DESIGN & OPTIMIZATION OF CENTRIFUGAL PUMP USING ANSYS® & GENETIC ALGORITHM

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENT FOR THE AWARD OF THE DEGREE OF

MASTER OF TECHNOLOGY

(COMPUTATIONAL DESIGN)

TO

DELHI TECHNOLOGICAL UNIVERSITY

SUBMITTED BY

ABHAY THAPLIYAL

ROLL NO.:2K15/CDN/18

UNDER THE GUIDANCE OF

Mrs SUSHILA RANI

ASSISTANT PROFESSOR

DELHI TECHNOLOGICAL UNIVERSITY

DEPARTMENT OF MECHANICAL, PRODUCTION & INDUSTRIAL AND AUTOMOBILE ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi college of Engineering)

Shahbad Daulatpur, Baawana Road,

Delhi-110042

STUDENT'S DECLARATION

I, **Abhay thapliyal,** hereby certify that the work which is being presented in this thesis entitled **"Design & Optimization of Centrifugal Pump Using ANSYS®** & **Genetic Algorithm "** is submitted in the partial fulfilment of the requirement for degree of **Master of Technology (Computational Design)** in Department of Mechanical Engineering at **Delhi Technological University** is an authentic record of my own work carried out under the supervision of **Mrs Sushila Rani.** The matter presented in this thesis has not been submitted in any other University/Institute for the award of Master of Technology Degree. Also, it has not been directly copied from any source without giving its proper reference.

Signature of Student

This is to certify that the above statement made by the candidate is correct to the best of my knowledge.

Signature of Supervisor

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi college of Engineering)

Shahbad Daulatpur, Baawana Road,

Delhi-110042

CERTIFICATE

This is to certify that this thesis report entitled, **"Design & Optimization of Centrifugal Pump Using ANSYS® & Genetic Algorithm"** being submitted by **Abhay Thapliyal (Roll No. 2K15/CDN/18)** at Delhi Technological University, Delhi for the award of the Degree of Master of Technology as per academic curriculum. It is a record of bonafide research work carried out by the student under my supervision and guidance, towards partial fulfilment of the requirement for the award of Master of Technology degree in Computational Design. The work is original as it has not been submitted earlier in part or full for any purpose before.

Mrs Sushila Rani

Assistant Professor

Mechanical Engineering Department

Delhi Technological University

Delhi-110042

ACKNOWLEDGEMENT

First and foremost, praises and thanks to the God, the Almighty, for his showers of blessings throughout my research work to complete the research successfully.

I would like to extend my gratitude to **Prof. R.S. Mishra, Head,** Department of Mechanical Engineering, Delhi Technological University, for providing this opportunity to carry out the present thesis work.

The constant guidance and encouragement received from **Prof. Vikas Rastogi M.Tech Coordinator** & **Dr. A.K. Agarwal, M.Tech. Coordinator and Associate Professor,** Department of Mechanical Engineering, Delhi Technological University, has been of great help in carrying out the present work and is acknowledge with reverential thanks.

I would like to express my deep and sincere gratitude to my research supervisor, **Mrs Sushila Rani,** Department of Mechanical Engineering, Delhi Technological University, for giving me the opportunity to do research and providing invaluable guidance throughout this research her dynamism, vision, sincerity and motivation have deeply inspired me. She has taught me the methodology to carry out the research and to present the research works as clearly as possible. It was a great privilege and honour to work and study under her guidance. I am extremely grateful for what she has offered me. I would like to thanks her for hers friendship, empathy, and great sense of humour. Without her wise advice and able guidance, it would have been impossible to complete the thesis in this manner.

I would like to extend my thanks to **Mr. Ashish Gupta,** PhD Scholar, Delhi Technological University, without the help of whom the project would not have been completed. I am also grateful to all the faculty members of the Mechanical Engineering Department for moulding me at correct time so that I can have a touch at final destination.

I am extremely grateful to my parents and family for their love, prayers, caring and sacrifices for educating and preparing me for the future.

ABHAY THAPLIYAL

M.Tech (COMPUTATIONAL DESIGN)

2K15/CDN/18

ABSTRACT

Centrifugal pump is the most widely used pump in the world today as it has simple design, with wide range of capacity and head. The ability of hydraulic structure or element to conduct fluid with minimum energy loss is the most important property of a pump system.

The present work deals with the Computation Fluid Dynamics study of the hydrodynamic characteristics (i.e. pressure and velocity) of a centrifugal pump using Ansys®-CFX and optimize the design with the help of pattern search method (i.e. Genetic Algorithm) using MATLAB®. In this work the pump is designed analytically with the use of characteristics curve which are further used in the validation of the design obtained by Vista-CPD. The pump is designed to obtain maximum value of hydraulic efficiency at the (BEP) Best Efficiency Point.

The work includes CFD simulation of centrifugal pump in baseline & optimized design variables. The optimized parameters have been evaluated for the design using genetic algorithm.

Keywords: Centrifugal pump, Ansys®-CFX, Optimization, Genetic algorithm, MATLAB®, Vista CPD

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

P_{th}	Theoretical power of centrifugal pumps
H_{th}	Theoretical head of centrifugal pumps
V	Absolute velocity
U	Rotational velocity (u =ωR)
W	Relative velocity
T	Centrifugal pump torque
β2	Blade angle at exit of impeller
t	Blade thickness
N	Blade count
β1	Blade angle at the internal of impeller
D1	Inlet diameter of impeller
D2	Outlet diameter of impeller
Θ	Wrap angle
B1	Inlet blade width
B2	Outlet blade width
H	Head
Q	Mass flow rate
η	Efficiency
βm	The average vane angle
N	Rotational speed
Cu	Absolute tangential velocity
Cm	Absolute meridional velocity
Ω	Angular velocity [rad/s]
ρ	Density [kg/m3]
g	Gravity acceleration [m/s2]

ABBREVIATIONS

CFD	Computational Fluid Dynamics.
LE	Leading edge
TE	Trailing edge
BL	Boundary layer
K-ε	K-epsilon
RPM	Revolution per minute
R1	Rotor
S1	Stator
Pa	Pascal
Atm	Atmosphere
CPD	Centrifugal pump design

CHAPTER 1

INTRODUCTION

This thesis is devoted towards research study of selection of Centrifugal pump as fluid machinery is occupying huge role in human life. These days' pumps are used from agricultural units to industrial units. The usage of pumps is introduced in chemical industry also.

Industries which involve fluids in the processes require hydraulic machines to provide energy to fluid or produce energy by the fluid. Specifically, the machine which utilizes hydraulic energy is defined as turbine. However, the machine which consumes energy to transport fluid is defined as pumps, compressors and fans (blowers). Pumps are used to give energy to liquids, on the other hand compressors and fans are used to give energy to gases. Pumps often find applications in the critical units of a plant, for example, a condensate pump in the Rankine cycle. Hence, pumps have a crucial role in the processes of industries.

For an application, pump should be selected for the suitable variety of operations which also depends on flow rate and the pressure of the system. Since, centrifugal pumps have a broad range of operational conditions in comparison with the positive displacement; these pump type dominates in industries.

1.1 Centrifugal pump

The centrifugal or radial pump type changes the fluid flow direction from horizontal to vertical or radial direction. The flow direction of centrifugal pumps generally changes from axial at the inlet to radial as the fluid moves toward the outlet. The flow direction of the axial type pumps is parallel with the shaft whereas for the mixed flow type pumps, the fluid flows both axially and radially. For axial pumps, the centrifugal effect has no contribution in energy transfer.
Dynamic pumps have a wide flow rate range at a particular speed but have poor performance with high viscosity fluid operations. On the other hand, positive displacement pump operates at a very high pressure for a certain flow rate; the

pressure is regulated with a pressure relief valve for the prevention of pump damage. A positive displacement pump has sound performance even with high-viscosity fluid.

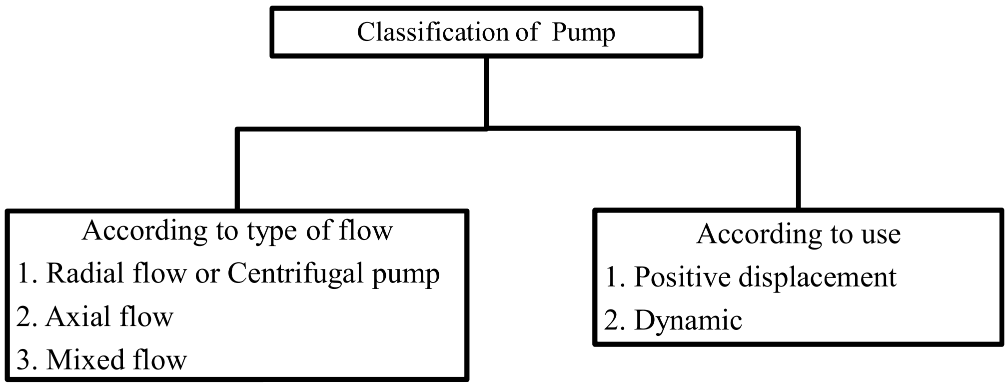

Fig 1.1 Classification of Pump

Generally, the main parts of centrifugal or radial pumps are a combination between the rotating parts which are vanes or impeller and shaft, and stationary part which is the diffuser or volute (fig.1.2). The working principle of a centrifugal pump is the liquid axially flows toward impeller inlet through the impeller eye. Then, the rotor rotation causes a pressure drop at the impeller inlet. Consequently, the fluid flows from the reservoir tank toward the impeller inlet. Then, the fluid flows radially outward in the passage to the impeller exit. The fluid kinetic energy increases because it experiences the centrifugal force which is caused by the impeller rotation and radius increase. When the fluid moves toward the impeller outlet, fluid pressure also increases due to the diffuser geometry of the impeller passage. The fluid then is guided to the discharge through the volute or the diffuser. In the volute, the kinetic energy is converted into fluid pressure also because of the diffuser geometry (fig.1.2).

Based on the casing type, centrifugal pumps can be classified as single volute or double volute. The double volute type is usually designed for reducing the radial load of the pump. However, it is noted that designing a centrifugal pump using a double volute type decreases the efficiency slightly because the double volute casing increases the contact surface area between the fluid and stationary part so the losses increase. Most of the centrifugal pump applications in industry still use the single volute type.

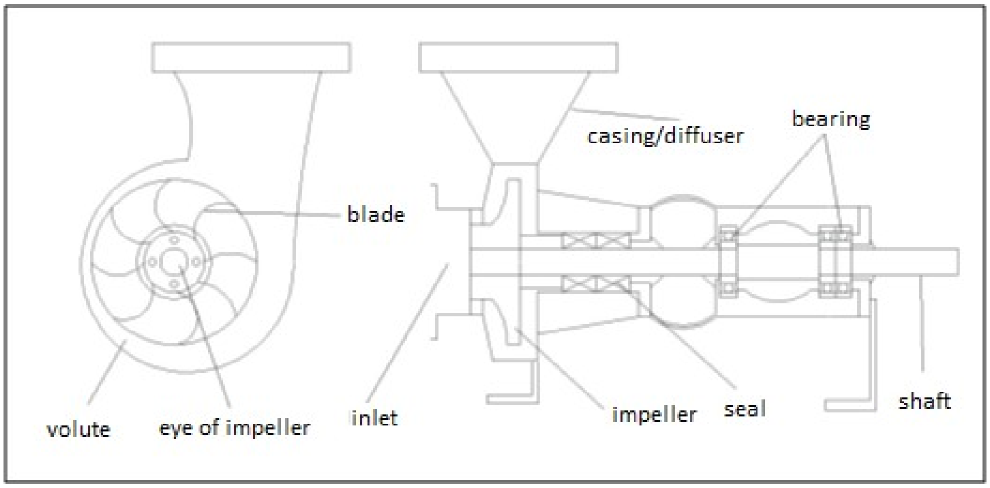

Figure 1.2: Schematic picture of general centrifugal pump [1]

Based on the suction type, the centrifugal pump can be categorized as single suction and double suction pump. The double suction type is designed to reduce the axial thrust of the pump and usually used for high flow rate operating conditions.

The impeller of the centrifugal pumps can be classified as open and closed type impeller based on the mechanical construction. There is no shroud or side wall on the top of the vanes for the open type impeller, whereas the closed type has the shroud. Engineers design the open type impeller for high flow rate and low pressure; whereas the closed type impellers can be used for high pressure and high flow rate. Open type impellers are also designed to deliver very dense liquids.

1.2 Centrifugal pump performance

Engineers identify centrifugal pumps performance through several parameters which are fluid flow rate (Q), head (H), shaft power (P), efficiency (η) and noise generated by the machine (dB). Head is the pressure rise of the centrifugal pump between the suction and the discharge of a pump and it is usually expressed in unit of height. Shaft power is the power required to deliver the required fluid flow rate. Generally, centrifugal pump performance is drawn as seen in fig.1.3. The centrifugal pump performance curve is obtained by doing an experiment (performance tests). Typically, centrifugal pump head decreases as the flow rate increases while the power required increases. Centrifugal pump efficiency has a maximum value at a certain flow rate called Best Efficiency Point (BEP)or it is usually known as rating design of a centrifugal pump.

Rotational speed (n), fluid density (ρ), fluid viscosity (μ) and pump geometry are other parameters which have an influence on centrifugal pump performance. Fig.1.4 shows the effect of the rotational speed and the impeller size on centrifugal pump performance.

Head and power increase if the speed of the impeller increases and the outlet diameter of the centrifugal pump impeller is larger at a certain flow rate. Theoretically, centrifugal pump performance can be analysed through a fixed control volume analysis. The relationship between centrifugal power and head is described in the Euler turbo machine equations as follows

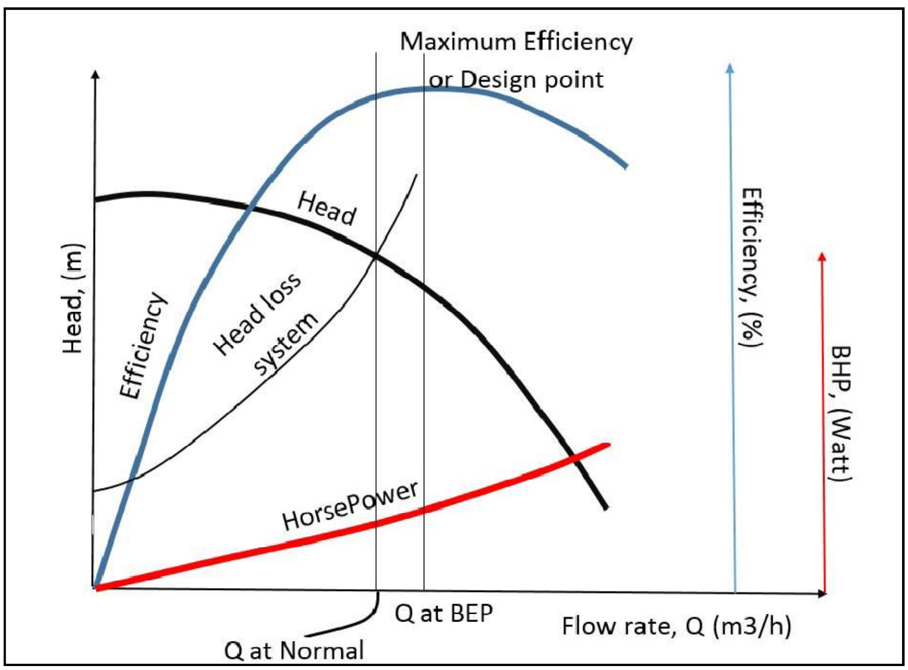

Figure 1.3: Schematic picture of general centrifugal pump [1]

$$Pth = \rho . Q(u2vt2 - u1vt1) \qquad 1.1$$

$$H_{th} = \frac{(u_2 v_{t2} - u_1 v_{t1})}{g} \qquad 1.2$$

$$Q = vn2\pi rb \qquad 1.3$$

The detailed description of the terms in the above equations can be seen in Table 1.1. The Euler turbo machine equation 1.1and 1.2above are derived from angular momentum and the velocity triangle built in the impeller inlet and outlet as seen in Fig.1.5.

The detailed descriptions of velocities mentioned in Fig. 1.5are presented in Table 1.1. The assumptions used to derive the equations above are the fluid flow is assumed ideal. Ideal fluid flow means that the fluid flows without any disturbance which might cause separation flow in the impeller, in other words, the fluid flow follows the blade curvature and there are an infinite number of blades.

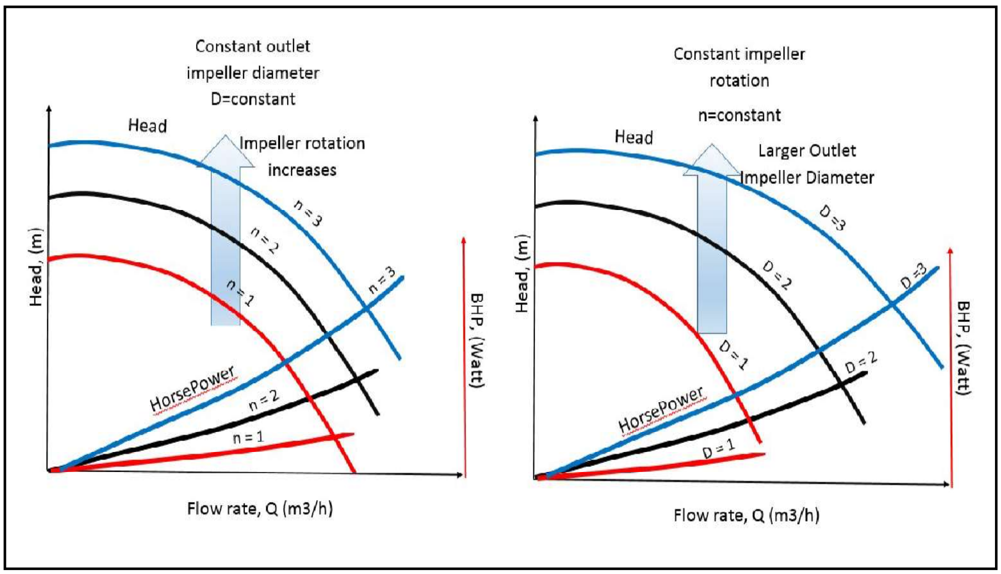

Figure 1.4: Performance curve (a) constant speed (b) Constant outlet diameter [1]

The v_n is related to the flow rate of the pump, while v_t is related to the pump torque (τ). W is the fluid velocity relative to the impeller which is always parallel to the blade angle (β) in the ideal Euler approach. In summary, ideal performance of a centrifugal pump can be obtained through the fluid and impeller velocity and impeller geometry. However, actual performance of a centrifugal pump still has to be obtained through experiment.

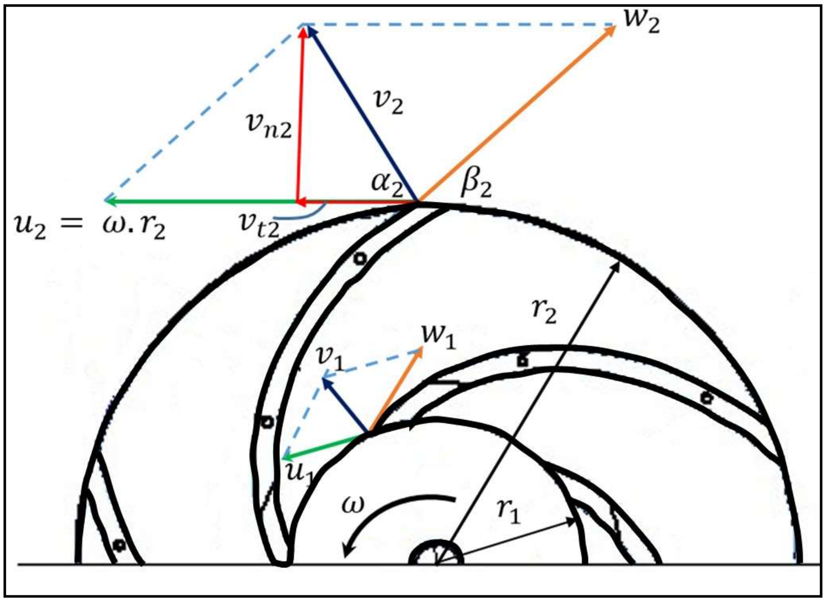

Figure 1.5: Velocity triangles at inlet and outlet [1]

Table 1.1: Descriptions of fluid velocity inside the impeller passage

P_{th}	Theoretical power of centrifugal pumps
H_{th}	Theoretical head of centrifugal pumps
v	Absolute velocity
u	Rotational velocity (u =ωR)
W	Relative velocity
v_t	Tangential component of absolute velocity
v_n	Normal / meridional component of absolute velocity
τ	Centrifugal pump torque
α	Angle between absolute velocity (v) and rotational velocity (u)
β	Angle between relative velocity (W) and negative rotational velocity (u)

1.3 Dimensionless pump performance

The relation of the physical process to the pump performance is described in the dimensionless parameters of the pump. The dimensionless parameters of a turbo machine are as follows:

$$Flowcoefficient(\phi') = \frac{Q}{\omega D^3} \quad 1.4$$

$$Headcoefficient(\psi') = \frac{gH}{\omega^2 D^2} \quad 1.5$$

$$Torquecoefficient(\tau) = \frac{\tau}{\rho D^5 \omega^2} \quad 1.6$$

$$Flowcoefficient(\phi) = \frac{v_{n2}}{u_2} \quad 1.7$$

$$Headcoefficient(\psi) = \frac{gH}{{u_2}^2} \quad 1.8$$

$$SpecificSpeed(n_s) = \frac{\phi'^{\frac{1}{2}}}{\psi'^{\frac{3}{4}}} \quad 1.9$$

$$SpecificSpeed(n_s) = \frac{\omega Q^{\frac{1}{2}}}{(gH)^{\frac{3}{4}}} \quad 1.10$$

$$SpecificDiameter(D_s) = \frac{\phi'^{\frac{1}{4}}}{\psi'^{\frac{1}{2}}} \quad 1.11$$

$$SpecificDiameter(D_s) = \frac{D(gH)^{\frac{3}{4}}}{Q^{\frac{1}{2}}} \quad 1.12$$

Φ and ψ are preferable when engineers design the centrifugal pump impeller. On the other handψ',ϕ' and τ are used to apply the affinity laws in turbomachines. The specific speed is often to be used for determining the appropriate type of centrifugal pump in the operating conditions.

