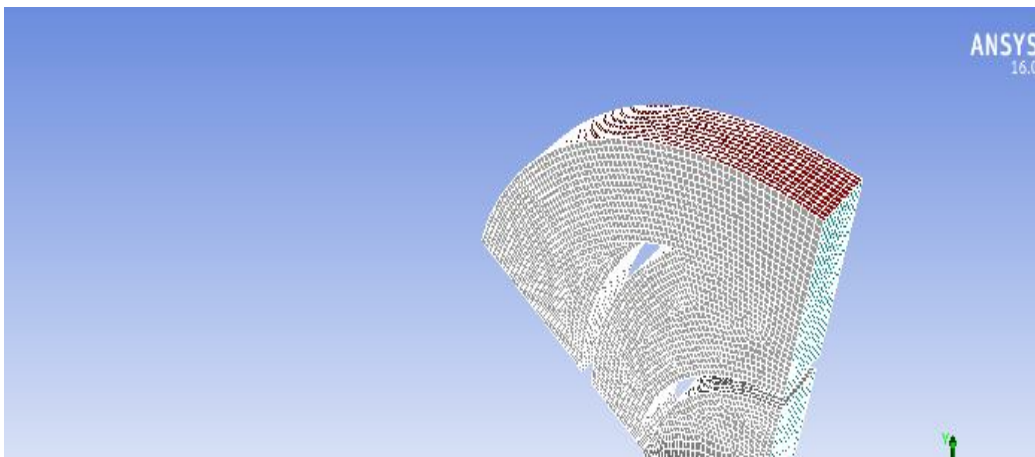


Figure 3-1 the mesh of impeller

Step 2: Models

Retain the default solver general parameters (3D, pressure-based solver, absolute velocity formulation). Enable the Realizable k-e model with

standard wall functions. Enable the mixture multiphase model, and disable the Slip Velocity.

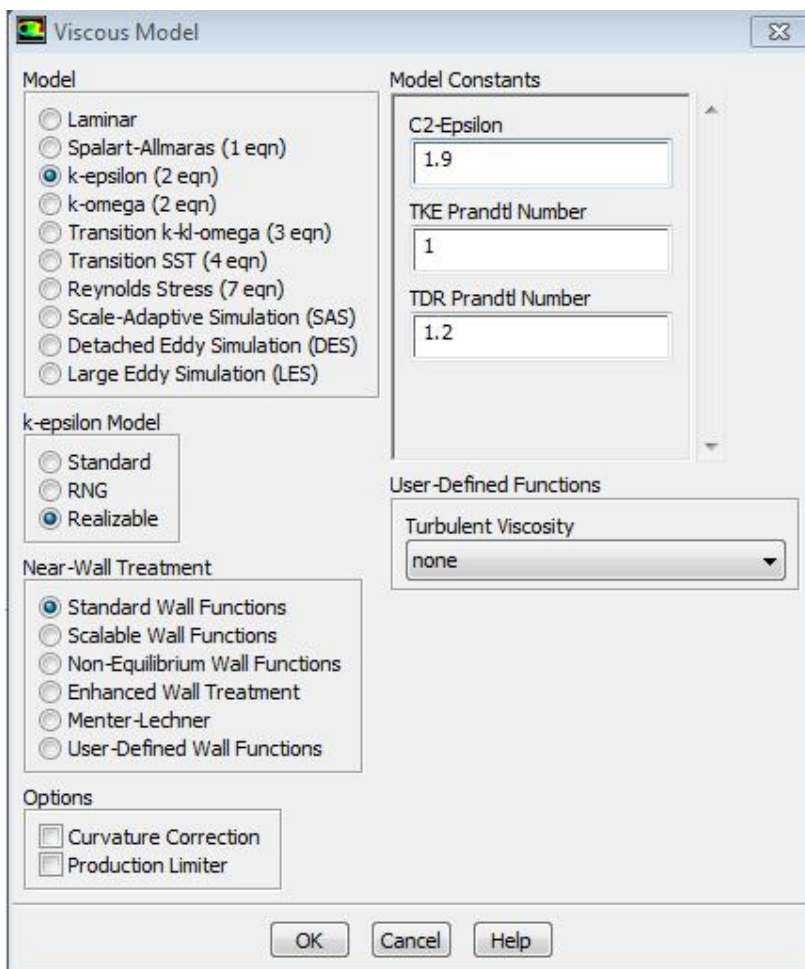
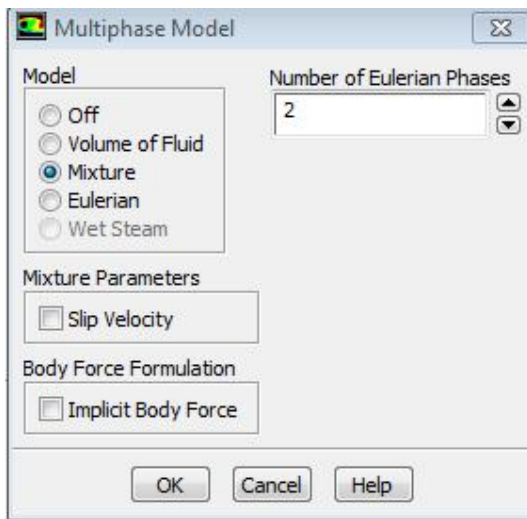


Figure 3-2 step 2

Step 3: Materials

Define the fluid material “water-liquid” with a density and viscosity set to 1000 kg/m³ and 0.001 kg/m-s, respectively. Define the fluid material water-vapor with a density and viscosity set to 0.01927 kg/m³ and 8.8e-6 kg/m-s, respectively.

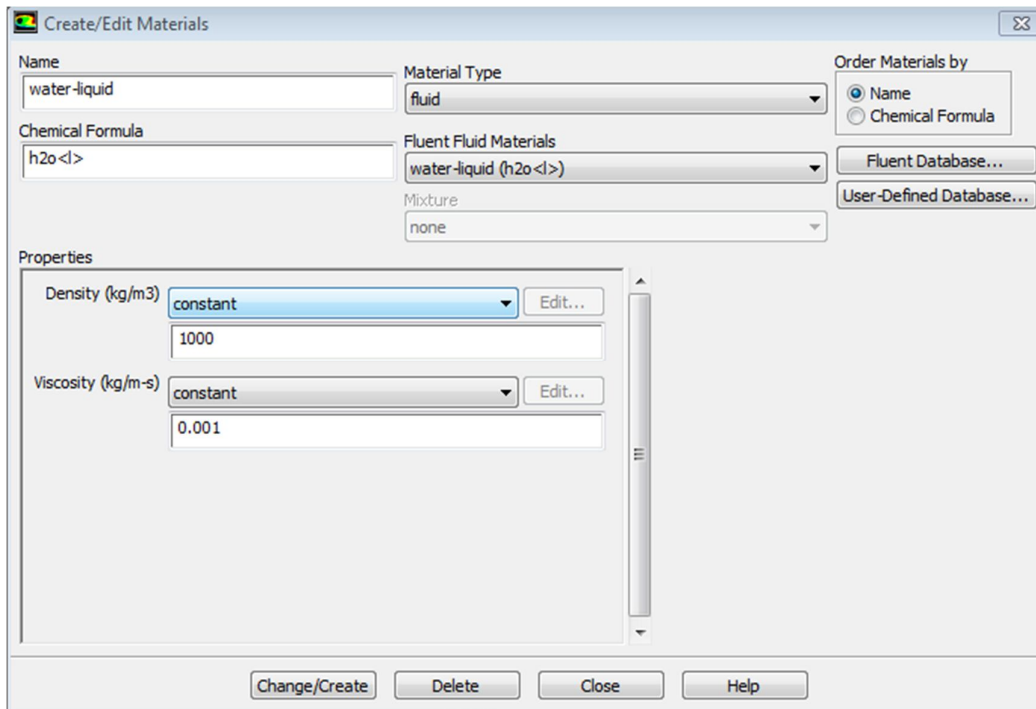


Figure 3-3 water phase

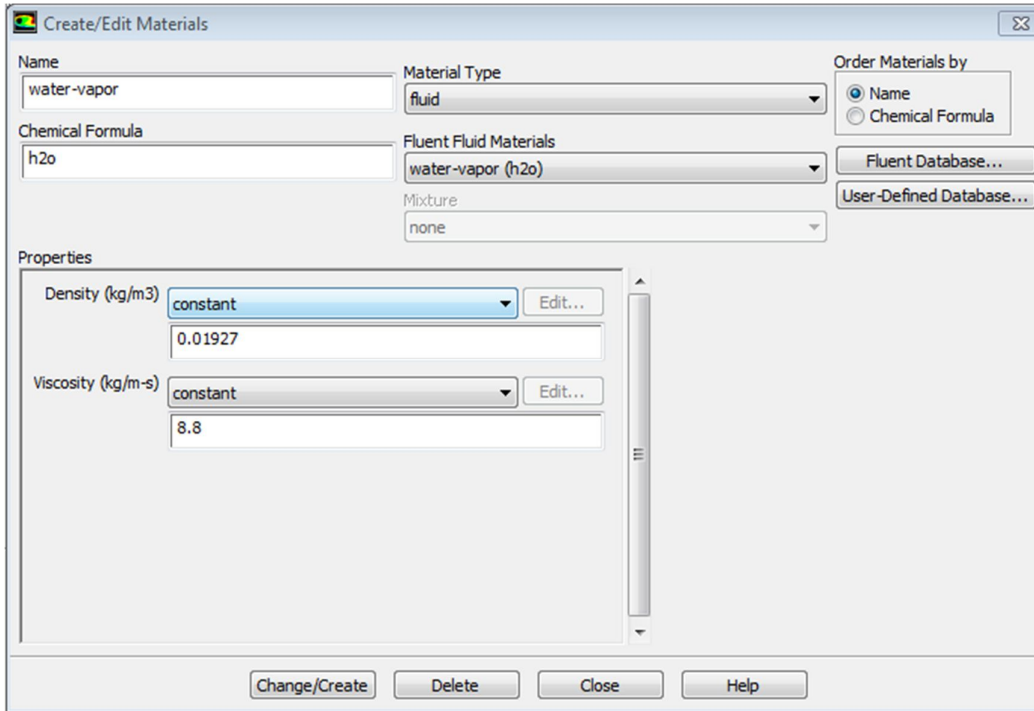


Figure 3-4 water vapor

Step 4: operations conditions:

Set the Operating Pressure to 0 Pa

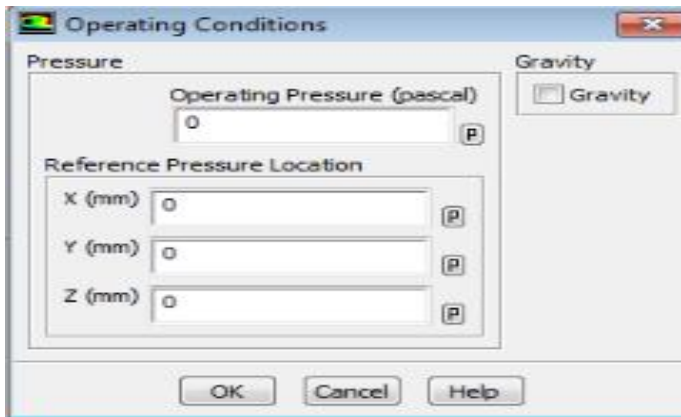


Figure 3-5 operation pressure

Step 5: Cavitation Model Setup

a- Define the phases Set phase-1 (primary phase) as water-liquid. Set phase-2 (secondary phase) as water-vapor.

b- Define interaction between the phases (see Figures 3-6). Click on Interaction to open the Phase Interaction panel. Under the Mass tab, enable Cavitation. Click on the Edit button to reveal the Cavitation Model panel. Set Vaporization Pressure to 3540 Pa and retain the default values for the other parameters. We will be using the Schnerr-Sauer cavitation model.

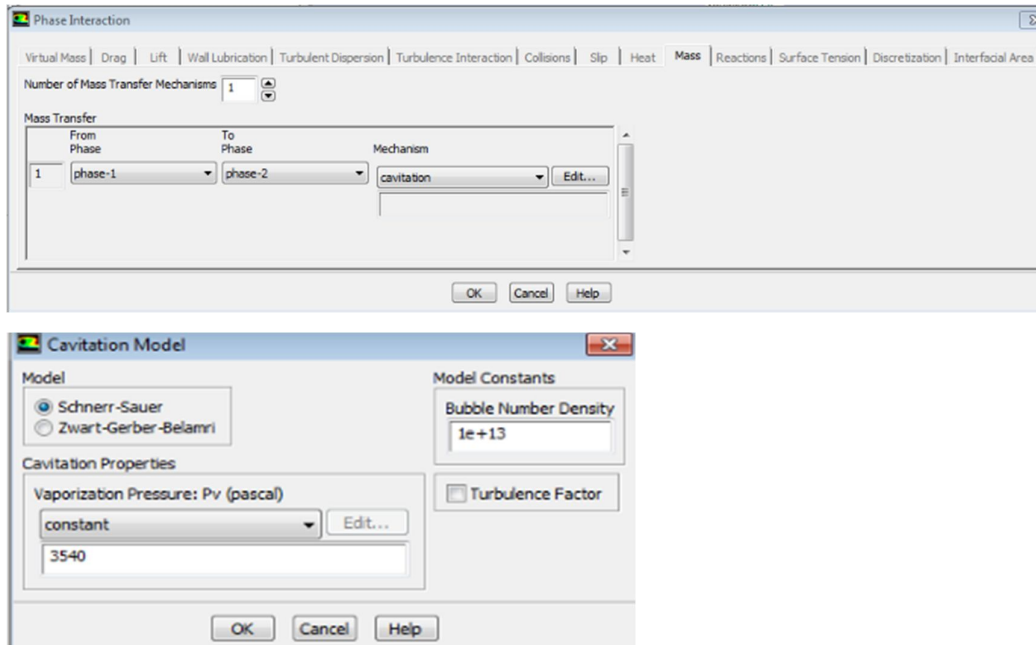


Figure 3-6 cavitation model setup

Step 6: Cell Zone Conditions:

The set up for moving reference frames in the cell zones panel has changed for ANSYS FLUENT 16 and is reproduced below.

A- Select “Frame Motion” and set the parameters for fluid as shown in the table 3-1.

Table 3-1 (cell zone conditions)

Parameter	Setting
Motion Type	Frame Motion
Rotation Axis Origin	(0, 0, 0)

Rotation-Axis Direction	X=0, Y=0, Z=1
Speed	3000 rpm

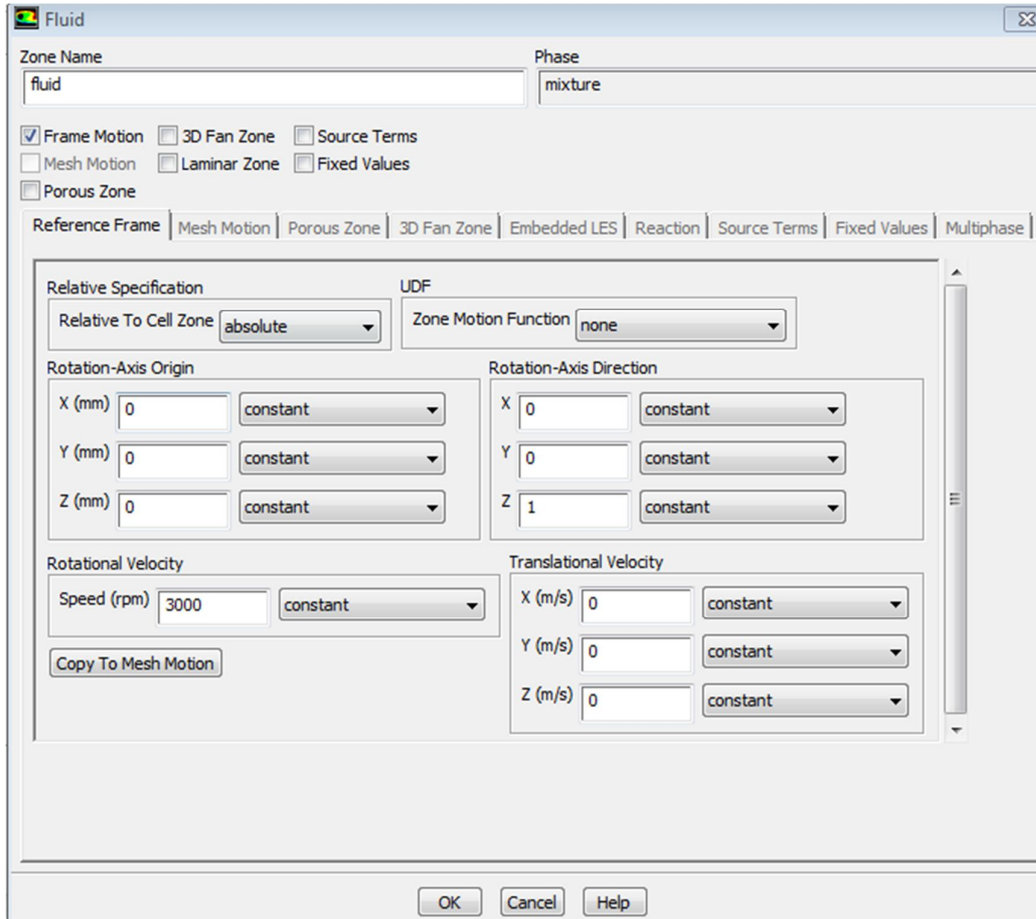


Figure 3-7 cell zone

Step 7: Boundary Conditions

(a) Set the parameters for inlet as shown in the table below and Figure 3-8).

Table 3-2: (boundary conditions inlet)

Parameter	Setting
Velocity magnitude	7 m/s
Velocity Specification method	Magnitude, Normal to Boundary

Turbulence Specification Method	Intensity and Hydraulic Diameter
Turbulence Intensity	5%
Hydraulic Diameter	103 mm
Phase-2 Volume Fraction	0

