بسم الله الرحمن الرحيم

Sudan University of science and Technology

College of graduate studies

The Cavitation Effect in Centrifugal Pumps

تاثير التكهف على مضخات الطرد المركزي

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

Prepared by:

Eng. Al Hadi Emam Abdalla Elshich

Supervisor:

Dr. Mohyedin Ahmed Abdelghadir

July 2018

الآية

بسم الله الرحمن الرحيم

ٹٱڈ

ٱٱبهتجتحتختمتهثمـ جم حم خم سم

لجلحلخلملهمجمحّ

صدق الله العظيم

الرعد: ١٧

Dedication

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

Acknowledgements

First, I am very grateful to allah for guiding me to finish this work. Also , I want to thank my father and mother for standing by my side and my family members for their precious time to help me and encourage me a lot during this work. special thanks to Supervisor Dr. **Mohyedin Ahmed Abdelghadir** for his assistance and a valuable ideas throughout the work . I also want to thank my commetee members participation and help especially in the theoretical areas. I hope, to thank the staff of the section of at Sudan University science and Technology, be the academic administration or technical, for their efforts ,cooperation and love. I am very appreciate to my fellow colleagues at Mechaical Engineering Faculity .

ABSTRACT

Cavitation in a centrifugal pump has a significant effect on pump performance. Cavitation degrades the performance of a pump, resulting in a fluctuating flow rate and discharge pressure, that reduces pumps efficiency and the operating life of the pump, In this research the effects of cavitation on pump performance was studied on the power required ,efficiency , volume flow rate ,and the head , using a 3-D simulation by ANSYS (fluent flow) software for two phase flow (the water and its vapor) . By reducing the values of exit pressure from (350000 to 100000) Pa at constant (R. P. M) and constant velocity inlet of water . It was found that the performance of a pump is reduce with increasing the level of cavitation.

المستخلص

التكهف في مضخات الطرد المركزية له تاثيركبيرعلي اداء المضخة .التكهف يقلل من اداء المضخة مما يؤدي الي تذبذب معدل التدفق وضغط التدفق. وذالك يسبب انخفاض كفاءة المضخة والعمر التشغيلي للمضخة.تمت دراسة تاثير التكهف على اداء المضخة كالقدرة المطلوبة ,الكفاءة ,معدل التدفق والارتفاع باستخدام برنامج الحاسوب (ANSYS) للسريان ثنائي الطور (الماء وبخاره) بتقليل قيم ضغط الخروج من (350000 إلى 100000) باسكال عند ثبوت سرعة الدوران وثبوت سرعة دخول الماء وجد ان أداء المضخة يقل بزيادة معدل التكهف .

List of Contents

Title	Page
Dedication	II
Acknowledgments	III
Abstract	IV
المستخلص	V
List of Contents	VI
List of Tables	VIII
List of Figures	IX
List of Apreviation	X
CHAPTER I	
1-1 Back ground	1
1-2 Problem Statement	2
1-3 Objective	2
CHAPTERII	
2-1 Back ground	3
2-2. Performance Prediction Analysis	3
2-3. Different Turbulence Model	4
2-4. Various Flow Conditions	6
2-5 Computational Fluid Dynamics	8
2-6 Impeller	9
2-7 Pump performance	10
CHAPTER III	
3-1 Methodology	14

3-2 Power Calculation	14
3-3 Program ANSYS (fluent flow) Steps	15
3-4 Calculation	27
CHAPTER IV	
4.1 Results	34
4.2 Relation	36
CHAPTER V	
5.1 Concolosion	41
5.2 Recomondation	41

List of figures

Figure 3.1 model impeller	15
Figure 3.2 project schematic	16
Figure 3.3 Fluent launcher	16
Figure 3.4: Phase Interaction panel	18
Figure 3-5: Cavitation Model panel	18
Figure3-6: Fluid panel for cell zone conditions illustrating new moving frame set up.	19
Figure3- 7: Velocity Inlet BC panels	21
Figure 3-8: Pressure Outlet BC panels	22
Figure 3-9: Rotating wall BC set up	23
Figure3- 10: Non-rotating wall BC set up	24
Figure 3.11 volume fraction at exit pressure 350000 Pa	27
Figure 3.12 volume fraction at 300000 Pa	29
Figure 3.13 volume fraction at 250000 Pa	30
Figure 3.14 volume fraction at 200000 Pa	31
Figure 3.15 volume fraction at 150000 Pa	32
Figure 3.16 volume fraction at 100000 Pa	33
Figure 4.1 head carve	36
Figure 4.2 P-out carve	37
Figure 4.3 volume flow rate carve	38
Figure 4.4 hydruolic efficiency carve	39
Figure 4.5 total efficiency carve	40

List of Tables

Table 3-1 set the parameters for fluid	19
Table 3 -2 set the parameters for inlet	20
Table 3-3 the parameters for outlet	22
Table 3-4 the solution methods	25
Table 3-5 the solution controls	25
Table 3-6 Moments	27
Table 4.1 design points	34
Table 4.2 results of cases	35

List of Apreviation

Symbol	Name	Unit
NPSH	Net positive suction head	m
NPSHa	Net positive suction head cuale	m
NPSHr	Net positive suction head Required	m
RPM	Revolution per minute	-
CFD	Computational Fluid Dynamics	-
N	Number of blade	-
H	Head	M
g	Gravity	m/s^2
Q	Volume flow rate	m^3/s
ρ	Density	Kg/m^3
P_1	Supplied power	W
P_2	Shaft power	W
P_{hyd}	Hydraulic power	W
η_{hyd}	Hydraulic efficiency	-
η_{tot}	Total efficiency	-
P_{dyn}	Dynamic pressure	N/m^2
V	Velocity	m/s

CHAPTER I

INTRODUCTION

1-1 Background:

Cavitation is the formation of vapor cavities in a liquid i.e. small liquid-free zones ("bubbles" or "voids") – that are the consequence of forces acting upon the liquid. It occurs when a liquid is subjected to rapid changes of pressure that cause the formation of cavities where the pressure is relatively low. When subjected to higher pressure, the air bubble collapses leading to heavy shockwaves. Cavitation is a major cause of wear of internal parts. Collapsing bubbles that implode on metal surface cause stress through repeated implosion. This result in surface fatigue of the metal causing a type of wear also called "cavitation". The most common examples of this kind of wear are to pump impellers and bends where a sudden change in the direction of liquid occurs in closed cavity. [1]

Cavitation is usually divided into two classes of behaviour: inertial (or transient) cavitation and non-inertial cavitation.

Inertial cavitation is the process where a void or bubble in a liquid rapidly collapses, leading a shock wave. Inertial cavitation occurs in nature in the strikes of waterfall, river jet as well as in the vascular tissues of plants. In man-made objects, it can occur in control valves, pumps, propellers and impellers.

Non-inertial cavitation is the process in which a bubble in a fluid is forced to oscillate in size or shape due to some form of energy input, such as an acoustic field. Such cavitation is often employed in ultrasonic cleaning baths and can also be observed in pumps, propellers, etc.

Since the shock waves formed by collapse of the voids are strong enough to cause significant damage to moving parts, cavitation is usually an undesirable phenomenon. It is very often specifically avoided in the design of machines such as turbines or propellers, and eliminating cavitation is a major field in the study of fluid dynamics. However, it is sometimes useful and does not cause damage when the bubbles collapse away from metal surface

1-2 Problem Statement:

The term cavitation refers to the formation and collapse of vapor bubbles or cavities in a fluid, generally due to localized reductions in the dynamic pressure. The collapse of vapor cavities can produce extremely high pressures that frequently damage adjacent surfaces and cause material loss. Cavitation is a major problem for the operation of hydraulic equipment such as hydroelectric turbines, valves and fittings, flow meters, hydrofoils, pumps, and ship propellers. Cavitation frequently contributes to high maintenance and repair costs; revenue lost due to downtime and cost of replacement power; decreased operating efficiencies; and reduction of equipment service life. The most commonly used method for cavitation repair is the fusion process (i.e., welding).This method involves removing material from the damaged areas and filling the space by welding.

1-3 Objective:

The objective of this research is:

study the effect of cavitation phenomenon in a centrifugal pump performance

As the power required ,efficiency , volume flow rate , pressure outlet and the head

.

CHAPTER II

LITERATURE REVIEW

2-1 Background

This section describes the research work done by the investigators, with CFD as a numerical simulation tool to carry out effects of cavitation, in the performance of centrifugal pump for performance enhancement.

2-2. Performance Prediction Analysis

Salem presented information about the experimental work which has been done to monitor cavitation in centrifugal pump using acoustic signals. Analysis of head, efficiency, flow rate are shown graphically with the amplitude of the signals. Standard deviation methods are implemented for analyses the influence of flow rate in pump[3]. Jafarzadeh et al. presented a general three-dimensional simulation of turbulent fluid flow to predict velocity and pressure fields for a centrifugal pump. Investigation on the effect of number of blades on the efficiency head coefficient as the selection criteria showed that the impeller with 7 blades has the highest head coefficient when compared with 5 and 6-blade pumps at all range. Finally, it was observed also that the position of blades with respect to the tongue of volute has great effect on the start of the separation[4]. Houlin Liu et al. carried out impeller analysis by using different turbulence model. The cavitational study was done for standard *k-e* model and modified *k-e* model. Occurrence of vapor content at different pressure ranges is illustrated. The author compared the derived analysis data with experimental result and showed that the modified turbulence model has good agreement with the results[5]. Dazhuan carried out simulations for different transient flow for the determination of cavitation effect. The author found that the breakdown of pump performances is mainly due to the development of cavitation[6]. Weidong Shietal . investigated the flow feature in the design condition in a turbulence model. Head flow and back flows were analyzed. It was found that when the flow rate decreased below a certain value of the design flow rate, backflow occurred near the pressure surface of the pump impeller and cavitation is occurred take place at this stage. Three different pump models were investigated using the turbulence model[7]. Shalin Marathe et al. presented the effect of outlet blade angle on cavitation in centrifugal pump. Modeling of the centrifugal pump along with the different configuration of the impeller having different exit blade angles was carried out using Creo Parametric. Numerical simulation was carried out using ANSYS and standard k-ε turbulence model is implemented for the analysis purpose. From the results, the author found that the pump having low value of the blade exit angle will have less chances of getting affected by the cavitation phenomenon.[8]

2-3. Different Turbulence Model

Spyridon Kyparissis studied the effect of flow conditions and the blade leading edge angle on the cavitation and performance of a centrifugal pump. A pump test rig has been developed and rotational speed of 1200 rpm and flow rate of $35m^3/h$ were applied. Studying the effect of the blade leading edge angle on the cavitation, it was observed that as the blade leading edge angle increases, both the total head and the total efficiency increase for the examined operating conditions. It is noticed that as the net positive suction head decreases the total head decreases slowly for all the examined blade leading edge angle[9]. Li et al. developed cavitation model and algorithm for the prediction of the hydrodynamic performance of a centrifugal pump under cavitating flow condition. The developed centrifugal pump was analyzed using a single phase Reynolds averaged Navier stokes solver[9]. The mathematical model has good agreement with results Masao Oshima et al showed the location of cavitation in centrifugal impellers vary with the flow rate. This study pointed out that higher suction performance does not always cause a higher initial recirculation flow. The minimum continuous flow should be determined by taking all the factors that affecting the pump operation into account, including the energy received by the inlet reverse flow. In addition, inlet blade angle was the controlling factor for altering the initial inlet reverse flow during pump design[10]. Xianwu Luo et. al. analysed the 3-D turbulent flow inside those pumps using on k-ε turbulence model and cavitation model. It wasfound that impeller inlet geometry had an important influenceon performance improvement in the case of centrifugal pump.Favorite effects on performance improvement have been achieved by both extending the blade leading edge and applying much larger blade angle at impeller inlet. The extended leading edge have favorite effect for improving hydraulic performance, and the much larger blade angle at impeller inlet have favorite effect for improving cavitation performance for the test pump and uniform flow up stream of impeller inlet is helpful for improving cavitation performance of the pump[11].Pande et al used a general three dimensional simulation of turbulent fluid flow to predict velocity and pressure fields for a centrifugal pump. CFD was used to solve the governing equations of the flow field. This study described a finite volume method to examine the pump cavitation at the pressure drop region on the blade. It easy found that there was a significant spike in residuals, in the outlet pressure difference, and absolute pressure is low enough to induce cavitation[12]. Mouhammed khuladhar abbas et al. numerically studied the flow through the blade passage with CFD code, ANSYS in order to detect formation of cavitation in centrifugal pump. Head drop curve has a kneeshape that head remain constant while NPSH decreased and head will be rapidly decreased at critical point. The beginning of cavitation in the blade passage can be detected and shown in quality and quantity with numerical simulation. The inception of cavitation is take place on the suction surface where the leading edges meet the tip.

