



# **CFD ANALYSIS OF A CENTRIFUGAL PUMP**



**A PROJECT REPORT**

*Submitted by*

**ABISHEK CHANDRASEKHAR 1010104001**

**JAYANTHLS 1010104016**

**SRUTHI.M 1010104050**

*In partial fulfillment for the award of the degree of*

**BACHELOR OF ENGINEERING**

*In*

**MECHATRONICS ENGINEERING**

**KUMARAGURU COLLEGE OF TECHNOLOGY,  
COIMBATORE - 641 049**

**(An Autonomous Institution Affiliated to Anna University, Chennai)**

**APRIL -2014**

**KUMARAGURU COLLEGE OF TECHNOLOGY  
COIMBATORE - 641049**

**(An Autonomous Institution Affiliated to Anna University, Chennai)**

**BONAFIDE CERTIFICATE**

Certified that this project report “**CFD ANALYSIS OF A CENTRIFUGAL PUMP**” is the bonafide work of “**ABISHEK CHANDRASEKHAR, JAYANTHI, SRUTHI**” who carried out the project work under my supervision.

**SIGNATURE**

**Dr. S.A. PASUPATHY**  
**HEAD OF THE**  
**DEPARTMENT**  
Mechatronics Engineering,  
Kumaraguru College of  
Technology,  
Coimbatore – 49.

**SIGNATURE**

**Mr. T.SURESH**  
**SUPREVISOR**  
**Assistant Professor**  
Mechatronics Engineering,  
Kumaraguru College of  
Technology,  
Coimbatore – 49.

Submitted for the end semester examination held on **23/04/2014**.

**INTERNAL EXAMINER**

**EXTERNAL EXAMINER**

## ACKNOWLEDGMENT

We are greatly indebted to our Founder Chairman **PADMABUSHAN ARUTSELVAR** **Dr.N.MAHALINGAM**, BSc, FIE, Co-Chairman **Dr.B.K.KRISHNARAJ VANAVARAYAR**, BCom, BL and our Joint-Correspondent **SHRI SHANKAR VANAVARAYAR** for their dynamic leadership and inspiring guidance throughout our engineering course.

We have esteemed respect and gratitude to our beloved and respected Principal **Dr.R.S KUMAR**, MTech., PhD, FIE, for his invaluable support, motivation and guidance, and also for providing us all the necessary facilities required for carrying out this project work and complete the same in time.

We are very grateful to our respected Head of the Department, Mechatronics Engineering, **Dr.S.A.PASUPATHY**, PhD, for his constant and continuous motivation, review and cooperation throughout this project work.

We wish to record my profound happiness and gratitude to our Project Coordinator & Project Guide **Mr.T.Suresh**, Assistant Professor, Department of Mechatronics for his constant and continuous effort, guidance, valuable time and suggestions throughout this project work without whom this project could not have been completed in time successfully. We also thank our Department faculty for their guidance and the non-teaching staff for their support.

We sincerely thank M/S.PIONEER PRODUCTS for their technical guidance and support in making this project a success.

## **ABSTRACT**

This project deals with the study based on centrifugal pump. Initially, a study on materials was conducted using Hypermesh and Radioss. It was shown that Aluminium Bronze alloy was a suitable substitute for Cast Iron impellers. This was proven to improve the wear resistance of the impeller. The next objective was to improve the efficiency of the pump. The overall efficiency of the pump was calculated theoretically and this was verified by conducting practical tests. The efficiency of the pump is influenced by the design of the impeller. The project deals with analysing fluid flow inside the impeller and casing of a centrifugal pump using Computational Fluid Dynamics. This is done in order to determine the efficiency of a centrifugal pump through computer simulation and compare them to theoretical values obtained from design calculations. Six impeller models were designed and their assemblies were tested in ANSYS Fluent. The simulation results for one of the six cases were in accordance with the theoretical values. Thus, this project serves as a viable method of testing pump assemblies and also reduces cost of product development.

## Table of Contents

CHAPTER NO.	TITLE	PAGE NO.
	ABSTRACT	4
	LIST OF FIGURES	6
1	1.1 INTRODUCTION	7
	1.2 LITERATURE REVIEW	8
	1.3 PROBLEM STATEMENT	13
	1.3.1 OBJECTIVE	13
2	2.1 METHODOLOGY	14
	2.2 DESIGN PROCEDURE	16
3	3.1 MODELLING	23
	3.2 CFD ANALYSIS	29
	3.2.1 CFD ANALYSIS USING ACUSOLVE	29
	3.2.2 CFD ANALYSIS USING ANSYS	33
	3.2.3 SIMULATION RESULTS	35
	3.3 RESULTS	42
	3.3.1 FUTURE SCOPE	42
	3.4 REFERENCES	43

## LIST OF FIGURES

<b>FIGURE NO.</b>	<b>TITLE</b>	<b>PAGE NO.</b>
2.1	Velocity triangles	22
3.1	Inlet Blade Angle	23
3.2	Outlet Blade Angle	24
3.3	Volute Casing	24
3.4	Pump Casing Assembly	25
3.5	ASE300 Impeller- PIONEER PRODUCTS	26
3.6	Modified 9 Blade Impeller	26
3.7	Loft 6 Blade Impeller	27
3.8	Loft 9 Blade Impeller	277
3.9	Straight 6 Vane Impeller	288
3.10	Straight 9 Vane Impeller	288
3.11	Manual Mesh Correction of Casing	31
3.12	Manual Mesh Correction of Impeller	31
3.13	ANSYS Workbench Geometry	33
3.14	Meshed Pump Assembly	34
3.15	Velocity Contour for Impeller 1	35
3.16	Velocity Contour for Impeller 2	36
3.17	Fluid Velocity in Impeller 1	37
3.18	Pressure Distribution in Impeller 1	387
3.19	Mass Flow Rate for Impeller 1	399
3.20	Mass Flow Rate Variation for Impeller 1	40
3.21	Fluid Velocity in Impeller 1	40
3.22	Pressure Distribution Variation for Impeller 1	41
3.23	Velocity Distribution Variations in Impeller 1	41

# CHAPTER I

## 1.1 INTRODUCTION

Fluid mechanics is a major discipline in the field of engineering. Also, Pump industries are one of the oldest industries in manufacturing world. Pumps are used in a variety of different fields including home, industrial, agricultural etc. Based on their application pumps are classified into positive displacement pumps, gravity pumps, impulse pump, velocity pumps, steam pumps etc. The most commonly used pump is the centrifugal pumps. They find their application in variety of fields. In the first phase of the project, a material study was conducted. The effect of impeller wear was proven to be because of cast iron. Alternative materials such as Gunmetal, Steel, Aluminium, PET Plastic and Aluminium Bronze were studied. Simulations were carried out using Hypermesh and Radioss. Aluminium Bronze was found to be a suitable alternative to Cast Iron. This phase of the project deals with analysing the parameters of a centrifugal pump using CFD software. One of the factors that influence the efficiency of the pump is the vane angle of the impeller. Impellers with various vane angles were modelled and their assemblies were tested using CFD software.

## 1.2 LITERATURE REVIEW

### DESIGN STUDY

[1]**Khin Cho Thin, Mya Mya Khaing, and Khin Maung Aye** have modelled and analyzed the performance characteristics of a 1 H.P. centrifugal pump impeller with nine blades. In this study, the steps involved in designing the impeller are discussed in detail. They used The BERMAN method to design the impeller. The various parameters which decide the performance of the pump are explained. The first step involved in designing the impeller is calculating the specific speed of the pump,  $n_s$ . Specific speed is a standard dimensionless value which can be defined as the speed at which water is transferred at one gallon per minute at a head of one foot. The specific speed is given as

$$N_s = 3.65n(\sqrt{Q} \div (H)^{3/4})$$

Where,  $n$  = pump speed in rpm

$Q$  = flow rate in  $m^3/sec$

$H$  = head in meters.

The formulae for capacity, water and shaft power were described next. The overall efficiency depended on the motor efficiency, the pump efficiency and the hydraulic efficiency. The calculation procedures for the inlet diameter, outlet diameter, diameter of the eye of the impeller, hub diameter, shaft diameter, hub length, inlet and outlet flow angles, inlet and outlet areas were explained using the nine blade impeller. The velocity triangles for the inlet and outlet flow angles of the vanes were also illustrated. The performance of the pump was described by plotting the head generated by the pump over a range of flow rates. The formulae for various friction losses were explained and the friction losses were plotted against the flow rate. Finally, the theoretical head and the actual head values were compared.

[2]**ERIK DICK, JAN VIERENDEELS, SVEN SERBRUYNS AND JOHN VANDE VOORDE** followed a similar procedure in the performance prediction of their centrifugal pump using FLUENT, a CFD software. Two pumps were tested, one of low specific speed and the other of medium specific speed. It was discussed that FLUENT provides three methods of flow solving in turbo machinery, which are Multiple Reference Frame method (MRF), Mixing Plane (MP) method and Sliding Mesh (SM) method. In all three methods, the flow in the rotor was calculated using a rotating reference frame whereas the flow in the stator was calculated using an absolute reference frame. In the first two methods, steady flow equations were solved whereas in the sliding mesh method unsteady flow equations were solved. It was shown that the unsteady flow method was much more expensive when compared to the other two, but the steady flow methods were erroneous which makes them unsuitable for predicting flows. In the analysis part, the meshing was done differently for the two pumps. For the first pump, a two dimensional mesh was generated using GAMBIT. The mesh consisted of hexahedral and wedge cells. In the second pump, a more complex mesh was generated, which were made of tetrahedral pyramid like cells. They also included hexahedral and wedge cells. In the conclusion the basic reasons for the erroneous behaviours of steady flow methods was discussed, it was because of the inability of the methods to represent the fluid leaving the impeller. The distribution of pressure, absolute and radial velocities inside the volute was illustrated.

In his paper, [3]**Joe Evans** describes the basic workings of a centrifugal pump. The principles of operation (centrifugal force) are described first. Next, the relationship between the head and the peripheral velocities are explained. The relationship is given as

$$h=v^2/2g$$

where,  $v$ = velocity of the water as it leaves the impeller

$g$ = acceleration due to gravity

The flexibility of the centrifugal pump was also described. The performance of the pump is easily alterable by either changing its impeller diameter or its rotational speed. When the speed and the diameter are altered, the performances changes in accordance with three fundamental laws called "Laws of Affinity". They state that:

- Capacity varies directly as the change in speed or diameter
- Head varies as the square of change in speed or impeller diameter
- Brake Horse power varies as the cube of change in speed or impeller diameter.

Therefore, it can be inferred that although the head and capacity may increase by increasing the speed and the diameter, but the power required to operate such a pump is tripled. Thus, the diameter of the impeller and the change in operating speed are not advisable as means of increasing the efficiency of the pump. The suction conditions of the pump are also described here. It is seen that the head depends upon the suction pressure (in psi) and the specific gravity of the liquid (the s.g for water is 1.0). The relation is such that

$$\text{Head}=\text{PSI} \times 2.31/\text{Specific Gravity}$$

The NPSH (Net Positive Suction Head) is also explained as a function of a specific pump design.