The classification of the centrifugal pump based on n_s at the BEP condition can be seen in fig.1.6. From fig.1.6, it can be inferred that it is better to design an axial type pump compared to a centrifugal pump at a higher specific speed. The high specific speed means that the operating condition has more flow rate than the head increase. If the operating condition needs to have high pressure, the radial or centrifugal pump type should be chosen.

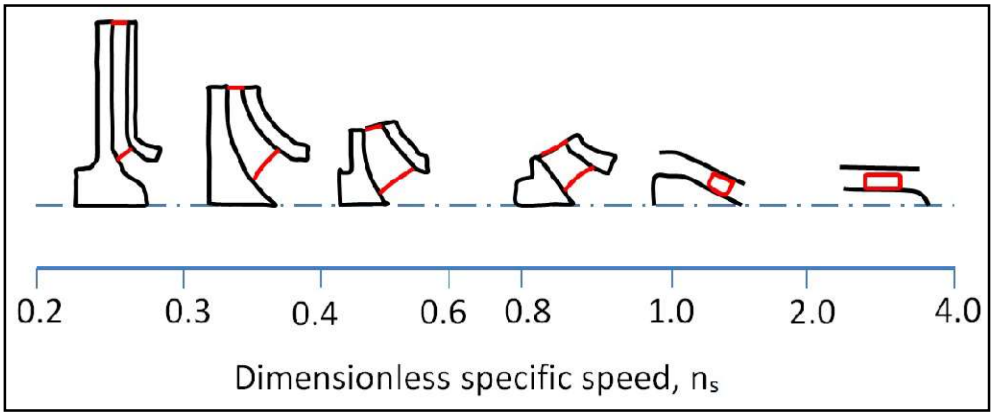

Figure 1.6: The classification of the centrifugal pump [2]

1.4 Centrifugal Pump Efficiency and Losses

Similar to other machines, centrifugal pump efficiency is obtained by calculating the ratio between the input power to the output power. The output power is obtained by extracting the power from the fluid energy which is usually known as water horsepower (WHP). The input power is the torque required of the pump to deliver fluid at a certain speed and called brake horsepower (BHP). The centrifugal pump efficiency relation can be seen in equation 1.13

$$\eta = \frac{WHP}{BHP} = \frac{\rho g Q H}{\tau \omega} \qquad 1.13$$

Several losses affecting centrifugal pump efficiency are impeller circulatory flow losses, turbulence losses and friction losses. The friction loss occurs as a result of the friction between the fluid and passage surfaces. The friction loss increases quadratically when the flow rate is increased since the flow inside the impeller is turbulent. The circulatory flow loss is caused by the slip flow or mismatch flow between the blade and flow inside the impeller passage. At low flow rates, the slip losses are predicted to be higher compared to high flow rates. Turbulence losses occur at low and high flow rates and are a minimum at the design flow rate. Fig. 1.7 shows several losses which cause the decreasing of the centrifugal pump performance.

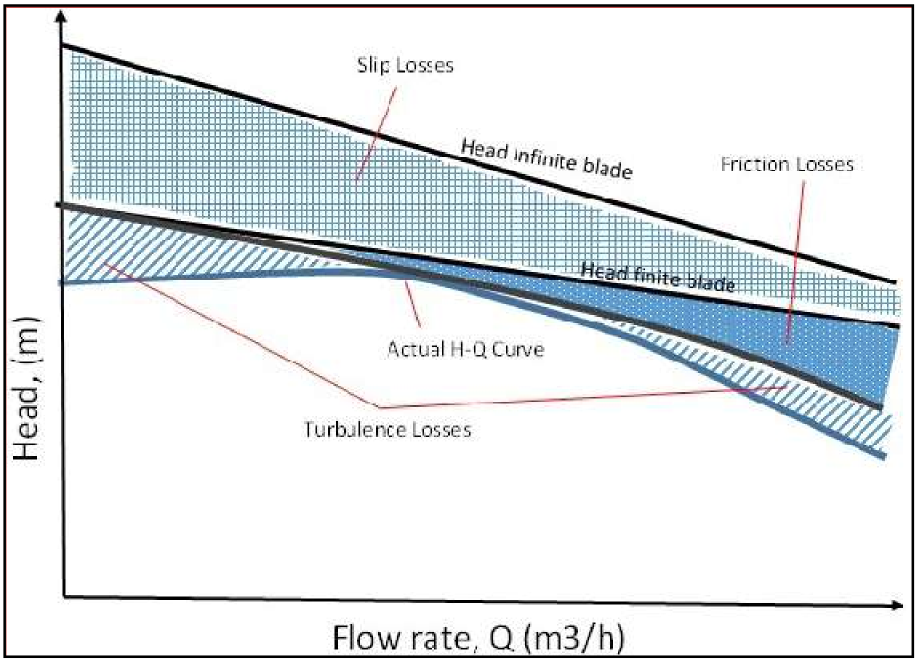

Figure 1.7: The losses occur in the centrifugal pump [3]

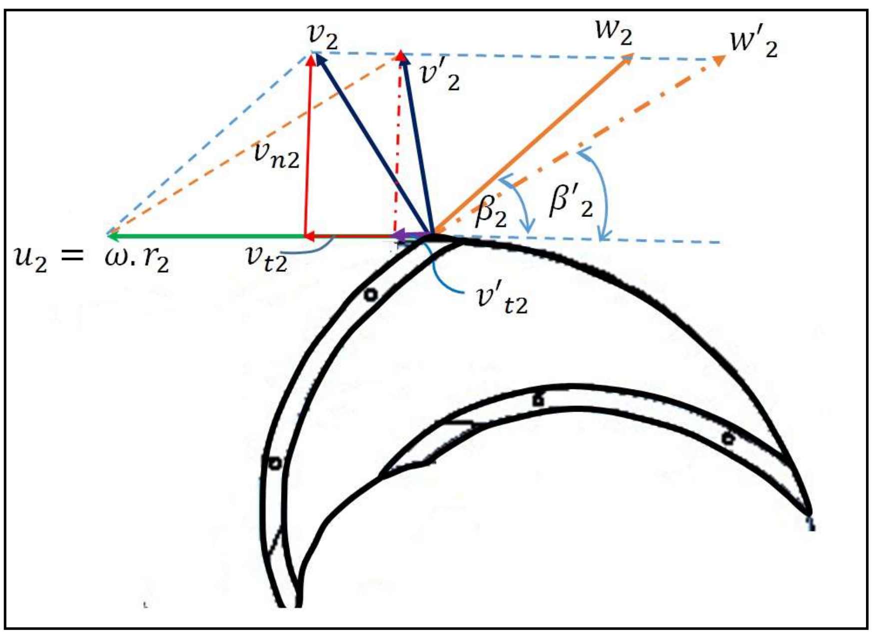

Figure 1.8: The slip model velocity triangle of the backward swept blade [4]

As mentioned previously, in an ideal flow model, flow inside a centrifugal pump follows the blade curvature, so the fluid exit at the discharge is parallel to the exit blade angle.

However, past studies modelled fluid flow inside the impeller passage and noted that the outlet fluid relative velocity (W_2) is shifting and creating angle (β_2') to the impeller velocity (u_2) which is known as slip (Fig. 1.8). The slip condition causes the normal component of the absolute velocity decreases to be v'_{t2}. Consequently, the head and power of the centrifugal pump decreases from the ideal condition. The relation between the ideal and the actual normal component of the absolute velocity is defined as the slip factor (equation 1.14)

$$\sigma = \frac{v_{t2}}{v'_{t2}} \qquad 1.14$$

Several approaches have been developed to obtain the slip factor. If the slip factor is defined, it can be used to approximate the actual centrifugal pump performance from impeller geometry. One of the slip factor approaches developed by Weisner is given in equation 1.15.

$$\sigma = \frac{1-\sqrt{\sin(\beta_2)}}{Z^{0.7}} \qquad 1.15$$

Where Z is the number of impeller blades. The correlation of the head and the flow coefficient with the slip factor can be derived as follows

$$\psi = \sigma - \phi\cot(\beta_2) \qquad 1.16$$

Volumetric efficiency (η_v)

$$\eta_v = \frac{g*H}{T*u_2^2} \qquad 1.17$$

1.5 Influence of impeller geometry on centrifugal flow behaviour

It is widely known that fluid flow inside the centrifugal pump is very complex and the geometry of the impeller has an important role in centrifugal pump performance. It can be inferred from the theoretical Euler prediction, see eq.1.2 that the head of the pump increases if the exit blade angle is larger because the larger exit blade angle causes the larger value of the tangential component of absolute velocity. Fig. 1.9shows the influence of the impeller shape on the centrifugal pump head. However, the characteristic of the centrifugal pump shown in Fig.1.14 only occurs if an infinite blade number and inviscid flow are assumed.

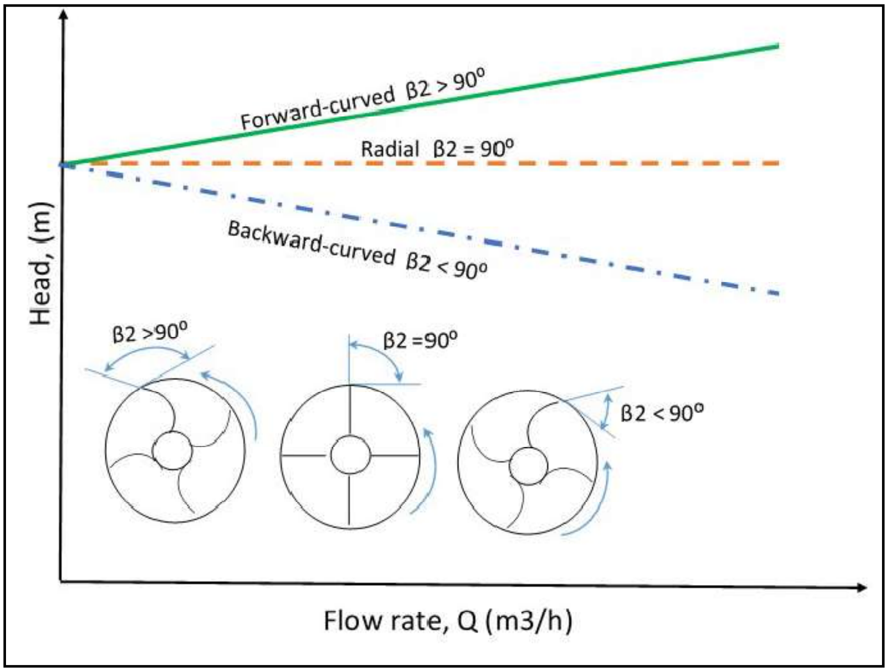

Figure 1.9: Influence of the blade shape on the theoretical head [4]

The number of blades becomes one of the factors contributing to centrifugal pump performance. Several studies observed the influence of blade numbers on the flow structure inside the impeller and the centrifugal pump performance. The influence of the number of blades on the centrifugal pump performance can be seen in Figure 1.10.

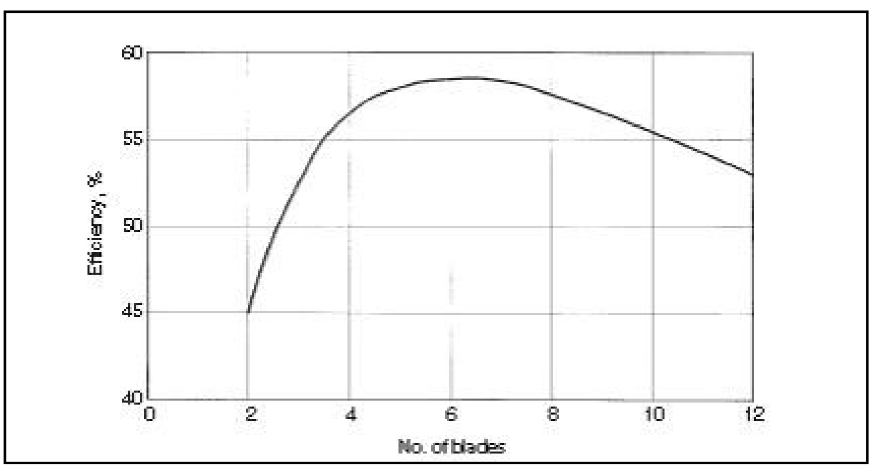

Figure 1.10: maximum efficiency curve; as a result of changing blade numbers [4]

1.6 Motivation

With the rapid development of the modern technology, marine equipment increasingly develops towards complication, precision and automation. The centrifugal pump is one of the most important equipment influencing normal sailing. Various kinds of centrifugal pumps are widely employed, such as fire fighting pump, hot water circulating pump, sanitary water pump and so on.

In order to make the manufacturing of centrifugal pumps more flexible and feasible studies & investigation has been conducted on the internal flow in centrifugal pump especially through the impeller. The main motivation behind this project is to increase the efficiency of pumps so as to increase the overall life expectancy of the centrifugal pumps. It is carried out by using advanced software codes to perform CFD analysis. As a result the performance of the pumps has been improved with reduced cost of producing pumps that have high durability.

1.7 Research Objective

The main objective of this thesis is to perform an examination of the pressure and velocity characteristics of flow through the impeller of a centrifugal pump using Computational Fluid Dynamics, and to optimize the design of the centrifugal pump impeller using Genetic algorithm in order to increase the efficiency and enhance the pump performance.

The objectives are listed as follows:

- Develop a mathematical approach for impeller design.
- Simulate the design and obtain our results.
- Hydrodynamic characteristic analysis of flow through the centrifugal pump impeller using CFD.
- Optimize the design of impeller using Genetic algorithm

1.8 Organisation of Thesis

The chapters in the thesis are arranged in the following manner.

Chapter 1 presents the background of this project which includes the research objectives, motivation and scope that is a frame line for this project. The primary aim of this chapter is to provide the reader with a basic idea of the work presented in the thesis.

Chapter 2 presents the literature survey performed in the field of centrifugal pump CFD analysis, their computational and experimental work and the algorithms involved in these literatures.

Chapter 3 demonstrates the design of the centrifugal pump. It contributes the analytical & computational approach to the design of pump.

Chapter 4 elaborates the computational modelling which involved ANSYS® modules explained.

Chapter 5 provides the results obtained by the simulation study and discuss the performance of centrifugal pump before and after optimisation.

Chapter 6 concludes the thesis and provides suggestions for future work.

CHAPTER 2

LITERATURE REVIEW

In recent times, centrifugal pump are finding their application in every field from chemical industries to daily household utility. This chapter contributes a literature survey on various applications of centrifugal pump & the different methodology used for their optimization. It also includes the different complex features involved in modelling & simulation of the software used for the project.

2.1 Numerical Analysis

Barrio et.al [5] conducted an investigation on the unsteady flow behaviour near the tongue region of a single-suction volute-type centrifugal pump with a specific speed of 0.47. The numerical predictions of velocity and pressure, obtained at several reference positions located at the near-tongue region, showed that:

1. The flow pulsation for medium and high flow rates is directly associated to the passage of the blades in front of each reference position.
2. The relative flow at the outlet of the impeller channels close to the tongue for the lowest flow rate showed a large counter- rotating vortex.

Shojaeefard et.al [6] conducted an experiment showing that the performance of centrifugal pumps drops sharply during the pumping of viscous fluids. Changing some geometric characteristics of the impeller in these types of pumps improve their performance.

The following conclusions have been made:

1. The friction on the discs including the wheel in the case of oil decreases the head and efficiency and increases the power consumption compared with the case of water.
2. Numerical results show that the impeller blade with the angle of 30_ and passage width of 21 mm produces a higher head relative to the other five blade settings

3. Increasing the blade outlet angle and the passage width decreases the pump efficiency

Huang et.al [7] used CFD to study the three-dimensional unsteady incompressible viscous flows in a centrifugal pump during rapid starting period (0.12 s). The computational transient performances qualitatively agree with published data, indicating that the present method is capable of solving unsteady flow in a centrifugal pump under transient operations.

Fuxiang et.al [8] used 3-D dynamic meshing technique for numerical simulation of unsteady flow fields in a centrifugal pump using the fluent software. Its results suggest that the Dynamic Mesh technique for flow simulation in centrifugal pumps, defined in an inertial reference frame, yields substantially greater computing efficiency than the Sliding Mesh method involving comprehensive data transfer among multiple reference frames.

Shou-qi et.al [9] has done Numerical simulation and 3-D periodic flow unsteadiness analysis for a centrifugal pump. The pressure fluctuation intensity coefficient (PFIC) based on the standard deviation method, the time-averaged velocity unsteadiness intensity coefficient (VUIC) and the time-averaged turbulence intensity coefficient (TIC) are defined by averaging the results at each grid node for an entire impeller revolution period. The following results are obtained:

1. The flow velocity unsteadiness intensity is larger near the blade suction side than near the pressure side.
2. Strong turbulence intensity can be found near the blade suction side, the impeller shroud side as well as in the side chamber.
3. The leakage flow has a significant effect on the inflow of the impeller, and can increase both the flow velocity unsteadiness intensity and the turbulence intensity near the wall.

Derakhshan et.al [10] used a global optimization method based on the Artificial Neural Networks (ANNs) and Artificial Bee Colony (ABC) algorithm has been used along with a validated 3D Navier–Stokes flow solver to redesign the impeller geometry and improve the performance of a Berkeh 32-160 pump as a case study.

The new impeller geometry presents much more changes in the meridional channel and blade profile. The results indicate a reasonable improvement in the optimal design of pump impeller and a higher performance using the ABC algorithm.

2.2 Cavitations in Centrifugal Pump

Abbas et.al [11] has done design, operation and refurbishment of centrifugal pumps are strongly related to cavitations' flow phenomena, which may occur in either the rotating runner-impeller or the stationary parts of the centrifugal pumps. The conclusions are as follows:

1. The inception of cavitations' is taking 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.
2. 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.

Cunha et.al [12] studied the Phenomenon of cavitation in centrifugal pump. Cavitation can be described as the vapour bubbles formation in an originally liquid flow, this change of phase is carried through at constant temperature and local drop pressure, generated by flow conditions.The cavitation pockets was observed in the place that was expected (low pressure places, first on the blades inlet) & a drop of 3% of Head with the critical Npsh value 1,44[m] and inlet pressure 17750[Pa] was observed.

Bing et.al [13] presented a paper to clarify the cavitation suppression mechanism of the gap structure impeller based on the analysis of cavitation characteristics in a low specific speed centrifugal pump. The conclusions are as follows:

1. It was confirmed that the new gap structure impeller has a very good characteristic of inhibiting cavitation, especially in large flow area.

2. The present numerical method can effectively capture the major internal flow features in the centrifugal pump, through the comparison of the two type impeller flow fields, the cavitation suppression mechanism of the gap

impeller may be the combination effects of the small vice blade's guiding flow and gap tunnel's auto-balancing of pressure.

Muthu et.al [14] analysed the cavitation in centrifugal pump through various turbulence models in CFD to get accurate numerical results.

The conclusions are as follows:

1. The effects of cavitation in the performance of centrifugal pump is identified, certain number of parameters is derived from the literatures which may act as causes for cavitation in the centrifugal pump. They are analysed using CFD.