In pressure distribution plot shows that the cavitation zone expanding to the trailing edge especially in super cavitation case. Author concluded that the available NPSH of the system must be equal to or greater than the NPSH required by the centrifugal pump in order to avoid cavitation difficulties.[12]

2-4. Various Flow Conditions

Myung Jin Kim et al. accurately predicted cavitation of a centrifugal pump in various flow conditions. In this study, numerical analysis was compared with experimental results modeled on a small industrial centrifugal pump for reliable prediction on cavitation of a centrifugal pump. To improve validity of the numerical analysis, transient analysis was conducted on the calculated domain of full type geometry. The numerical analysis from the results wasconsidered to be a reliable prediction of cavitation. Ragavendramuttali et al[13]. analyzed the flow through a centrifugal pump using commercial CFD package. CFD analysis was carried out at design and off design condition and was reported. The simulation results were obtained at different operating speed with different mass flow rates for transportation of fluid. The Simulation was performed by using turbulent modeling k-ε and cavitation analysis, pressurecontours and velocity vector contour were predicted. The performance results were satisfactorily matching with test data[14]. Li newly developed cavitation model and algorithm for the prediction of the hydrodynamic performance of a centrifugal pump under cavitating flow condition is presented. The cavitation model is implemented in an RANSflow solver method. The liquid or vapour interface is tracked and obtained by an iteration procedure between the flow field computation and the interface updating. The capability of the model to capture some important phenomena of cavitation flows in a centrifugal pump impeller is demonstrated. The iterations of the cavity shape updating are quite stable the cavity size, thus the blockage of the channel, increases dramatically with the decrease in the cavitation number and sodoes the loss of the head[14] riseLiu Houlin et al. stated the blade number of impeller is an important design parameter of pumps, which affects the characteristics of pump heavily. The model pump has a design specific speed and an impeller with 5 blades. The blade number is varied with the casing and other geometric parameters keep constant. The inner flow fields and characteristics of the centrifugal pumps with different blade number are simulated and predicted in non cavitation and cavitation conditions by using commercial code FLUENT.The impellers with different blade number are made by using rapid prototyping, and their characteristics are tested in an open loop. The comparison between prediction values and experimental results indicates that the prediction results are satisfied. With the increase of blade number, the head of the model pumps increases too, the variable regulation of efficiency and cavitation characteristics are complicated, but there are optimum values of blade number for each one[15].Chakra borty et al. shows a general three dimensional simulation of turbulent fluid flow to predict velocity and pressure fields for a centrifugal pump. ANSYS is used to solve the governing equations of the flow field. This study describes a method to examine the pump cavitation at the pressure drop region on the blade. The method is based on an analysis of the blade number variations in the pump suction and discharge in a simple pump model. There is a significant spike in residuals, in part due to the outlet pressure difference and absolute pressure is low enough to induce cavitation[16]. Chakraborty et al. evaluated the performance of impellers with the same outlet diameter having different blade numbers for centrifugal pumps. Centrifugal pump with impeller blades 5, 6 and 7 was prepared and its efficiency at 3000 rpm was assessed by FLUENT 6.3 software. The numerical analysis revealed that with the increase of the head ,blade number, the head and static pressure of the model become greater, but the efficiency of centrifugal pump varies with number of blades.[16]

2-5 Computational fluid dynamics:

Computational Fluid Dynamics is a very powerful engineering tool, enabling a wide variety of flow situations to be simulated, reducing the amount of testing required, increasing understanding and accelerating development. It can be applied to a very wide range of applications and this breadth of application means that personnel from a wide range of different backgrounds come into contact with CFD; be they managers, engineers (mechanical, chemical, biomedical, civil or even electronic) or people involved in sales or marketing. The use of CFD jargon can therefore be particularly frustrating.[17]

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.[18]

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. [19]

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", 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 book.[19]

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.this group developed a variety of numerical methods to simulate transient two-dimensional fluid flows, such as Particle-in-cell method [6] Fluid-in-cell method (Gentry, Martin and Daly, [7]Vorticity stream function method [8] and Marker-and-cell method (Harlow and Welch. 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. Smith of Douglas Aircraft in . 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 . panel method for airfoil design

2-6 Impeller:

The blades of the rotating impeller transfer energy to the fluid there by increasing pressure and velocity. The fluid is sucked into the impeller at the impeller eye and flows through the impeller channels formed by the blades between the shroud and hub, The design of the impeller depends on the requirements for pressure, flow and application. The impeller is the primary component determining the pump performance. Pumps variants are often created only by modifying the impeller.

2-7 pump Performance:

2-7-1 Pressure:

Pressure (p) is an expression of force per unit area and is split into static and dynamic pressure. The sum of the two pressures is the total pressure .Static pressure is measured with a pressure gauge, and the measurement of static pressure must always be done in static fluid or through a pressure tap mounted perpendicular to the flow direction, Total pressure can be measured through a pressure tap with the opening facing the flow direction. The dynamic pressure can be found measuring the pressure difference between total pressure and static pressure. Such a combined pressure measurement can be performed using a pitot tube. Dynamic pressure is a function of the fluid velocity. The dynamic pressure can be calculated with the following formula, where the velocity (V) is measured and the fluid density (ρ) is know:

$P_{dyn} = 0.5\ \rho\ .\ V^2$ (Pa)

Pressure is defined in two different ways: absolute pressure or relative pressure. Absolute pressure refers to the absolute zero, and absolute pressure can thus only be a positive number. Relative pressure refers to the pressure of the surroundings. A positive relative pressure means that the pressure is above the barometric pressure, and a negative relative pressure means that the pressure is below the barometric pressure .The absolute and relative definition is also known from temperature measurement where the absolute temperature is measured in Kelvin [K] and the relative temperature is measured in Celsius [°C]. The temperature measured in Kelvin is always positive and refers to the absolute zero. In contrast, the temperature in Celsius refers to water’s freezing point at 273.15K and can

Therefore be negative .The barometric pressure is measured as absolute pressure. The barometric pressure is affected by the weather and altitude. The conversion from relative pressure to absolute pressure is done by adding the current barometric to the measured relative pressure In practice, static pressure is measured by means of three different types :

• An absolute pressure gauge, such as a barometer, measures pressure relative to absolute zero.

• An standard pressure gauge measures the pressure relative to the atmospherich pressure. This type of pressure gauge is the most commonly used.

• A differential pressure gauge measures the pressure difference between the two pressure taps independent of the barometric pressure

2-7-2 pressure gauges:

• An absolute pressure gauge, such as a barometer, measures pressure relative to absolute zero.

• An standard pressure gauge measures the pressure relative to the atmospherich pressure. This type of pressure gauge is the most commonly used.

• A differential pressure gauge measures the pressure difference between the two pressure taps independent of the barometric pressure.

2-7-3 NPSH, Net Positive Suction Head:

NPSH is a term describing conditions related to cavitation, which is undesired and harmful.Cavitation is the creation of vapour bubbles in areas where the pressure locally drops to the fluid vapour pressure. The extent of cavitation depends on how low the pressure is in the pump. Cavitation generally lowers the head and causes noise and vibration .Cavitation first occurs at the point in the pump where the pressure is lowest, which is most often at the blade edge at the impeller inlet, The NPSH value is absolute and always positive. NPSH is stated in meter [m]like the head. Hence, it is not necessary to take the density of different fluids into account because NPSH is stated in meters [m].Distinction is made between two different NPSH values: NPSHR and NPSHA.

NPSHA stands for NPSH Available and is an expression of how close the fluid in the suction pipe is to vapourisation.

2-7-**3 Head:**

The different performance curves are introduced on the following pages.

A QH curve or pump curve shows the head (H) as a function of the flow (Q). The flow (Q) is the rate of fluid going through the pump. The flow is generally stated in cubic metre per hour [m^3/h] but at insertion into formulas cubic metre per secon [m^3/s] is used.

The Q-H curve for a given pump can be determined using the setup

The pump is started and runs with constant speed.(Q) equals 0 and (H) reaches its highest value when the valve is completely closed. The valve is gradually opened and as(Q) increases(H) decreases.(H) is the height of the fluid column in the open pipe after the pump.

2-7-4 Power :

The power curves show the energy transfer rate as a function of flow. The power is given in Watt [W]. Distinction is made between

Three kinds of power:

• Supplied power from external electricity source to the motor and controller (P_1)

• Shaft power transferred from the motor to the shaft (P_2)

• Hydraulic power transferred from the impeller to the fluid (P_{hyd})

The power consumption depends on the fluid density. The power curves are generally based on a standard fluid with a density of 1000 kg/m^3 which corresponds to water at 4°C. Hence, power measured on fluids with another density must be converted.

In the data sheet (P_1) is normally stated for integrated products, while(P_2) is typically stated for pumps sold with a standard motor.

2-7-5 Hydraulic power :

The hydraulic power(P_{hyd}) is the power transferred from the pump to the fluid. As seen from the following formula, the hydraulic power is calculated based on flow, head and density:

$P_{hyd} = \rho . g . H . Q.$ [W]

2-7-6 Efficiency:

The total efficiency (η_{tot}) is the ratio between hydraulic power and supplied power.

The hydraulic efficiency refers to (P_2) , whereas the total efficiency refers to(P_1) **[2].**

CHAPTER III

METHODOLOGY

This chapter is interested in how to cavitation effect in centrifugal pump by using Computational fluid dynamics (ANSYS FLUENT FLOW) by applying reduceing the exit pressure from (350000 to 100000) Pa in six cases to obtain for values of volume flow rate ,head from program and calculate the power ,iffiecincy from the equations and then compare with it.