### **Computational Fluid Dynamics:**

The unsteady flow of fluid in a Centrifugal pump was studied by [4]**K.W Cheah, T.S. Lee, and S.H Winoto**. A centrifugal pump with a twisted six blade impeller was chosen. The analysis was done by using a k- 2-equations turbulence model. Modelling was done using multiple frames of reference. A

tetrahedral mesh interface was created and the impeller and the volute casing were connected using the sliding mesh technique. The analysis showed a recirculation zone near the suction end. The flow discharged from the impeller into the volute was shown to be distorted and a spiral flow pattern was observed. [5]**Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H. Winoto** used the finite volume method in a commercial three dimensional Navier Stokes equation called CFX to simulate a turbulent model. Three pumps models were investigated, one with four straight blades and the other two with six twisted blades. The study showed that the twisted blade impellers were more efficient when compared to the straight blade impellers. The mathematical models used in the software were illustrated in the paper. The inlet boundary, the solid walls and the outlet boundary and their velocity flow rates were defined. The paper shows that if the flow rate decreases to a certain value below the design flow rate, backflow occurs, this causes the path to become narrower. [6]**S.Rajendran and Dr.K. Purushothaman** conducted similar experiments in the analysis of a centrifugal pump impeller using ANSYS-CFX, which was used to develop a three dimensional, fully turbulent model of the compressible flow across the geometry of an impeller. A similar six blade impeller was used for the study. The geometry and the mesh of the impeller domain was generated using Ansys workbench. The unstructured mesh which contained tetrahedral cells was used for parts of the impeller and volute. Around the blades structured hexahedral cells were generated to obtain better boundary layer details. The boundary layers defined were, the impeller was considered as a rotating frame of reference with a rotational speed. The rotating fluid was water at 25 deg C. And finally, the inlet static pressure and the outlet mass flow rate were given as boundary conditions. Finally the performance values were plotted in tables. The pressure distributions were illustrated. The head value was predicated for the design flow rate. Stress analysis of a centrifugal fan was performed. This was done using Finite Element method (FEM). The program was coded in

FORTTRAN. Triangular shell elements with six degrees of freedom were used to create a mesh of the fan. It was then found out that the element used in the program was a non-conforming element and for acceptable results, a finer mesh was needed. Based on Mindlin's plate theory Triangular and Quad elements were used to create a finer mesh. These elements were proven to give better results compared with the earlier elements. The same principle can be used to mesh the centrifugal impeller.

### **1.3 PROBLEM STATEMENT:**

The overall efficiency of the chosen pump is only 38%. The efficiency is influenced by the design of the impeller and the volute casing of the pump. Studies show that the design of the chosen impeller has lower efficiency compared to other impeller models.

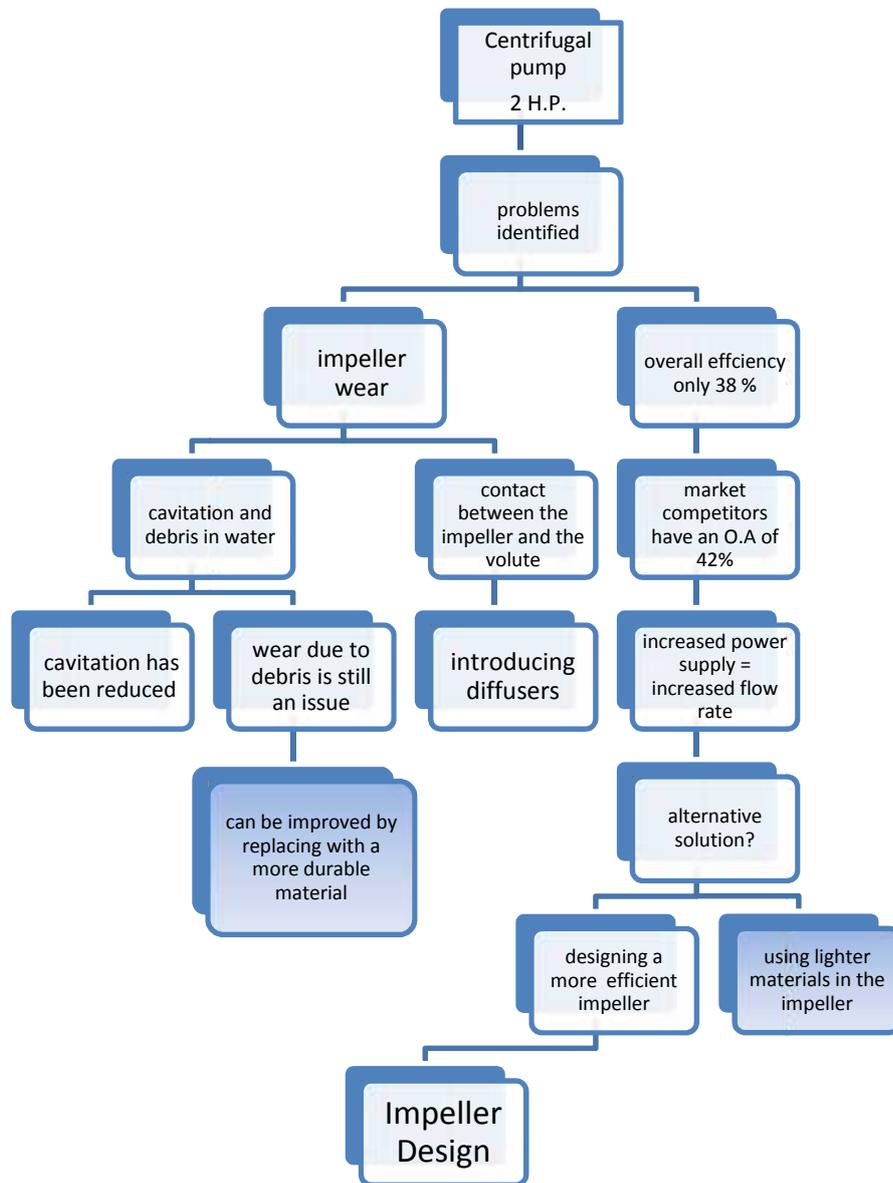
#### **1.3.1 OBJECTIVE:**

- To improve the overall efficiency of the centrifugal pump by modifying the blade angle of the impeller.
- To perform CFD analysis for centrifugal pump.

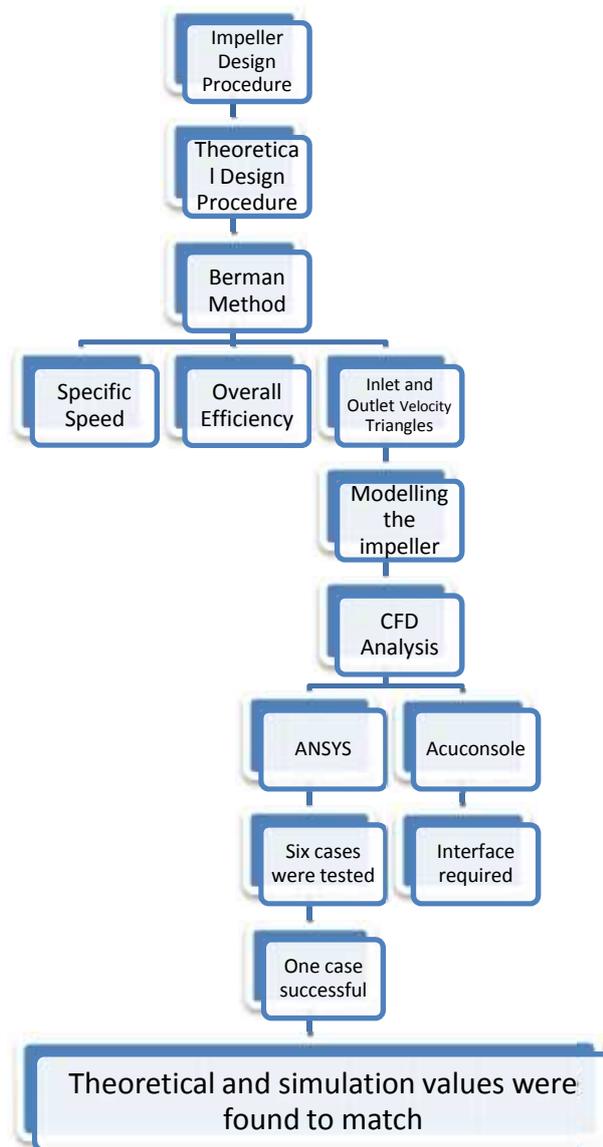
# CHAPTER II

## 2.1 METHODOLOGY

### BASIC METHODOLOGY



## DESIGN METHODOLOGY



## 2.2 DESIGN PROCEDURE

The Berman method [1] is used to design the centrifugal impeller. The various parameters involved and the steps to be followed are described below:

The specifications of the pump chosen are:

<i>Pipe size:</i>	80x80mm
<i>Rated voltage:</i>	230V
<i>Rated power:</i>	1.5kW/ 2.0H.P
<i>Rated head:</i>	10m
<i>Rated discharge:</i>	8.0 lps
<i>Rated Speed:</i>	2800 rpm
<i>Head Range:</i>	8 to 12 m
<i>Maximum current:</i>	12.0 A
<i>Overall Efficiency:</i>	38.00%
<i>No. Of blades:</i>	6
<i>Rated Frequency:</i>	50.0 Hz

The first step in the design process is to find out the specific speed of the pump.

“Specific speed is a standard dimensionless value. It is the speed at which one gallon of liquid is transferred per minute, at a head of one foot.” The value of specific speed depends on the pump speed, flow rate and the head value. It is given by the formula (1).

$$^{[1]}N_s = 3.65n(\sqrt{Q} \div (H)^{3/4}) \quad (1)$$

Where, n = pump speed in rpm

Q= flow rate in m<sup>3</sup>/sec

H= head in meters.

The specific speed was calculated for the chosen pump. It was found out to be

$$N_s = \mathbf{162.55}$$

The second step is to calculate the capacity of the pump. Capacity is given by

$$Q = AV \quad (2)$$

Where, A= area of the pipe

V= volume flow rate

Volume Flow rate is defined as the amount of liquid (water) pumped per unit time.

Then, the power inputs to the pump are calculated. The efficiency of the pump depends upon the amount of power required at the shaft. The overall efficiency is  $\eta_o$

$$^{[1]}\text{Overall efficiency, } \eta_o = \eta_m \times \eta_v \times \eta_r \quad (3)$$

Where,  $\eta_m$  = Motor efficiency

$\eta_v$  = Pump efficiency

$\eta_r$  = Hydraulic efficiency

*Hydraulic Efficiency:*

The hydraulic efficiency of the pump is given by the formula

$${}^{[1]}\eta_r = \frac{1-0.42}{(\log D_0 - 0.1772)^2} \quad (4)$$

Where,  $D_0$  = the diameter of the eye of the impeller

The value of  $D_0$  is substituted in (4).

For the chosen pump, the Hydraulic efficiency was calculated and was found to be,

$$\text{Hydraulic Efficiency } \eta_r = 85.52\%$$

The pump efficiency was calculated using the following procedure,

$$\text{Given; Head } H = 10\text{m} = 32.808 \text{ feet}$$

$$\text{Capacity} = 8.0 \text{ litres per second} = 126.802 \text{ gallons per minute}$$

$$\text{Specific gravity of water} = 1$$

$$\text{Weight of one gallon of water} = 8.33 \text{ lbs}$$

The horse power  $H.P._{out}$  is calculated using the formula

$$H.P._{out} = \frac{\text{head} \times \text{capacity}(\text{gpm}) \times 8.33 \left( \frac{\text{lbs}}{\text{gallon}} \right) \times 5.6}{33000 \text{ foot pounds/sec}} \quad (5)$$

The values were substituted in (5) and the output power of the pump was calculated. It was found to be

$$H.P._{out} = 1.050 \text{ H.P}$$

The input power from the motor is,

$$H.P._{in} = 2.0 \text{ H.P}$$

Therefore the efficiency was calculated as a ratio of the output power to the input power.

$$\text{Efficiency} = \frac{H.P \text{ out}}{H.P \text{ in}} = \frac{1.050}{2.0} = 0.522 \quad (6)$$

Therefore, the pump efficiency was calculated to be

$$\text{Pump efficiency } \eta_v = 52.2\%$$

The motor efficiency  $\eta_m$  is assumed to be 85% (as this is the general efficiency for an electric motor). The motor efficiency, (4), (5) and (6) are substituted in (3).