2. Obtained optimum value to reduce the effect of cavitation has been represented in the work.

2.3 CFD Analysis

Li et.al [15] redesigned the impeller blades by using the method, and the three-dimensional turbulent viscous flows inside the original and redesigned impellers were calculated numerically by means of a CFD code Fluent.

The conclusions are as follows:

1. It is represented that the blade shape and flow pattern on the blade can be controlled easily by altering the density function of bound vortex intensity.

2. The CFD outcomes confirmed that the original impeller hydraulic efficiency was improved by 5% at the design duty, but 9% at off-design condition.

Ghorab et.al [16] presented a paper on improving the performance of an axial flow impeller with blades that wrap in a helix around a central hub and concluded that:

1. Usually inducers have between 2 and 4 vanes, although they may be more, the inducer imparts sufficient head to the liquid so that the NPSH requirement of the adjacent main impeller is satisfied.
2. The inducer usually has a lower NPSH requirement than the main impeller & it often cavitates during normal operation, the key is that there is so little

horse power involved with an inducer that there is virtually no noise, vibration, or resulting mechanical problems.

Margaris et.al [17] presented a paper describing the simulation of the flow into the impeller of a laboratory pump in a parametric manner. In this study, the performance of impellers with the same outlet diameter having different outlet blade angles is thoroughly evaluated. The one-dimensional approach along with empirical equations is adopted for the design of each impeller.

Voorde et.al [18] presented a paper onflow analysis of two test pumps of end-suction volute type: one of low specific speed and one of medium specific speed. For both, head as function of flow rate for constant rotational speed is known from experiments. It is found that the Multiple Reference Frame method (MRF) and Mixing Plane method (MP) methods lead to completely erroneous flow field predictions for flows far away from the best efficiency point. This makes the steady methods useless for general performance prediction.

Cheng et.al[19] used LabVIEW programming software from American NI Corporation to monitor a centrifugal pump system.

The conclusions are as follows:

1. A lot of testing experiments have been conducted. He proved that the system can acquire the performance parameters, analyze these data, draw the conclusion, and help engineer manage the pumps.

Ynag et.al [20] studied the transient characteristics of a closed-loop pipe system with room temperature water through experiments which were carried out based on different pump stopping periods from rate rotational speed to zero. The following results were obtained:

1. Rapid change of the pump operating conditions occurs during the stopping period and transient flow rate of the pipe system and characteristics of the pump depend largely on the way of stopping.

2. The kinetic energy stored in the pump can drive the impeller keeping rotating for more time after the motor is shutdown.

3. Due to the kinetic energy stored in the loop pipe, the flow rate does not reach zero immediately after the rotational speed reaches zero.
4. The inertia of pump rotor and fluid inertia affect the impact of fluid flow and the duration of the loop during pump stopping period.

Brenner et.al [21] demonstrated the applicability of an eddy resolving turbulence model in a turbomachinery configuration. The model combines the Large Eddy Simulation (LES) and the Reynolds Averaged Navier Stokes (RANS) approach. Its results demonstrate that both models are able to predict the major stall frequency at part load. Results are similar for URANS and SAS, with advantages in predicting minor stall frequencies for the turbulence resolving model.

Shah et.al [22]. reviewed the CFD analysis of centrifugal pumps along with the future scopes for further improvement is presented in this paper. Unsteady Reynolds-averaged Navier-Stokes equations together with two equation k-ϵ turbulence model were found to be appropriate for CFD analysis of Centrifugal pump. Volute flow study and impeller-volute interaction appeared as an interesting research fields for the further improvement in the pump performance.

Yan et.al [23] developed a constrained least-squares reconstruction method to improve spacial accuracy. A parallel, accurate, robust and grid-transparent CFD solver was developed to solve compressible flow on 3D arbitrary polyhedral grids. To improve spacial accurate, a constrained least-squares reconstruction method is developed. To accelerate convergence, a matrix free implicit method GMRES+LU-SGS, and a parallel method using OpenMP are presented on shared-memory parallel systems with help of a special grid reordering technique. Several typical test cases, including subsonic, transonic and hypersonic flows, prove that the resulting solver performs fast, accurate and robust.

Samad et.al [24] conducted a study on fitting the Inner guide vanes at the entrance of centrifugal fan impeller to resolve the non-uniformity of the flow and to get rid of the vortices that are generated by the existence of Inner distortion.

The following results are observed:

1. Poor efficiency and lack of capacity control range are observed when performance tested in various applications.
2. Improvement in the stable operating range and reduction in part load power consumptions, particularly in off-design conditions was achieved by upgrading the aerodynamics and fitting Inner guide vane control to the machines in this work.

Dribssa et.al [25] studied with the aid of computational fluid dynamics, the complex internal flows in water pump impellers. From the simulation results it was observed that the flow change has an important effect on the location and area of low pressure region behind the blade inlet and the direction of velocity at impeller inlet. From the study it was observed that FLUENT simulation results give good prediction of performance of centrifugal pump and may help to reduce the required experimental work for the study of centrifugal pump performance.

Gupta et.al [26] studied the dense slurry flow through centrifugal pump casing that has been modeled using the Eulerian-Eulerian approach with Eulerian multiphase model in FLUENT 6.1®. It is observed that despite the difference in the turbulence models (k-ε Vs. Mixing length), the results are found to be within 12% difference.

Gawade et.al [27] studied with the aid of CFD the complex internal flow in horizontal split case pump. Efficiency of the pump from CFD results is coming 82 % and by actual performance test efficiency is coming 81.37%, by which it is confirmed that CFD analysis is clearly validated. Traditional volute design is based on two-dimensional analysis, and the emphasis is on collection (Impeller) and less on the diffusion (Volute) function. However, with the use of advanced fluid modeling tools, it is possible to design a volute using three-dimensional analysis. This shows that CAD and CAE tools are very useful in hydraulic design

Sun et.al [28] presented a paper using INVENTER software to establish the model of main valve and the servo valve is simulated by FLUENT software. He concluded that Outlet valve is prone to cavitation as it has sharp edge .If we adopt curve or chamfer it is easy to reduce cavitation.

Zeng et.al [29] presented a paper on three-dimensional flow field of the whole flow passage of a mixed-flow pump. He observed that the flow rate of the pump can be increased by:

1. Reducing the number of blades,
2. Increasing the blade inlet structure angle
3. Reducing the blade outlet structure angle

2.4 Experimental Investigation

Dou et.al [30] studied the performance of centrifugal pumps during transient operating periods. This research result shows that there exists clear transient effect during the rapid startup period, and the quasi-steady analysis is unable to accurately assess the transient flow.

Dong-xi et.al [31] studied the development of attached sheet cavitation in centrifugal pumps. He observed that the cavitation bubbles were observed in the entrance of the impeller. He concluded that FBM model and the modified Zwart model are effective for the numerical simulation of the cavitating flow in centrifugal pumps.

Nourbakhsh et.al [32] studied the influence of slip phenomenon on the performance of centrifugal pumps. It was observed that in the design-point condition of the pumps, the experimental values are in a good agreement with the theoretical values. However, there are significant disagreements between the theoretical and experimental values at off-design regiments. The difference is more apparent at low flow rates. It is also found that the slip factor depends on the impeller-outlet velocity profile. Finally, he presented a correlation between slip factor & geometry of centrifugal pumps impeller..

Villanueva et.al [33] analysed the failure mode of the six impellers of a centrifugal pump in an irrigation system used for street washing. The results show a very high level of torsional vibrations induced by severe pulsations of engine torque. These vibrations were mainly responsible for damage to the impellers.

Villanueva et.al [34] presented a comprehensive theoretical non-linear torsional dynamic model of a pump-coupling engine assembly. He observed that reduction of the coupling stiffness leads to a decrease in the first torsional natural frequency

2.5 Design Optimisation

Kozmar et.al [35] proposed a method to improve efficiency of centrifugal pump. He proposed a method of impeller trimming and tested the new designed impeller successively & concluded that the influence of disregarded similarity can be estimated to ±3.94% for the pump head and to ±5.24% for the power, both with a 95% statistical certainty.

Mendiratta et.al [36] has developed a work to study the performance analysis of a centrifugal radial flow pump designed to deliver 0.0074 m^3/s of water with a head of 30 m at a speed of 2870 rpm using ANSYS® CFX (ver.14.0). The performance of the pump was first determined using the existing number of the blades and then, the number of blades has been varied to analyze the pump's performance. The results show that for the optimized value, pump efficiency increased by 2.23%.

Issac et.al [37] conducted a study using Computational Fluid Dynamics (CFD) approach to investigate the flow in the centrifugal pump impeller using the Ansys® Fluent. Impeller is designed for the head (H) 70 m; discharge (Q) 80 L/sec; and speed (N) 1400 rpm. He concluded that the backward curved vanes have better performance than the forward curved vane.

Mendriratta et.al [38] conducted a study on the performance analysis of a centrifugal mixed flow pump designed to deliver 0.25 m3/s of water with a head of 20 m at a speed of 1450 rpm using ANSYS® CFX (ver.14.0). PTC Creo (ver. 2.0) has been used to model the pump unit. The results show that for an initial inlet angle 21.08˚, outlet angle 16.28˚ and blade thickness as 10mm, the efficiency of the pump was 84%. However, the efficiency of pump rises to 89.19% for the optimized angles and blade thickness.

Chen et.al [39] proposed a new multi-objective optimization method for a family of double suction centrifugal pumps with various blade shapes, using a

Simulation-Kriging model-Experiment (SKE) approach. The Kriging Meta model is established to approximate the characteristic performance functions of a pump, namely, the efficiency and required net positive suction head (NPSHr). Hence, the two objectives are to maximize the efficiency and simultaneously to minimize NPSHr. The results show that the solution of the proposed multi-objective optimization method is in line with the experiment test.

Wang et.al [40] conducted a study in order to improve internal unsteady flow in a double-blade centrifugal pump (DBCP), major geometric parameters of the original design as the initial values, heads as the constraints conditions, and the maximum of weighted average efficiency at the three conditions as the objective function. Performance characteristic test results show that the weighted average efficiency of the impeller after the three-condition optimization has increased by 1.46% than that of original design. PIV measurements results show that vortex or recirculation phenomena in the impeller are distinctly improved under the three conditions

2.6 Summary of the chapter

In this chapter, the literature review has been represented from the various research paper studied. The topic wise discussion was represented for numerical analysis, cfd analysis, experimental investigation & design optimization. In next chapter the pump design is presented.

CHAPTER 3

PUMP DESIGN

In this chapter the centrifugal pump is designed in ANSYS® (VISTA CPD) and the results obtained are further validated through theoretical approach which includes characteristics curves of the pump which have developed by Lobanoff [41].

The classification of pump designed has been classified as follows:

1. Analytical approach
2. Computational approach

3.1 Analytical approach

This topic discusses in detail the design of a centrifugal pump impeller with the help of characteristics curves. The described design factors are based on a theoretical approach and collecting experimental test results over a time period of several years.

3.1.1 Impeller Design Methodology

It is the rotating part of the centrifugal pump that is used to supply energy to the fluid. The design of pump impeller is as follows:

Step 1: Calculate Specific speed(N_s).

$$N_s = \frac{RPM \times (GPM)^{0.5}}{H^{0.75}} \qquad 3.1$$

$$= \frac{3,600 \times (2,100)^{0.5}}{450^{0.75}} = 1668.51$$

Step 2: Select Vane number and discharge angle

The desired rate of head rise from point of highest efficiency to zero discharge is 20% continuously rising from Fig.3.1 and to produce the required rate of rise in head, 6 equally spaced number of vanes having a 25° discharge angle are assumed with the help of Fig. 3.2.

Step 3: Calculate impeller diameter

The value of head constant (K_u) is found by Fig.3.3 and this value is used to calculate the impeller diameter (D_2) by equation 3.2.

Head constant $K_u = 1.075$

$$D_2 = \frac{1840 \times K_u \times (H)^{0.5}}{RPM} \tag{3.2}$$

$$= \frac{1{,}840 \times 1.075 \times (450)^{0.5}}{3600} = 11.66 \text{ in.}$$

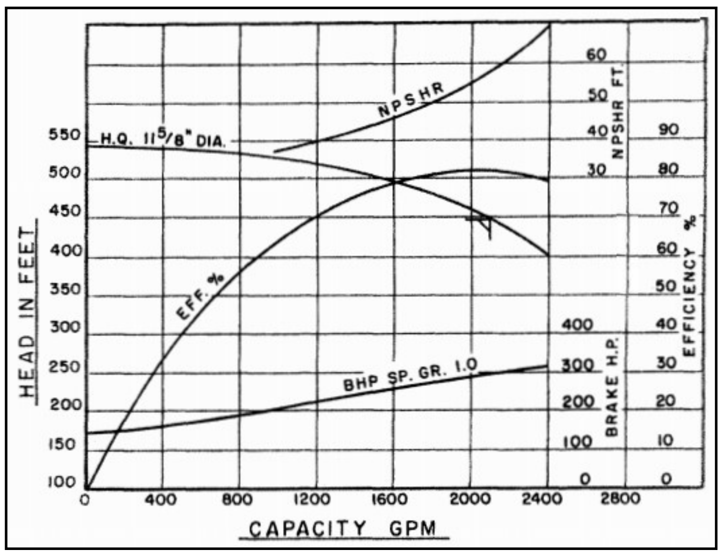

Fig. 3.1: Head, NPSHR, Brake power &efficiency Vs. Capacity [44]

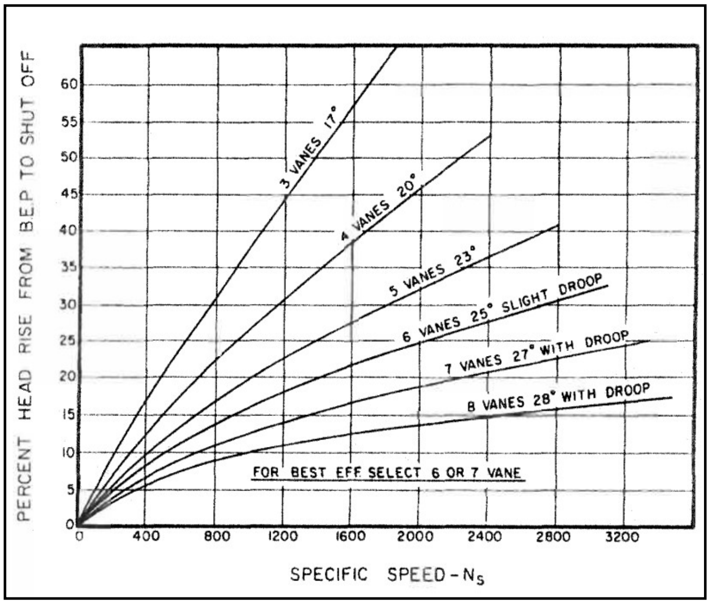

Fig. 3.2: % Head rise Vs Specific Speed[44]

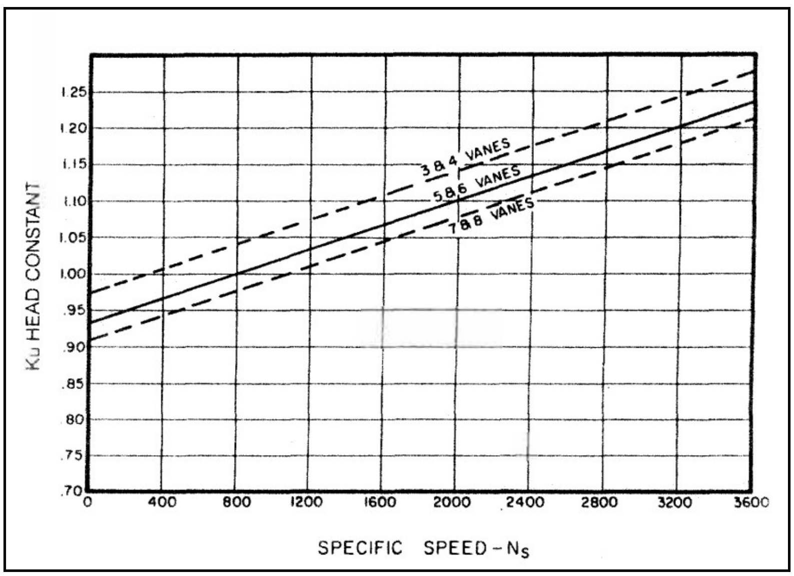

Figure 3.3: Head constant Vs Specific Speed [44]

Step 4: Calculate impeller width (b_2)

The value of capacity constant (K_{m2}) is found by Fig.3.4 and this value is used to calculate the radial velocity at impeller discharge (C_{m2}) by equation 3.2 and finally b_2 by equation 3.3.

$$K_{m2} = 0.125$$

$$C_{m2} = K_{m2} \times 2gH^{0.5} \qquad 3.2$$

$= 0.125 \times 170 = 21.3$ ft/sec

$$b_2 = \frac{GPM \times 0.321}{C_{m2} \times (D_2 \pi - ZS_u)} \qquad 3.3$$

Estimated $S_u = \frac{1}{2}$ in.

$$b_2 = \frac{2{,}100 \times 0.321}{21.3 \times (11.3 \times \pi - 6 \times 0.5)} = 1.09 \text{ in.}$$

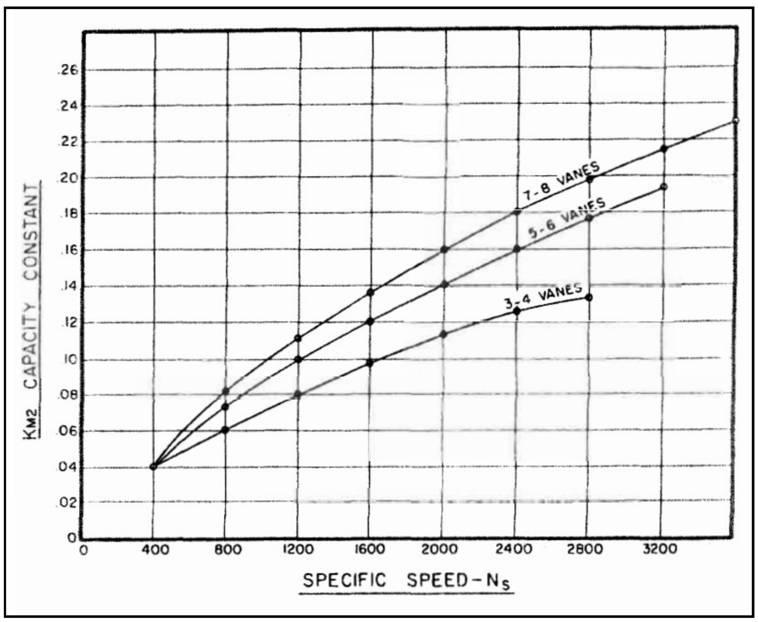

Figure 3.4: Capacity constant Vs Specific Speed [44]

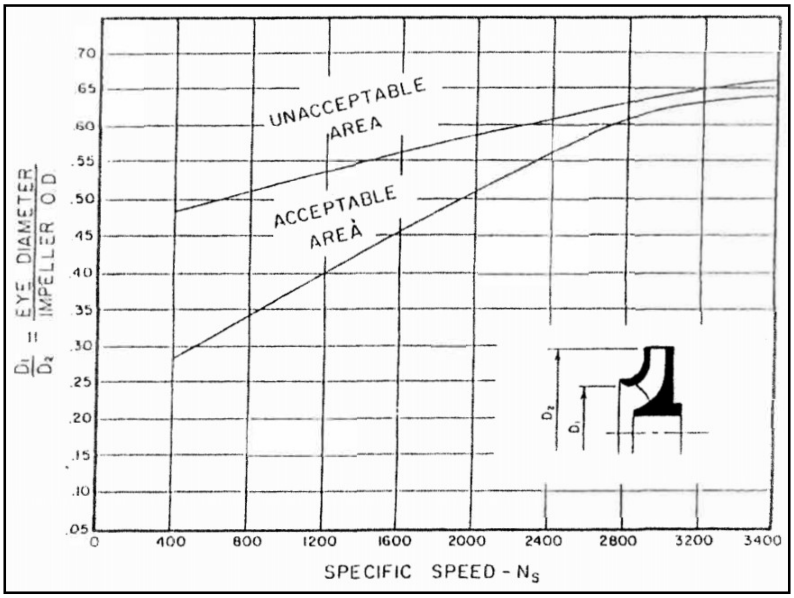

Figure3.5: D_1/D_2 Vs Specific Speed [44]

Step 5: calculate eye diameter

The value of ratio of D_1/D_2 is found by Fig 3.5 and as we already know the value of D_2, so we can easily find the value of $D_{1.}$

$$\frac{D_1}{D_2} = 0.47\,;\; D_1 = 11.66 \times 0.47 = 5.5\; in.$$

Step 6: Determine shaft diameter under impeller eye (D_{shaft})

The shaft diameter under the impeller eye is assumed to be 2 in.