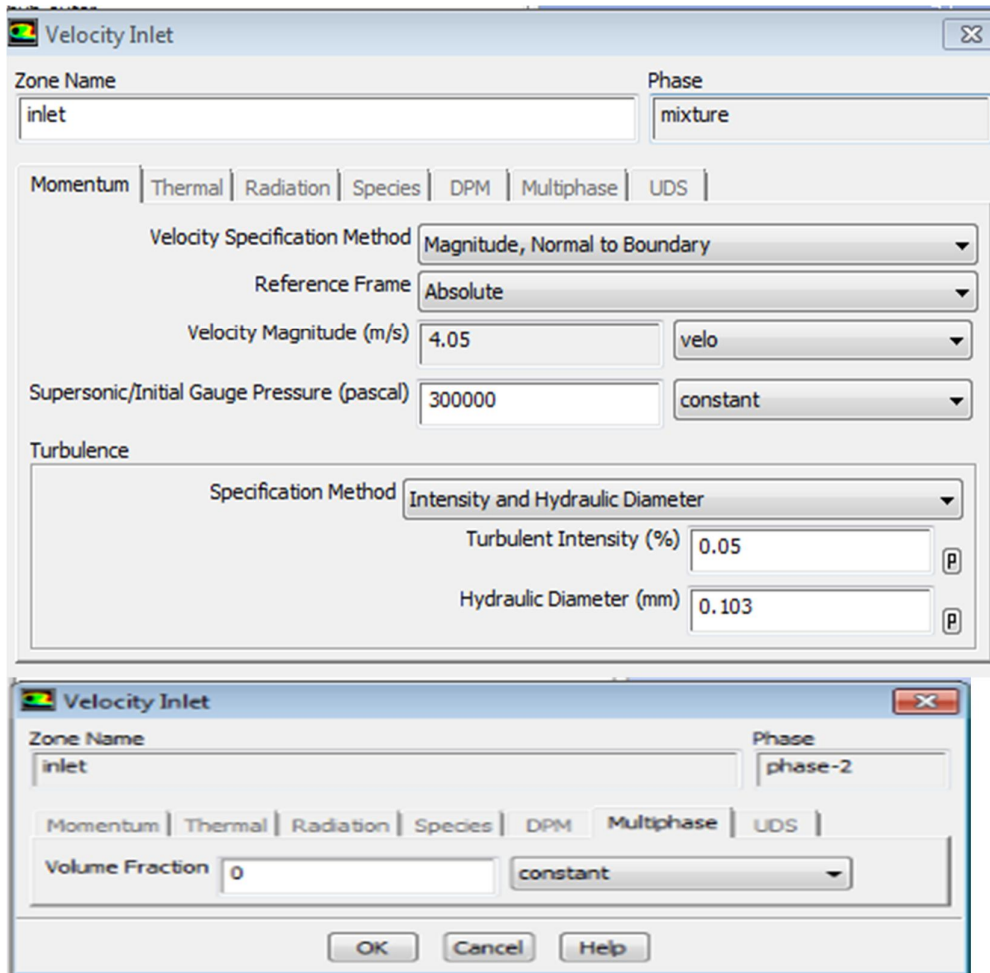


Figure 3-8 inlet parameter

(b) Set the parameters for outlet as shown in the table (3-3) and figure (3-9).

Table 3-3 (boundary conditions outlet)

Parameter	Setting
Gauge Pressure	300000 Pa

Turbulence Specification Method	Intensity and Viscosity Ratio
Backflow Turbulence Intensity	5%
Backflow Phase-2 Volume Fraction	0

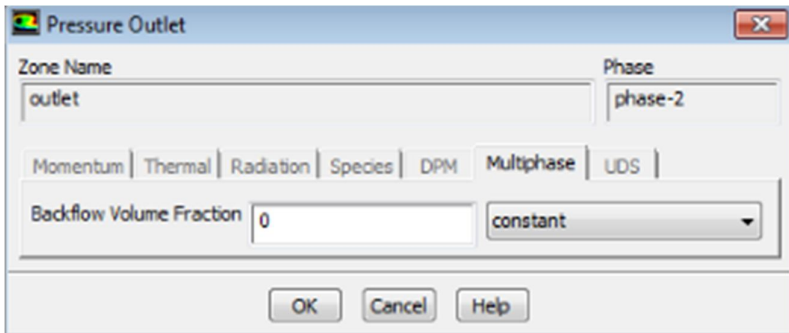
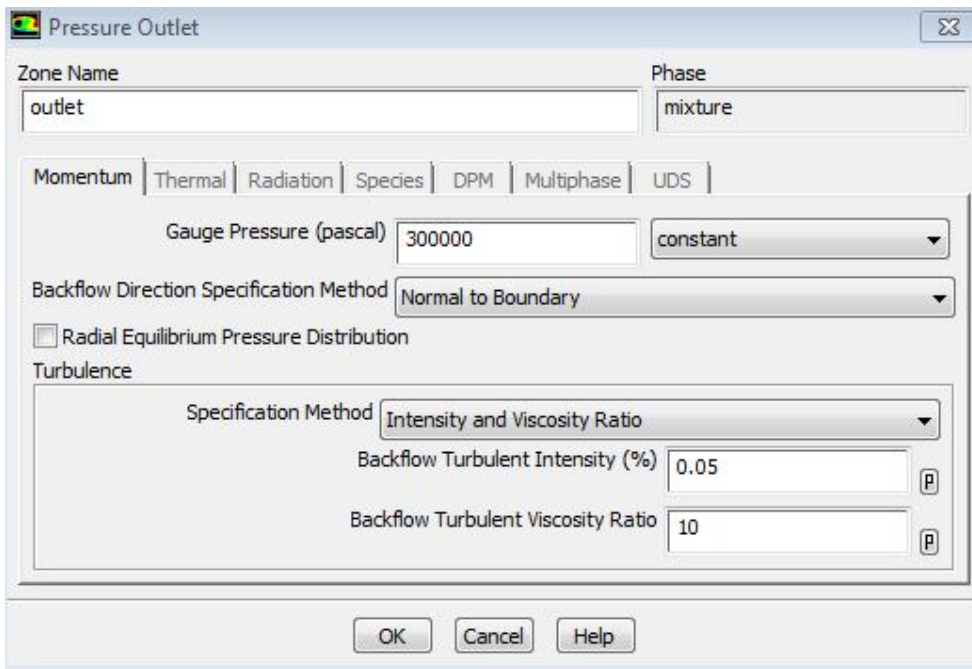


Figure 3-9: outlet parameter

Set the zones periodic.10 and periodic.11 as Rotational

Set the parameters for rotating walls (blade, hub, and shroud) as illustrated in Figure (3-10).

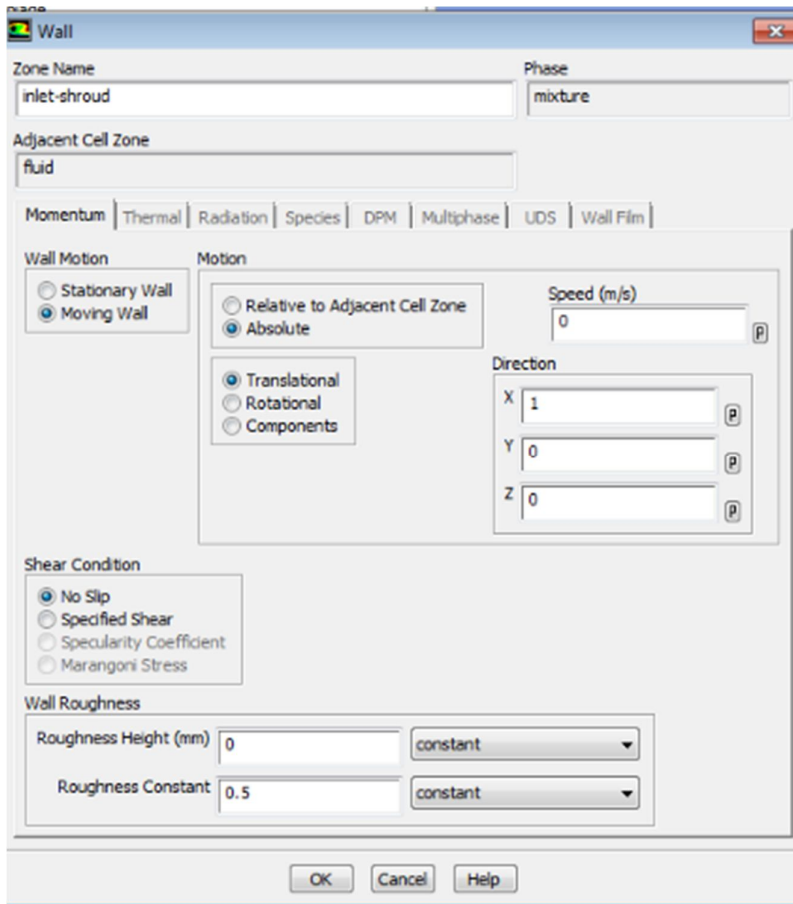


Figure 3-10 inlet-shroud

Set the parameters for non-rotating walls (blade, hub, and shroud) as shown in Figure 11.

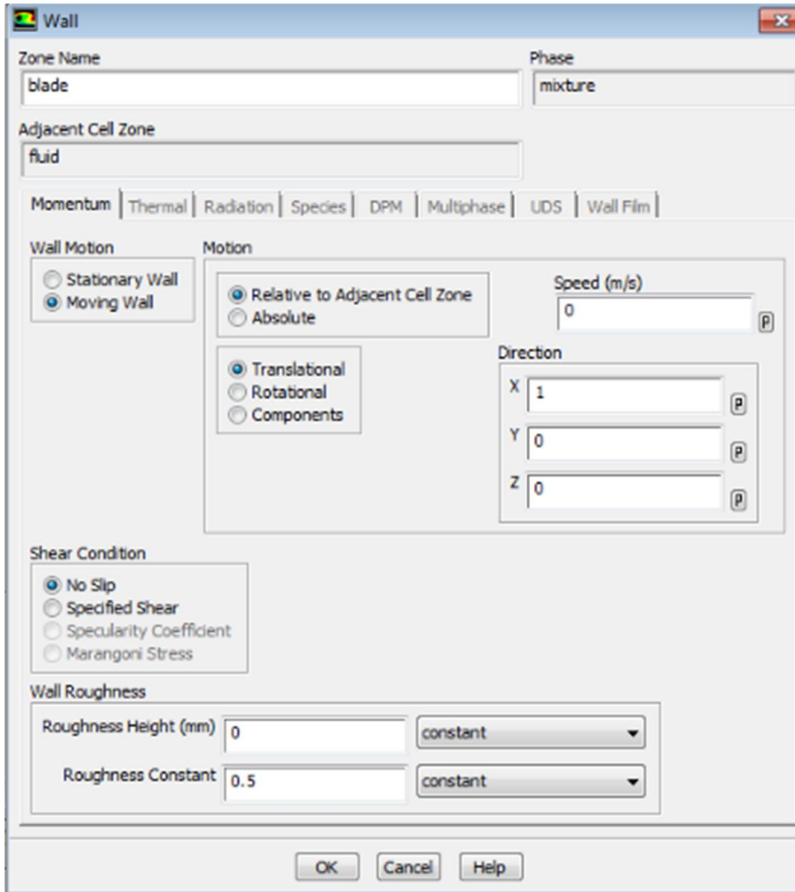


Figure 3-11 blade motion

Step 8: Solver Discretization, Controls, and Monitors set the solution methods under Solve Methods according to the table 3-4:

Table 3-4: solver discretization

Parameter	Setting
Scheme	Coupled
Gradient	Least Squares Cell Based
Pressure	PRESTO!
Momentum	Second Order Upwind
Volume Fraction	First Order Upwind
Turbulence Kinetic energy	First Order Upwind
Turbulence Dissipation Rate	First Order Upwind

Set the solution controls under Solve Controls according to the table 3-5:

Table 3-5 solution controls

Parameter	Value
Courant Number	200
Explicit Relaxation Factors (both values)	0.5
Density	1
Body Forces	1
Vaporization Mass	1
Volume Fraction	0.5
Turbulence Kinetic Energy	0.5
Turbulence Dissipation Rate	0.5

(a) Create a monitor for the area-averaged static pressure at the inlet boundary. This will be used to determine when the pressure rise through the pump has converged

Step 9: Solution

(b) Initialize the solution using values computed from the inlet values. Select Absolute for the Reference Frame.