Therefore, the overall efficiency of the pump is calculated to be

$$\text{Overall efficiency } \eta_o = \eta_m \times \eta_v \times \eta_r$$

$$\eta_o = 0.85 \times 0.522 \times 0.855$$

$$\eta_o = 0.379 = 37.9 \%$$

The overall theoretical efficiency of the pump is **37.9 %**. This is found to match the measured value which is **38 %**.

*Velocity triangles:*

The efficiency of the pump can be improved by altering several parameters. One of which is the velocity angle of the impeller blade. The procedure of finding the fluid velocities at the inlet and the outlet of the impeller are as follows:

The first step is to calculate the inlet and the outlet area of the impeller.

The inlet area depends on the diameter of the eye of the impeller. The formula for calculating the inlet area is given by

$$\text{Inlet area } A_1 = 2 \quad r_1 b_1 \quad (7)$$

The eye of the impeller is 74mm. When this value is substituted in (7), the inlet area is

$$A_1 = 0.00325\text{m}^2$$

The outlet area depends upon the outer diameter and the width of the impeller at the outlet.

$$\text{Outlet area } A_2 = 2 r_2 b_2 \quad (8)$$

The values are substituted in (8) and the outlet area is found to be

$$A_2 = 0.00401\text{m}^2$$

Inlet angle:

The inlet angle of the vane was found out by following the steps below

$$\text{Tangential velocity of the impeller at the inlet } u_1 = R_1 \quad (9)$$

Where,  $\omega$  = angular velocity

$$R_1 = \text{Inlet radius}$$

Angular velocity is found out by

$$\omega = r_1 \times n \times 2 \pi / 60 \quad (10)$$

The angular velocity at the inlet is found by substituting (10) in (9)

$$= 293.06 \text{ rad/sec}$$

Therefore, the inlet velocity of the impeller was found to be

$$U_1 = 10.84 \text{ m/sec}$$

The inlet angle is found using the formula

$$\tan \alpha_1 = c_{1m}/U_1 \quad (11)$$

Where,  $c_{1m}$  = velocity of fluid along the radius

Substituting the values for inlet velocity and the velocity of fluid along the radius in (11), the inlet angle was calculated.

Therefore, the inlet angle was found to be

$$\alpha_1 = 12.782$$

The optimum value for the inlet angle of a six blade centrifugal impeller is supposed to be 17 degrees.

A similar triangle is constructed and the outlet velocities of the impeller are found.

The outlet area was first calculated in (8) and was found to be  $A_2 = 0.0042 \text{ m}^2$ .

The velocity  $c_{2m}$ , the velocity along the radius at the outlet was calculated,

$$c_{2m} = \frac{Q_{\text{impeller}}}{A_2} \quad (12)$$

$$c_{2m} = 1.995 \text{ m/s}$$

The tangential velocity  $u_2$  is calculated

$$U_2 = 18.756 \text{ m/s}$$

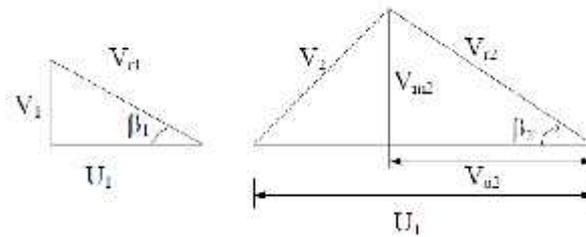
The blade angle was assumed to be between 15 degrees and 35 degrees. These values were substituted to find the outlet velocity.

$$W_2 = \frac{c2m}{\sin B2} \quad (13)$$

The velocity was calculated

$$W_2 = 3.503\text{m/s to } 7.71 \text{ m/s}$$

The velocity triangles are shown below in Figure 2.1



**Figure 2.1 Velocity triangles**

The velocity of fluid in the inlet and outlet pipe was calculated using the following equation

$$\text{Velocity } v = \frac{1.274 * q}{d^2} \quad (14)$$

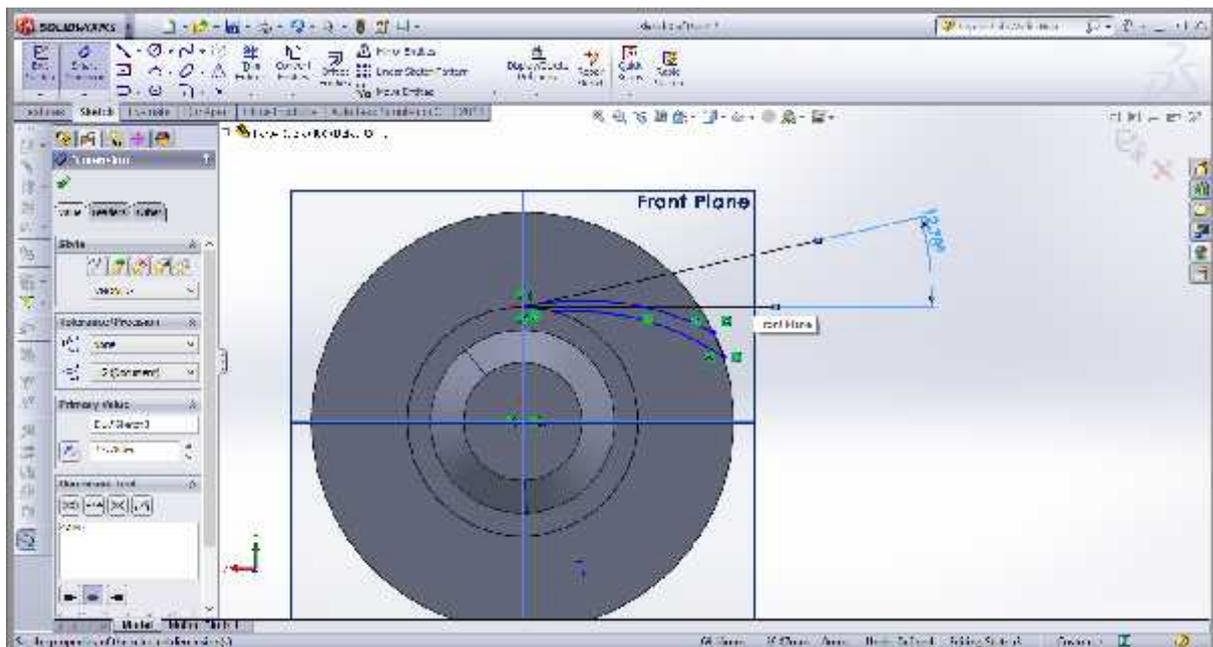
The inlet velocity was found to be **2.02m/s**

The outlet velocity was found to be **2.87m/s**.

## CHAPTER III

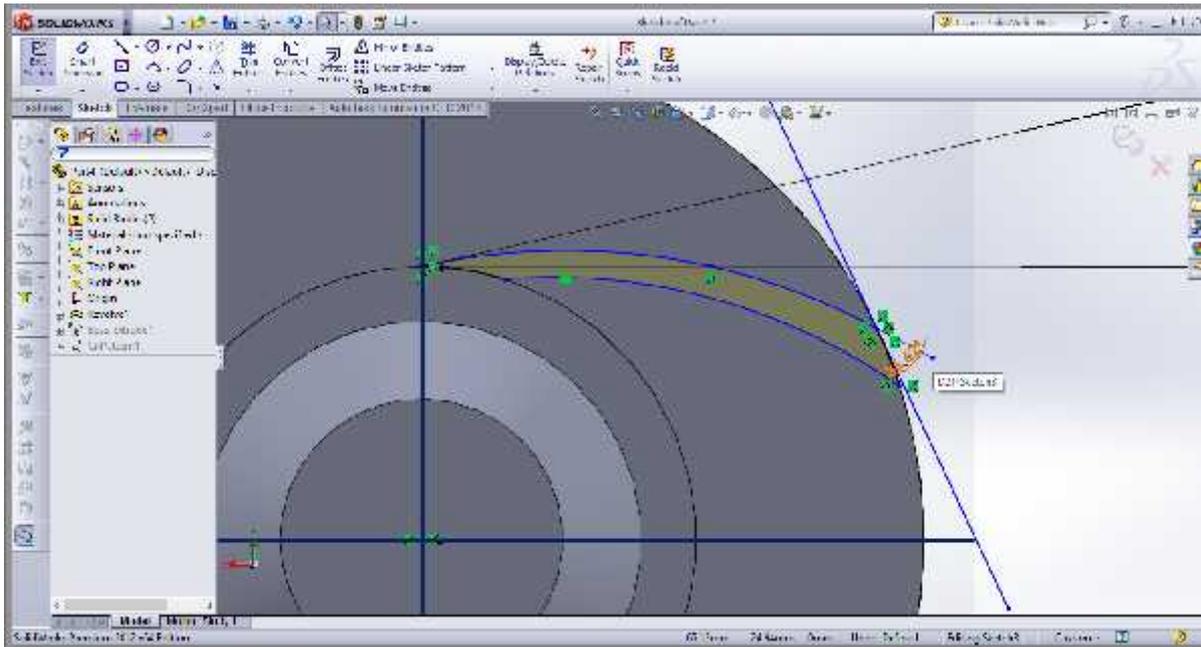
### 3.1 MODELLING

The model of the impeller was created using SolidWorks 12. The vanes were designed using the calculated vane angles as a reference. The dimensions of the impeller were measured from the actual component using vernier callipers. The impeller inlet vane angle of 12.78 degrees is shown below in Figure 3.1



**Figure 3.1 Inlet Blade Angle**

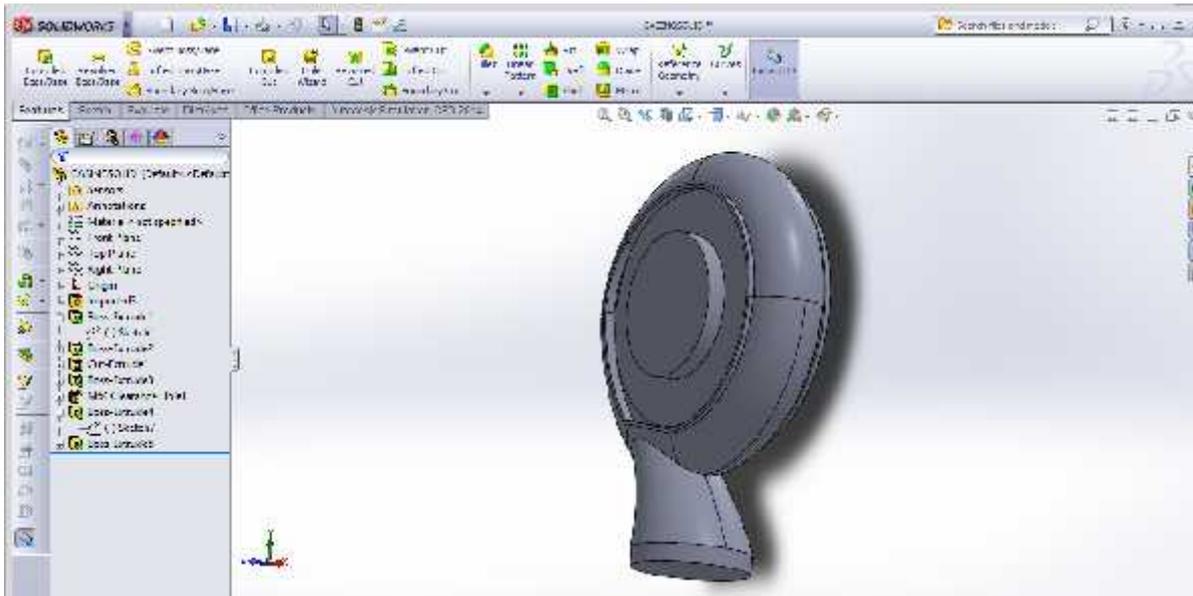
The vane is designed based on the inlet angle and the outlet angle. The tangent lines are drawn first, such that they are at an angle of 12.78 degrees. An arc is drawn such that they touch the eye and the outer diameter of the impeller. The outer arc is also drawn at an approximate angle of 17 degrees. The outlet vane angle is shown below in Figure 23.2.