3-1 **Power Calculation** :

$$P_{hyd} = N \,.\, H \,.\rho \,.g \,.Q \quad (W) \qquad (3.1)$$

Where:

P_{hyd} :hydraulic power (W)

N: number of blade

H: head (m)

ρ :density of water (1000kg/ m^3)

Q: volume flow rate (m^3/s)

g =9.81m/s^2

$$\text{shaft power } (P_2) = N * \text{Torque} * \text{spead} \quad (W) \qquad (3.2)$$

$$\text{hydraulic effiecincy} = P_{hyd} / P_2 \qquad (3.3)$$

$$\text{total effiecincy} = P_{hyd} / P_1 \qquad (3.4)$$

Where : P_1 is supplied power from external electricity source to the motor and controller ,this power obtain from table

After that plotted the relationship between the fraction and **(** the power required ,efficiency , volume flow rate ,and the head

3-3 Program ANSYS(fluent flow) steps :

Setup and Solution:

Step 1 Impeller :

1. Start the 3D version of ANSYS FLUENT.Outlet- inlet
2. Read the mesh file, centrifugal pump.msh.

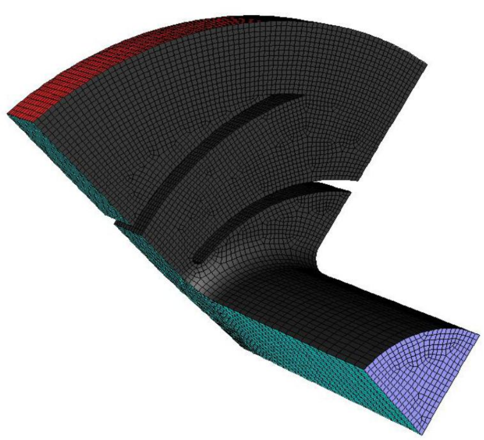


Figure 3.1 model impeller

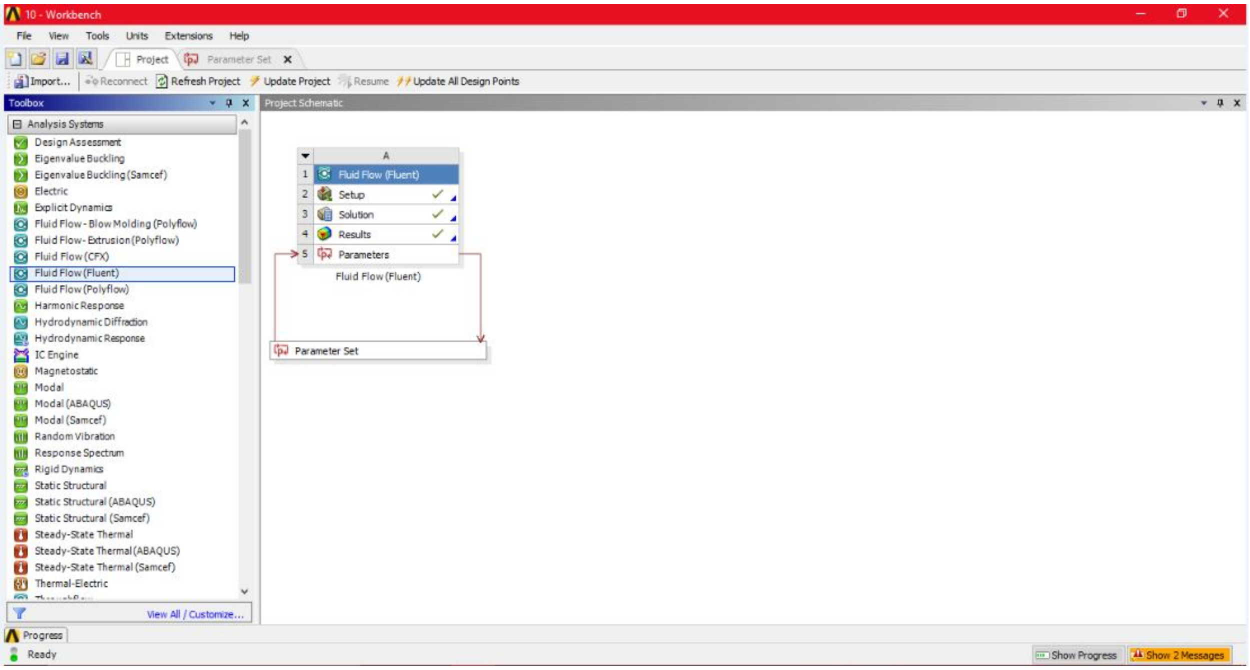


Figure 3.2 project schematic

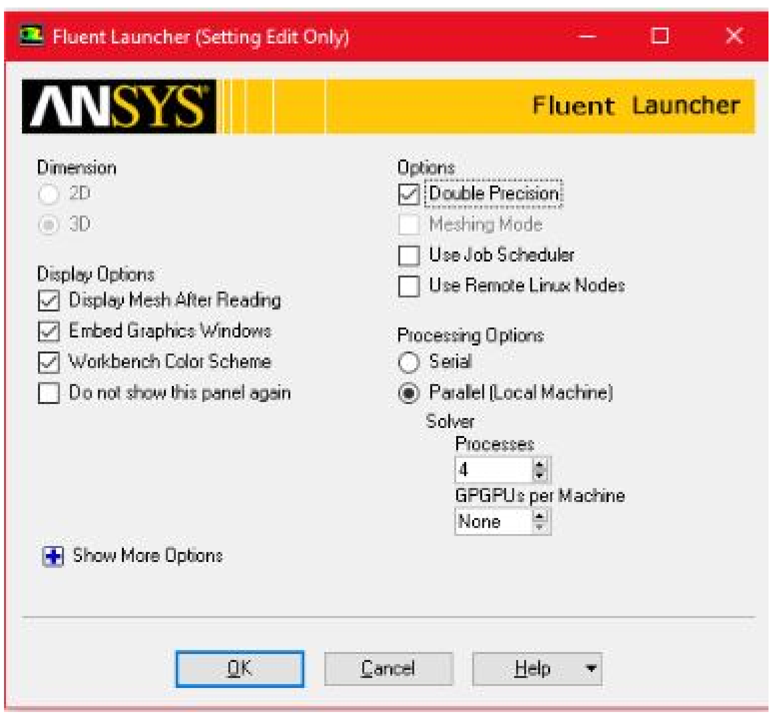


Figure 3.3 Fluent launcher

Step 2: Models

1. Retain the default solver general parameters (3D, pressure-based solver, absolute velocity formulation).

2. Enable the Realizable(k- e)model with standard wall functions.

3. Enable the mixture multiphase model, and disable the Slip Velocity.

Step 3: Materials

1. Define the fluid material “water-liquid” with a density and viscosity set to 1000 kg/m^3 and 0.001 kg/m-s, respectively.

2. Define the fluid material water-vapor with a density and viscosity set to 0.01927 kg/m^3 and 8.8e+6 kg/m-s, respectively.

Step 4: Operating Conditions.

Set the Operating Pressure to 0 Pa.

Step 5: Cavitation Model Setup

1. Define the phases (Define-Phases…).

(a) Set phase 1 (primary phase) as water-liquid.

(b) Set phase 2 (secondary phase) as water-vapor.

2. Define interaction between the phases (see Figure 3.4 and Figure 3.5).

(a) Click on Interaction... to open the Phase Interaction panel.

(b) Under the Mass tab, enable Cavitation.

(c) Click on the Edit… button to reveal the Cavitation Model panel.

(d) Set Vaporization Pressure to 3540 Pa and retain the default values for the other parameters. using the Schnerr-Sauer cavitation model

3. Check and display the grid.

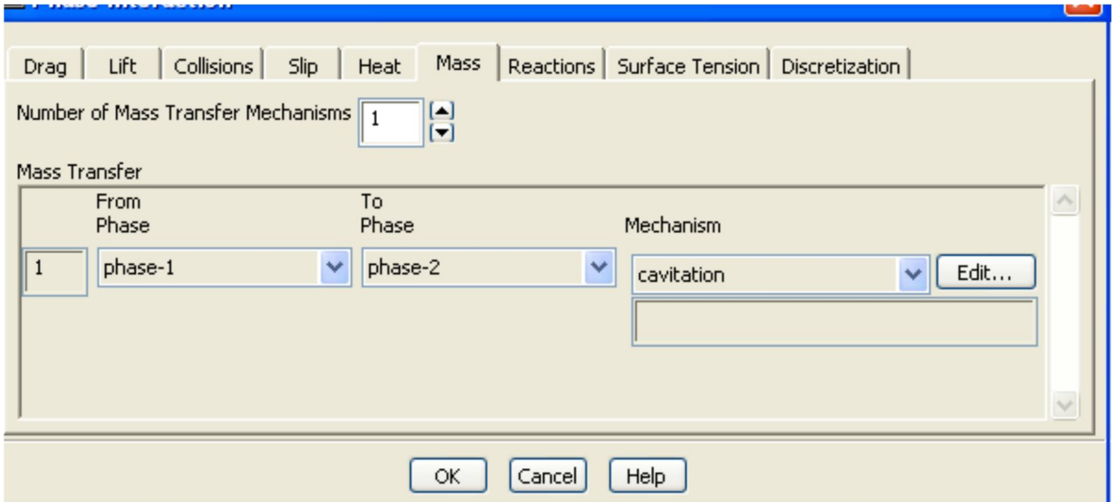


Figure 3.4: Phase Interaction panel.

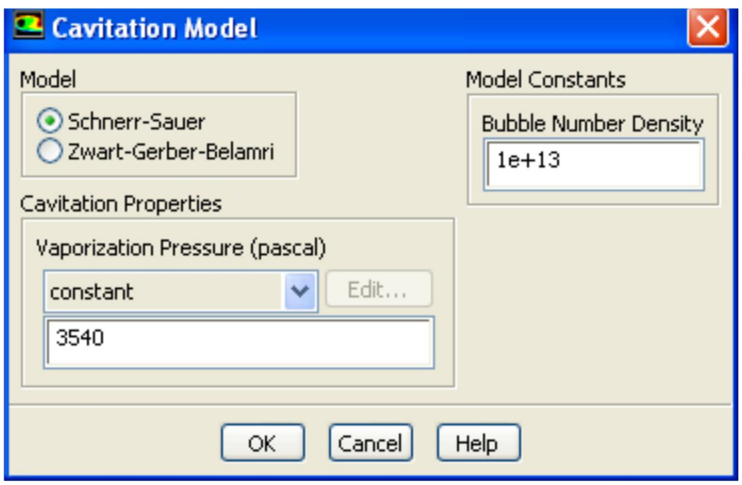


Figure 3-5: Cavitation Model panel.