(c) Solve for 1500 iterations. A

CHAPTER IV
RESULTS AND DISCUSSIONS

CHAPTER IV

RESULTS AND DISCUSSIONS

4-1 Effect of Rotational Speed (RPM) on Cavitation in Pumps and Flow Systems:

For a pump rotates at any running speed, the liquid entering the impeller eye turns and is split into streams by the leading edges of the impeller blades. The pump manufacturer provides the pressure in the pump suction exceeds the vapor pressure to prevent cavitation. This value is known as(NPSHr) NPSHr depends on the pump rotational speed as follows;
 $NPSHR \propto N$.

The present work considers five different values of rotational speed of (3000rpm;2500rpm; 2500rpm; 1750rpm; 1450rpm and constant discharge .(Increase in rpm = increase in NPSHr = increase in cavitation)

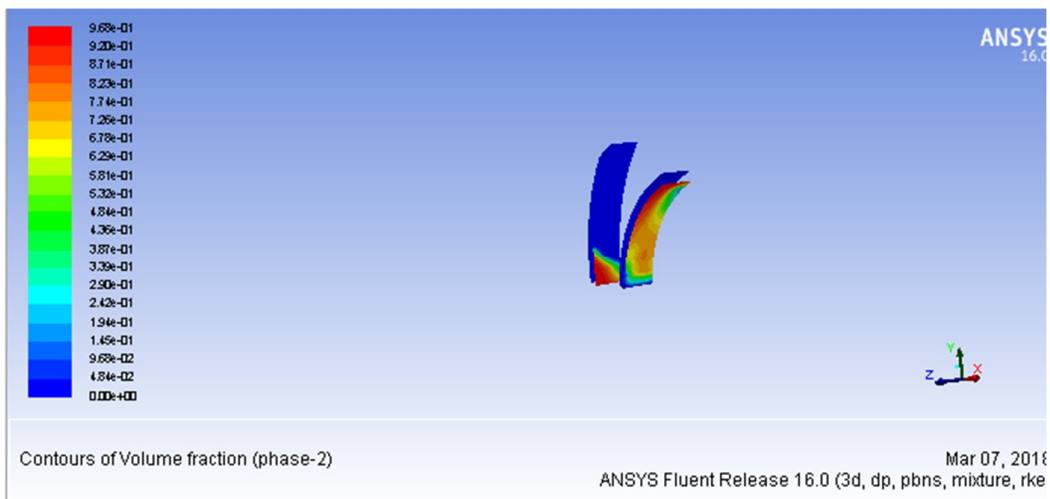


Figure 4-1 volume fraction at 3000rpm.

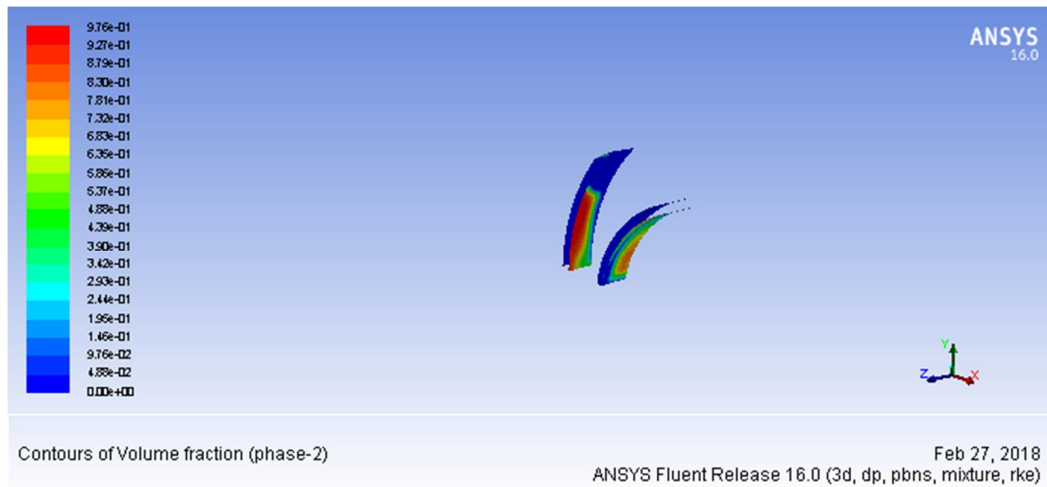


Figure 4-2 volume fraction at 2600rpm.

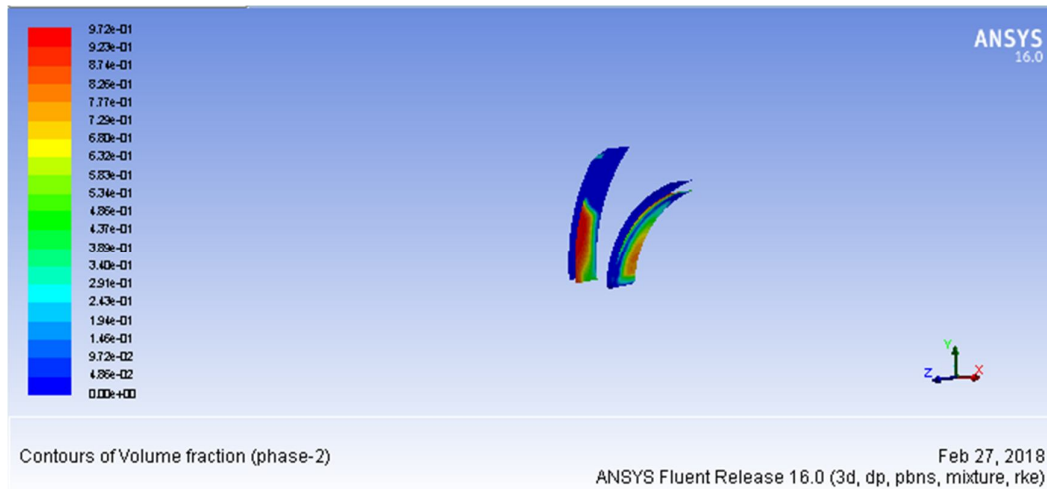


Figure 4-3 volume fraction at 2500rpm.

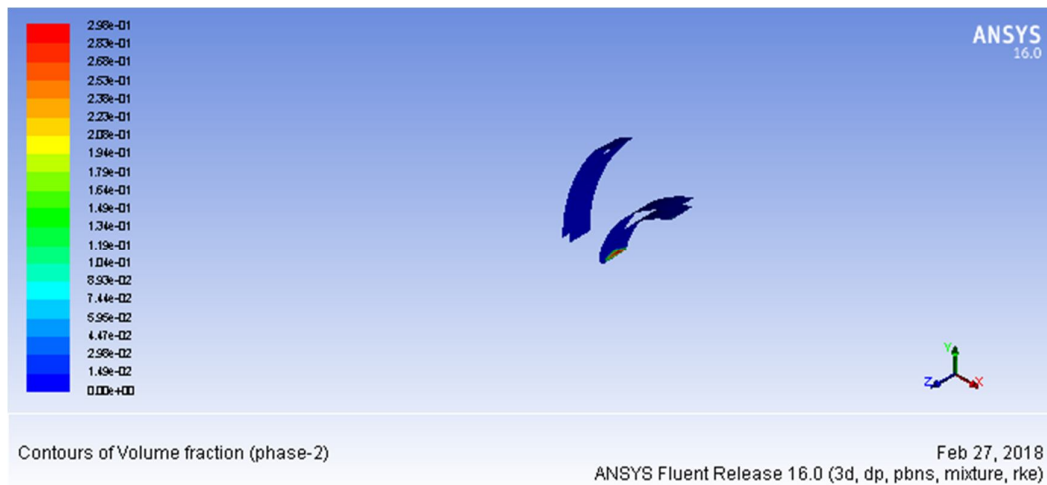


Figure 4-4 volume fraction at 1750rpm.