**Figure 2.2 Outlet Blade Angle**

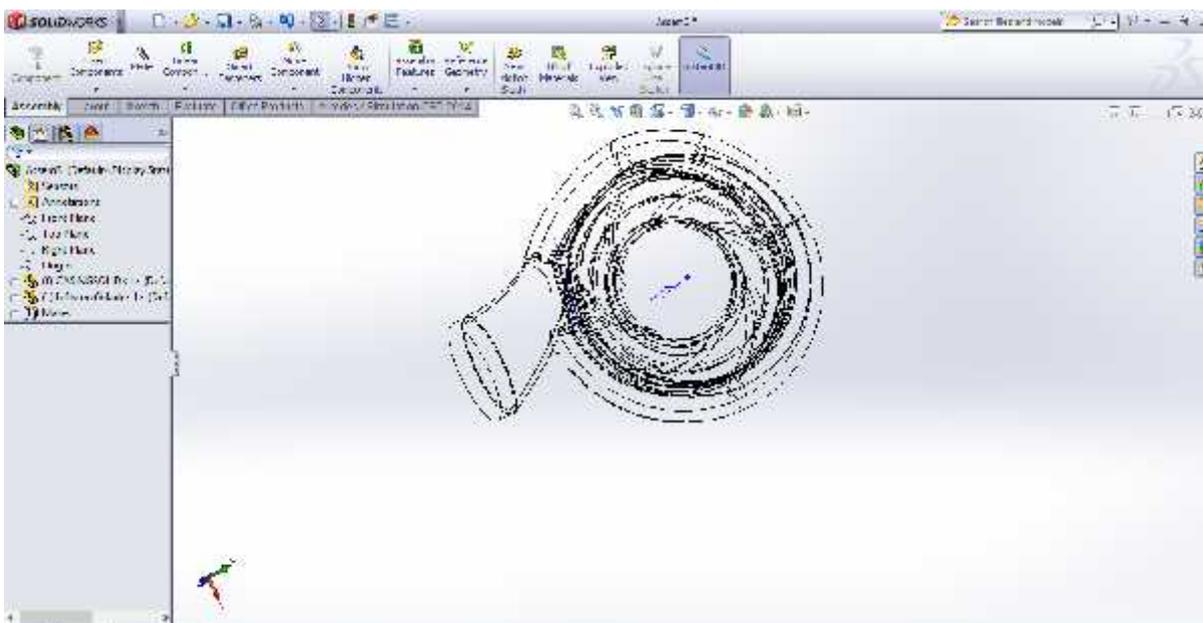
The volute casing is also designed using SolidWorks. The measurements were taken from the actual casing using vernier callipers. The tools used for creating the variable diameters at the outlet was the loft tool and the tool used for creating the variable arcs which define the curvature of the profile was created using the swept blend tool.

The inlet and the outlet were defined using extrude features. The model of the casing is given below in Figure 3.3.



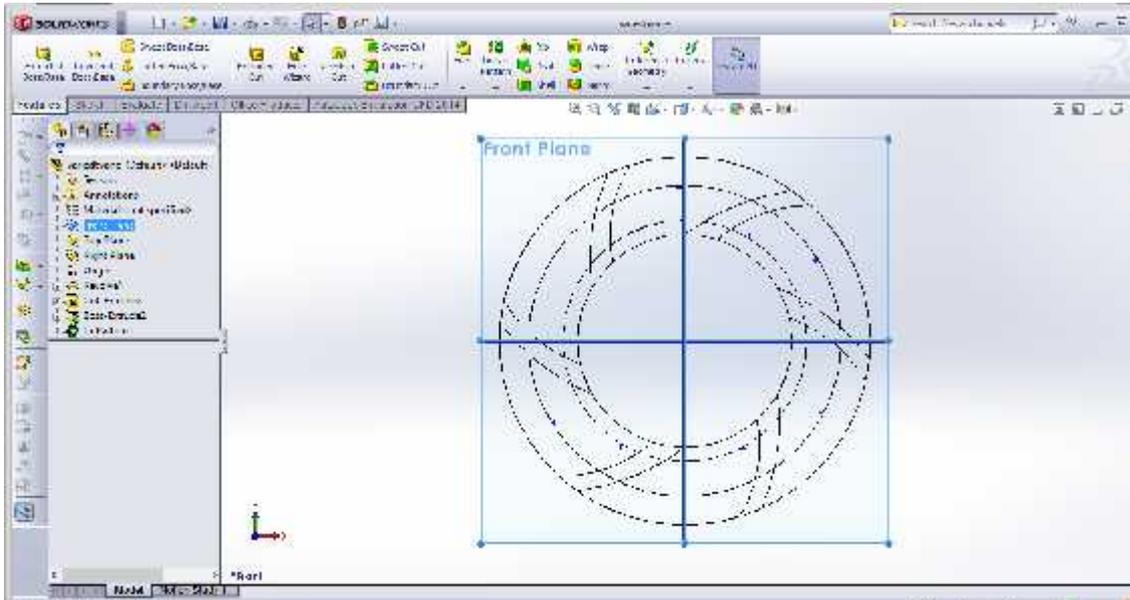
**Figure 3.3 Volute Casing**

The assembly of the casing and the impeller was done and is checked for interference. The assembly is saved in three formats. A Solid works assembly file, an IGES file (.igs) and a Parasolid file (.x\_t). The assembly of the centrifugal pump is given below in Figure 3.4.

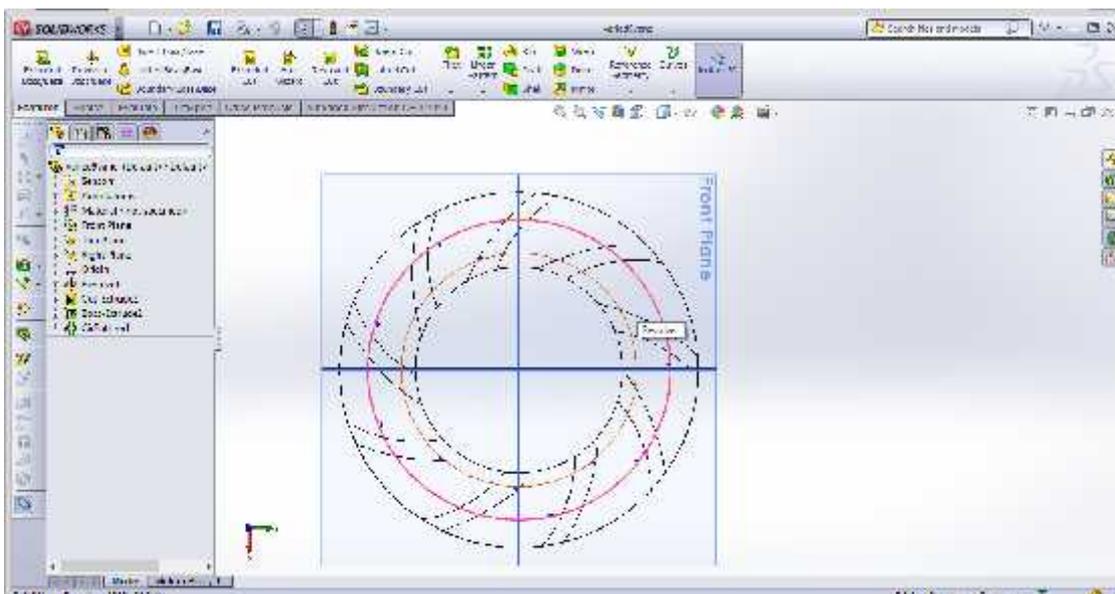


**Figure 3.4 Pump Casing Assembly**

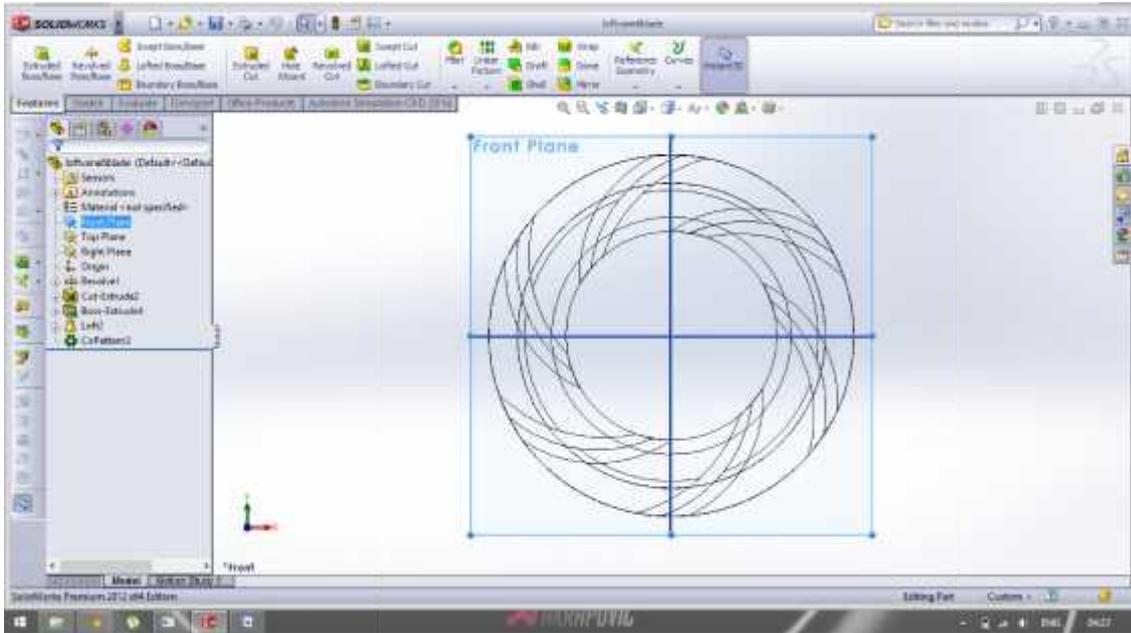
Six impeller models were created with different vane angles and vane configurations. These impellers were then assembled to the casing to create six different pump models. The models are shown below in Figure 3.5, Figure 3.6, Figure 3.6, Figure 3.7, Figure 3.8, Figure3.9 and Figure 3.10.