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. Select “Frame Motion” and set the parameters for fluid as shown in the table 3.1

Table 3-1 set the parameters for fluid:

Parameter	Setting
Motion Type	Frame Motion
Rotation Axis Origin	(0, 0, 0)
Rotation-Axis Direction	X=0, Y=0, Z=1
Speed	2160 rpm

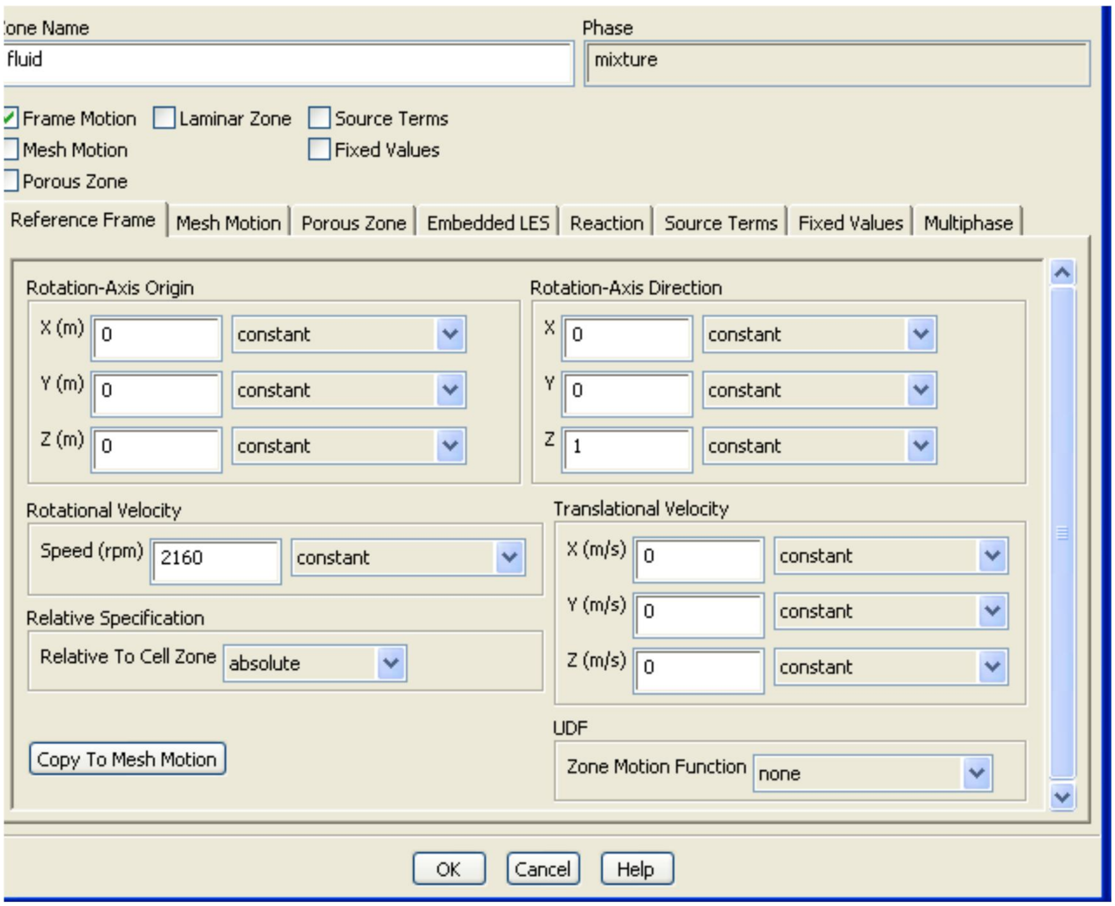


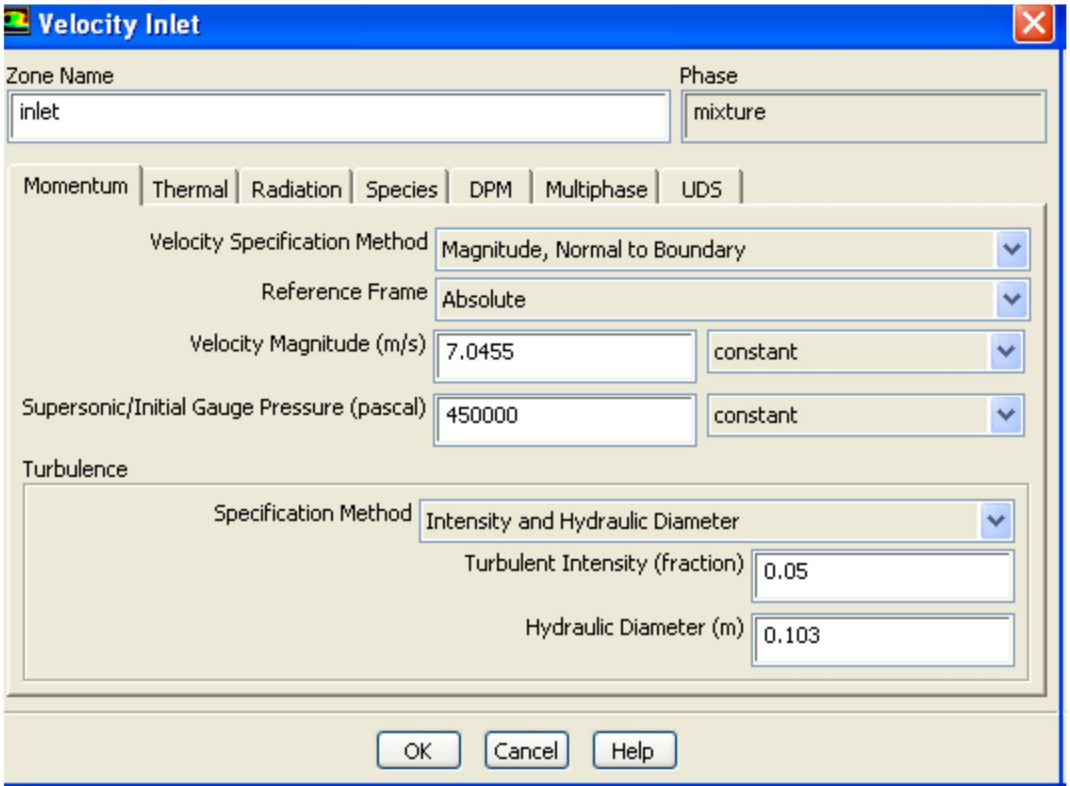
Figure3.6: Fluid panel for cell zone conditions illustrating new moving frame set up.

Step 7: Boundary Conditions :

Set the parameters for inlet as shown in the Table (3.2) and Figure (3.6)

Table 3.2 Set the parameters for inlet :

Parameter	Setting
Velocity magnitude	7.0455 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



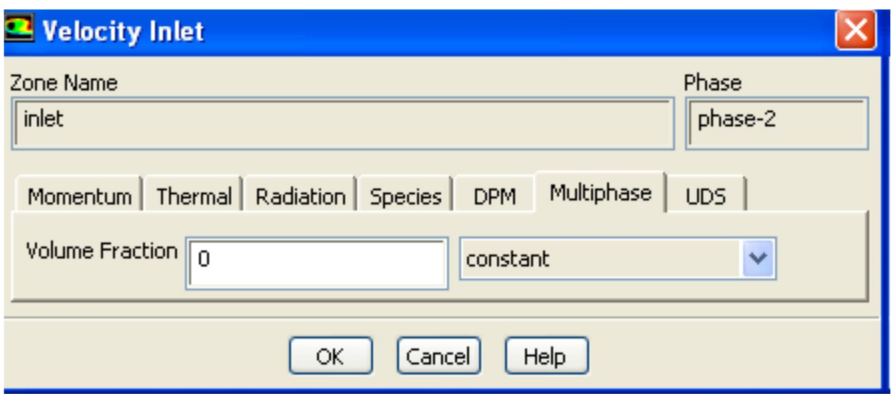


Figure3- 7: Velocity Inlet BC panels

Set the parameters for outlet as shown in the table below and Figure 3.7.

Table 3.3 Set the parameters for outlet :

Parameter	Setting
Gauge Pressure	450,000 Pa
Turbulence Specification Method	Intensity and Viscosity Ratio
Backflow Turbulence Intensity	5%
Backflow Phase-2 Volume Fraction	**0**