Step 7: Estimate Impeller eye area (A_e)

Eye area (A_{eye}) =Area at Impeller Eye – Shaft Area 3.4

=23.76 - 3.1

= 20.66 sq in.

Step 8 Estimate NPSHR

The value of Suction Eye velocity (C_{m1}) and peripheral velocity (U_t) is found by equation 3.5.

$$U_t = \frac{D_1 \times RPM}{229} \tag{3.5}$$

$$= \frac{5.5 \times 3{,}600}{229}$$

$= 86.5$ ft/sec

$$C_{m1} = \frac{0.321 \times GPM}{A_e} \qquad 3.6$$

$$= \frac{0.321 \times 2{,}100}{20.66}$$

$= 32.63$ ft/sec

From Fig. 3.6

NPSHR = 59 ft

$$N_{ss} = \frac{3{,}600 \times (2{,}100)^{0.5}}{59^{0.75}} = 7{,}749$$

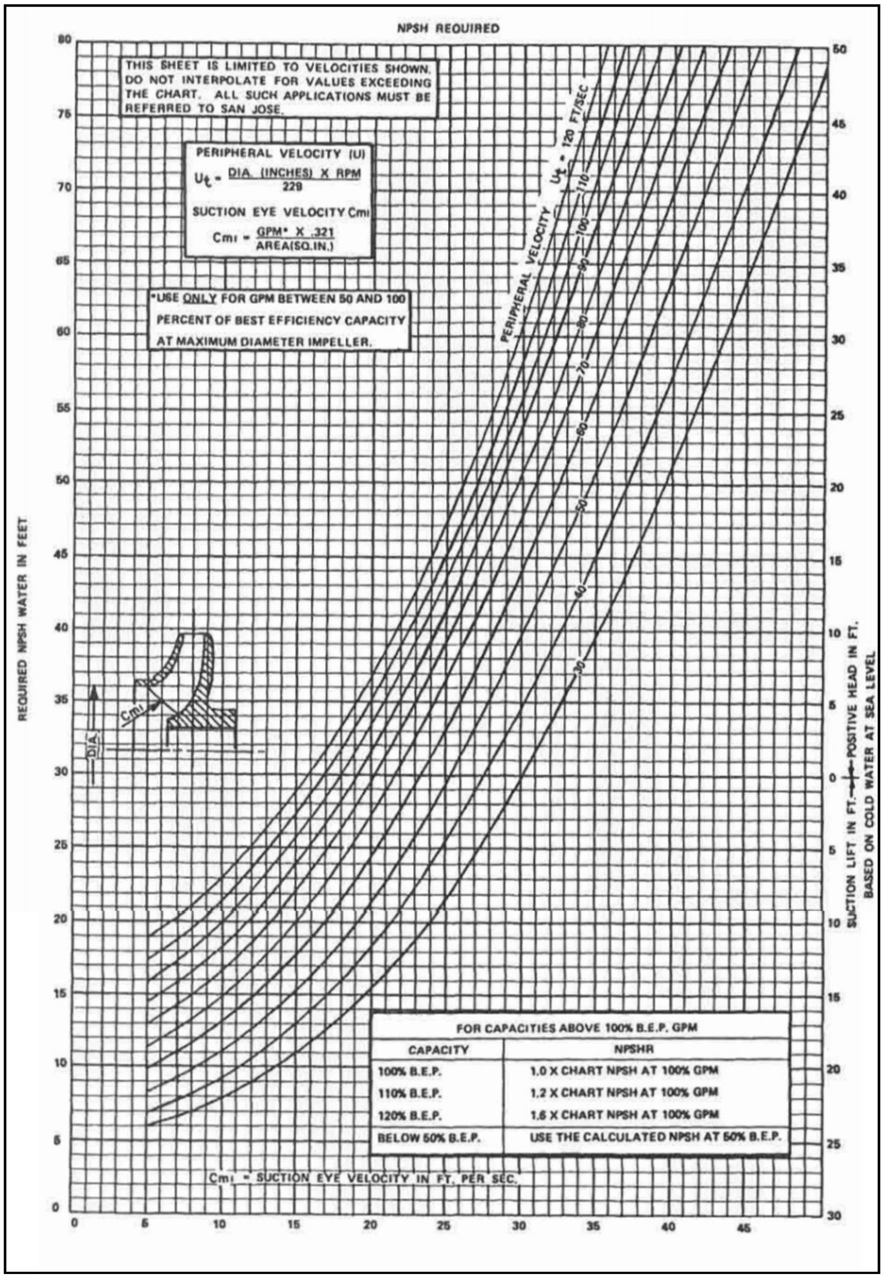

Figure 3.6: NPSH prediction chart [44]

3.1.2 Volute Design Methodology

It is the stationary part of the centrifugal pump that receives the fluid being pumped by the impeller. Its main purpose is to keep the velocity constant.

The design of pump volute casing is as follows:

Step 1 Volute area

Fig. 3.7 shows a number of curves for volute velocity constant K_3 .These represent the statistical gathering efforts of a number of major pump companies.

$$A_8 = \frac{0.04 \times GPM}{K_3 \times (H)^{0.5}} \qquad 3.7$$

$$= \frac{0.04 \times 2{,}100}{0.365 \times (450)^{0.5}}$$

$$= 10.85\ sq\ in.$$

The calculated volute area is the final area for a single volute pump.

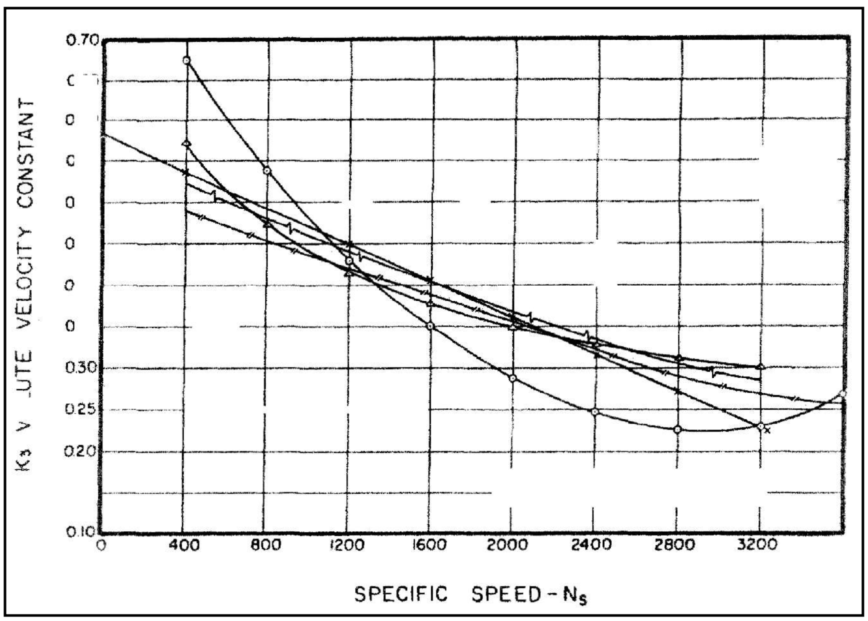

Figure 3.7: Volute velocity constant Vs Specific Speed [44]

Step 2: Establish volute width (b_3)

In determining the width of the volute, the need to accommodate impellers of different diameter and b_2 must be considered. Distance from the impeller shroud to the stationary casing should be sufficient to allow for casting inaccuracies yet still maintain a satisfactory minimum end play. The values shown in Table3.1 reflect those published by stepanoff and are reasonable guidelines.

Volute Width = $1.09 \times 1.75 = 1.9$ in.

Step 3: Establish cutwater diameter

A minimum gap must be maintained between diameter and volute lip to prevent noise, pulsation, and vibration, particularly at vane passing frequency. From Table 3.2:

$$D_3 = D_2 \times 1.07 = 11\frac{5}{8} \times 1.07 = 12\frac{7}{16}\text{ in.}$$

Table 3.1: Guidelines for Volute Width

Volute Width (b_3)	**Specific Speed (N_s)**
2.0 b_2	< 1,000
1.75 b_2	1,000 – 3,000
1.6 b_2	>3,000

Table 3.2: Guidelines for Cutwater Diameter

Specific Speed (N_s)	**Cutwater Diameter (D_3)**
600-1000	$D_2 \times 1.05$
1000-1500	$D_2 \times 1.06$
1500-2500	$D_2 \times 1.07$
2500-4000	$D_2 \times 1.09$

3.2 Computational approach

This topic discusses in detail the design of a centrifugal pump impeller with the help of ANSYS®-Vista CPD. The described design factors are based on pre-programmed design approach stored in Vista CPD. The pump

3.2.1 Baseline Pump Design

The baseline geometry of pump is designed through Vista CPD. The rotational speed (N), volume flow rate (Q), density of the pump fluid (ρ), height (H) and other geometry specification at BEP of the centrifugal pump are entered in Vista CPD interface. It utilises the above entered data to generate the basic pump parameters like flow coefficient, specific speed etc.

3.2.1.1 Baseline Impeller Design

The parameters obtained in Vista CPD are then used in BladeGen to generate a 3D geometry of the impeller. The basic parameters obtained in Vista CPD for e.g. number of blades, inlet & outlet flow angle & thickness of the blade can be edited to suit our requirements. The calculated design parameters are shown in Fig.3.8 to 3.14.

The 3D view of Impellers & its blade are shown in fig 3.15 & 3.16 & and the calculated design parameters are shown in table 3.3

3.2.1.2 Baseline Volute Design

The parameters obtained in Vista CPD are then used in volute modeller to generate a 3d geometry of the volute. The central section view of the volute casing is shown in fig. 3.17 and generated geometry is shown in fig. 3.18. Table 3.4 to 3.6 shows the specifications of the designed volute.

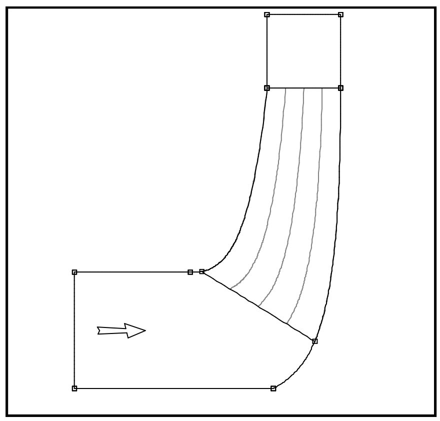

Figure 3.8: Meridional view of impeller blade

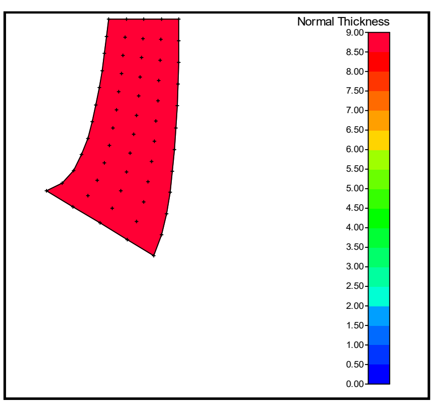

Figure 3.9 Normal thickness

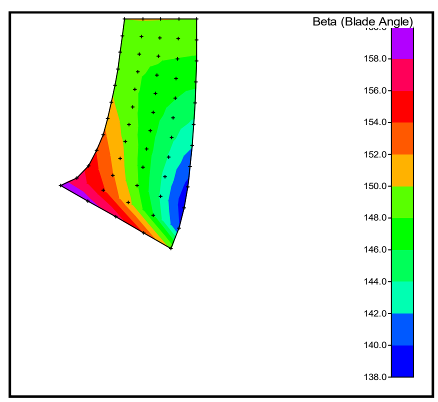

Figure 3.10 : Outlet Blade angle

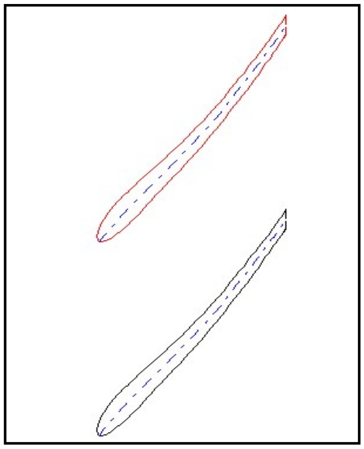

Figure 3.11: Blade to Blade view

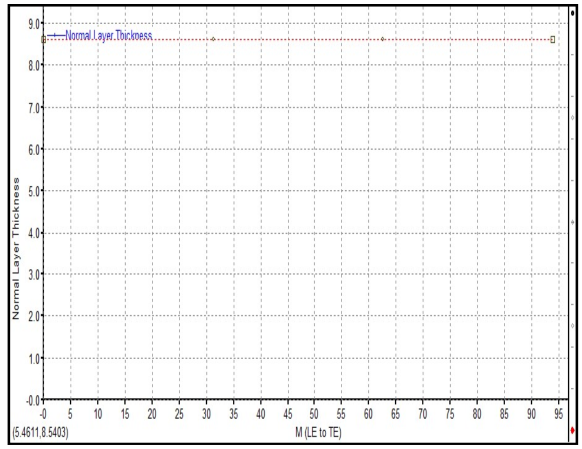

Figure 3.12: Normal layer thickness Vs M

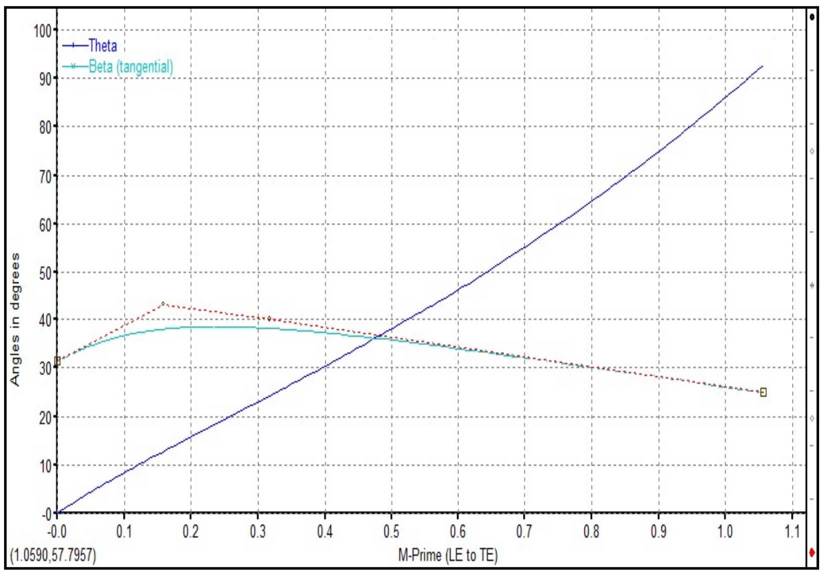

Figure 3.13: Blade angles vs M-prime

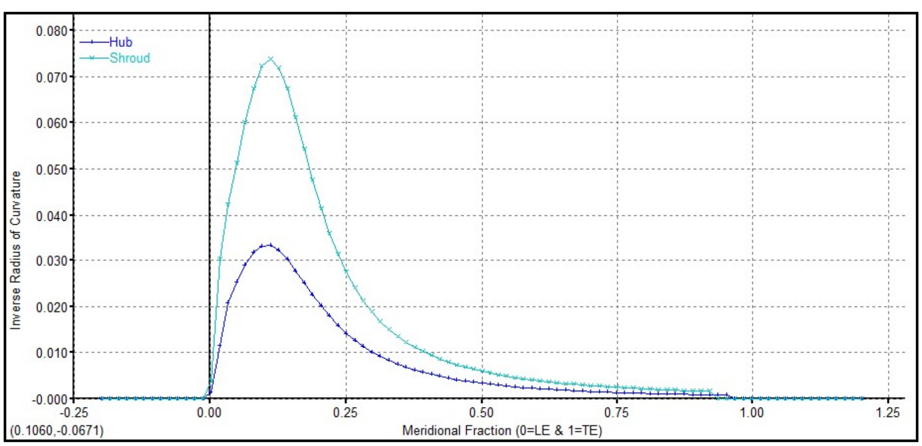

Figure 3.14: Inverse radius of curvature Vs M-factor

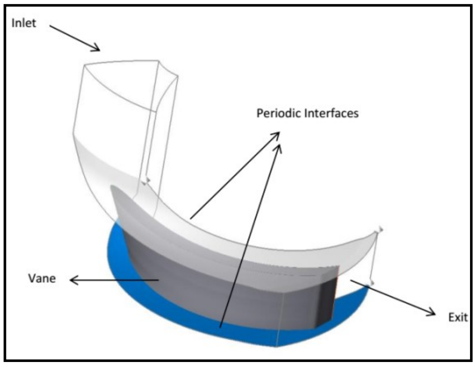

Figure 3.15 :3D view of impeller blade

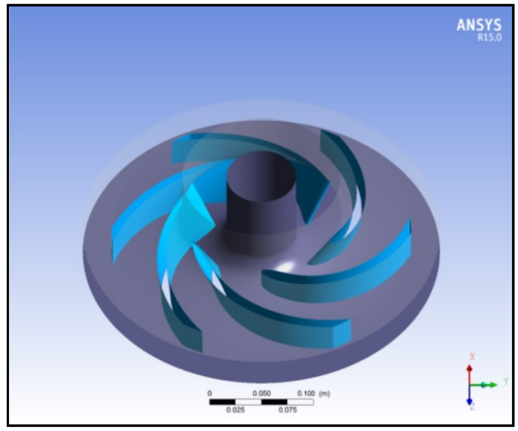

Figure 3.16 :3D view of impeller

Table 3.3: Impeller Specification

Design parameter	Analytical approach	Vista CPD
Head (in ft)	450 ft	450
Flow rate (in GPM)	2100	2100
Rotating speed (in RPM)	3600	3600
Hydraulic efficiency (in %)	66.58 %	70.96 %
Specific speed (N_s)	1688.51	1741
Inlet diameter (in mm)	139.7	152
Outlet diameter(in mm)	296.164	287.4
Hub diameter(in mm)	65.3	66.9
Outlet angle (in degrees)	25	25
Outlet width (in mm)	27.686	27.9
Blade thickness (in mm)	8.0	8.6
Blade number (n)	6	6
NPSHr (in ft)	59	54

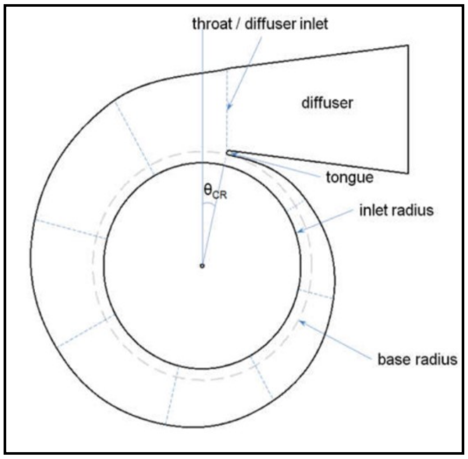

Fig. 3.17: Central section of volute casing

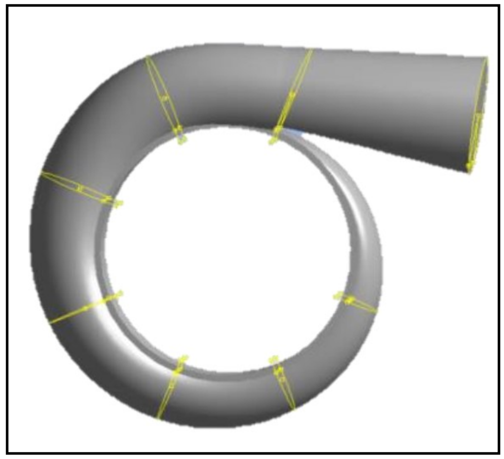

Fig. 3.18: 3D view of volute casing

Table 3.4: Volute Specification

Section detail	Analytical approach	Vista CPD
Inlet width	48.26 mm	55.8 mm
Base circle radius	157.96 mm	159.4 mm

Table 3.5: Diffuser Specification

Exit area	10737 mm^2
Exit hydrodynamic diameter	116.9 mm
Length	193.2 mm
Cone angle	7°

Table 3.6: Sections, Cutwater to Throat

No.	Area	Centroid radius	Outer radius	Major radius	Minor radius	
1	0	159.4	159.4	27.9	0.0	Cutwater
2	704	166.2	175.5	27.9	16.1	
3	1467	173.2	191.5	28.2	28.2	
4	2273	178.8	203.3	30.8	30.8	
5	3116	183.8	213.4	34.2	34.2	
6	3991	188.4	222.5	37.7	37.7	
7	4896	192.6	230.7	41.1	41.1	
8	5828	196.5	238.4	44.4	44.4	
9	6835	201.6	247.3	47.8	47.8	Throat

3.3 Summary of the chapter

In this chapter, the design methodology for impeller & volute casing of a centrifugal pump by analytical & computational approach has been discussed in detail. Next chapter will present the computational model of the centrifugal pump using ANSYS®.