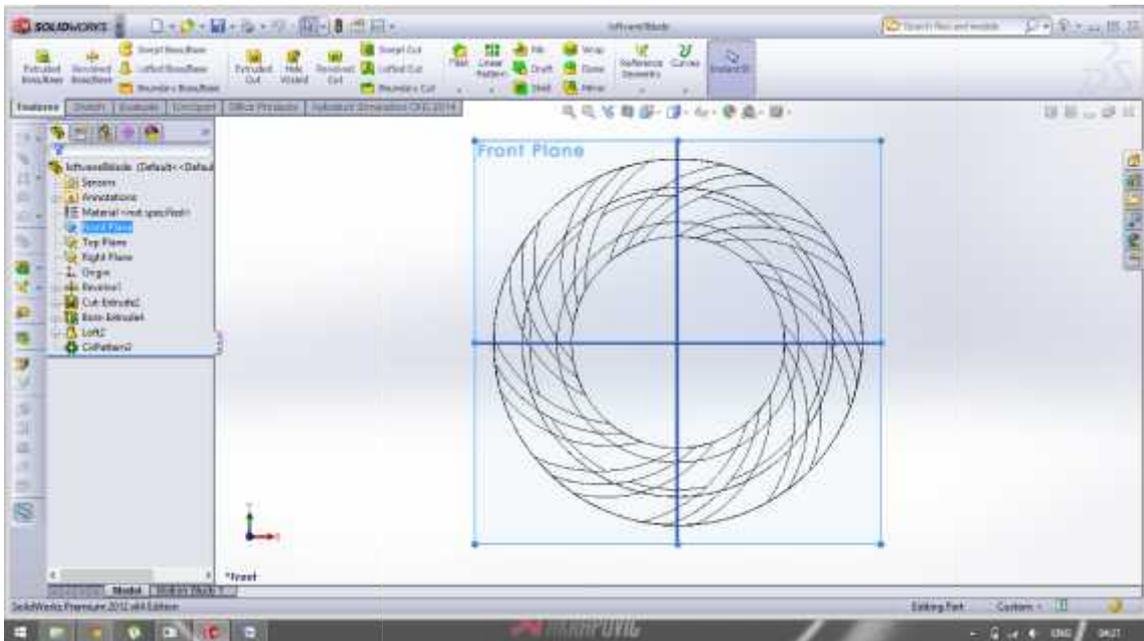
**Figure 3.5 ASE300 Impeller- PIONEER PRODUCTS**



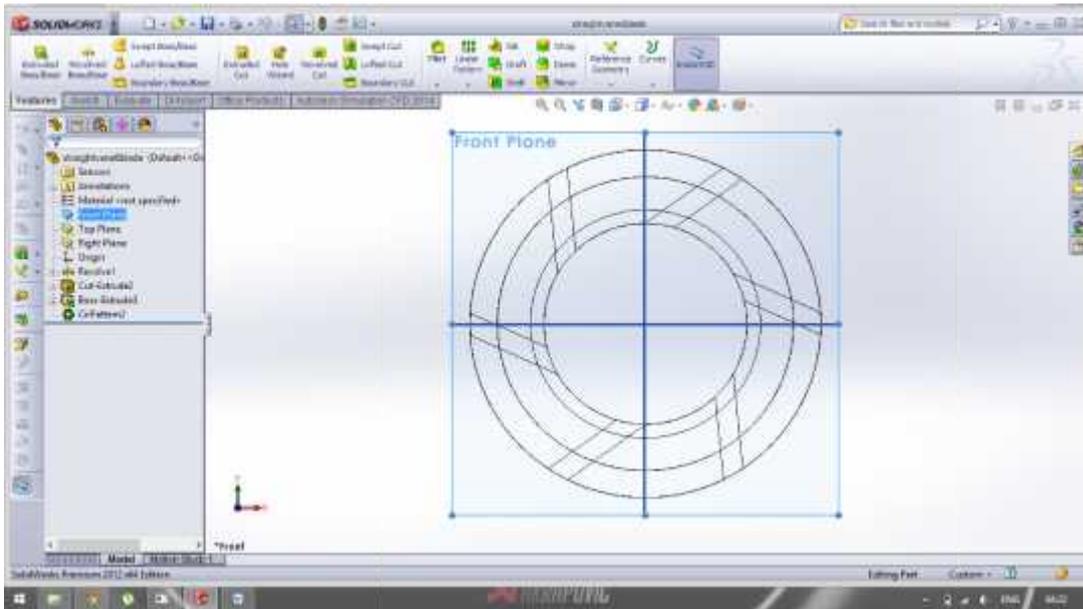
**Figure 3.6 Modified 9 Blade Impeller**



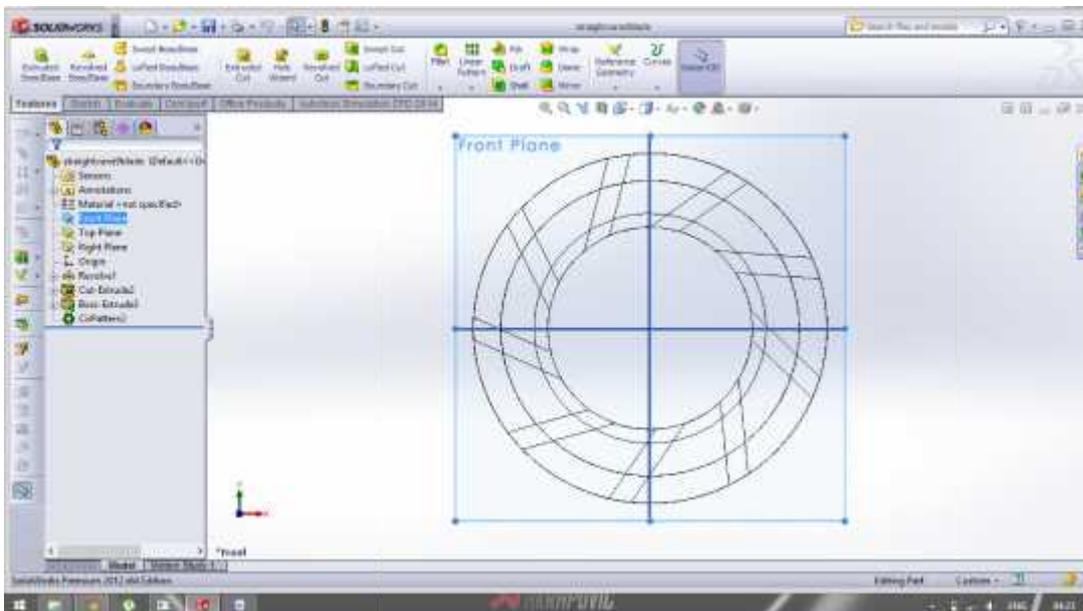
**Figure 3.7 Loft 6 Blade Impeller**



**Figure 3.8 Loft 9 Blade Impeller**



**Figure 3.9 Straight 6 Vane Impeller**



**Figure 3.10 Straight 9 Vane Impeller**

These models then were analyzed using CFD analysis.

## **3.2 CFD ANALYSIS**

To perform CFD analysis initially the basics of Flow Dynamics is studied. The cfd is computational mechanics which uses the numerical methods and algorithms that solves the fluid flow equations.

the cfd solution involves the basic steps:

- creation of the geometry
- grid generation
- choice of models
- boundary conditions
- flow field computation
- post processing

### **3.2.1 CFD ANALYSIS USING ACUSOLVE**

The acusolve tutorial of blower is studied. The tutorial explains in detail about importing geometry and creating surface and volumes sets of the blower model. Building of reference frame for the blower, defining global, surface mesh parameters and boundary conditions is explained. Generating all-tet mesh, viewing a mesh cut plane and solving the fluid flow equations in acusolve is studied in the tutorial.

#### **CREATION OF GEOMETRY:**

The model to be analyzed is created in modelling software. The created model is imported in acusolve.

## GRID GENERATION:

For generation mesh, the surfaces are defined and absolute mesh size is specified and the blower is meshed.

## CHOICE OF MODELS:

This is related for analysing the model in 2D or 3D. For the blower model, the choice of turbulence equation is the SPALART ALLMARAS equation which solves the transport equation for turbulent viscosity.

## BOUNDARY CONDITIONS:

Boundaries are typical inlets, outlet, walls, symmetry planes or axis for axisymmetrical computational. For turbulence compressible flow, velocity components/ total pressure and flow direction with temperature is specified. Then the nodal initial conditions are created. The blower is ready to solve in acusolve. The output is plotted in acuprobe.

## POST PROCESSING:

The surface mesh is visualized in acufiled . The discrete solution for the flow variable is available for the domain at each mesh element.

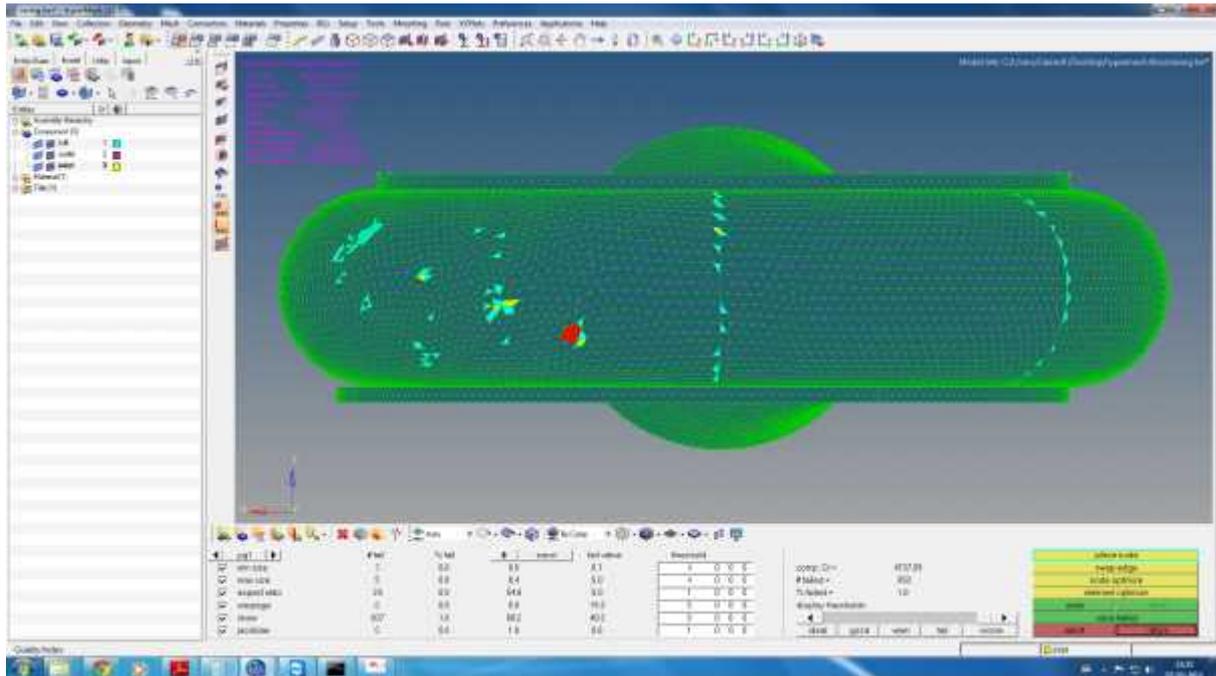
## CFD ANALYSIS USING ACUSOLVE:

Initially, the impeller-casing assembly was imported into acusolve. There were many difficulties that had to be overcome while importing the model. The assembly had to be saved as a parasolid (x\_t) file, or a ACIS (.SAT) file. Once the format was converted, the assembly was again imported. While doing the pre-processing procedures for the model, other problems arose. The separate part files of the pump i.e, the impeller and the casing could be imported separately without a problem, but as an assembly several error messages opened

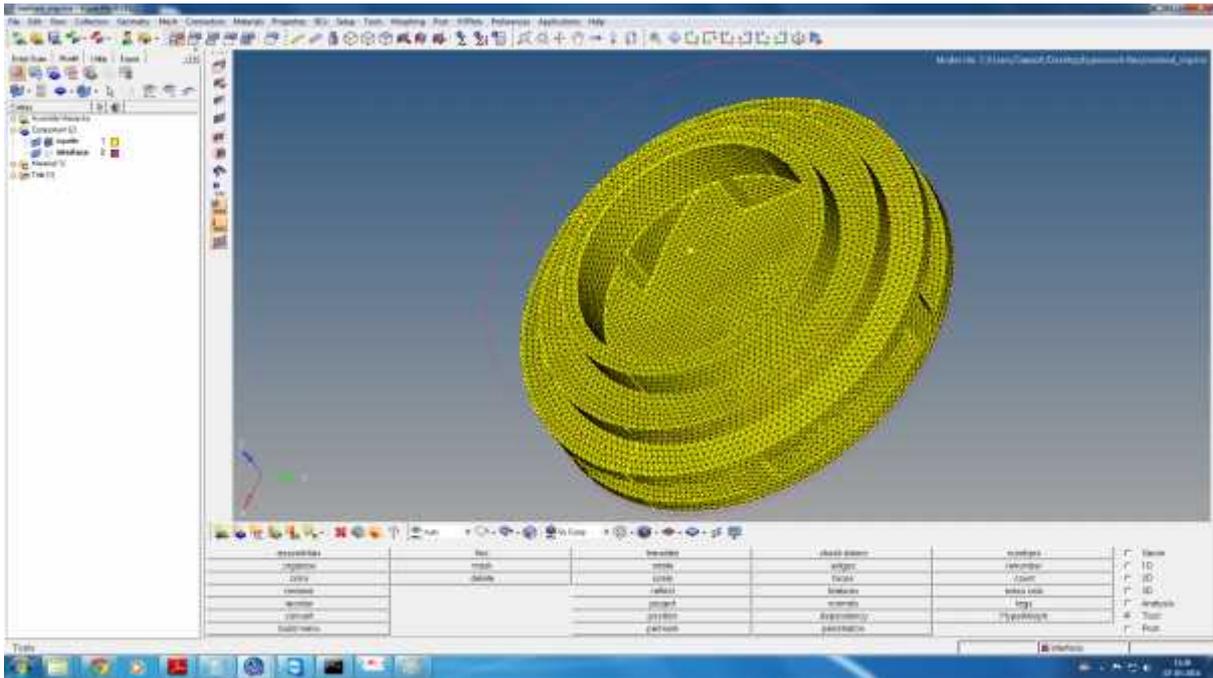
up. The Acusolve technician was contacted. After trial and error methods, the error was found to be in the geometry.

### GEOMETRY ERRORS:

The model was then imported to hypermesh and then checked for surface errors.



**Figure 3.11 Manual Mesh Correction of Casing**



**Figure 3.12 Manual Mesh Correction of Impeller**

A manual mesh was created for the impeller and the casing separately. This mesh was then checked to point out surface errors. The surface errors were rectified. This rectified assembly was then imported again to acusolve. On this instance as well, errors were notified. The error corrections are shown in Figure 3.11 and Figure 3.12.

**BOUNDARY ERRORS:**

The geometry of the assembly was rectified. On this instance, the errors were related to the boundary conditions of the pump assembly. The inlet and outlet of the pump assembly were initially set as openings. The ‘Acusolve’ technician pointed out that the inlet and outlet needed to be defined as separate bodies. The assembly was again modified and the boundary surfaces were set.

**ACUSOLVE:**

After making the above modifications, the model was again imported to acusolve. Errors were again pointed out. The acusolve technician was informed. The process was temporarily put on hold.

### **3.2.2 CFD ANALYSIS USING ANSYS**

Since we encountered several problems with Acusolve, we tried our analysis with ANSYS-FLUENT. ANSYS FLUENT software is a high-performance, general purpose fluid dynamics program that has been applied to solve wide-ranging fluid flow problems. The ANSYS fluid dynamics suite includes two powerful general purpose CFD software codes: ANSYS CFX and FLUENT CFD. The steps involved in ANSYS CFX are similar to Hyperwork's Acusolve.

The model was imported to ANSYS. The geometry was opened and a mesh was created. Then the various input parameters were entered. The transient flow condition is selected. The inlet velocity, the flow rate and the inlet pressure are entered into the software. The simulation is run for 100 iterations. The output graphs are plotted. The results obtained are compared with the theoretical calculations obtained earlier.

#### **PRE-PROCESSING:**

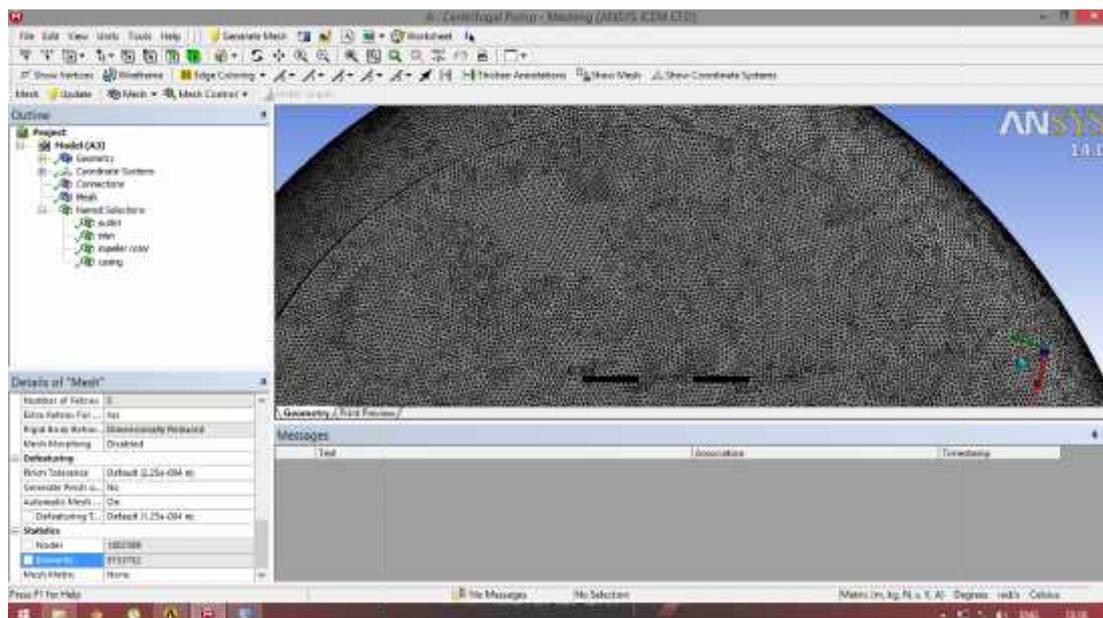
The first step in this process involves importing the assembly file into ansys. ANSYS-Workbench 14.0 is opened and a FLUENT system is selected. The geometry was imported. The Workbench page is shown in Figure 3.13.



**Figure 3.13 ANSYS Workbench Geometry**

The inlet and the outlet faces were selected and the named sections were created. The impeller and the casing were also defined.

The geometry was then meshed, the element size was given in the order of  $10^{-4}$ . The number of elements and the number of nodes were varied to obtain a fine mesh. The solver took four hours to mesh. The meshed impeller is shown in Figure 3.14.



**Figure 3.14 Meshed Pump Assembly**

## SOLVER SET-UP:

The mesh was generated and it was updated into the fluent case file. The solver set-up was opened and the parameters were set.

There were a set of solving equations, the k-  $\epsilon$  equation method was chosen.

The cell zone conditions were set. The impeller rotor was assigned to be a moving reference frame. The speed of rotation was set at 2800 rpm about the z-axis.

The boundary conditions were set. The impeller initial conditions were set as a moving reference frame with an initial rotational speed of 2800 rpm.

The inlet was chosen and set as a velocity-inlet. The inlet velocity of the fluid was entered.

## SOLUTION:

The set-up was initialized. The monitors were set to observe the mass flow rate at the outlet of the pump. The calculations were set in such a way that they used the inlet conditions as the base for the calculations. The Coupled equation was selected for calculations and the hybrid initialization was selected.

The solution was calculated after 1000 iterations took place.

### **3.2.3 SIMULATION RESULTS**

#### FLUID VELOCITY:

The inlet fluid velocity was calculated. Given below are the simulation results for the velocity of the liquid. The first two impeller models were

simulated. The results are shown below in Figure 3.153.15 and Figure 3.16.