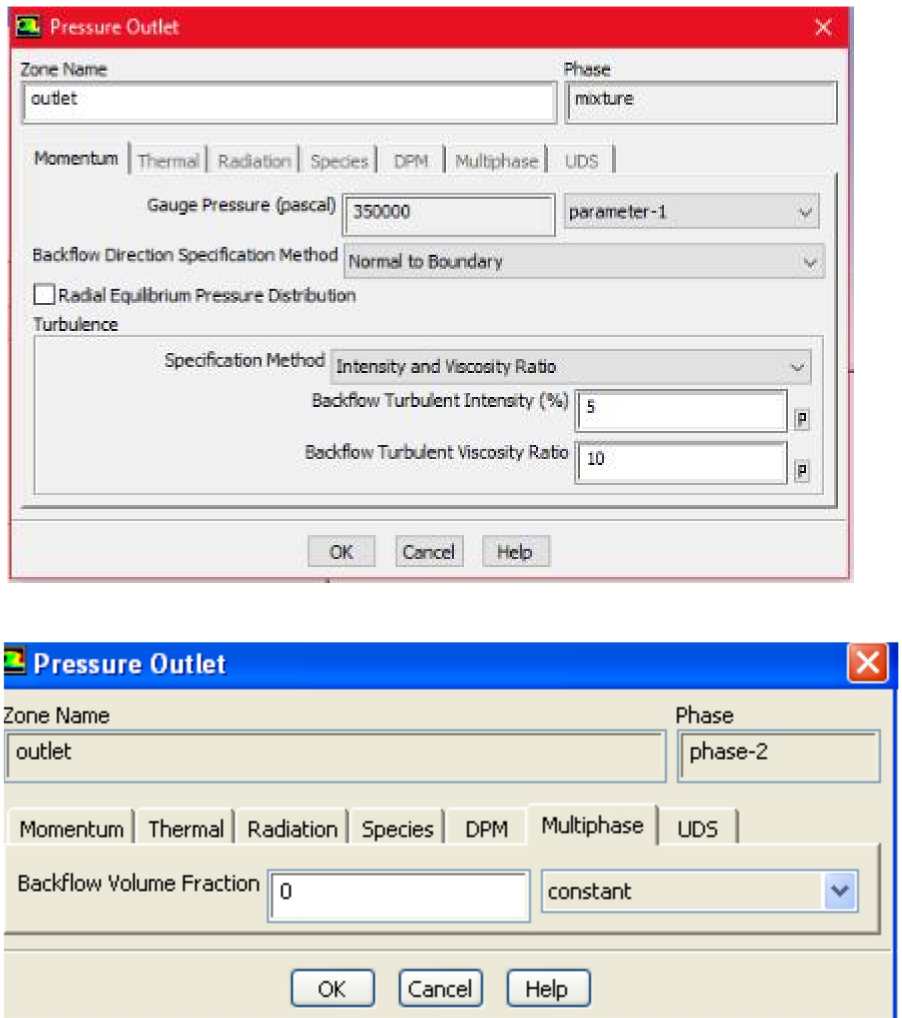


Figure 3.8: Pressure Outlet BC panels

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

4. Set the parameters for rotating walls (blade, hub, and shroud) as illustrated in Figure 3.9

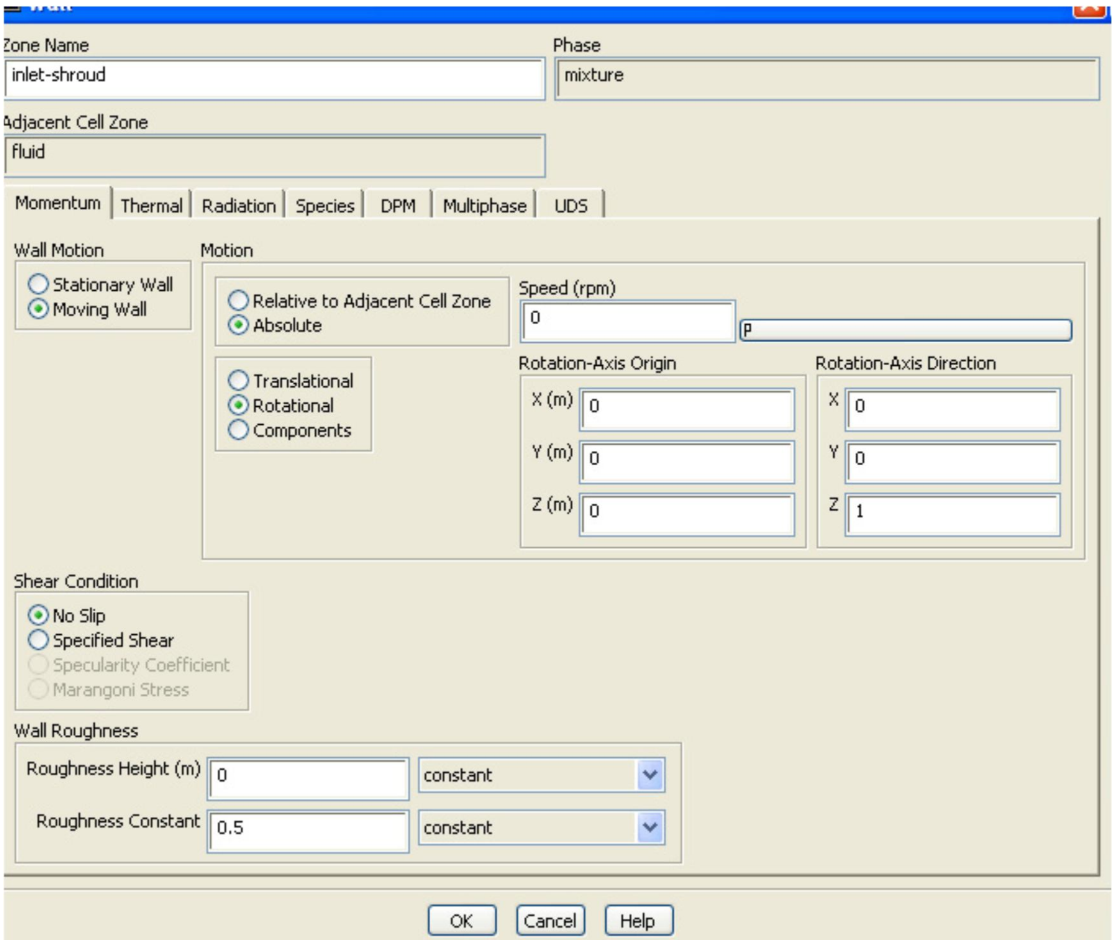


Figure 3-9: Rotating wall BC set up.

5. Set the parameters for non-rotating walls (blade, hub, and shroud) as shown in Figure3.10

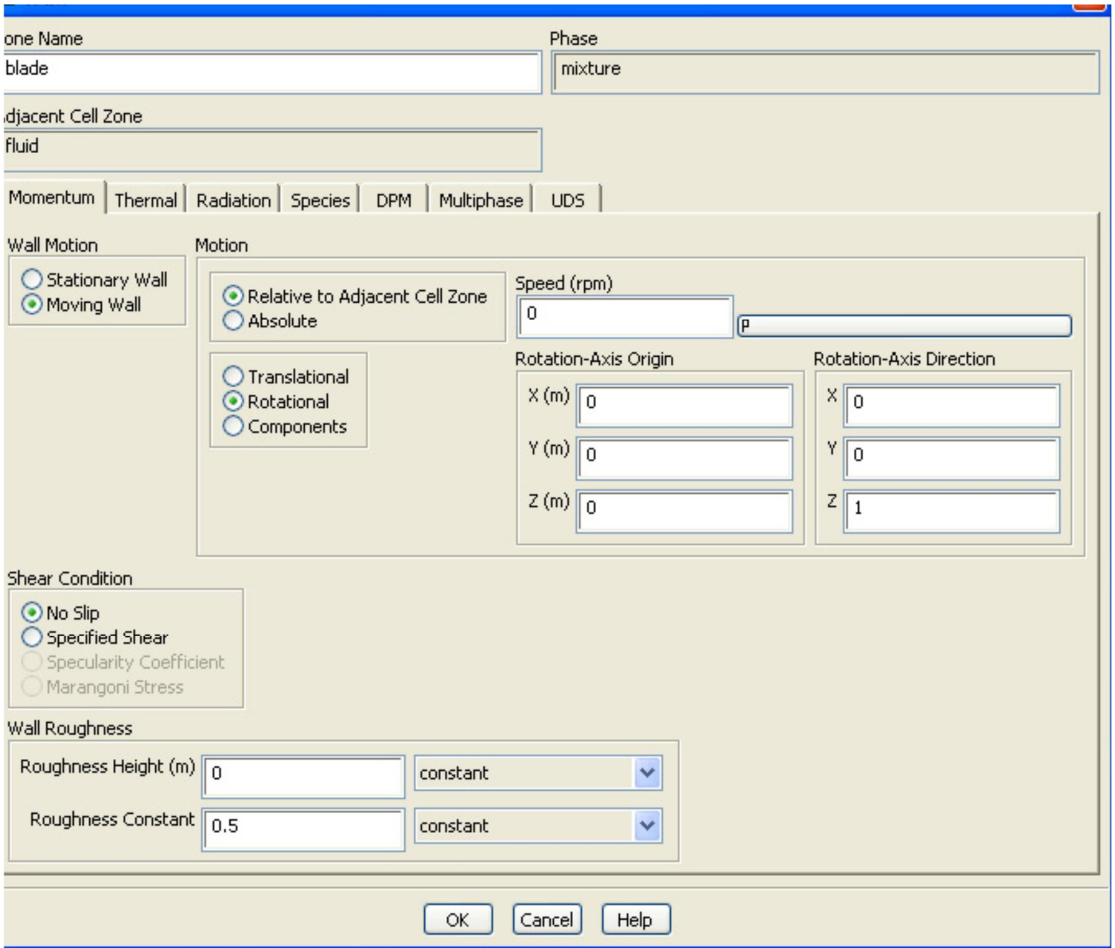


Figure3- 10: Non-rotating wall BC set up.

Step 8: Solver Discretization, Controls, and Monitors

Set the solution methods under Solve Methods according to Table 3.4

Table 3.4 Set the solution methods

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 below:

Table 3.5 the 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
Turbulent Viscosity	0.1

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

Step 9: Solution

1. Initialize the solution using values computed from the inlet values. Select Absolute for the Reference Frame.

2. Solve for 1500 iterations.

At 1500 iterations the monitor indicates that the inlet static pressure is no longer changing significantly with iteration (Figure 3.10). This, along with the residual history (Figure 3. 11) suggest the solution is well converged.

3. Save the case and data files (centrif-pump.cas.gz and centrif-pump.dat.gz).

3-4 Calculation :

Case 1 : exit pressure =350000 Pa

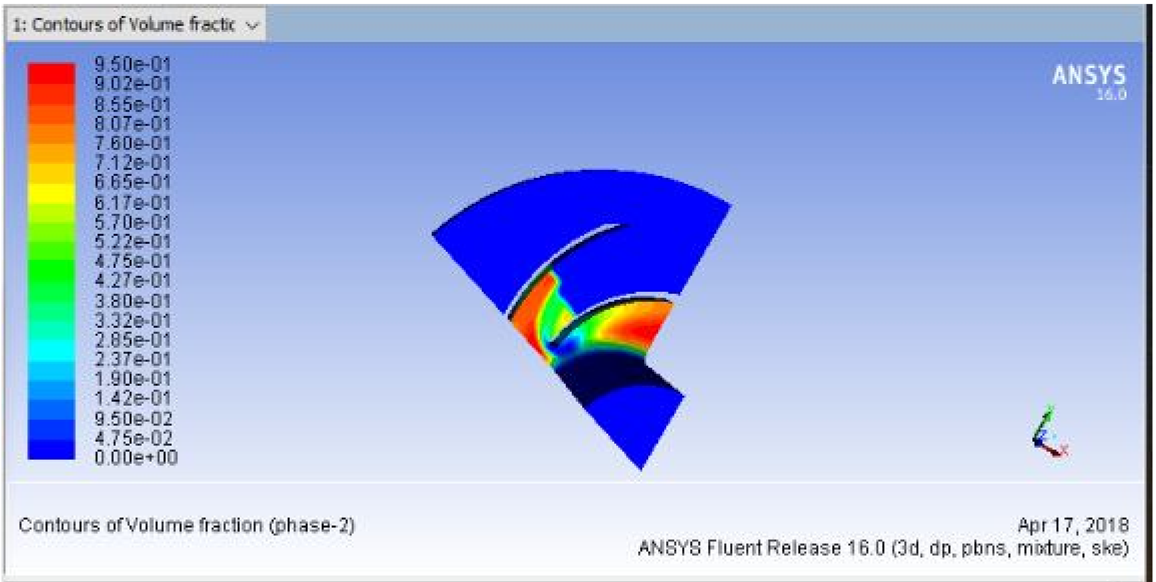


Figure 3.1 volume fraction at exit pressure 350000 pa

Volume fraction =0.9499022

Q =-0.012495754 m^3 /s

Table 3.6 Moments - Moment Center (0, 0, 0), Moment Axis (0, 0 ,1)

Zone	Pressure	vicous	Total
Blade	-23.205806	-0.45452017	-23.660326
Hub	2.620775e-05	-0.48799959	-0.48797338
Shroud	-0.00067329002	-0.30678488	-0.30745817
Net	-23.206453	-1.2493046	-24.455757