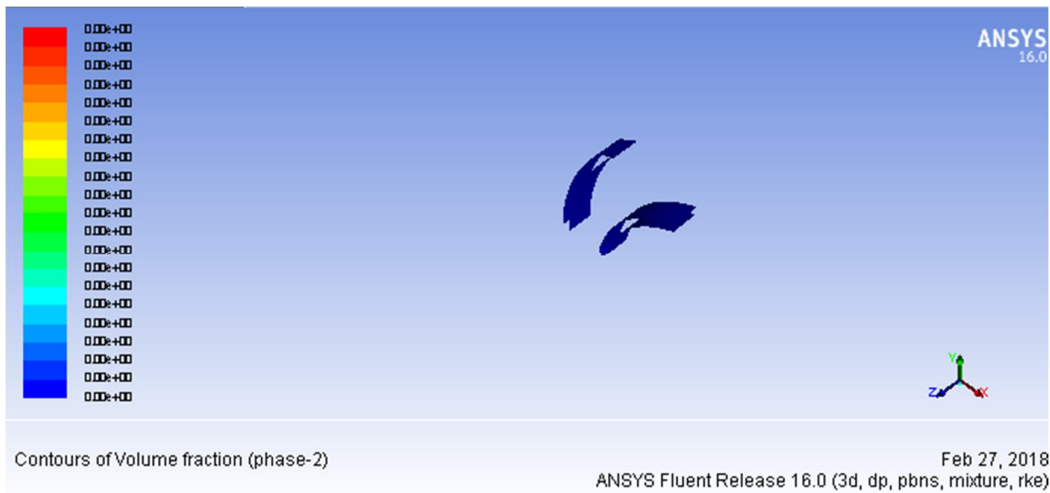


Figure 4-5 volume fraction at 1450rpm.

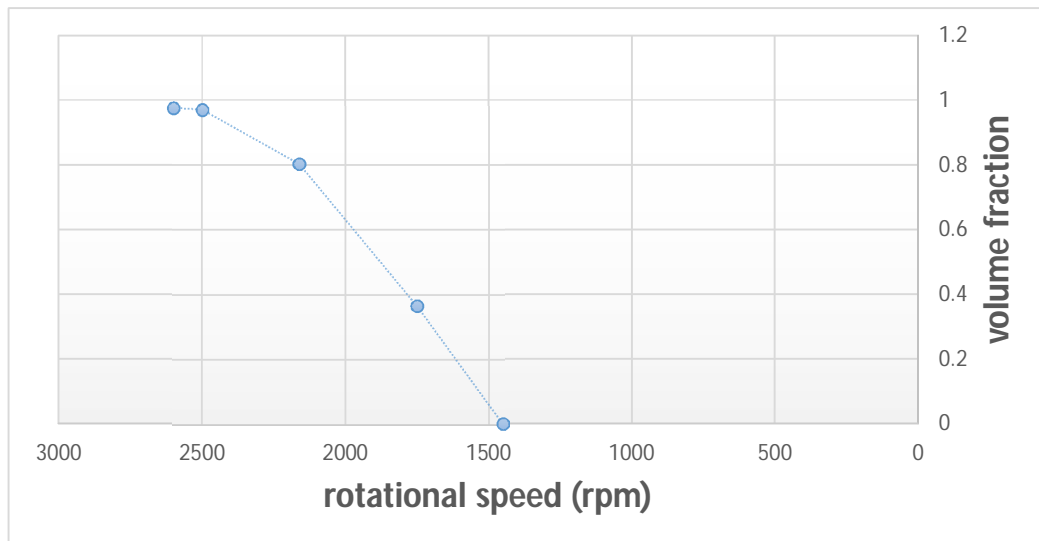


Figure 4-6 relation between the rotational speed and volume fraction.

4-2 Effect of Fluid Flow Rate (Q) on Cavitation in Pumps and Flow Systems:

The effects of flow rate on the (NPSH_r) is an increasing with increasing flow rate and cavitation increasing. The positive incidence is formed at the impeller inlet when the pump works under a low flow rate condition. the decrease of NPSH_a, a cavity first appears on the suction side of the blades and gradually grows upward.

The present work considers five different values of flow rate of (12.5; 10.5; 7; 5.046; 4.05m³/hr).



Figure 4-7 volume fraction at 12.5m³/hr

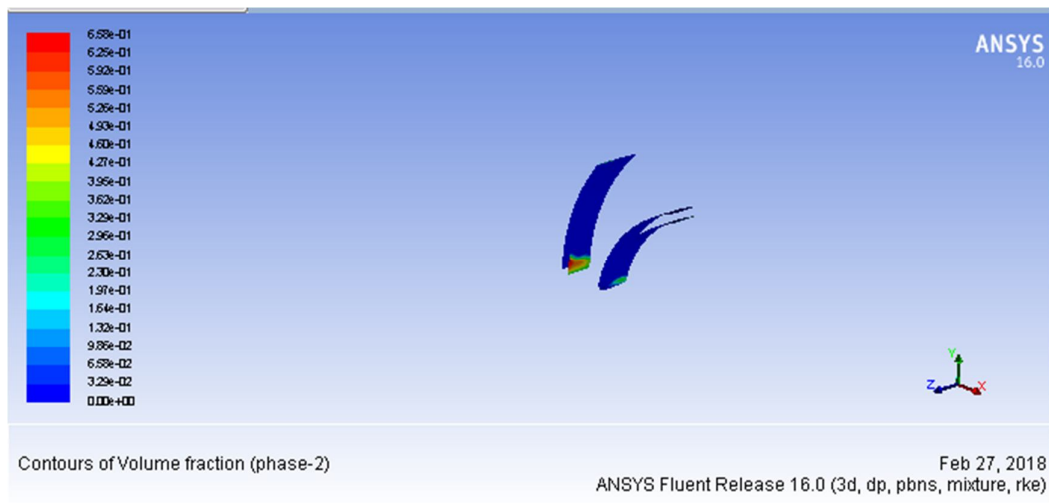


Figure 4-8 volume fraction at 10.5m³/hr

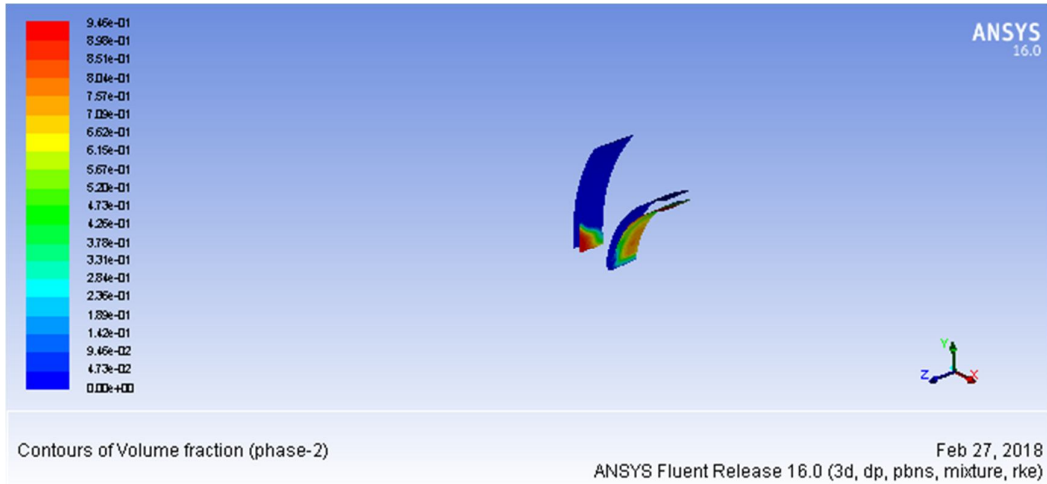


Figure 4-9 volume fraction at 5.046m³/hr

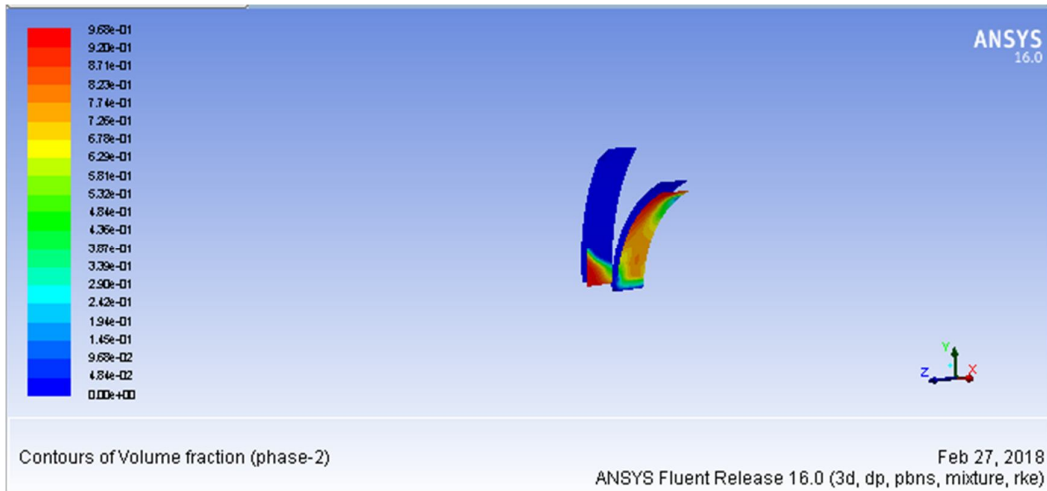


Figure 4-10 volume fraction at 4.05m³/hr

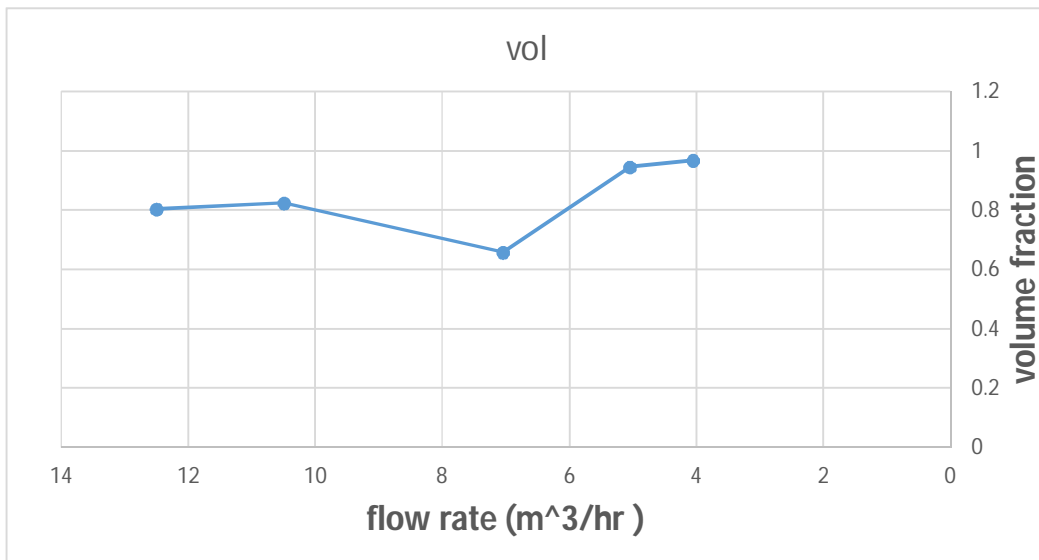


Figure 4-11 relation between the fluid flow rate and volume fraction

4-3 Effect of Fluid Temperature (T) on Cavitation in Pumps and Flow Systems:

The thermodynamic effect of cavitation for a pumps in any fluid is defined here in as the difference between the required net positive suction head for a given liquid and temperature this refers at constant rotational speed and fluid flow rate. When temperature of fluid increase net positive suction required decrease, head rise increase and cavitation in pumps decrease.

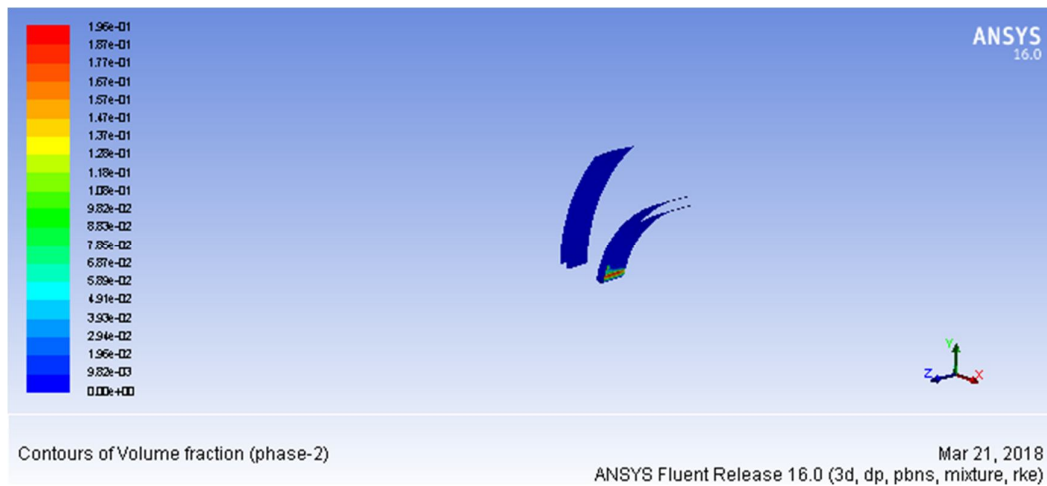


Figure 4-12 volume fraction at 250k

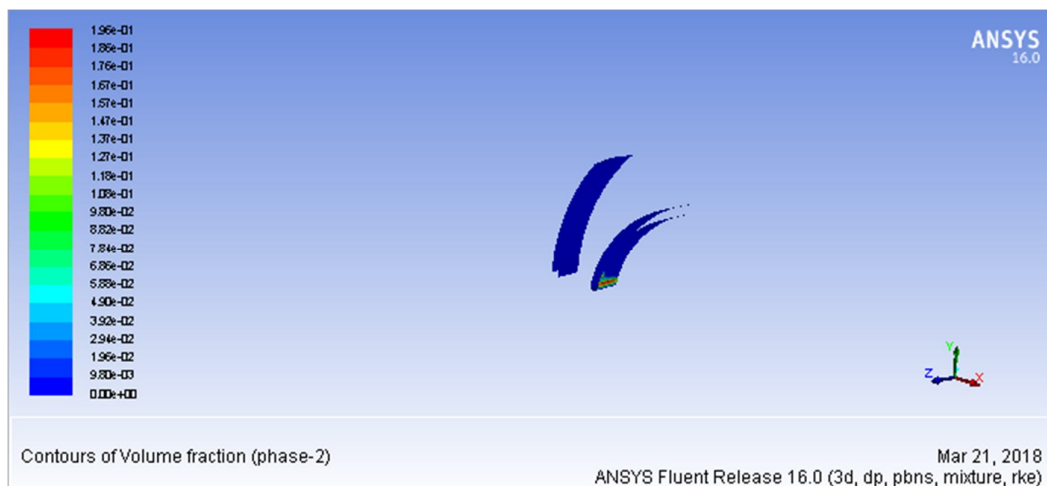


Figure 4-13 volume fraction at 280k

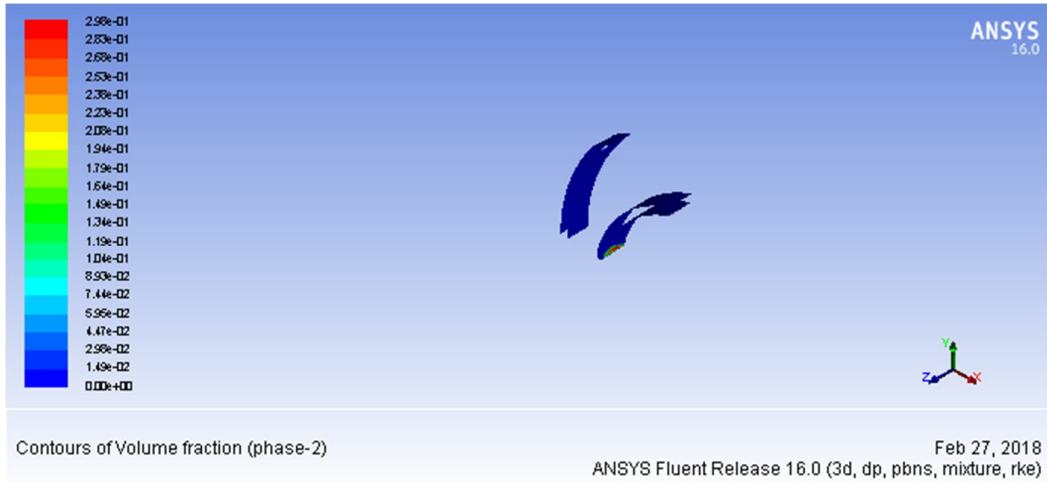


Figure 4-14 volume fraction at 350k

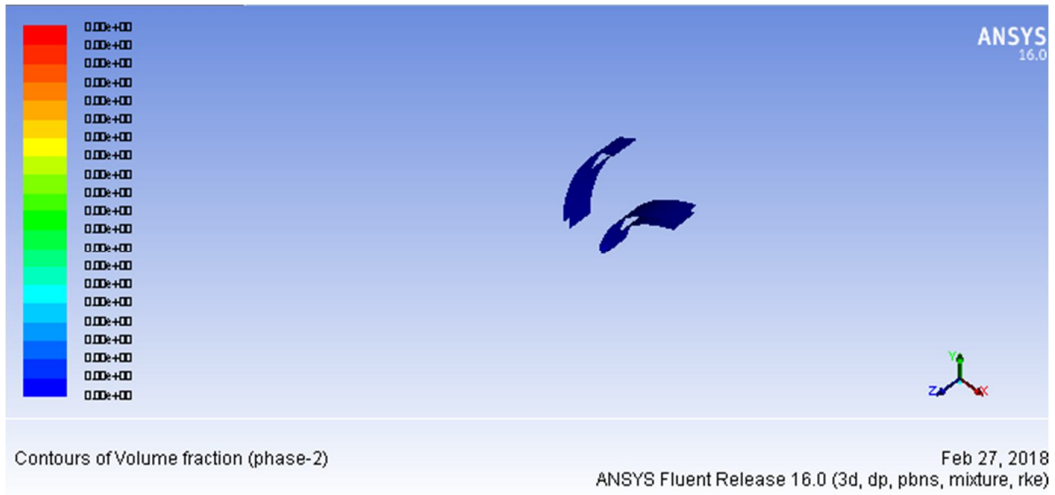


Figure 4-15 volume fraction at 400k

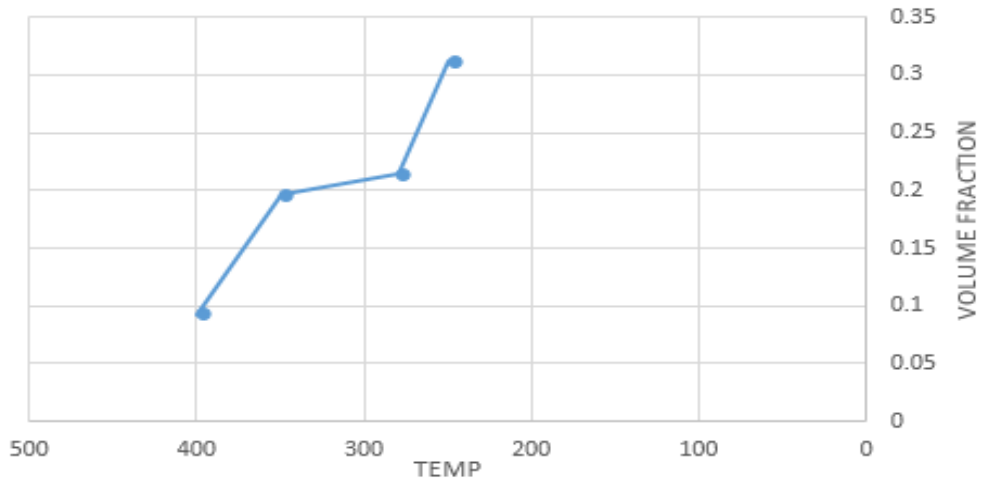


Figure 4-16 relation between volume fraction and temperature

4-4 Effect of Pressure (P) on Cavitation in Pumps and Flow Systems:

To reduce the level of cavitation, increase the exit pressure, the cavitating region of the flow will expand to include more of the blade passage. This will increase the head rise, leading to a reduce net positive suction head required (NPSHr). Conversely, by increasing the exit pressure, the cavitation bubble will shrink and, at a high enough pressure.