CHAPTER 4

COMPUTATIONAL MODELLING

Computational modelling is the use of computers to simulate and study the behaviour of complex systems using mathematics, physics and computer science.

There are three elements to CFD analysis. The first one is CFX- PRE processor used to case file and definition. The second one is CFX-Solver for results file, and the last is CFX-Post for resulting and data analysis. In addition, ANSYS® simulation software simplifies the numerical solution of turbo machinery impeller blades rows. Blade is drawn in a simple 1D mean line method in BladeGen after getting the dimensions from the Vista CPD design tools for centrifugal pump, then exported in Design Modeler to do the geometry of the impeller blade. In similarity, volute geometry is done using Modeler geometry and an unstructured mesh in order to be exported to CFX processor.

ANSYS® TurboGrid is used for meshing of the impeller blade then it is exported to CFX for physical model definition, solving, and post-processing.

This chapter will be focusing on the study of pre-processing as follows:

- ✓ Geometry of Pump
- ✓ Meshing
- ✓ Physical model definition
- ✓ CFD simulation
- ✓ Boundary condition

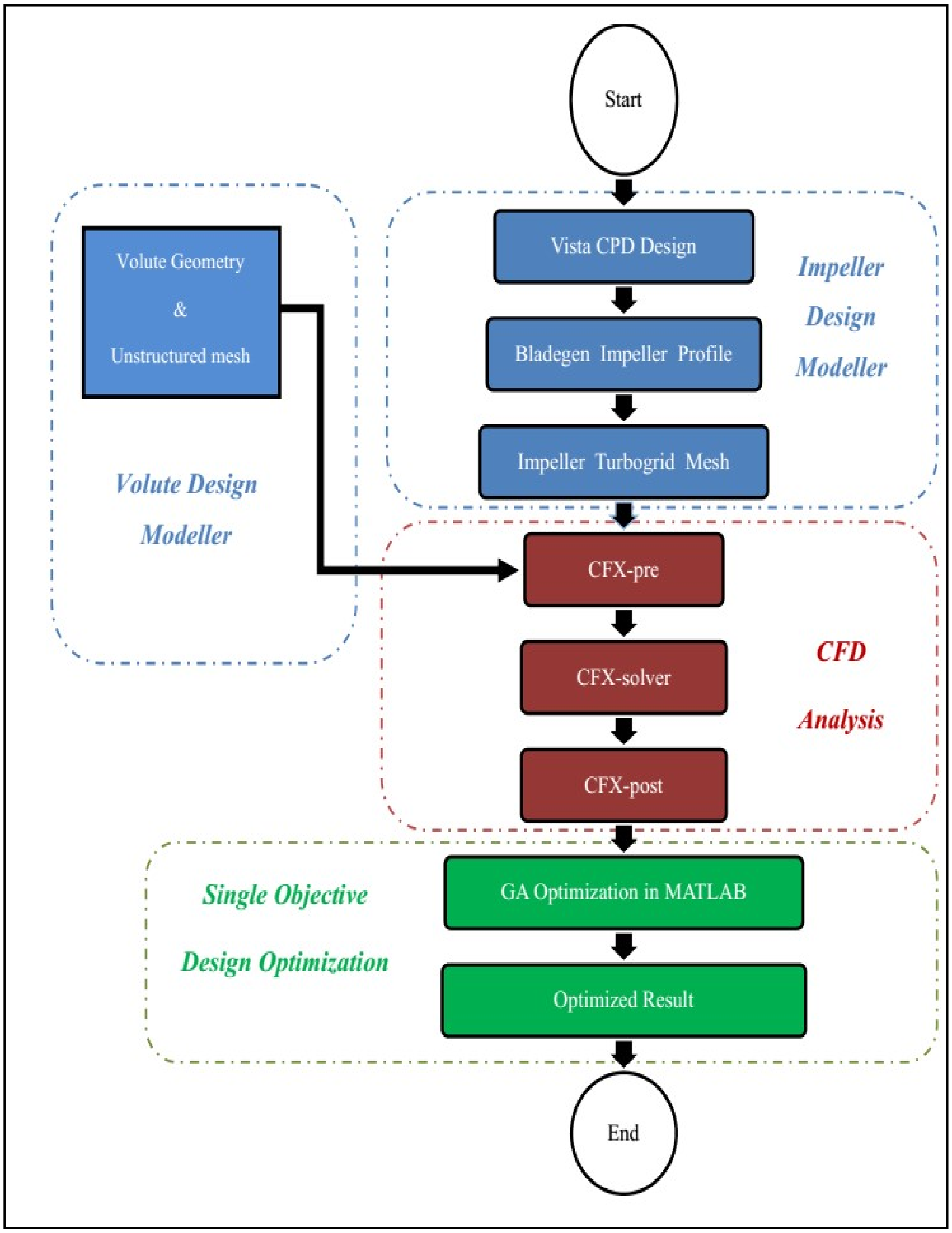

Fig.4.1: Flow chart of design process

4.1 Geometry of Pump

The impeller is considered as the rotor of the centrifugal pump [R1], while the volute is considered as the stationary part [S1]. The design of impeller & volute is done through BladeGen and Design Modeler respectively. Fig.4.2 and 4.3 show the geometry of the rotor &stationary part of the pump.

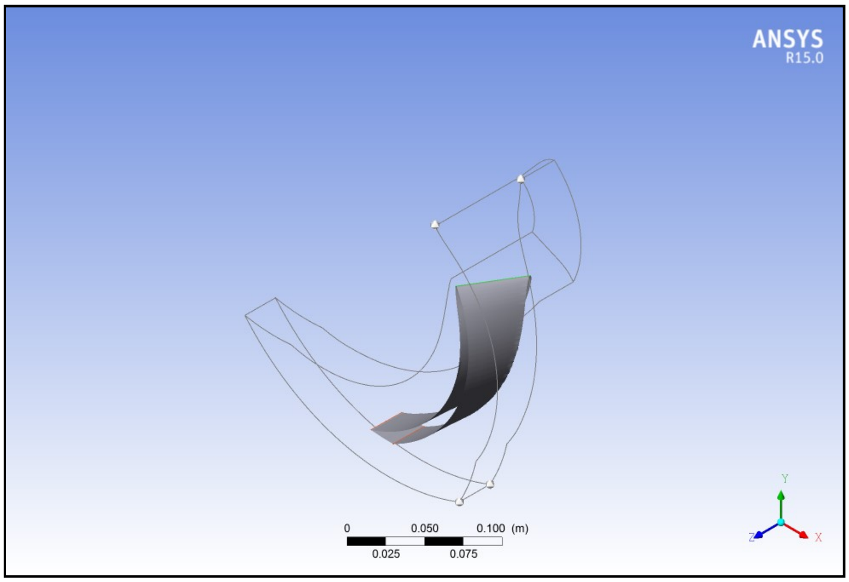

Fig.4.2: Blade Profile & outline

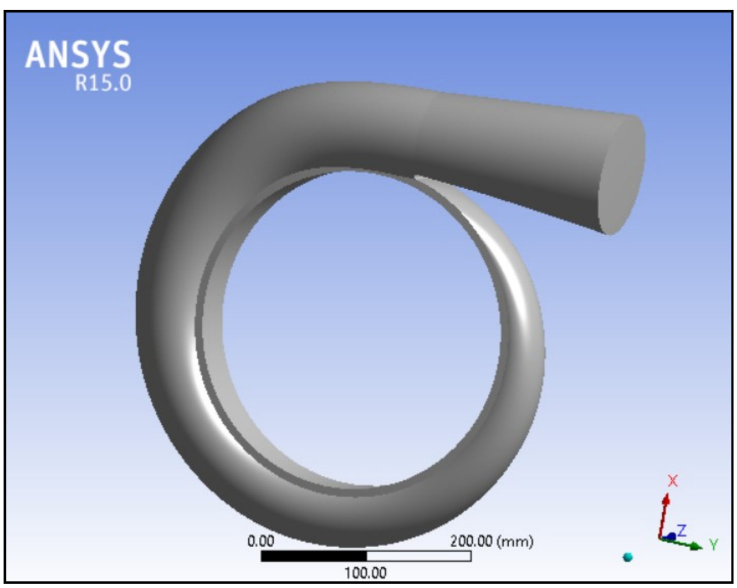

Fig.4.3: Volute casing

4.2 Mesh

4.2.1 Impeller Mesh

The impeller blade models were meshed by the use of ANSYS® TurboGrid as shown fig.4.4. Normal sized tetrahedron & hexahedron mesh was used in the meshing of the blade. The number of nodes & elements are shown in Table 4.1.

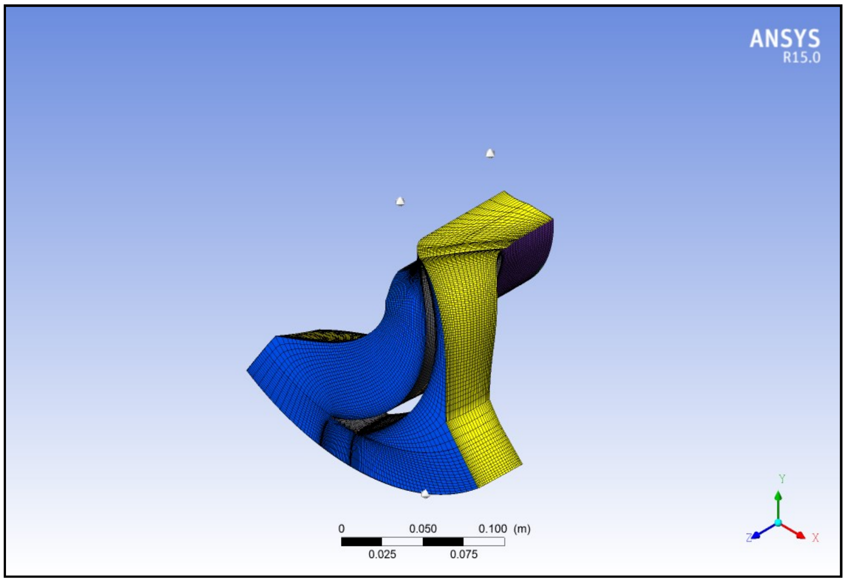

Fig.4.4: Impeller mesh

Table 4.1 Impeller mesh

Number of nodes	Number of elements
170534	154725

4.2.2 Volute (Scroll Casing) Mesh

The unstructured mesh of the volute was generated by finite element modeler as shown in fig.4.5. Unstructured grids used were concentrated near the cutwater areas. The number of nodes & elements are shown in Table 4.2.

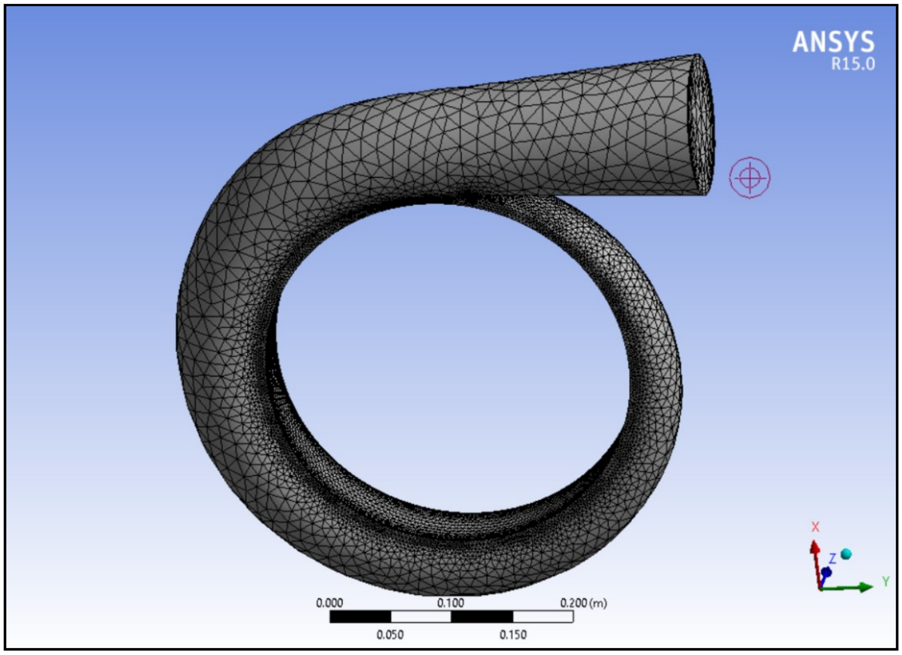

Fig.4.5: Volute mesh

Table 4.2 Volute mesh

Number of nodes	Number of elements
61975	182396

4.3 Physical model

The CFD simulations have been carried out with uniform inlet and outlet boundary conditions obtained from the mean-line analysis. The fluid flow is modeled as incompressible flow using water as a working fluid. The wall of solid is modeled as no-slip condition. The specifications of the CFX-preprocessing setup are provided in Table 4.3 and shown in fig.4.6

Table 4.3 Pre-processing specification

Analysis type	Steady state
Interference	Staged
Convergence criteria	K-Є
Reference pressure	1 atm
Residual type	RMS
Residual target	1E-6
Inlet[R1]	
Mass flow rate	0.1384 m^3/sec
Turbulence intensity	Medium (5%)
Outlet[S1]	
Static pressure	0 atm
Wall boundaries	
Mass and momentum	No slip condition
Wall roughness	Smooth

4.4 CFD simulation

Through CFD, the nature of the complex internal flow in the impeller of the pump can be predicted accurately. A steady state solution with k-Є turbulence model was used in CFX for both baseline and optimized models of centrifugal pump as it is very stable & converges accurately. CFX-post was used to find the pressure & velocity distribution, turbulence kinetic energy vectors &streamlines along the blade of the impeller.

4.5 Boundary condition

The following boundary conditions are used:

1. At the inlet of the pump mass flow rate is equal to 0.1384 m^3/sec

2. At the outlet of the pump static pressure is equal to 0 atm
3. No slip conditions are assumed at the walls of blades, hub & shroud of the pump
4. The interface between the rotor & stationary part of the model is staged

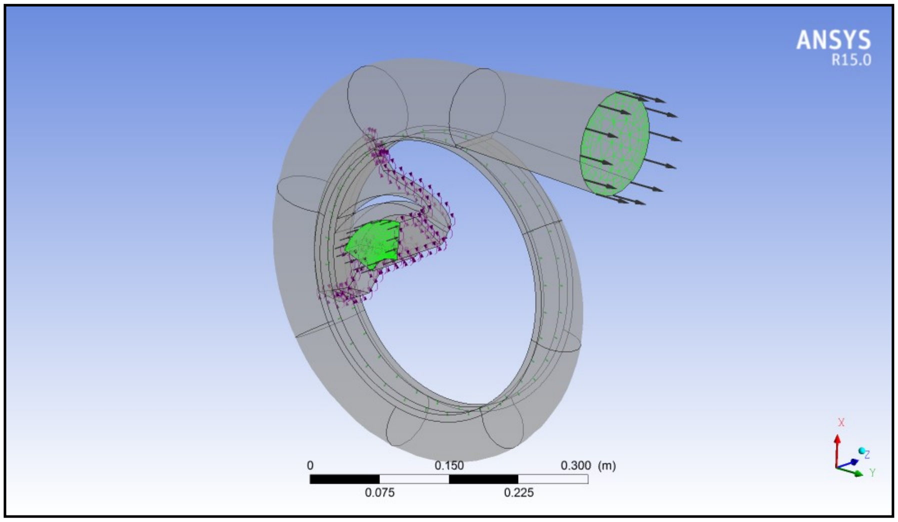

Fig.4.6: Interface between impeller blade & volute casing.

4.6 Summary of the chapter

In this chapter, the software technique & boundary conditions for solving the problem has been discussed. It includes the details of geometry of the pump its meshing, physical model definition, CFD simulation, boundary condition. The next chapter discusses about the results of the simulation before and after simulation.

CHAPTER 5

RESULTS & DISCUSSION

The physical model defined in CFX-pre was solved in CFX-solver and after the solution is converged the results are obtained in CFX-post. The hydrodynamic characteristics are studied & optimized through the use of genetic algorithm.
In this chapter the results and optimization are discussed as follows:

1. Hydrodynamic characteristics
2. Optimization with genetic algorithm
3. Conclusions

5.1 Hydrodynamic characteristics

In this we will discuss the various hydrodynamics distribution plots of the centrifugal pump. It is classified as follows:

5.1.1 Pressure Distribution

Pressure distribution curves of the baseline centrifugal pump are shown in fig.5.1 & 5.2. Pressure of the flowing fluid increases as it moves along the blades of the impeller & the minimum value of pressure in stationary frame is '-1.82996 x 10^6Pa' which is obtained at the leading edge of the impeller blades& maximum value of '1.87891x 10^6Pa' is obtained at the trailing edge of the impeller blades.

5.1.2 Velocity Distribution

Velocity distribution curves of the baseline centrifugal pump are shown in fig.5.3 &5.4.Velocity of the flowing fluid increases as it moves along the blades of the impeller. The minimum value of velocity in stationary frame is '18.9391 m/s ' which is obtained at the leading edge of the impeller blades & maximum value of '57.13 m/s ' is obtained at the trailing edge of the impeller blades.

5.1.3 Turbulence Kinetic Energy

Turbulence kinetic energy distribution curves of the baseline centrifugal pump are shown in fig.5.5 (a) & (b). The minimum value of turbulence kinetic energy is

‘2.613 x 10^{-2} m^2/s^2 ‘& maximum value of ’17.55 x 10^1 m^2/s^2 ’ is obtained along the impeller blades.