Moment =-24.455757 n-m

P_2 = N * Torque *Spead

P_2 = 5 *(-24.455757) *226.195 =27658.849773075 W

P_{hyd} = N *ρ *g *H*Q

P_{hyd} =5*1000*9.81*35.958 *0.012495754 =22039.2599103846 w

P_1 =42840.46 W

$\eta_{hyd} = P_{hyd} / P_2 = 0.79$

$\eta_{tot} = P_{hyd} / p_1 = 0.51$

Case 2: exit pressure = 30000 Pa

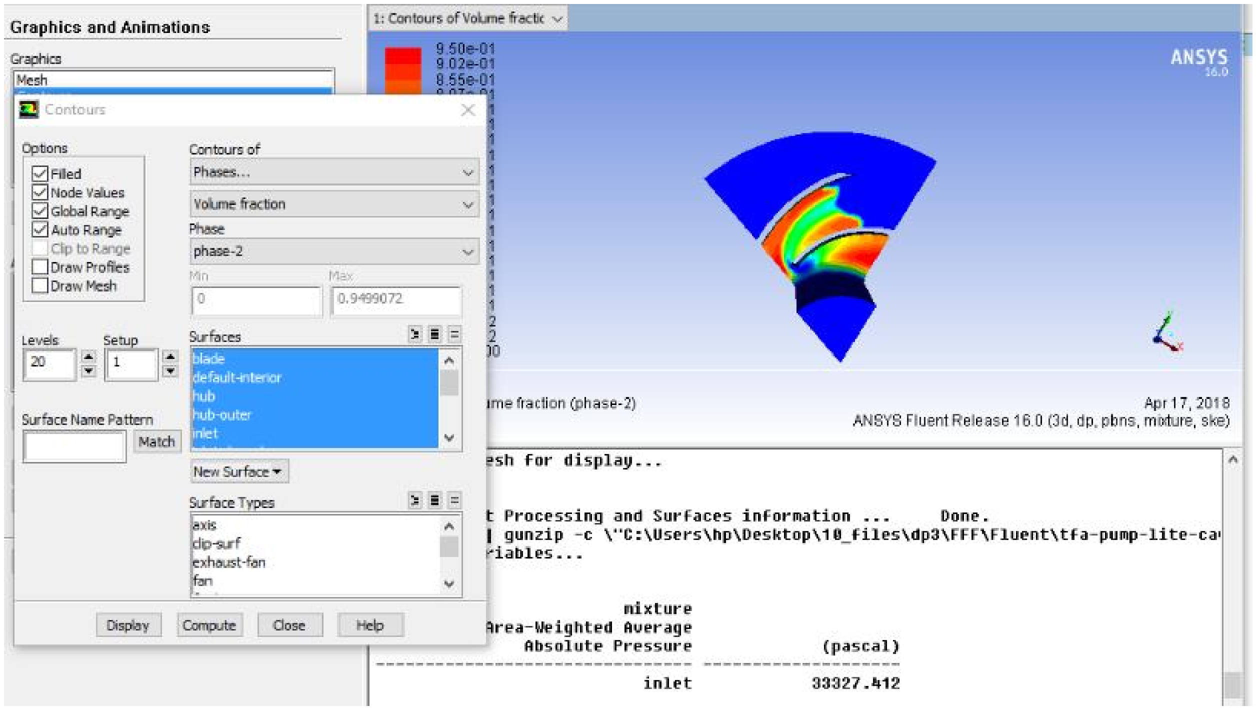


Figure 3.2 volume fraction at 300000 Pa

Volume fraction =0.9499072

Q=- 0.014109881 m^3/s

Moment=- 27.959621 N-m

H=28.938 m

P_2 =31621.632360475 W

P_{hyd} = 20027.6906693409 W

P_1 =40938.92 W

η_{hyd} =0.63 , η_{tot} = 0.48

Case 3: exit pressure =250000 Pa

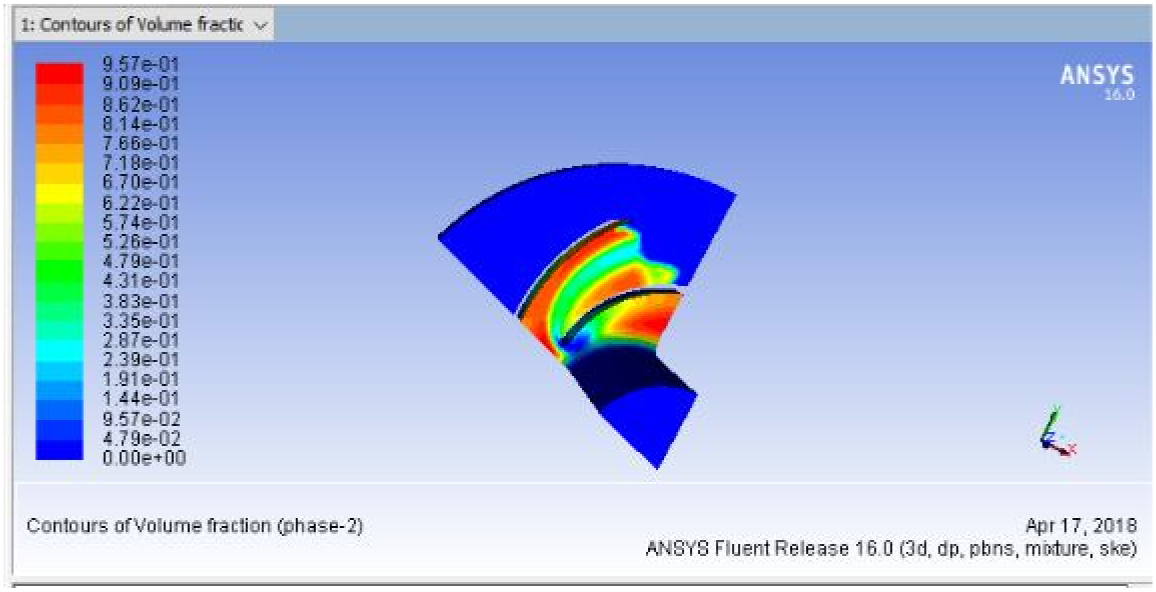


Figure 3.3 volume fraction at 250000 Pa

Volume fraction =0.9572629

Q=0.01006034 m^3/s

Moment =- 26.948519 N-m

H =21.484 m

P_2=30478.101276025 W

P_{hyd} =10601.487700668 W

P_1 = 19089.92 W

η_{hyd} =0.34

η_{tot} =0.55

Case 4 : exit pressure 200000 Pa

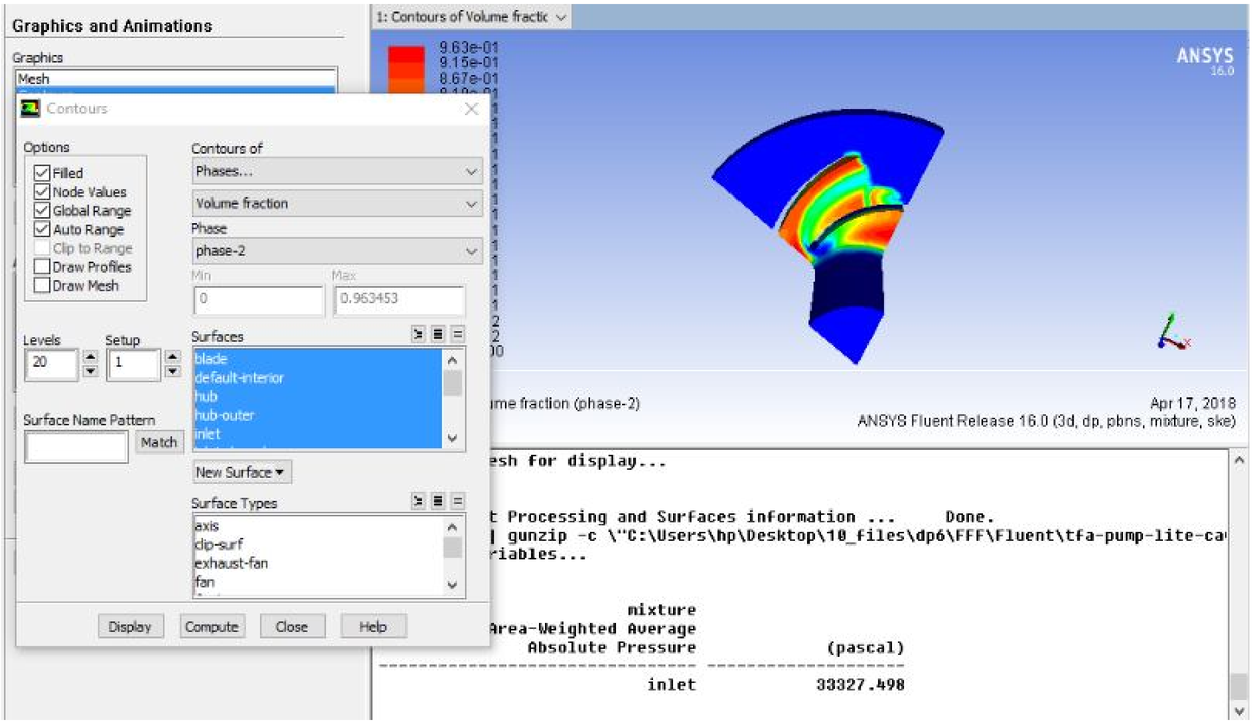


Figure 3.4 volume fraction at 200000 pa

Volume fraction =0.963453

Q =- 0.0078186877 m^3 /s

Moment =-21.180617 N-m

H =16.502 m

P_2=23961.52624475 W

P_{hyd} = 6328.62643606587 W

P_1=11931.2 W , η_{hyd} = 0.26 , η_{tot} =0.53

Case 5 : exit pressure 150000 pa

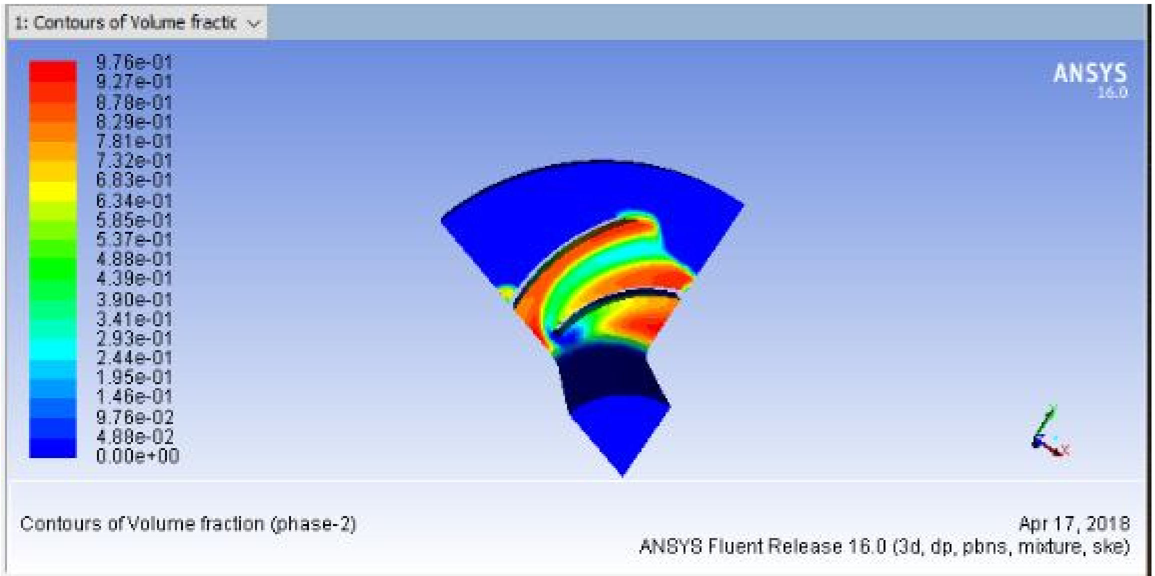


Figure 3.5 volume fraction at 150000 pa

Volume fraction =0.9757041

Q= -0.010807225 m^3/s

Moment = -18.200863 N-m

H=11.346 m

P_2 =20584.721031425 W

P_1 =10961.79 W

η_{hyd} =0.29

η_{tot} =0.54

Case 6 : exit pressure 100000 Pa

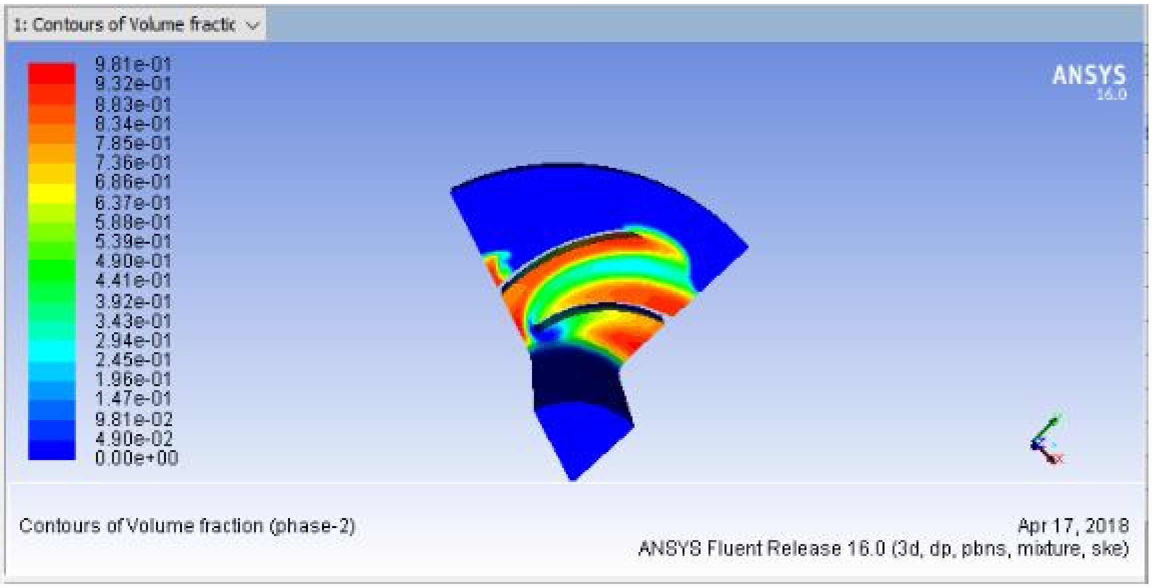


Figure 3.6 volume fraction at 100000 pa

Volume fraction =0.9806895

Q =- 0.012305322 m^3/s

Moment =- 12.097097 N-m

H =5.5406 m

P_2 =13681.514279575 W

P_{hyd} = 3344.17342994046 W

P_1 =8575.549 W

η_{hyd} =0.2

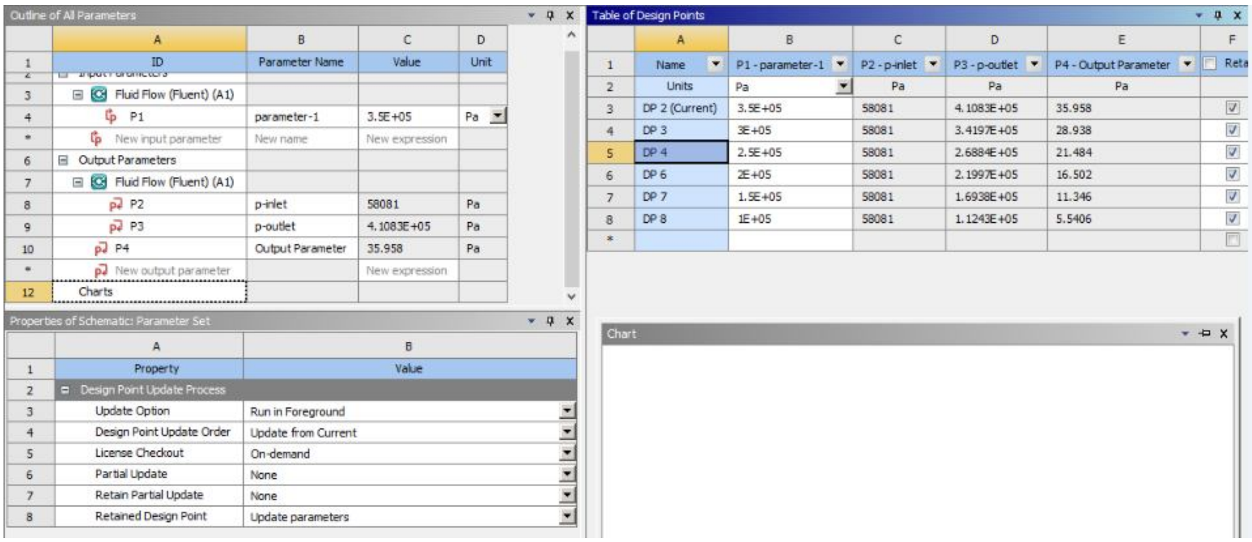
η_{tot} = 0.38

CHAPTER V

RESULTS AND DISCUSSIONS

4.1 Results:

Table 4.1 design points :



Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
3	Fluid Flow (Fluent) (A1)			
4	P1	parameter-1	3.5E+05	Pa
*	New input parameter	New name	New expression	
6	Output Parameters			
7	Fluid Flow (Fluent) (A1)			