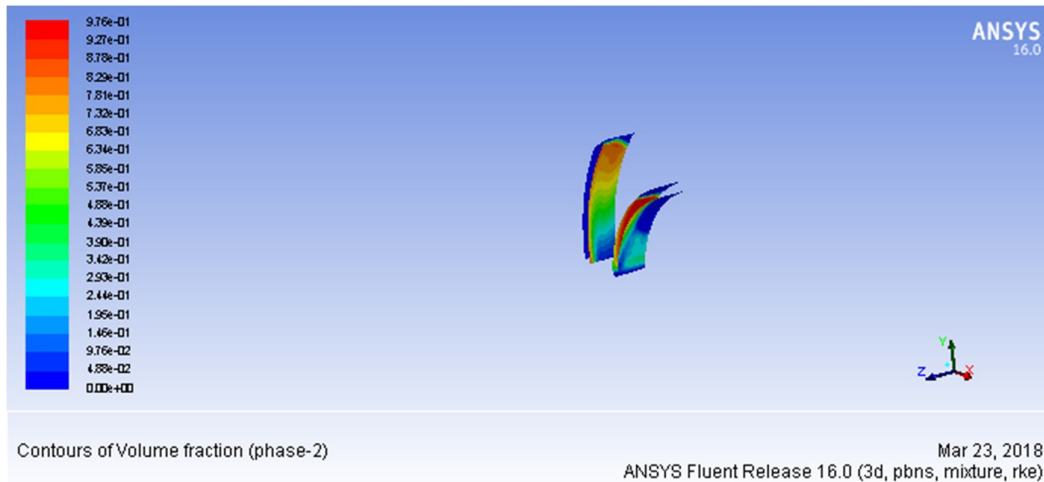


Figure 4-17 volume fraction at 2.5MPa

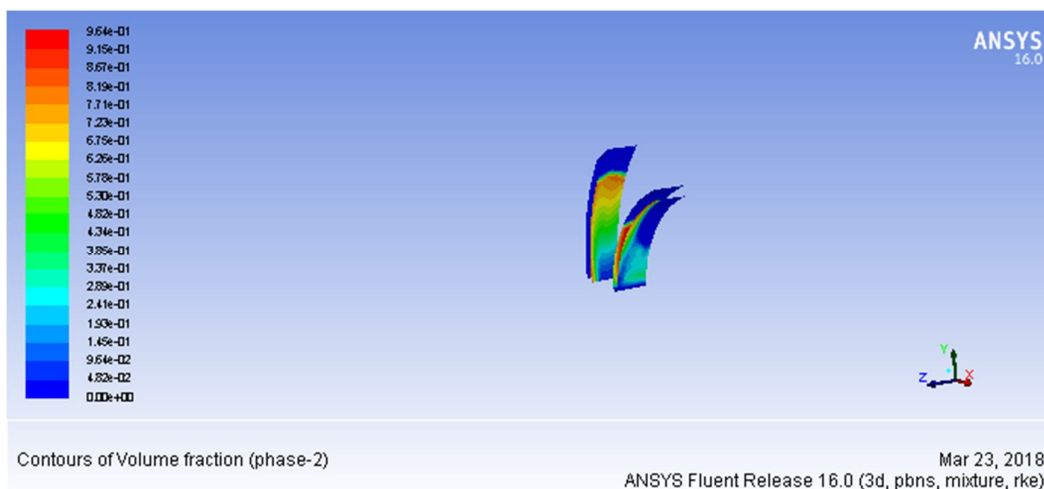


Figure 4-18 volume fraction at 3MPa

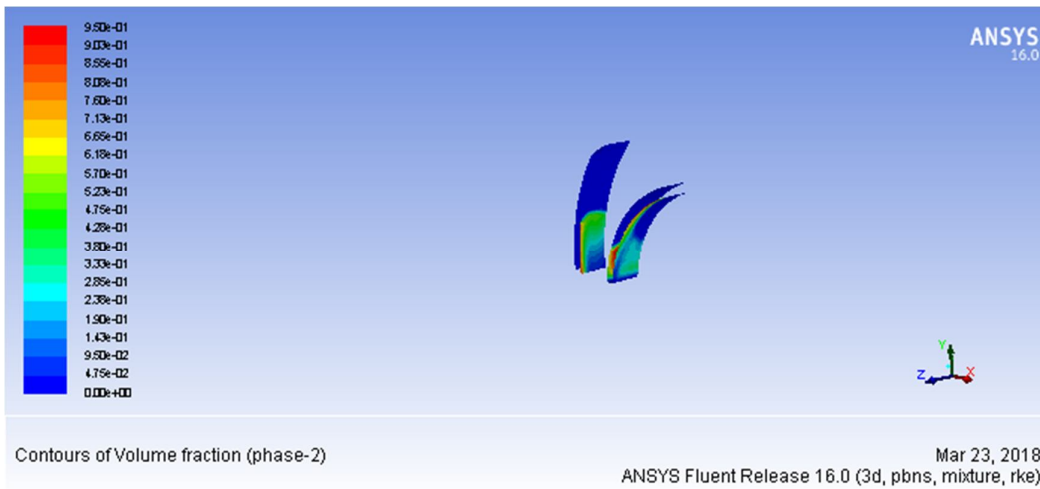


Figure 4-19 volume fraction at 3.5MPa

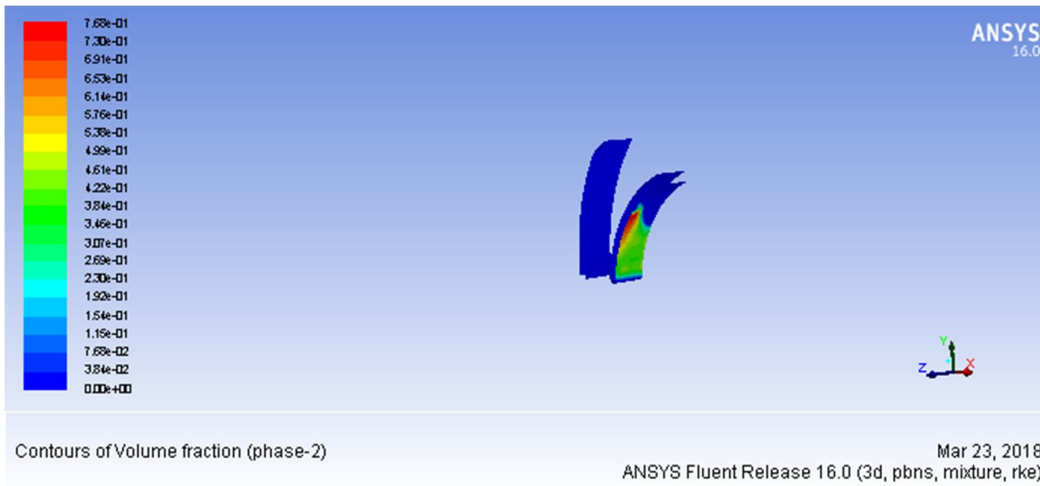


Figure 4-20 volume fraction at 4.5MPa

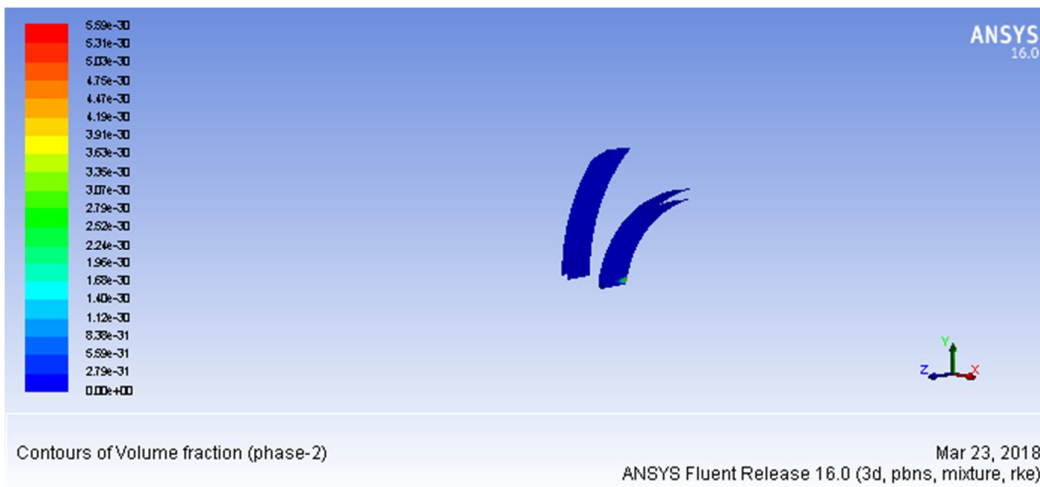


Figure 4-21 volume fraction at 5MPa

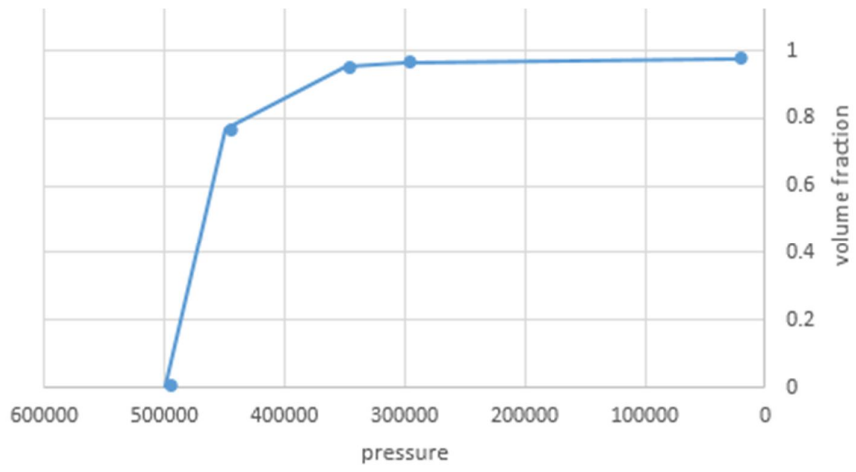


Figure 4-22 relation between pressure and volume fraction

CHAPTER V
CONCLUSIONS AND RECOMMENDATIONS

CHAPTER V

CONCLUSIONS AND RECOMMENDATIONS

5-1 Conclusion

In this study to reduce the cavitation in pumps and flow systems to figure outlet how change in rotational speed; fluid flow rate; temperature and exit pressure. The results obtained are summarized as follows:

- (a) Finally, through this research find NPSHr increases with increasing the flow rate and increasing rotational speed, whereas the NPSHa decreases, this mean cavitation in pumps increases.
- (b) NPSHa decreases with increasing the flow rate and temperature and it increases with decreasing the rotational speed, whereas NPSHr increases with increasing the flow rate and it decreases with increasing the temperature and decreasing the rotational speed.
- (c) Cavitation regions appear at the leading edge of impeller blades which represents the lowest pressure area inside the computational domain of the pumps, where these cavitation regions expand with increase the NPSHr and decreasing the temperature whereas they reduce with decreasing the rotational speed.

Recommendation

1- recommend that this research apply experimentally, and then evaluate it and compare the result to make sure that the best method use it in next researches.

2- recommend that the rising in temperature it lead to rising in pressure so cavitation is not occurring in this case.

References.

- [1] Sudsuansee T, Nontakaew U, Tiaple Y (2011) Simulation of leading edge cavitation on bulb turbine. *Songklanakarin J Sci Technol* 33: 51-60.
- [2] Mostafa NH, Rayan MA, Mahgob MM (1990) Energetic model for cavitation erosion prediction in centrifugal pump impeller. *Proceeding of ASME/CSME Cavitation and Multiphase flow forum*, Toronto, Canada.
- [3] Fukaya M, Okamura T, Tamura Y, Matsumoto Y (2003) Prediction of Cavitation Performance of Axial Flow Pump by Using Numerical Cavitating Flow Simulation with Bubble Flow Model. *Fifth International Symposium on Cavitation*, Osaka, Japan.
- [4] Sedlář M, Zima P, Němec T, Maršík F (2008) Analysis of Cavitation Phenomena in Water and its Application to Prediction of Cavitation Erosion in Hydraulic Machinery. *ICPWS XV*, Berlin.
- [5] De-sheng Z, Wei-dong S, Bin C, Xing-fan G (2010) Unsteady flow analysis and experimental investigation of axial-flow pump. *Journal of Hydrodynamics, Ser. B* 22: 35-43.
- [6] Tabar MTS, Poursharifi Z (2011) An Experimental Study of Tip Vortex Cavitation Inception in an Axial Flow Pump. *World Academy of Science, Engineering and Technology* 73: 527-530
- [7] Albraik, A., Althobiani, F., Gu, F. and Ball, A. (2012) Diagnosis of Centrifugal Pump Faults Using Vibration Methods. *Journal of Physics: Conference Series*, 364, Article ID: 012139. <http://dx.doi.org/10.1088/1742-6596/364/1/012139>
- [8] Suhane, A. (2012) Experimental Study on Centrifugal Pump to Determine the Effect of Radial Clearance on Pressure Pulsations,

Vibrations and Noise. International Journal of Engineering Research and Applications (IJERA), 2, 1823

Distribution” ASME J. Fluids Eng., vol. 115, pp. 243-248, 1993.

[9] Callenaere M., Franc J.P., Michel J.M. and Riondet M., “The Cavitation Instability Induced by the Development of a Re-entrant Jet”, J. Fluid Mech., vol 444, pp. 223-256, 2001.

[10] R.T. Knapp et al. Cavitation" Mc Craw-Hill Book Company, 197111.

[11] S. Lazariwicz, A.T. Trosklanski. "Impeller Pumps" Pergamon Press, 1965.