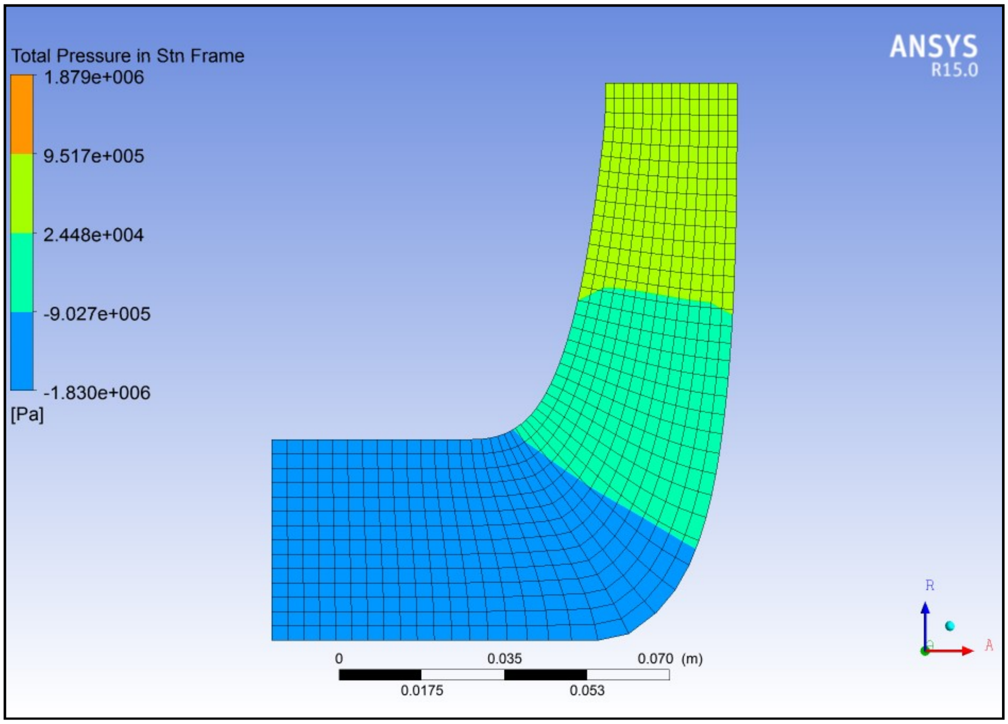

Fig.5.1 Pressure distribution through the impeller blades meridional view

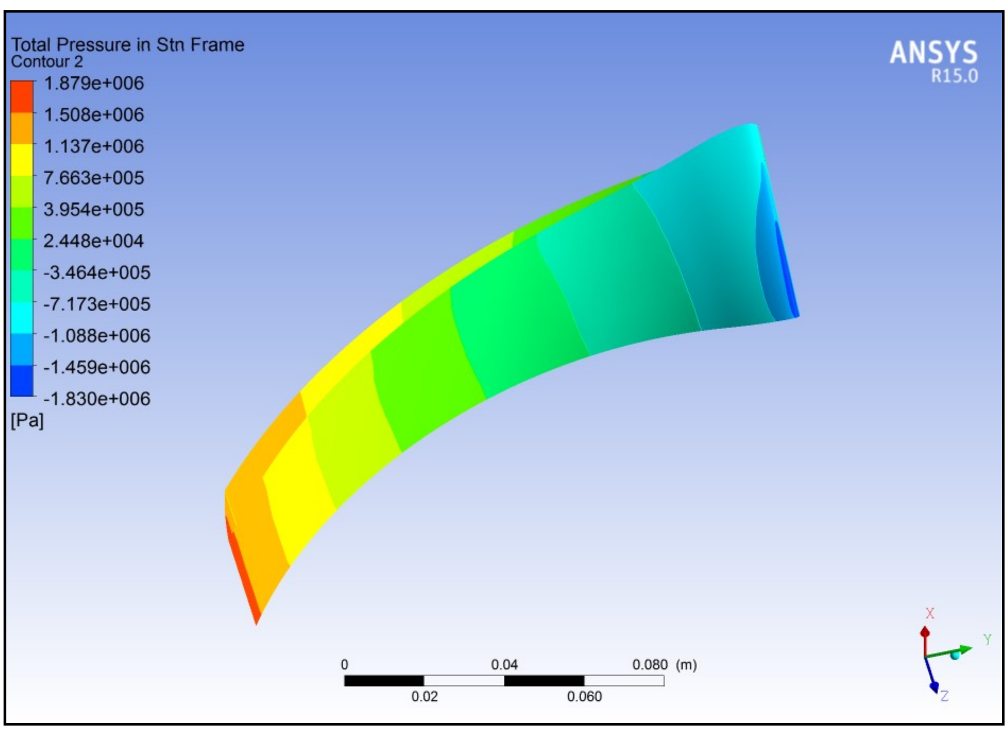

Fig.5.1 Pressure distribution along the impeller blades

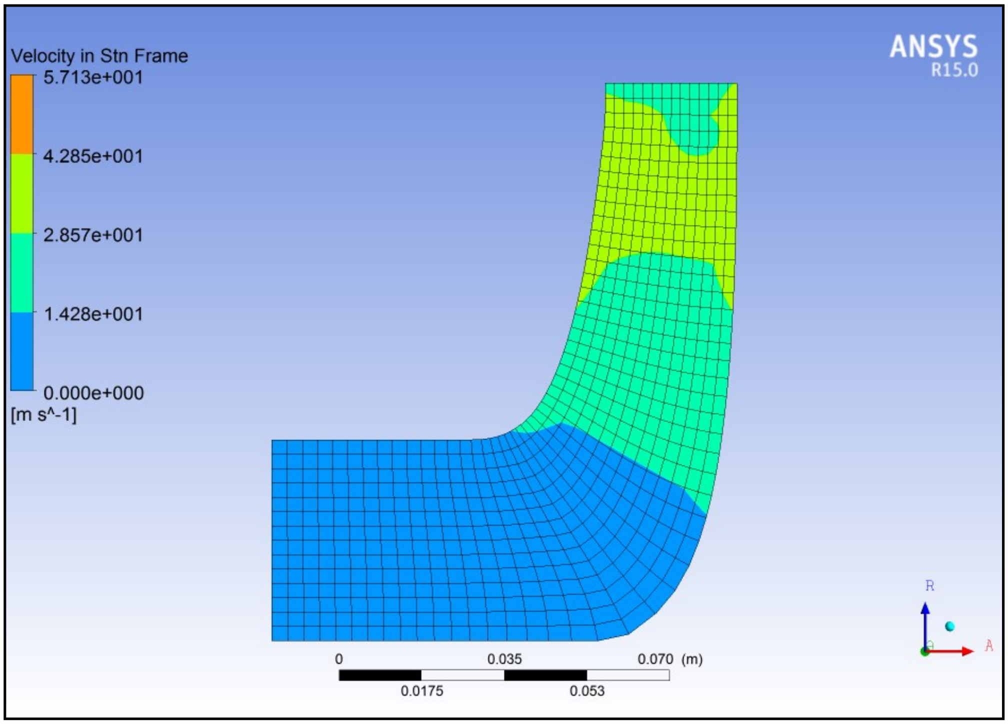

Fig.5.3 Velocity distribution through the impeller blades meridional view

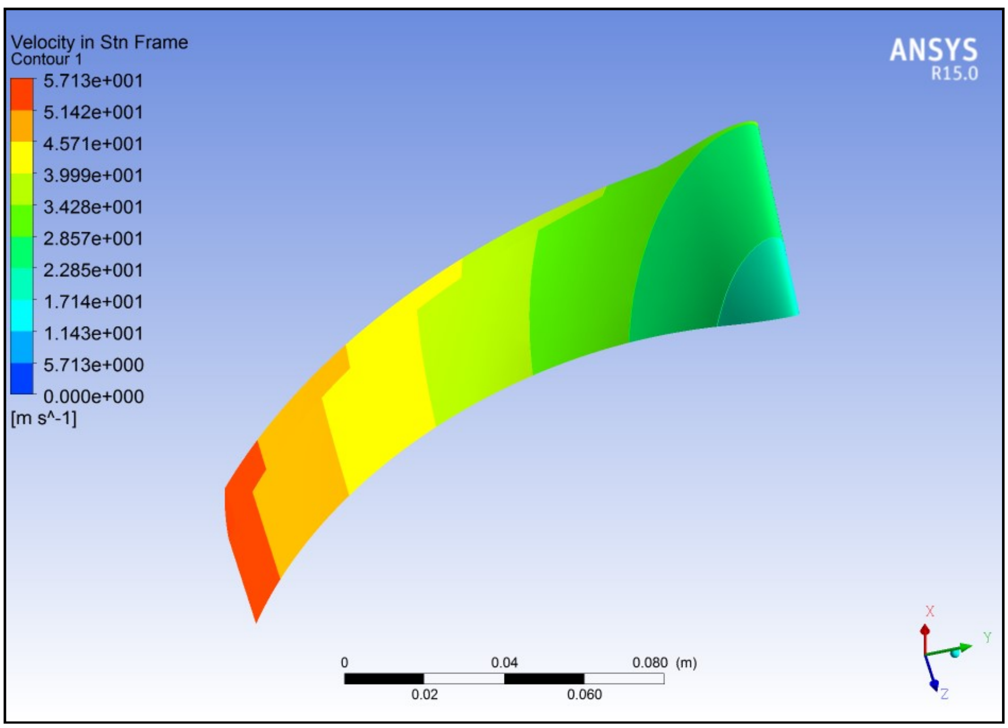

Fig.5.4 Velocity distribution along the impeller blades

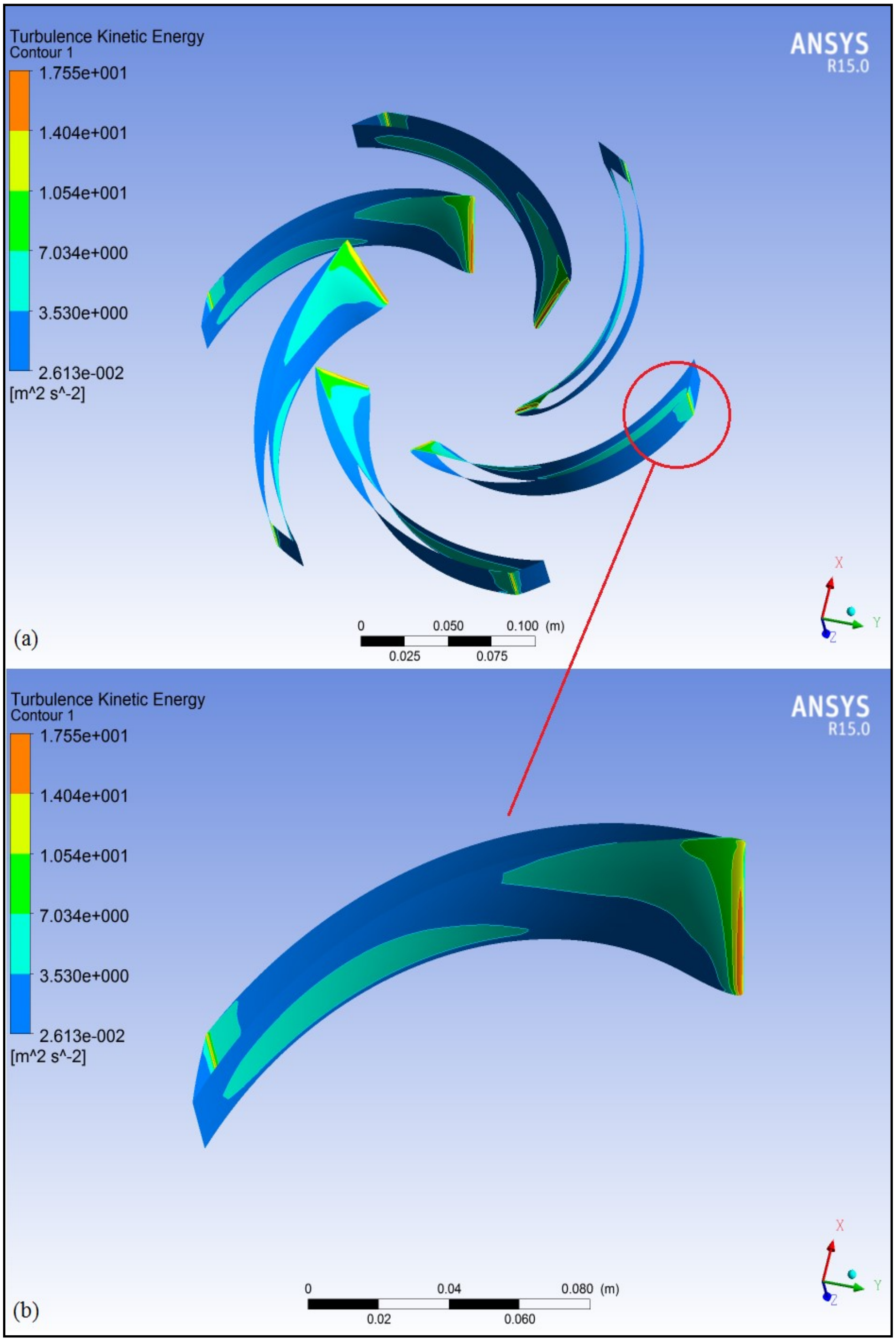

Fig.5.5 Turbulence kinetic energy distribution (a) Impeller (b) along the impeller blades

Fig.5.6 shows the velocity vector distribution of the centrifugal pump. Turbulence is generated at the trailing edges of the impeller blades as can be seen in the fig.5.6 which leads to vortices & causes loss of head due to acceleration of water flow. It also leads to increase in torque & hence power consumption.

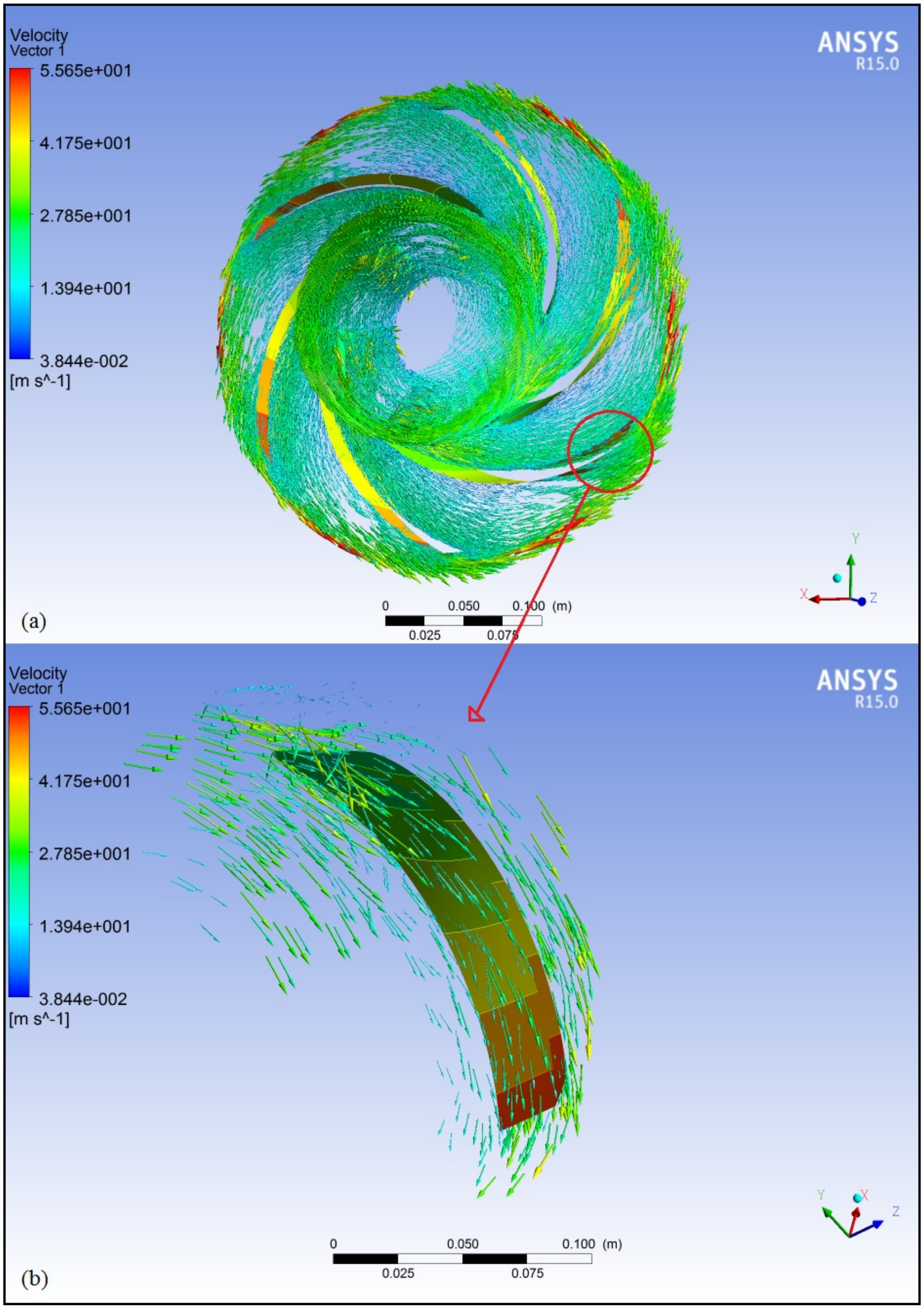

Fig.5.6 Velocity Vector distribution (a) Impeller (b) along the impeller blades

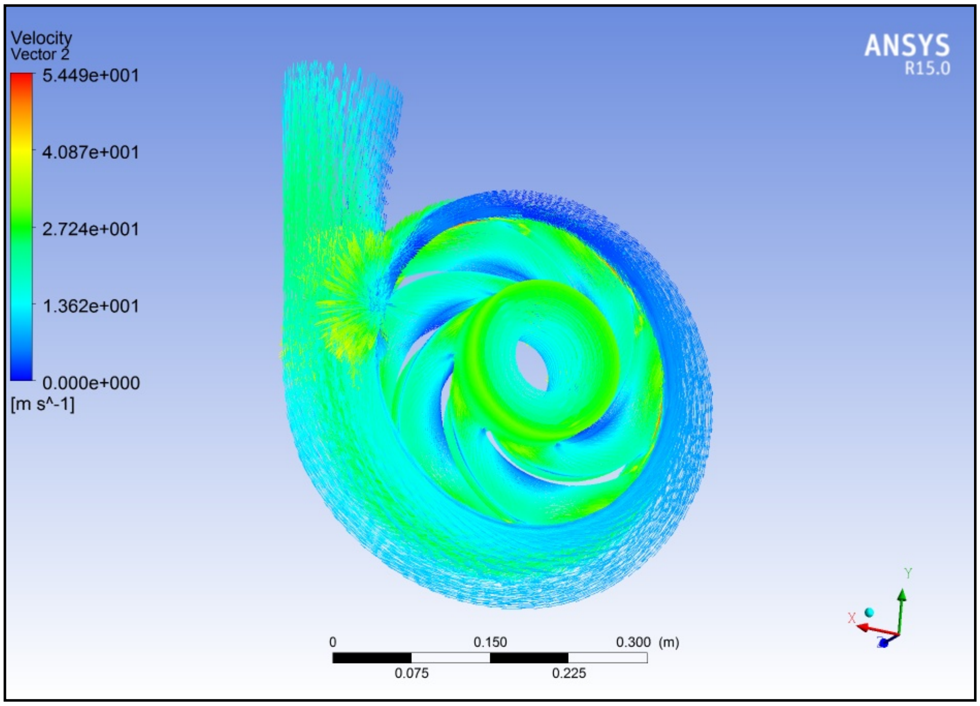

Fig.5.7: Velocity distribution vector of the centrifugal pump.

As can be seen in Fig.5.7 the velocity of the fluid in the cutwater region is greatly reduced. This is mainly due to the turbulence generated as the fluid passing through the impeller mixes with the fluid in the diffuser part of the volute casing.

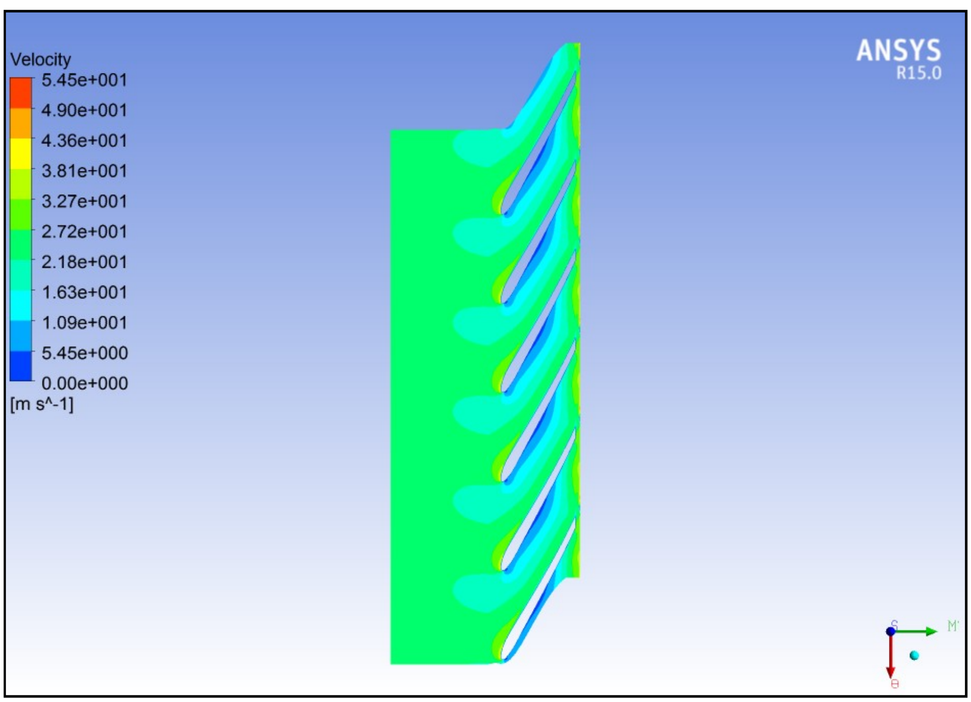

Fig.5.8 Velocity contour, blade to blade view

As can be seen in fig.5.8 the velocity of flow at the blade contact is lesser in comparison to fluid with greater boundary layer thickness and its value is maximum at the tip of the blade i.e. the trailing edge.

Table 5.1 & 5.2 shows the minimum & maximum value of the velocity and pressure respectively.

Table 5.1 Fluid Flow Velocity

Minimum velocity	Maximum velocity
18.9391 m/sec	57.13 m/sec

Table 5.2 Fluid Flow Pressure

Minimum pressure	Maximum pressure
-1.82996×10^{06} Pa	1.87891×10^{06} Pa

Fig.5.9 shows the streamline velocity of the fluid flow at various time intervals. Firstly the fluid enters through the inlet at time t= 0 sec (fig.5.9 a), as the fluid enters the impeller as shown in the next frame (fig.5.9 b) it gains rotational motion due to the rotation of the impeller which causes an increase in the total energy of the fluid (i.e. pressure & kinetic energy) (fig.5.9 c to g).

As the fluid is entering the volute casing as shown in fig.5.9 (h) the flow rate of the fluid decreases. This causes the conversion of kinetic energy of the fluid into pressure energy.

The fluid is entering the diffuser as shown in fig.5.9 (j) whose main function is the efficient conversion of velocity head into pressure head & also to provide a more controlled flow.

In fig.5.9 (l) fluid can be seen leaving the centrifugal pump from the diffuser with greater energy to reach the desired height.