8	P2	p-inlet	58081	Pa
9	P3	p-outlet	4.1083E+05	Pa
10	P4	Output Parameter	35.958	Pa
*	New output parameter		New expression	
12	Charts			

Properties of Schematic: Parameter Set

	A	B
1	Property	Value
2	Design Point Update Process	
3	Update Option	Run in Foreground
4	Design Point Update Order	Update from Current
5	License Checkout	On-demand
6	Partial Update	None
7	Retain Partial Update	None
8	Retained Design Point	Update parameters

Table of Design Points

	A	B	C	D	E	F
1	Name	P1 - parameter-1	P2 - p-inlet	P3 - p-outlet	P4 - Output Parameter	Reta
2	Units	Pa	Pa	Pa	Pa	
3	DP 2 (Current)	3.5E+05	58081	4.1083E+05	35.958	
4	DP 3	3E+05	58081	3.4197E+05	28.938	
5	DP 4	2.5E+05	58081	2.6884E+05	21.484	
6	DP 6	2E+05	58081	2.1997E+05	16.502	
7	DP 7	1.5E+05	58081	1.6938E+05	11.346	
8	DP 8	1E+05	58081	1.1243E+05	5.5406	
*						

Chart

Table 4.2 results of cases :

Case NO	Volume fraction	p_{exit} (kpa)	p_{inlet} (kpa)	p_{outlet} (kpa)	H (m)	Q (m^3/s)	P_2 (kw)	P_{hyd} (kw)	η_{hyd}	η_{tot}
1	0.9499022	350	58.081	410.830	35.958	44.98	27.658	22	0.79	0.51
2	0.9499072	300	58.081	341.070	28.938	50.79	31.6	20	0.63	0.48
3	0.9572629	250	58.081	268.840	21.484	36.21	30.4	10.6	0.34	0.55
4	0.963453	200	58.081	219.970	16.502	28.14	23.9	6.3	0.26	0.53
5	0.9757041	150	58.081	169.380	11.346	38.9	20.5	6	0.29	0.54
6	0.9806895	100	58.081	112.430	5.540	44.29	13.6	3.34	0.24	0.38

Frome Table 4.2 We note that all the values of (the power required ,efficiency , volume flow rate ,and the head) decrease with increase in cavitation.

.

4.2 Relation

Relation Between Volume Fraction and Head :

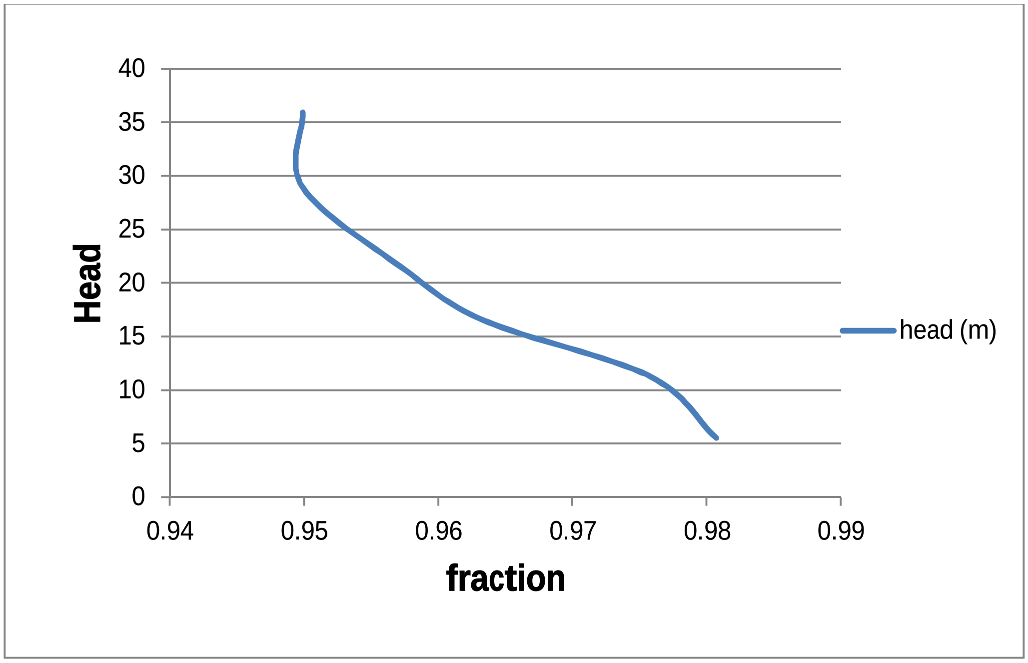


Figure 4.1 Volume Fraction and Head carve

It is found that the value of head decreasing from35.958 m to 5.540m

with increasing the Volume Fraction flow rate where the inlet pressure is constant

Relation between volume fraction and pressure outlet

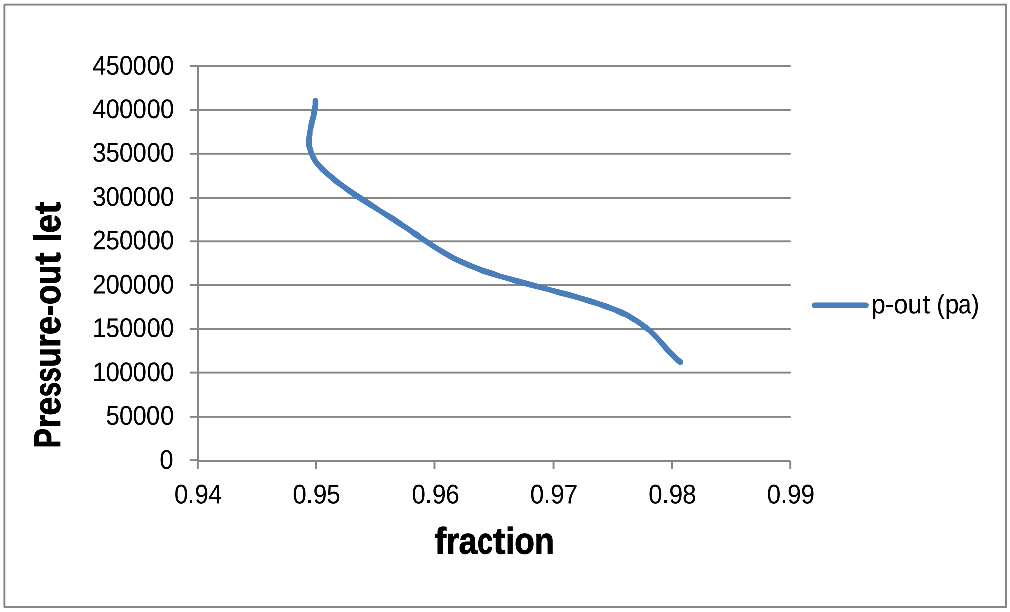


Figure 4.2 volume fraction and pressure outlet carve

It is found that the value of pressure out let decreasing with increasing the Volume Fraction where the inlet pressure is constant