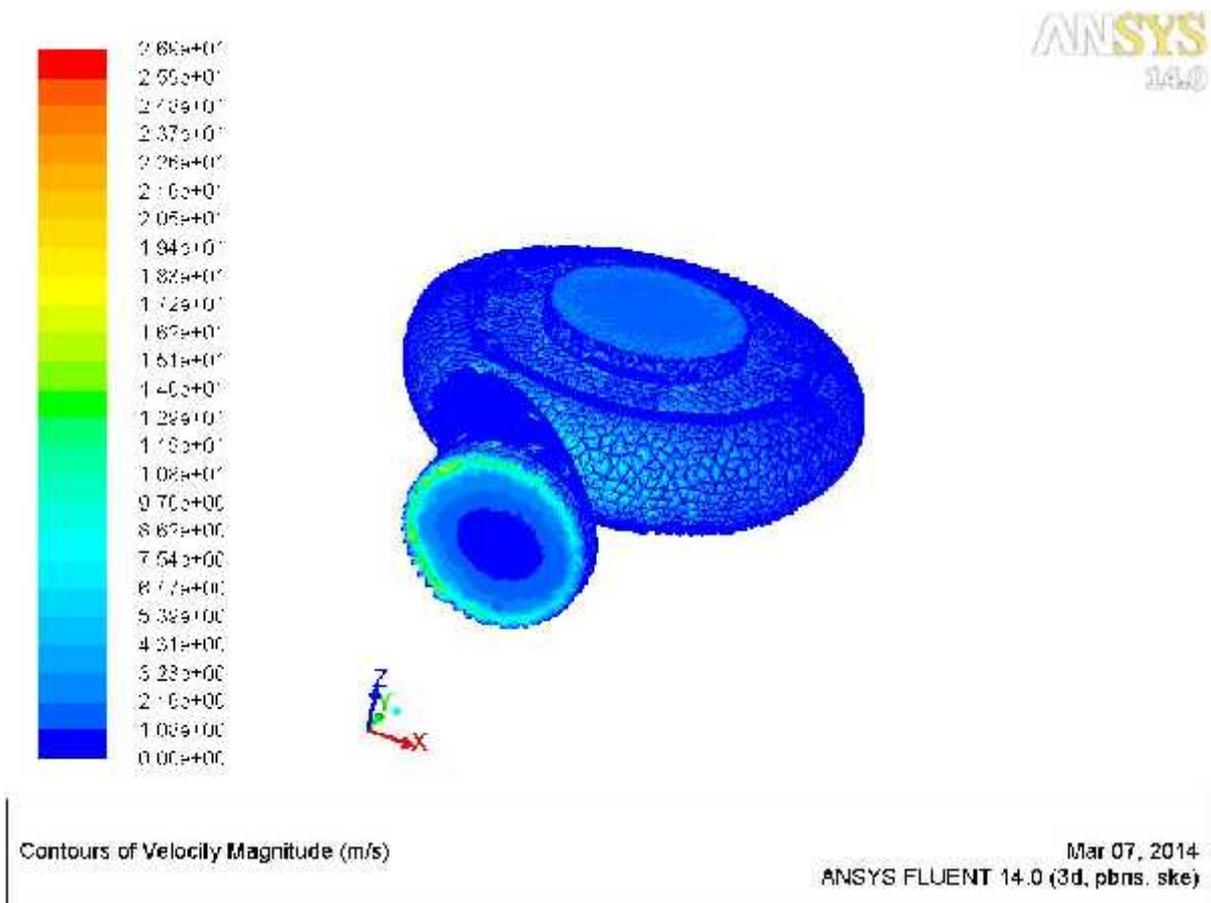
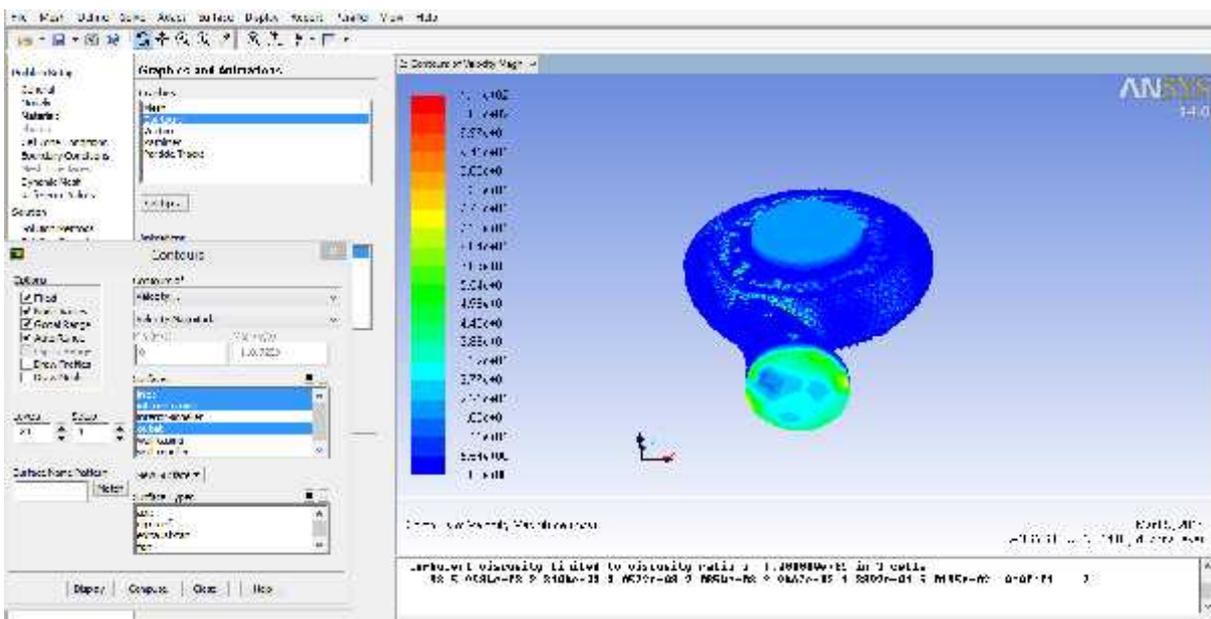


Figure 3.153 Velocity Contour for Impeller 1

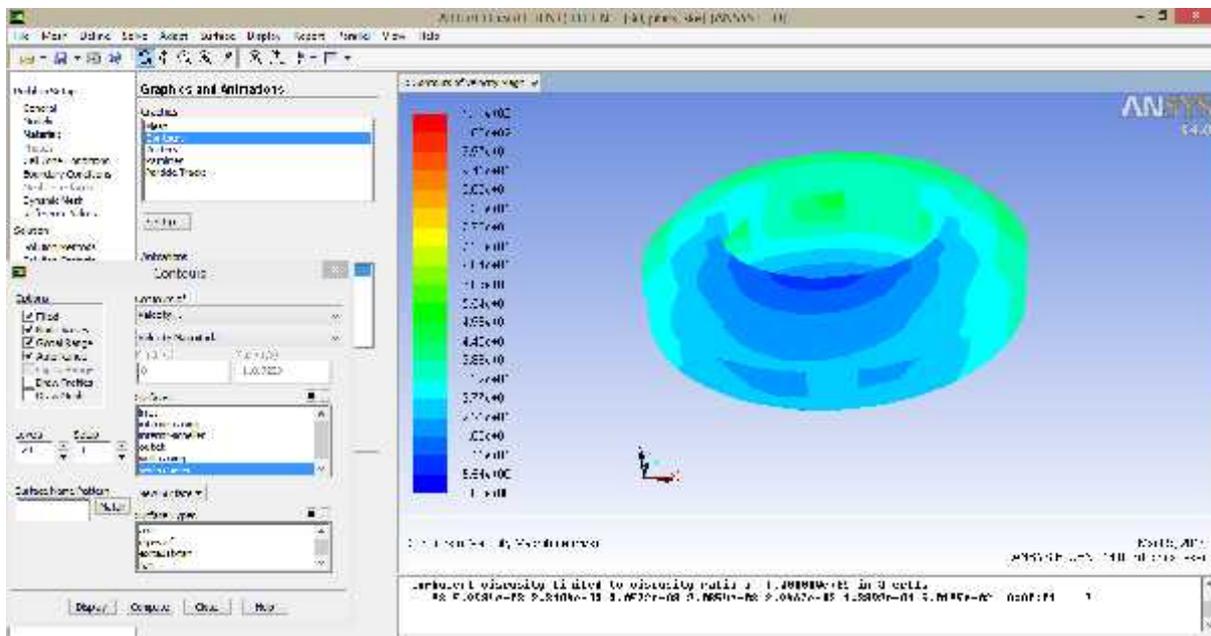


The inlet velocity from theoretical calculations was found to be 2.87m/s.

The formula for velocity is calculated from the discharge and the diameter of the pump.

The outlet velocity from theoretical calculations was found to lie between 3.5m/s to 7.7m/s.

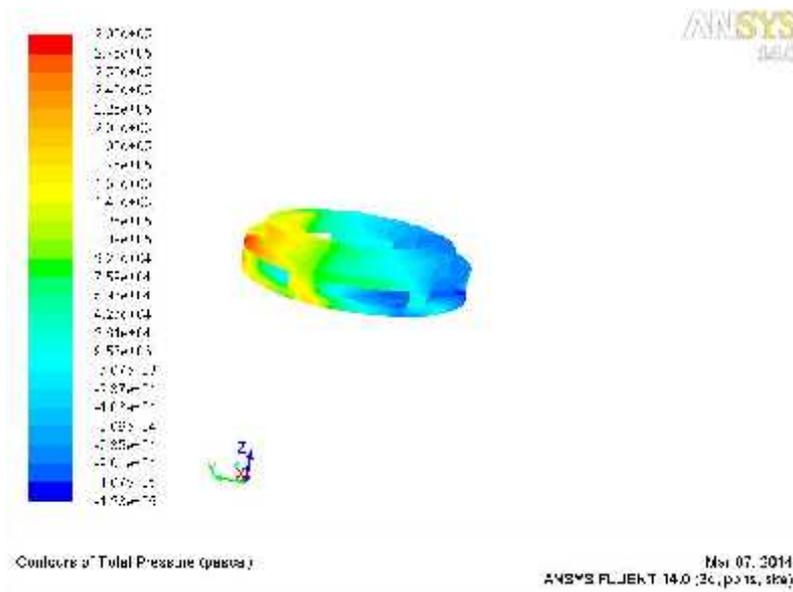
The velocity plot inside the impeller is shown in Figure 3.17.



**Figure 3.17 Fluid Velocity in Impeller 1**

### PRESSURE:

The inlet pressure was found to be 0.77atm. The pressure plot is given below:



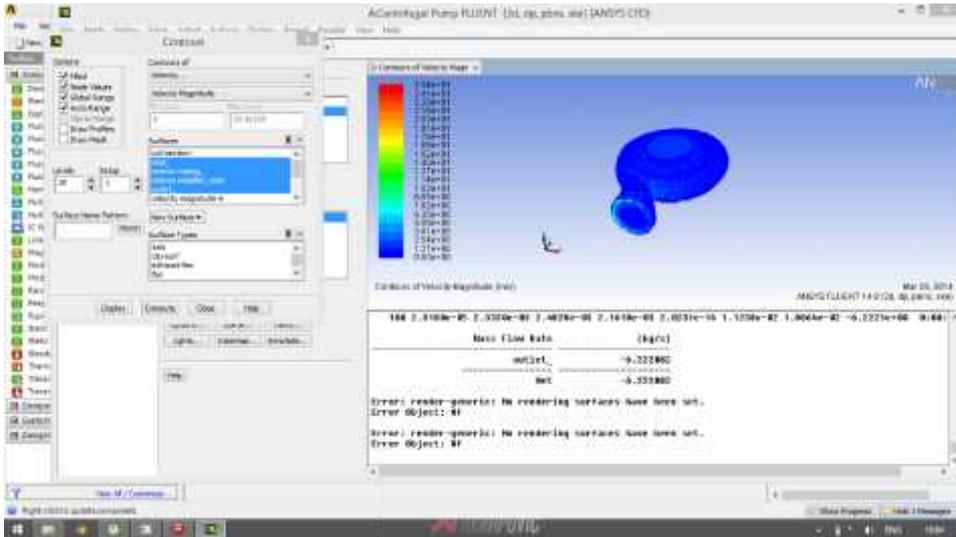
**Figure 3.18 Pressure Distribution in Impeller 1**

Given above is the pressure contour plot inside the impeller. The inlet pressure is 0.77atm. The maximum pressure in the impeller was found to be 3.8atm.

From the above simulation, it can be seen that the theoretical calculations of the pump are near the obtained values. So, it can be used to predict the changes in performance that are affected by the changes in design. The pressure plot is shown in Figure 3.18.

#### MASS FLOW RATE:

The mass flow rate for one of ten cases was found to match the theoretical values for the first case.