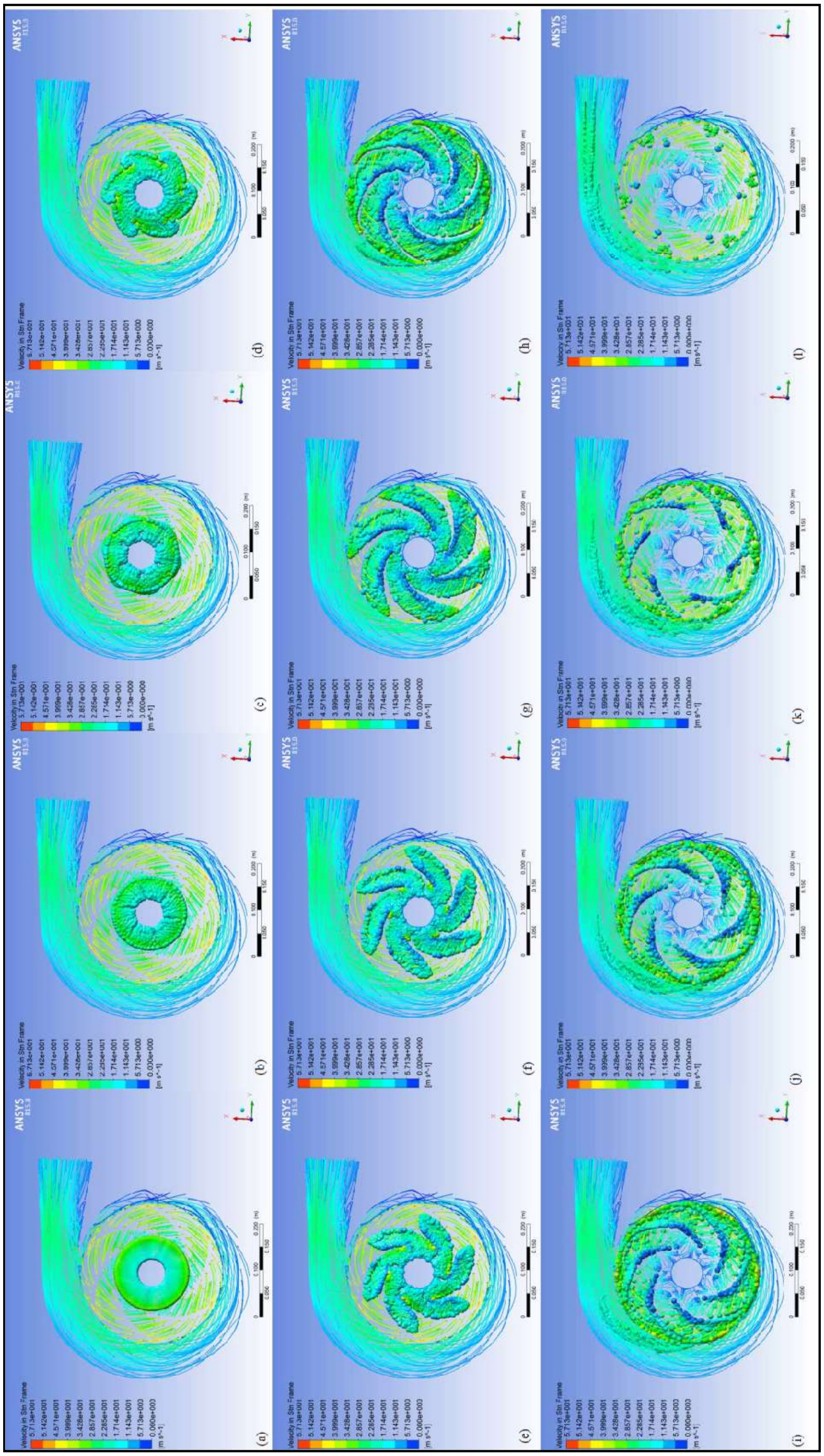

Fig.5.9 CFD streamlines simulation frame of a centrifugal pump

Fig.5.10 & 5.11 shows the pressure and velocity distribution between the hub and shroud of the centrifugal pump. It also shows that the change in load is smooth.

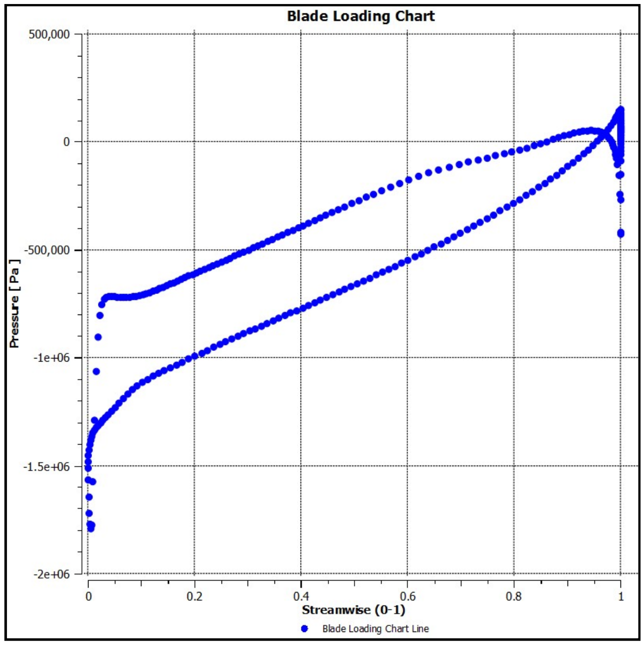

Fig.5.10 Pressure blade loading chart

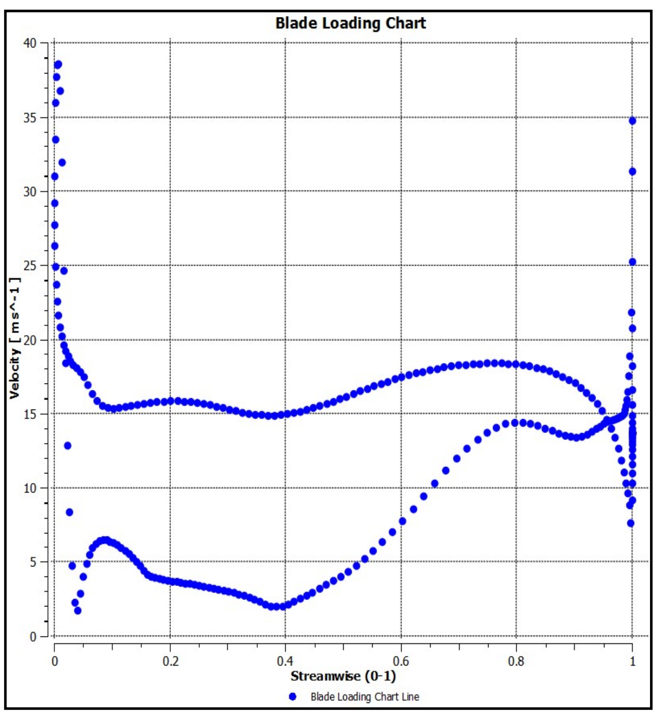

Fig.5.10 Velocity blade loading chart

Fig.5.11 & 5.12 shows the 2D projection of a 3D discretely sampled data of the pressure & velocity distribution of the centrifugal pump respectively.

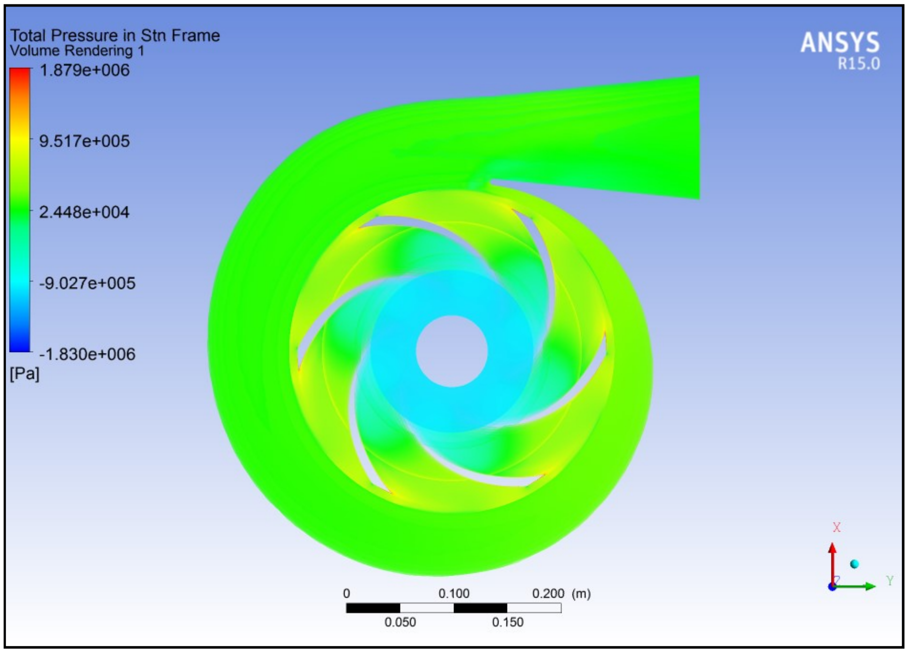

Fig.5.11: Volume rendering of total pressure distribution

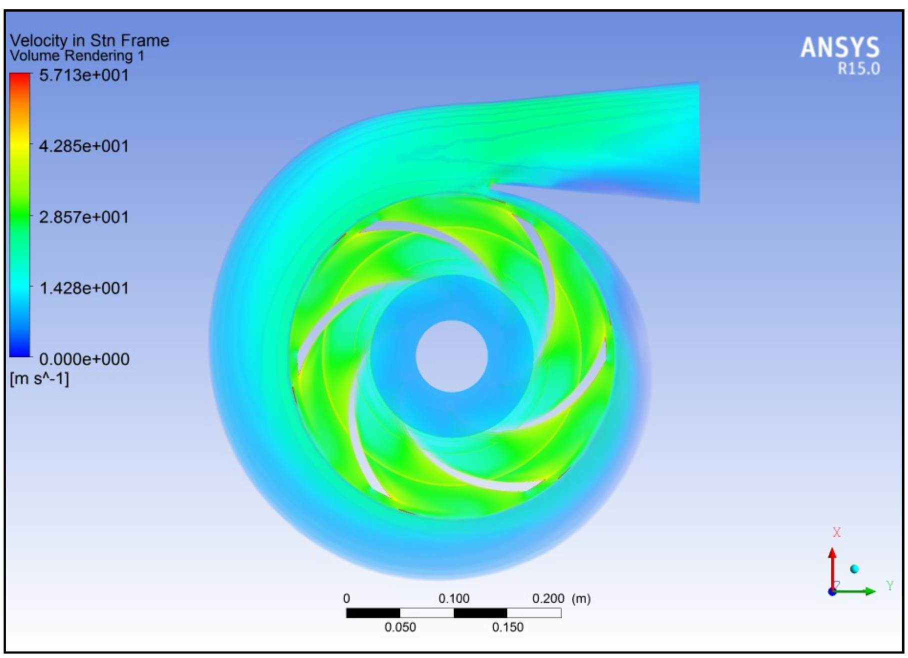

Fig.5.12: Volume rendering of total velocity distribution

5.2 Optimization with Genetic Algorithm

To calculate the optimum values of the design variables we have used the Genetic Algorithm pattern search method as it has the following advantages:

- There are multiple values of local optima
- As the fitness function is not smooth the derivative methods cannot be applied
- There are very large numbers of parameters
- The fitness function is noisy or stochastic

5.2.1 Optimization toolbox in MATLAB

MATLAB has a predefined optimization toolbox which can be used to find the optimum value of the design variable of the centrifugal pump. We have chosen 4 design variables that are as follows:

1. Number of impeller blades (n)
2. Outer diameter of the impeller (D_2)
3. Outer breadth of the impeller (B_2)
4. Outlet blade angle (b_2)

5.2.2 Optimization results

Hydraulic efficiency of centrifugal pump is the main performance parameter that has been selected for optimizing. Optimization is done by creating a fitness function (i.e. $1/\eta_h$) from equation (1.1) to (1.17) & minimising it to find the maximum efficiency.

5.2.2.1 Pressure Distribution

Pressure distribution curves of the optimized centrifugal pump are shown in fig.5.15& 5.16. Maximum pressure of the fluid flow is increased from 1.68107 x 10^{06} Pa to 1.87891 x 10^{06} Pa

5.2.2.2 Velocity Distribution

Velocity distribution curves of the optimized centrifugal pump are shown in fig.5.17& 5.18. Maximum velocity of the fluid flow is increased from 57.13 m/sec to 64.0865 m/sec

5.2.2.3 Turbulence Kinetic Energy

Turbulence kinetic energy distribution curves of the baseline centrifugal pump are shown in fig.5.15 (a) & (b). The maximum value of turbulence kinetic energy along the impeller lades is increased from17.55 m^2/s^2 to 37.48 m^2/s^2 is obtained.

Table 5.3 Fluid Flow Velocity

Minimum velocity	Maximum velocity
18.9172 m/sec	64.0865 m/sec

Table 5.4 Fluid Flow Pressure

Minimum pressure	Maximum pressure
-3.11656 x 10^{06} Pa	1.68107 x 10^{06} Pa

Table 5.3 Design variables

Design Variables	Initial values	Optimized values
n	6	5
D_2	0.2874 m	0.28 m
B_2	0.0279 m	0.025 m
b_2	25°	30°