Relation between volume fraction and volume flow rate

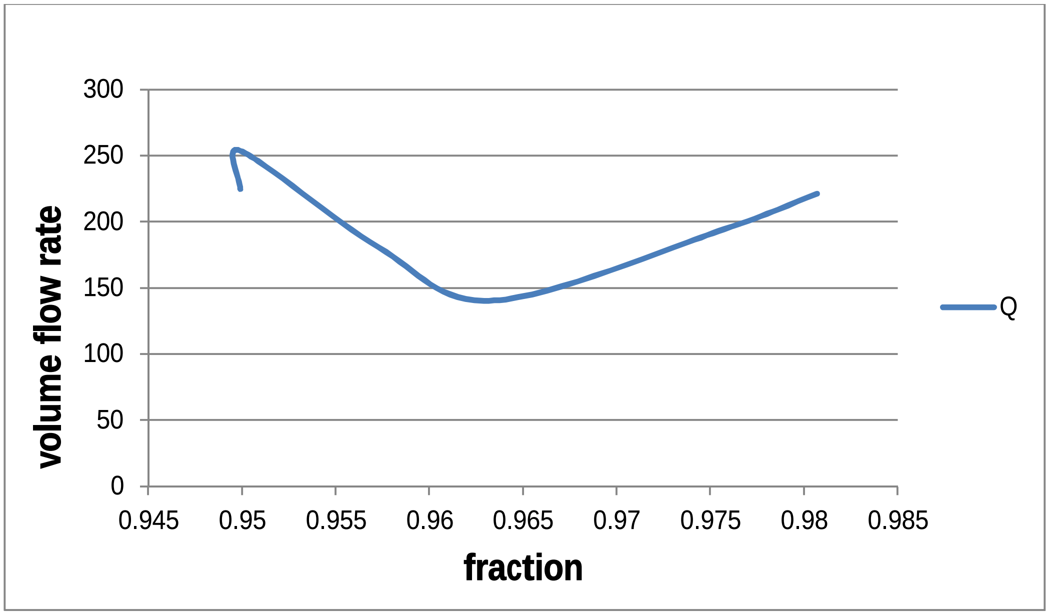


Figure 4.3 volume fraction and volume flow rate carve

It is found that the value of volume flow rate decreasing from 44.98m^3/s to 38.9m^3/s with increasing the Volume Fraction where the inlet pressure is constant it is found Error at at Volume Fraction 0.9499072 and 0.9806895

Relation between volume fraction and hydraulic efficiency:

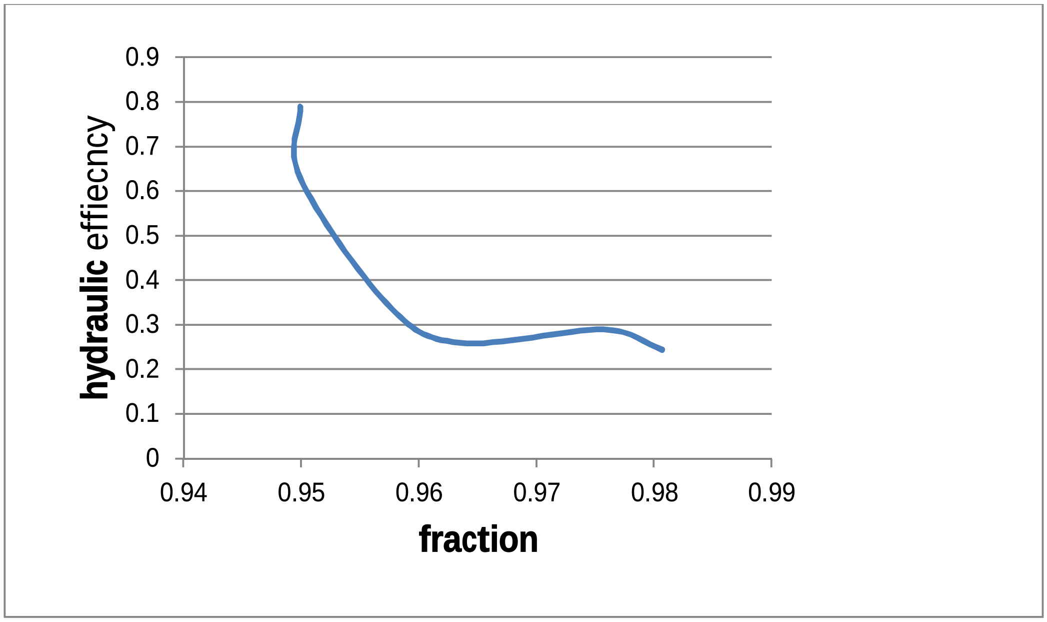


Figure 4.4 between volume fraction and hydraulic efficiency carve

It is found that the hydraulic efficiency decreasing with increasing the Volume Fraction where the inlet pressure is constant

Relation between volume fraction and total efficiency

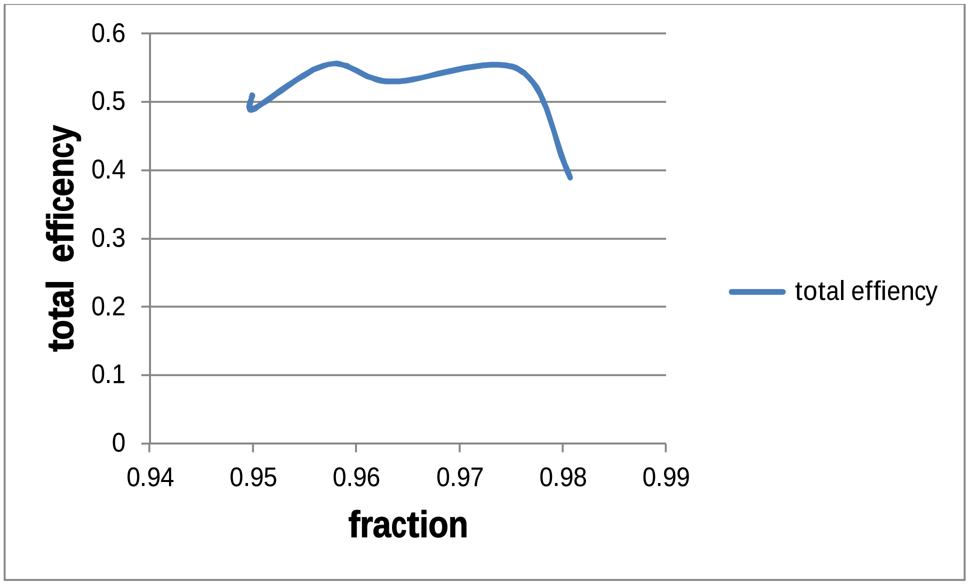


Figure 4.5 volume fraction and total efficiency curve

It is found that the Total efficiency decreasing with increasing the Volume Fraction where the inlet pressure is constant

Finally Table 4-2 indicate to influence in degrade of exit pressure that cuases grouth in the level of cavitation,

When reduce the exit pressure from 350,000 Pa. As the pressure is reduced, the cavitating region of the flow will expand to include more of the blade passage. Figures(4.1 to 4.5) indicate to reduce in all performance (head rise ,pressure outlet, volume flowrate, Hydraulic efficiency and total efficiency) .

5.1 Conclusions:

With decreasing cavitation level the pump head, flow rate and efficiency will decrease .

cavitation is serious undesirable phenomena which is affect the performance of the centrifugal pump and leads to machinery damage, which reduces life of pump.

cavitation region appear at the leading edge of impeller blades

5.2 Recomondations:

These results of enhancement of pump performance are given as responses for optimization method to find out the best combination of design parameters which will give the optimum performance of the pump.

1.the range of values for pressure exit used can be changed

2. Blade inlet angle and outlet angles to be changed

3. CFD software results compare with the actual tested results to get maximum head, Efficiency, Discharge and with low NPSH available.

References:

[1] Sudsuansee T, Nontakaew U, Tiaple Y (2011) Simulation of leading edge cavitation on bulb turbine.

[2] the Centrifugal pump research and technology European Association of Pump Manufacturers (1999), ”NPSH for rotordynamic pumps: a reference guide”, 1st edition

[3] A. Salem, and Al Hashmi,2013 “Statistical Analysis of Acoustic Signal for Cavitation Detection”,Int. J. Emerg. Technol. and Adv. Eng., vol.

[4] B. Jafarzadeh, A. Hajari, M.M. Alishahi and M.H. Akbari,2011 “The flow simulation of a low specific speed high speed centrifugal pump”, Appl. Math. Modell., vol. 35, pp. 242–249,

[5] Hou lin Liu, Dong xi Liu,2013“Application of modified *k*- ε model to predicting cavitating flow in centrifugal pump”,Water Sci. and Eng., vol. 6(3), pp. 331-339

[6] Dazhuan 2010,“Experimental study on hydrodynamic performance of cavitating centrifugal pump during transient operation”, J. Mech. Sci. and Technol., vol. 24, pp. 575-582.

[5] Weidong Shi, Zhimei Zhao,T. S. Lee, and S. H.Winoto 2014“Numerical Calculation on Cavitation Pressure Pulsation in Centrifugal Pump”, Adv. Mech. Eng., vol. 9, pp. 49–61

[6] Ashish J. Patel,Bhaumik B. Patel,2014 “Design and flow through cfd analysis of enclosed impeller”, Int. J. Eng. Res. Technol., vol. 3, pp. 4308-4314, .

[7] Myung Jin Km,Hyun Bae Jin,2012 “Study of prediction of cavitation for centrifugal pump”, World Academy of Science, Eng. and Technol., vol. 6, pp. 603-608,.
[8] Ragavendra muttali,Swetha agarwal,2014 “CFD simulation of centrifugal pump impeller using analysis CFX”, Int. J. Innov. Res. Sci. Eng. Techmol, vol. 3, pp.553-561,.
[9] Brennen, C.E., 1994, "Hydrodynamics of Pumps", Concepts NREC and Oxford University Press.
[10]Cernetic, J., Prezelj, J., and Cudina, M., 2008, "*Use of Noise and Vibration* Signal for Detection and Monitoring of Cavitation in Kinetic Pumps", University of Ljubljana, Faculty of Mechanical Engineering, Slovenia, pp. 2199-2204,
[11]Girdhar, P., and Moniz, O., 2005, first Edition, "Practical Centrifugal Pumps", IDC Technologies,

[12] Harihara, P.P., and Parlos, A.G., 2006, "Sensorless Detection of Cavitation in Centrifugal Pumps", ASME International Mechanical Engineering Congress and Exposition, November 5-10, Chicago, USA.
[13] Houlin, L., Yong, W., Shouqi, Y., Minggao, T., and Kai, W., 2010, "Effects of Blade Number on Characteristics of Centrifugal Pumps", Chinese Journal of Mechanical Engineering, Vol. 23, pp. 1-6.
[14] "Kubota Pump Hand Book", 1972, First Edition, Volume.

[15] Xianwu Luo, Wei Wei, Bin Zi, Wenchao Zhou 2013“Comparison of Cavitation prediction for a centrifugal pump with or without volute casing Published in “Journal of mechanical science & technology” 27 (6) 2013 Pages 1643-1648

[16] D. Somashekar, Dr. H.R. Purushothama, Siddganga institute of technology, Karnataka "Numerical simulation of Cavitation inception on radial flow impeller" 2012in "IOSR Journal of Mechanical & civil engineering, (IOSRJMCE)" in July-A
[17] Mohammad T Shervani and Navid shervani in2012 "Movement of location of tip vortex Cavitation along blade edge due to due to reduction of flow rate in axial flow pump" "World academy of science, engineering & technology" 61 IN
[18] Dazhuan Wu, Leqin Wang, Zongrui Hao, Zhifeng Li, Ziren Bao Published in2010 "Experimental study of hydrodynamic performance of cogitating centrifugal pump during transient operation "Journal of mechanical science & technology" 24 (2) Pages 575-582
[19] Zhou, Q Guo and Z W 2012 "Analysis of cavitation behavior in centrifugal pump" Wand published in "26th IAHR Symposium on Hydraulic machines & systems" .