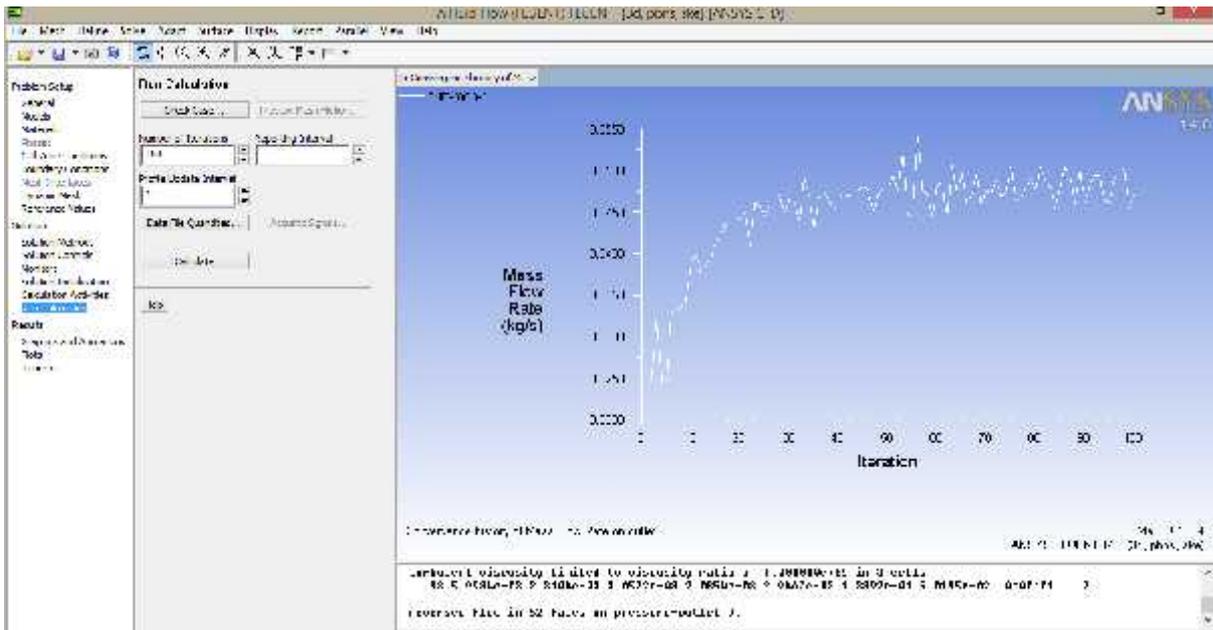


**Figure 3.19 Mass Flow Rate for Impeller 1**

The theoretical calculations indicated a flow rate of 8kg/sec. The simulation was carried out and a flow rate of 6.2kg/sec was achieved. The Mass Flow Rate is shown in Figure 3.19.

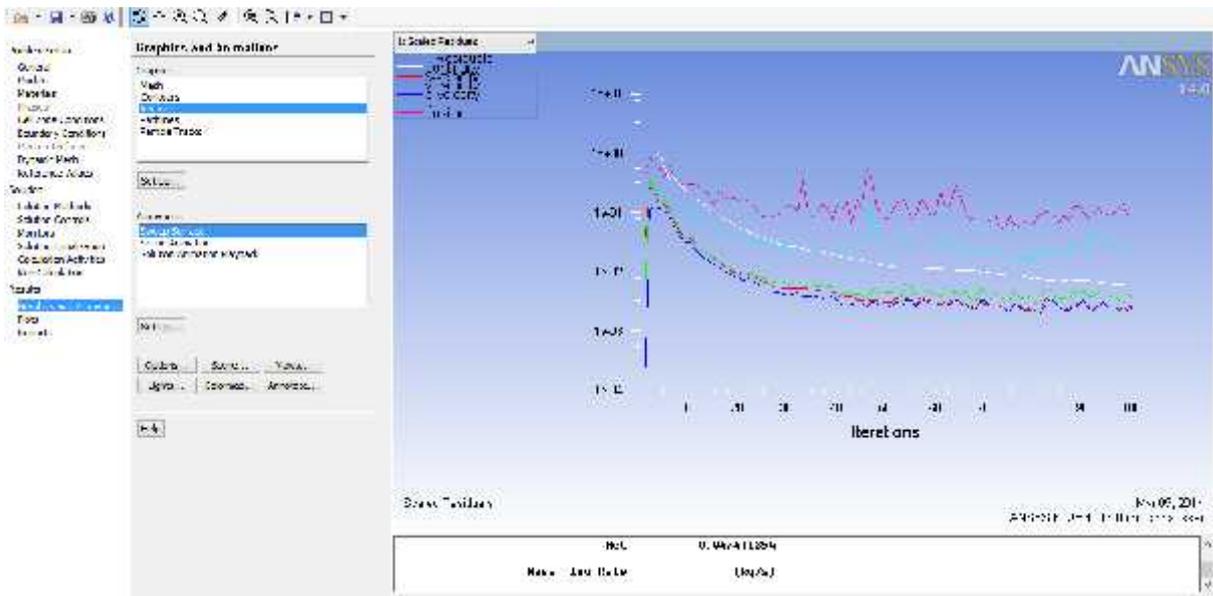
The case was repeated for the same model, but the plot shows that the flow rate plot is found to be much lower.

The simulation was carried out again for different models, it shows a varied result. The results are shown in Figure 3.20.



**Figure 3.20 Mass Flow Rate Variation for Impeller 1**

The graph showing the velocities at three different axes is given below.



**Figure 3.21 Fluid Velocity in Impeller 1**

If mass flow rate is entered as 8kg/sec, then the pressure plot is incorrect.

The velocity plot was also found to be varied. The theoretical values do not match the simulation results. The varied results are shown in Figure 3.22 and Figure 3.23.

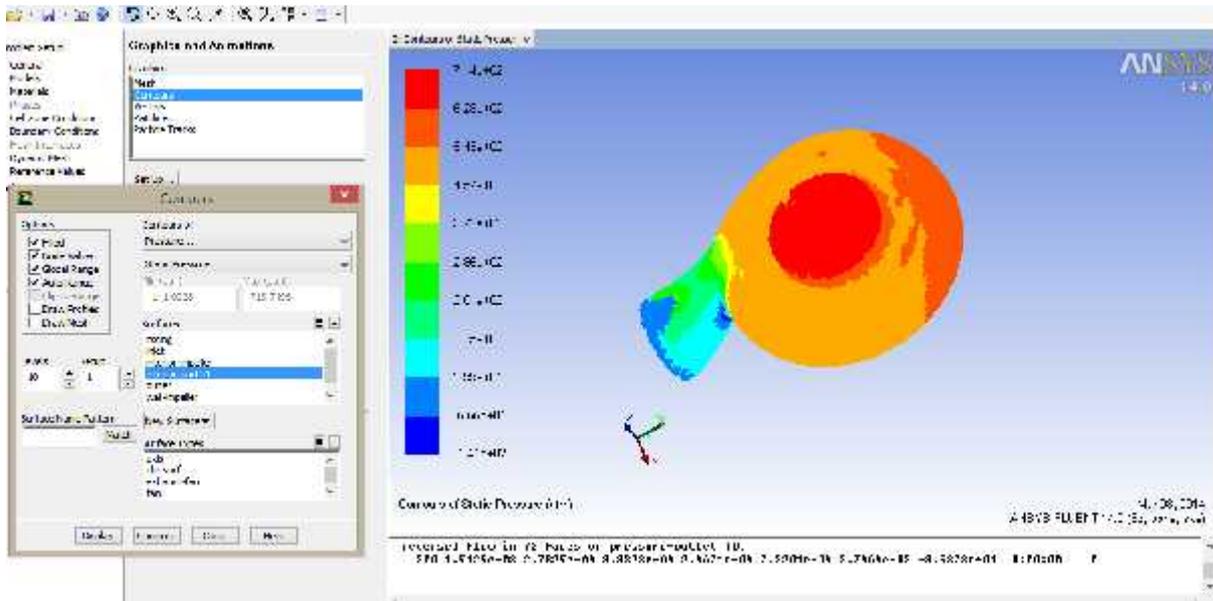


Figure 3.22 Pressure Distribution Variation for Impeller

1

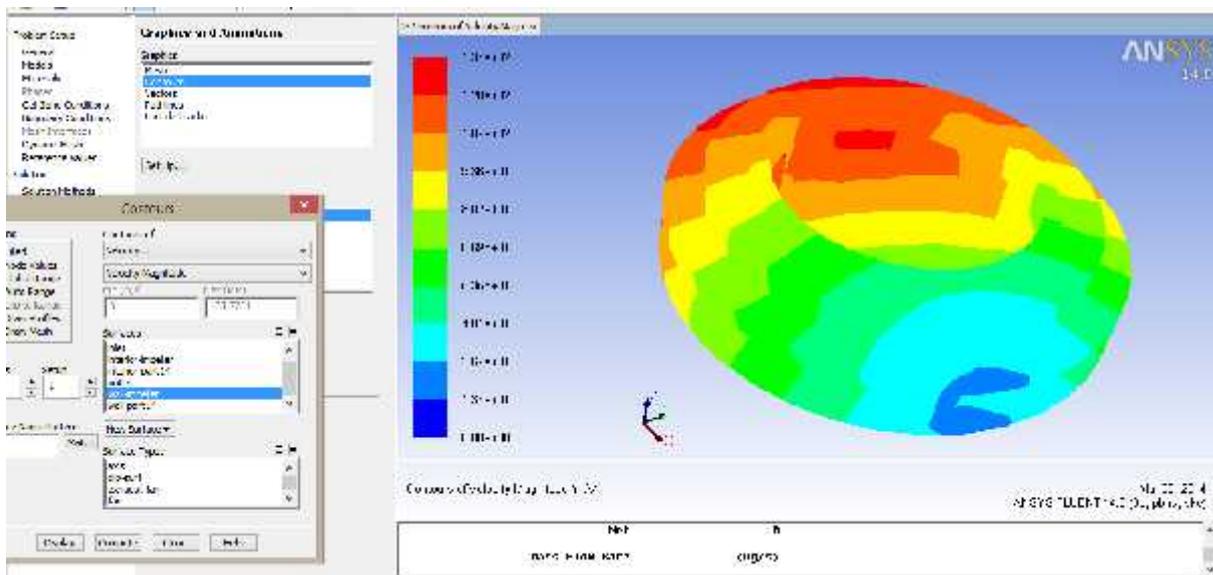


Figure 3.23 Velocity Distribution Variations in Impeller 1

### **3.3 RESULTS:**

The theoretical calculation for the given pump assembly were done. Based on the literature collected, six impeller models with different vane angles and vane designs were created using Solidworks. The models were analysed in ANSYS FLUENT software. The mass flow rate found to deviate from the calculated values for all except one case. Analysis was also carried out using ACUCONSOLE, but the results could not be obtained.

#### **3.3.1 FUTURE SCOPE:**

Analysis of new impeller and volute casing designs of pumps can be simulated before fabrication.

This method will be economical and time saving compared to traditional methods of design and development.

By analysing the fluid flow new models different and more efficient from the existing conventional models can be developed.

## REFERENCES:

- [1] K. C. Thin, M. M. Khaing, and K. M. Aye, “Design and Performance Analysis of Centrifugal Pump,,” *Proc. World Acad. Sci. Eng. Technol.*, vol. 48, 2008.
- [2] E. Dick, J. Vierendeels, S. Serbruyns, and J. V. Voorde, “Performance prediction of centrifugal pumps with CFD-tools,” *Task Q.*, vol. 5, no. 4, pp. 579–594, 2001.
- [3] “A BRIEF INTRODUCTION TO CENTRIFUGAL PUMPS - pumpintro.pdf.” .
- [4] K. W. Cheah, T. S. Lee, S. H. Winoto, and Z. M. Zhao, “Numerical flow simulation in a centrifugal pump at design and off-design conditions,” *Int. J. Rotating Mach.*, vol. 2007, 2007.
- [5] W. Zhou, Z. Zhao, T. S. Lee, and S. H. Winoto, “Investigation of flow through centrifugal pump impellers using computational fluid dynamics,” *Int. J. Rotating Mach.*, vol. 9, no. 1, pp. 49–61, 2003.
- [6] S. Rajendran and K. Purushothaman, “Analysis of a centrifugal pump impeller using ANSYS-CFX,” in *International Journal of Engineering Research and Technology*, 2012, vol. 1.