Table 5.4 Performance of the centrifugal pump

Performance parameter	Initial values	Optimized values
η_h	70.96 %	76.37 %

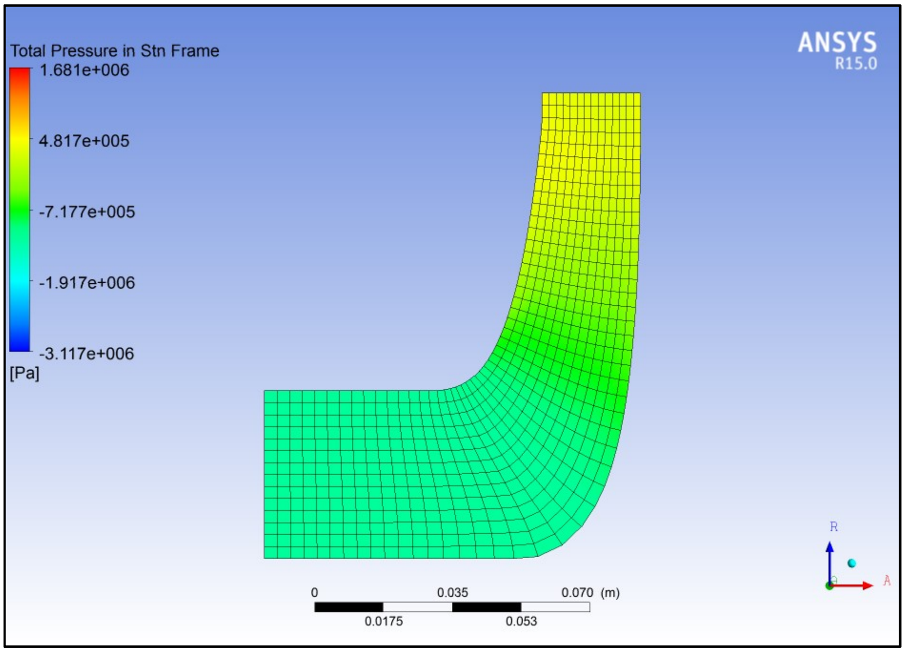

Fig.5.15 Pressure distribution through the impeller blades meridional view

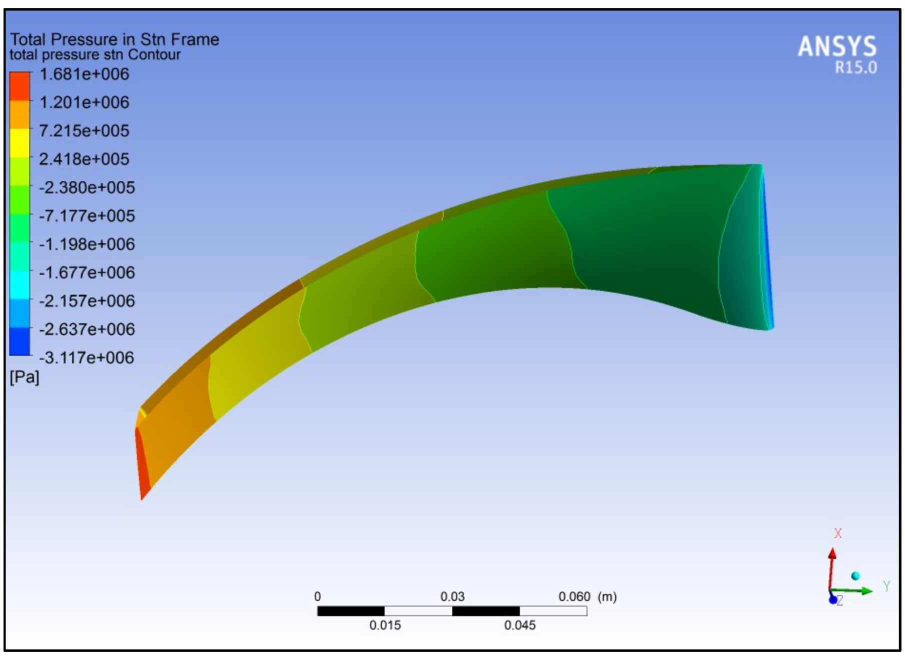

Fig.5.16 Pressure distribution along the impeller blades

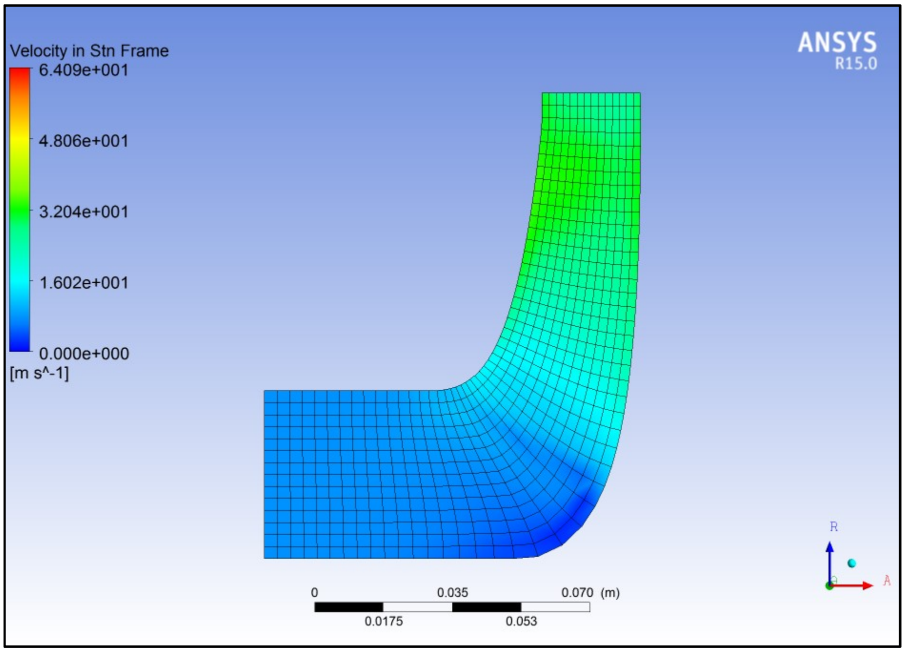

Fig.5.13 Velocity distribution through the impeller blades meridional view

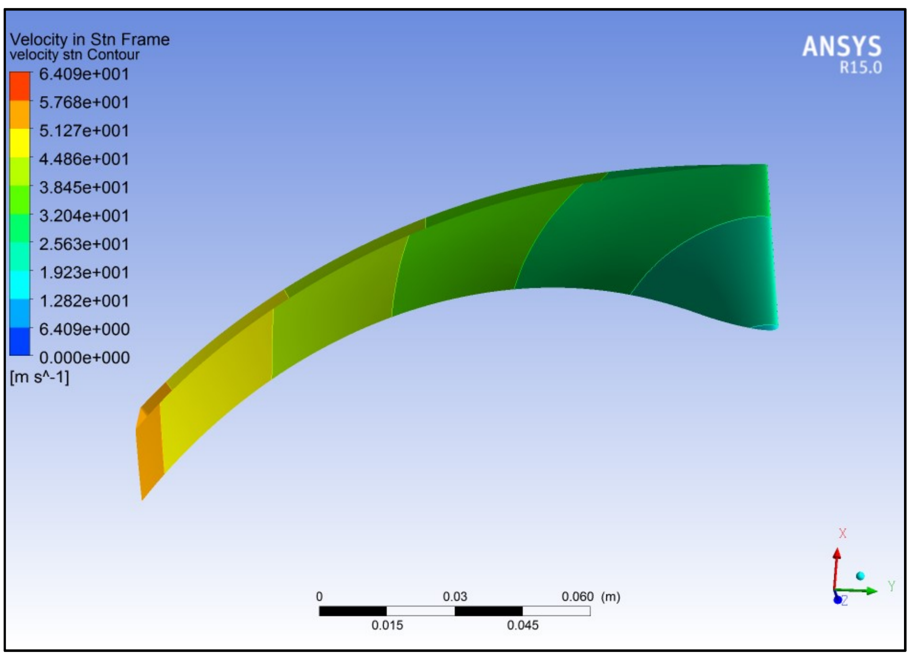

Fig.5.14 Velocity distribution along the impeller blades

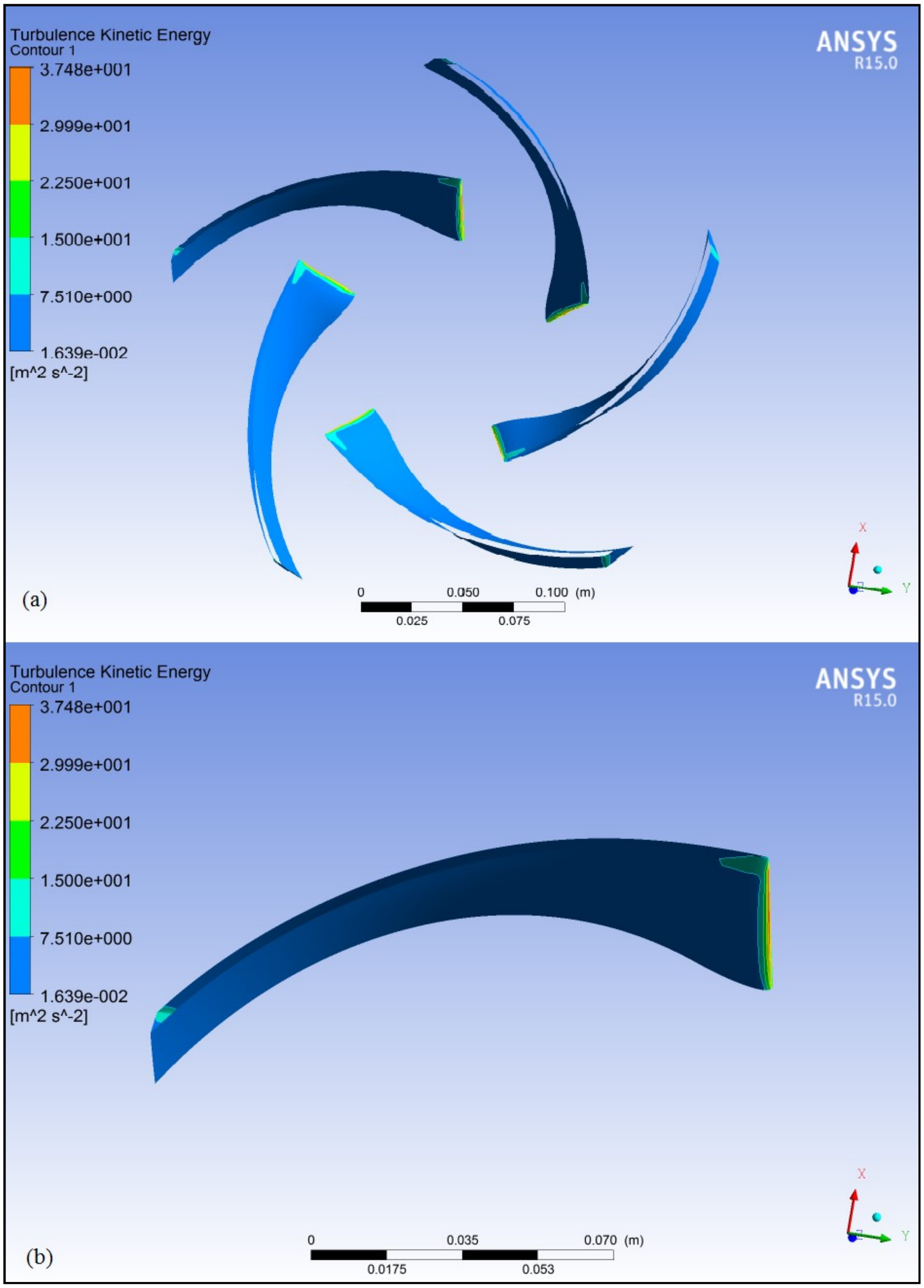

Fig.5.15 Turbulence kinetic energy distribution (a) Impeller (b) along the impeller blades

CHAPTER 6

CONCLUSIONS & FUTURE SCOPE

This dissertation work has been attempted to obtain the hydrodynamic characteristics of a centrifugal pump using ANSYS® & optimize its performance using GA (genetic algorithm).

In this research work it is observed that hydraulic efficiency of the centrifugal pump can be increased by decreasing its number of vanes & outlet diameter. Therefore it reduces the cost of material used in the making of pump and also increases the life of the pump.

By performing optimization of centrifugal pump along with hydraulic efficiency other performance parameters like pressure, velocity & turbulence kinetic energy have been improved.

6.1 Conclusions

To get improved hydraulic efficiency the following changes were incorporated in the impeller design:

- Impeller diameter was decreased by 2.57%.
- Outer breadth of impeller was decreased by 10.39%.
- Number of vanes was decreased from 6 to 5.
- Outlet blade angle was increased from 25° to 30°.

The following hydrodynamic characteristics were observed in the optimized design:

- Maximum pressure was increased by 11.77 %.
- Maximum velocity was increased by 12.177 %.
- Maximum value of turbulence kinetic energy was increased from 17.55 m^2/s^2 to 37.48 m^2/s^2.
- Hydraulic efficiency was increased by 5.41 %.

6.2 Future Scope

- Structural & vibrational analysis will be evaluated for a centrifugal pump in future.
- Volute flow study and impeller-volute interaction has great application in future designing processes.
- The optimization can be extended to the design of volute casing.
- Parameterization of other design parameters can also be included
- Different turbulence model can also be included.

REFERENCES

[1] Frank M. White. *Fluid Mechanics*. McGraw Hill, Reading, Massachusetts, 2011.

[2] R.W. Fox, A.T. McDonald, and P.J. Pritchard. *Introduction to Fluid Mechanics*. Wiley, 2009.

[3] David A Johnson. *Turbomachines ME 563 Lecture Notes*. University of Waterloo,2014.

[4] George F. Round. *Incompressible Flow Turbomachines: Design, Applications, and Theory*. Butterworth Heinemann, 2004

[5] Raúl Barrio, Jorge Parrondo, Eduardo Blanco, (2010) "Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points", *Computers & Fluids 39 (2010) 859–870.*

[6] M.H.Shojaeefard, M.Tahani, M.B.Ehghaghi, M.A.Fallahian, M.Beglari,(2012) "Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid", *Computers & Fluids 60 (2012) 61–70*

[7] Dazhuan Wu, Bin Huang, (2010) "Numerical Simulation of the Transient Flow in a Centrifugal Pump During Starting Period", *Journal of Fluids Engineering:August 2010*

[8] Huamg Si, Yang Fuxiang, Guo Jing, (2013) "Numerical simulation of 3d unsteady flow in centrifugal pump by dynamic mesh technique", *Procedia Engineering 61(2013)270 – 275*

[9] Pei Ji, Yuan Shou-Qi , Li Xiao-Jun, Yuan Jian-Ping, (2014) "Numerical prediction of 3-D periodic flow unsteadiness in a centrifugal pump under

part-load condition", *Science Direct Journal Of Hydrodynamics 2014,26(2):257-263*

[10] Shahram Derakhshan, Maryam Pourmahdavi, Ehsan Abdolahnejad, Amin Reihani, Ashkan Ojaghi, (2013) "Numerical shape optimization of a centrifugal pump impeller using artificial bee colony algorithm" *Computers & Fluids 81 (2013) 145–151*

[11] Mohammed Khudhair Abbas, (2010) "Cavitation in Centrifugal Pumps" *ISSN 1999-8716 22-23 December. 2010, pp. 170-180*

[12] Marco Antonio Rodrigues Cunha, Helcio Francisco Villa Nova, (2013) "Cavitation Modeling of a Centrifugal Pump Impeller" *ISSN 2176-5480 (2013)*

[13] ZHU Bing, CHEN Hong-xun, (2012) "Cavitating Suppression of Low Specific Speed Centrifugal Pump with Gap Drainage Blades" *Science Direct Journal Of Hydrodynamics 2012,24(5):729-736*

[14] Alex George, Dr. P Muthu, (2013) "CFD Analysis Of Performace Charectristics Of Centrifugal Pump Impeller To Minimising Cavitation" *ICCREST-2016*

[15] Wen-guang Li, (2011) "Inverse design of impeller blade of centrifugal pump with a singularity method" *JJMIE Volume 5, Number ISSN 1995 2, April 2011 -6665Pages 119 – 128*

[16] M.A. El Samanody, Ashraf Ghorab, Mamdoh Aboul Fitoh Mostafa, (2014) "Investigations on the performance of centrifugal pumps in conjunction with inducers" *Ain Shams Engineering Journal (2014) 5, 149–156*

[17] Evangelos Bacharoudis, Michalis Dimitrios, Mentzos Dionissios, P. Margaris,(2008) "Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle" *The Open Mechanical Engineering Journal, 2008, 2, 75-83*

[18] Erik Dick, John Vande Voorde, (2001) "Performance prediction of centrifugal pumps with CFD-tools" TASK QUARTERLY 5 No 4 (2001), 579–594

[19] Bo Ningaa, Xiang-xin Chenga, Shuo Wua, (2011) "Research on centrifugal pump monitoring system based on virtualization technology*" Procedia Engineering 15 (2011) 1077 – 1081*

[20] Dazhuan Wu, Peng Wu, Shuai Yang, (2014) "Transient Characteristics of a Closed-Loop Pipe System During Pump Stopping Periods" *Journal of Pressure Vessel Technology· February 2014*

[21] A. Lucius, G. Brenner, (2010) "Unsteady CFD simulations of a pump in part load conditions using scale-adaptive simulation" *International Journal of Heat and Fluid Flow 31 (2010) 1113–1118*

[22] S R Shah, S V Jain, R N Patel, V J Lakhera, (2013) "CFD for centrifugal pumps: a review of the state of the art" *Procedia Engineering 51 (2013) 715 – 720*

[23] Zhongliang Kang, Chao Yan, (2012) "Accurate and robust CFD algorithms applied to 3D arbitrary polyhedral grids" *Procedia Engineering 31 (2012) 9 – 15*

[24] Md.Abdul Raheem Junaidia, N.B.V Laksmi Kumarib, Mohd Abdul Samadc, G.M. Sayeed Ahmedd, (2015) "CFD Simulation to Enhance the Efficiency of Centrifugal Pump by Application of Inner Guide Vanes Materials" *Proceedings 2 (2015) 2073 – 2082*

[25] Abdul kadir Aman, Sileshi Kore and Edessa Dribssa, (2011) "Flow Simulation And Performance Prediction Of Centrifugal Pumps Using Cfd-Tool" *Journal of EEA, Vol. 28, 2011*

[26] Krishnan V. Pagalthivarthi, Pankaj K. Gupta, Vipin Tyagi, M. R. Ravi, (2011) "CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings" *International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:5, No:3, 2011*

[27] V.S. Kadam, S.S. Gawade, H.H. Mohite, N.K.Chapkhane, (2011) "Design and Development of Split Case Pump Using Computational Fluid Dynamics" *International Conference On Current Trends In Technology, 'NUiCONE – 2011'*

[28] Zhi-fei Peng, Chun-geng Sun, Rui-Bo Yuan, Peng Zhang, (2012) "The CFD analysis of main valve flow field and structural optimization for double-nozzle flapper servo valve" *Procedia Engineering 31 (2012) 115 – 121*

[29] Jidong Lia, Yongzhong Zeng, Xiaobing Liu, Huiyan Wang, (2012) "Optimum design on impeller blade of mixed-flow pump based on CFD" *Procedia Engineering 31 (2012) 187 – 195*

[30] Yuliang Zhang, Hua-Shu Dou, Yi Li "Experimental and Theoretical Study of a Prototype Centrifugal Pump during Startup Period" *Int.J.Turbo Jet-Engines 2013; 30(2): 173-177*

[31] Liu Hou-Lin, Liu Dong-Xi, Wang Yong, Wu Xian-Fang, Wang Jian, Du Hui, (2013) "Experimental investigation and numerical analysis of unsteady attached sheet cavitating flows in a centrifugal pump" *Science Direct Journal Of Hydrodynamics 2013,25(3):370-378*

[32] Mohamad Memardezfouli, Ahmad Nourbakhsh, (2009) "Experimental investigation of slip factors in centrifugal pumps" *Experimental Thermal and Fluid Science 33 (2009) 938–945*

[33] F.Jiménez Espadafor, J.Becerra Villanueva, M.Torres García, E.Carvajal Trujillo, (2011) Experimental and dynamic system simulation and optimization of a centrifugal pump-coupling-engine system. Part 1: Failure identification" *Engineering Failure Analysis 18 (2011) 1–11*

[34] F. Jiménez Espadafor, J.Becerra Villanueva, M.Torres García, E.Carvajal Trujillo, (2010) "Experimental and dynamic system simulation and optimization of a centrifugal pump coupling engine system Part 2: System design" *Engineering Failure Analysis 17 (2010) 1551–1559*

[35] Mario Šavar, Hrvoje Kozmar, Igor Sutlović, (2009) “Improving centrifugal pump efficiency by impeller trimming” *Desalination 249 (2009) 654–659*

[36] Krishna Kumar Yadav, Karun Mendiratta, V K Gahlot, (2016) “Optimization Of The Design Of Radial Flow Pump Impeller Though Cfd Analysis” *IJRET Volume: 05 Issue: 11 | Nov-2016*

[37] Ajith M S, Dr Jeoju M Issac, (2015) “Design And Analysis Of Centrifugal Pump Impeller Using Ansys® Fluent” *IJSETR, Volume 4, Issue 10, October 2015*

[38] Krishna Kumar Yadav, Karun Mendiratta, V.K. Gahlot, (2016) “Optimization Of Design Of Mixed Flow Centrifugal Pump Impeller Using Cfd” *IJRET Volume: 05 Issue: 08 | Aug-2016*

[39] Yu Zhang, Sanbo Hu, Jinglai Wu, Yunqing Zhang, Liping Chen, (2014) “Multi-objective optimization of double suction centrifugal pump using Kriging metamodels” *Advances in Engineering Software 74 (2014) 16–26*

[40] Houlin Liu, Kai Wang, Shouqi Yuan, Minggao Tan, Yong Wang, Liang Dong, (2013) “Multicondition Optimization and Experimental Measurements of a Double-Blade Centrifugal Pump Impeller” *Journal of Fluids Engineering ,Vol. 135 / 011103-1*

[41] Val. S. Lobanoff, Robert R. Ross, “Centrifugal pumps Design and application”, second edition, A practical reference stressing hydraulic design, 1985, pg 28-36

APPENDIX A

Software Details

In this thesis ANSYS® Workbench 15.0 was used in the design & analysis of the centrifugal pump and MATLAB was used in optimizing its geometry.

1.7.1 ANSYS® Workbench 15.0

This topic discusses in detail the design of a centrifugal pump with the help of ANSYS®. The following modules are involved in this project:

1. Vista CPD-It is a program used for the basic design of a centrifugal pump. It creates impeller and volute geometry data which can be used in BladeGen & Volute modeller.

2. BladeGen- It is a program used for the design of impeller. The geometry data generated in Vista CPD is used to generate the 3D model of the impeller.

3. Volute modeller- It is a program used for the design of volte casing. The geometry data generated in Vista CPD is used to generate the 3D model of the volute. After generating the 3D model of the volute the geometry is meshed so that it can be sent to CFX-pre module for analysis.

4. TurboGrid-It is a program used for meshing the geometry of the turbo-machines. The geometry generated in BladeGen is used to create a mesh model. This model can then be used in CFX-pre module for analysis

5. CFX- "It is a high-performance, general-purpose fluid dynamics program that engineers have applied to solve wide-ranging fluid flow problems for over 20 years. At the heart of **CFX** is its advanced solver technology, the key to achieving reliable and accurate solutions quickly and robustly."

- CFX Pre-

The meshed model geometry generated in TurboGrid & Volute modeller is combined together. Boundary conditions are specified and the model is updated.

- CFX Solver-

This program solves the model along with the boundary condition until the results are converged.

- CFX Post-

This program is used to obtain the various results of the pre-defined parameters of the centrifugal pump.

1.7.2 MATLAB

The main objective of this project is to improve the efficiency of the centrifugal pump design. The efficiency of the pump is related to its geometry so by assuming the design parameters as variable we can utilise genetic algorithm to find the optimum value of design parameters.

MATLAB has a predefined Optimization tool with genetic algorithm module which can be utilised in optimizing the design geometry.

APPENDIX B

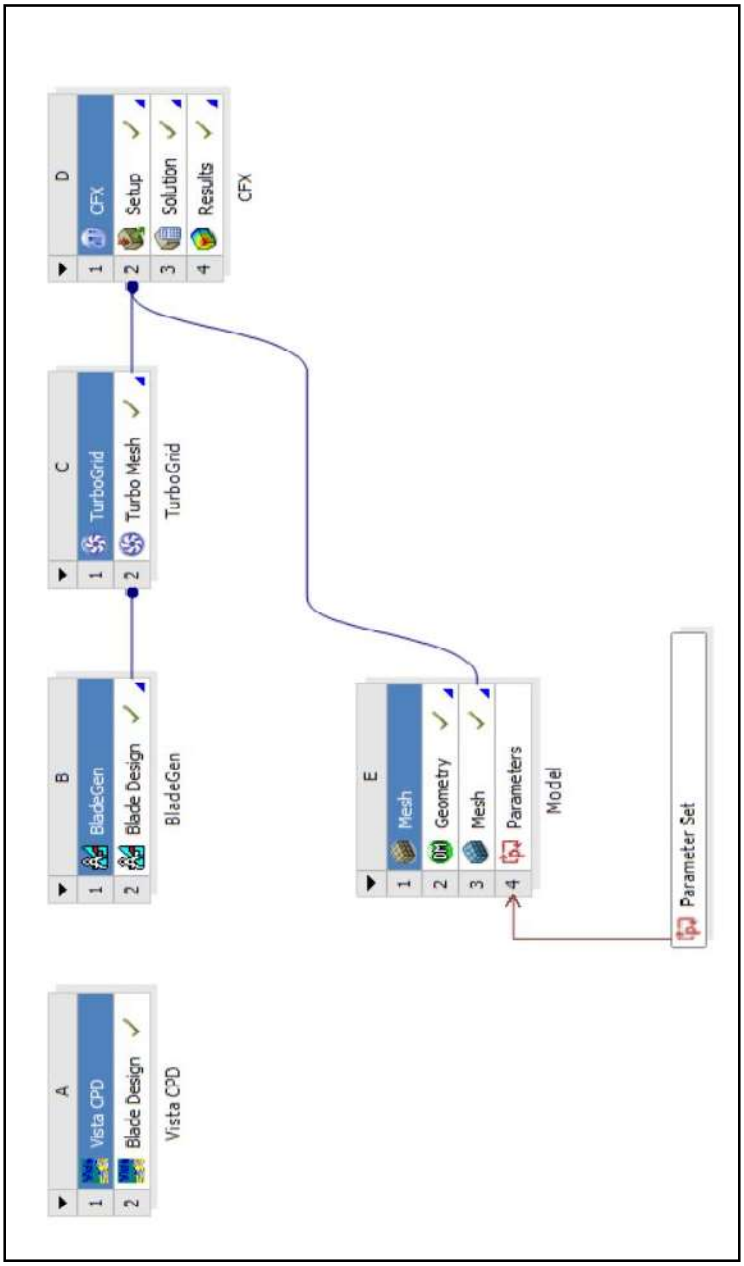

Figure A: Project Schematic Process in ANSYS® Workbench

APPENDIX C

Fitness function for hydraulic efficiency of the centrifugal pump:

```
function FITNESS=HYDRAULIC_EFFICIENCY(X)

z=X(1); %NUMBER OF VANES

D2=X(2); %OUTER DIAMETER OF IMPELLER

B2=X(3); % OUTER WIDTH OF IMPELLER

b2=X(4); %OUTER BLADE ANGLE

Q=0.132489; %FLOW RATE AT BEP

H=137.16; %PUMP HEAD AT BEP

N=3600; %ROTATIONAL SPEED OF PUMP AT BEP

g=9.81; %ACCELERATION DUE TO GRAVITY

u2=pi*D2*N/60; %IMPELLER TIP SPEED

F=Q/(pi*B2*D2*u2); %FLOW COEFFICIENT

S=1-(sin(pi/180*(90-b2)))^(0.5)/z^(0.7); %SLIP FACTOR
```

```
T= S- F*tan(pi/180*b2); %HEAD COEFFICIENT

n=g*H/(T*u2^2); %HYDRAULIC EFFICIENCY

FITNESS=1/n;

end
```