[12] J. Dally. "Lessons from a carrier in hydraulics" ASCE, Journal Hydraulic engineering, vol. 112, No 9, pp 779-793, Sep 1986.

[13] R.K. Turton. "Principles of turbomachinery" E & F.N. Spon, 1984.

[14] D.A. Taylor. Cavitation of Fluids Introduction to marine engineering" Butterworth Ltd, 1983.

[15] W.M. Deeprose et al. Cavitation noise, flow noise and erosion" Cavitation conference, Paper C118/74, 1974.

[16] M J Braun and W M Hannon.,“ Cavitation formation and modeling for fluid film bearings”, Proceedings of the Institution of Mechanical Engineers, Part J: Journal of Engineering Tribology.

[17] Chang,L., “Simulation of cavitating flow in circulating water Pumps in a power generation industry”, B.Sc.thesis, Faculty of Mechanical Engineering, Design&Innovation Department, Universiti Teknikal, Melaka,Malaysia, May 2008.

[18] Chen, Y., “Cavitation Erosion”, ASM Handbook Volume 11, Failure and Prevention (ASM International), CETIM Senlis, France,2002.

[19] Whitesides, R., “Understanding Net Positive Suction Head”, PE, PDH Online Center www.PDHcenter.com , May 2012.

- [20] Koivula, T., "On Cavitation in fluid power", PhD thesis, Tampere University of Technology Institute of Hydraulics and Automation, Hamburg, Germany, 2000
- [21] Pereira F, Salvatore F, Felice F (2004) Measurement and Modeling of propeller cavitation in uniform inflow. *J Fluid Eng- T ASME* 126: 671-679.
- [22] CFD-ACE+ V2008 (2008) Modules Manual - Volume
- [23] Kerr, B. M.; Day, M. S.; Howick, G. J.; Lesher, R. L.; and Timms, H. L.: Technology Utilization. Selected Technology for the Petroleum Industry. NASA SP-5053, 1966, pp. 155-166.
- [24] Alder, Thomas F. ; Ruggeri, Robert S. ; and Moore, Royce D. : Cavitation Similarity Considerations Based on Measured Pressure and Temperature Depressions in Cavitated Regions of Freon 114. NASA TN D-3509, 1966.
- [25] Os, M.J., Op de Woerd, J.G.H and Jonker, J.B. ,1997, "A Parametric Study of the Cavitation Inception Behaviour of a Mixed-Flow Pump Impeller Using a Three-Dimensional Potential Flow Model," Proc. ASME FEDSM97-3374.
- [26] Jump up ^ Milne-Thomson, L.M. (1973). *Theoretical Aerodynamics*. Dover Publications. ISBN 0-486-61980-X.
- [27] Jump up ^ Richardson, L. F.; Chapman, S. (1965). *Weather prediction by numerical process*. Dover Publications.
- [28] Jump up ^ Hunt (1997). "Lewis Fry Richardson and his contributions to mathematics, meteorology, and models of conflict". *Annual Review of Fluid Mechanics*. 30. Bibcode:1998AnRFM..30D..13H. doi:10.1146/annurev.fluid.30.1.0.
- [29] Jump up ^ "The Legacy of Group T-3". Retrieved March 13, 2013.
- [30] Jump up ^ Harlow, F. H. (2004). "Fluid dynamics in Group T-3 Los Alamos National Laboratory:(LA-UR-03-3852)". *Journal of*

- Computational Physics. Elsevier. 195 (2): 414–433.
Bibcode:2004JCoPh.195..414H. doi:10.1016/j.jcp.2003.09.031.
- [31] Jump up ^ F.H. Harlow (1955). "A Machine Calculation Method for Hydrodynamic Problems". Los Alamos Scientific Laboratory report LAMS-1956.
- [32] Jump up ^ Gentry, R. A.; Martin, R. E.; Daly, J. B. (1966). "An Eulerian differencing method for unsteady compressible flow problems". Journal of Computational Physics. 1 (1): 87–118. Bibcode:1966JCoPh...1...87G. doi:10.1016/0021-9991(66)90014-3.
- [33] Jump up ^ Fromm, J. E.; F. H. Harlow (1963). "Numerical solution of the problem of vortex street development". Physics of Fluids. 6: 975. Bibcode:1963PhFl....6..975F. doi:10.1063/1.1706854.
- [34] Jump up ^ Harlow, F. H.; J. E. Welch (1965). "Numerical calculation of time-dependent viscous incompressible flow of fluid with a free surface" (PDF). Physics of Fluids. 8: 2182–2189. Bibcode:1965PhFl....8.2182H. doi:10.1063/1.1761178.
- [35] Jump up ^ Hess, J.L.; A.M.O. Smith (1967). "Calculation of Potential Flow About Arbitrary Bodies". Progress in Aerospace Sciences. 8: 1–138. Bibcode:1967PrAeS...8....1H. doi:10.1016/0376-0421(67)90003-6.

