Fall 11-2014

A CFD Investigation of the Hydrodynamic Characteristics of Fluid Flow through an Impeller and Multi-Objective Design Optimization of a Centrifugal Pump

Mohammed Khammat Sagban
Embry-Riddle Aeronautical University

A CFD INVESTIGATION OF THE HYDRODYNAMIC CHARACTERISTICS OF FLUID FLOW THROUGH AN IMPELLER AND MULTI-OBJECTIVE DESIGN OPTIMIZATION OF A CENTRIFUGAL PUMP

by

Mohammed Khammat Sagban

A Thesis Submitted to the College of Engineering and the Department of Mechanical Engineering in Partial Fulfillment of the Requirements for the Degree of Master of Science in Mechanical Engineering

Embry-Riddle Aeronautical University

Daytona Beach, Florida

November 2014

A CFD INVESTIGATION OF THE HYDRODYNAMIC CHARACTERISTICS OF FLUID FLOW THROUGH AN IMPELLER AND MULTI-OBJECTIVE DESIGN OPTIMIZATION OF A CENTRIFUGAL PUMP

by

Mohammed Khammat Sagban

This thesis was prepared under the direction of the candidate's Thesis Committee Chair, Dr. Sathya N. Gangadharan, Professor, Daytona Beach Campus, and Thesis Committee Members Dr. Sandra K. Boetcher, Professor, Daytona Beach Campus, and Dr. Ilteris Demirkiran, Professor, Daytona Beach Campus, and has been approved by the Thesis Committee. It was submitted to the Department of Mechanical Engineering in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering

Thesis Review Committee:

Sathya N. Gangadharan, Ph.D.
Committee Chair

Sandra K. Boetcher, Ph.D.
Committee Member

Ilteris Demirkiran, Ph.D.
Committee Member

Jean-Michel Dhainaut, Ph.D.
Graduate Program Chair
Mechanical Engineering

Charles F. Reinholtz, Ph.D.
Department Chair
Mechanical Engineering

Robert Oxley, Ph.D.
Associate Vice President of Academics

Maj Mirmirani, Ph.D.
Dean, College of Engineering

11-14-14
Date

Acknowledgments

I would like to thank my adviser Dr. Sathya N. Gangadharan for all his invaluable guidance, help and support for my Master's research.

In addition, I would like to thank the rest of my thesis committee: Dr. Sandra K. Boetcher and Dr. Ilteris Demirkiran for their encouragement and assistance.

Special thanks also go to my family as well as my friends for their help and support.

I also would like to thank everyone (students, faculty) of Mechanical Engineering Department at Embry-Riddle Aeronautical University especially Dr. Darris White, Dr. Jean-Michel Dhainaut, and Dr. Charles Reinholtz for their continued support to get this thesis done. Thanks to Embry-Riddle Aeronautical University for providing the computational resources needed for this research. Special thanks are to my sponsor HCED (The Higher Committee for Education Development in Iraq) for their funding, assistance, support, and encouragement for pursuing this research.

Abstract

Researcher: Mohammed Khammat Sagban

Title: A CFD Investigation of the Hydrodynamic Characteristics of Fluid Flow through an Impeller and Multi-Objective Design Optimization of a Centrifugal Pump

Institution: Embry-Riddle Aeronautical University

Degree: Master of Science in Mechanical Engineering

Year: 2014

In this thesis, the first part is conducting an investigation of the hydrodynamic characteristics (pressure and velocity) of fluid flow (water) through an impeller of a centrifugal pump. To illustrate a one-dimensional model is used to identify an impeller of a GDM 10 x 12 HD centrifugal pump using Vista CPD design and BladeGen tools. A three-dimensional model is then built with a commercial software package ANSYS Blade Modeler. The structured mesh of the impeller blade is obtained by using TurboGrid system. The volute of the centrifugal pump is built in ANSYS geometry, and an unstructured mesh is accomplished by using a very fine mesh. Afterwards, Computational Fluid Dynamics (CFD) is applied in ANSYS-CFX to set output parameters of objectives and constraints of the centrifugal pump. A multi-objective design optimization (MDO) process is defined for a centrifugal pump GDM 10 x 12 HD impeller and volute to get optimum values of efficiency, total head and performance. In addition, the objective of this research is to maximize the efficiency and the total head for the centrifugal pump GDM 10 x 12 HD, making sure that the performance target does not exceed the value of the relative ratio of outlet and inlet pressure so that the head losses do not increase. The CAD modeler of the impeller, with optimal inlet and outlet angles, are

modeled using BladeGen modeler. The efficiency and the head of the standard centrifugal pump is observed to be 61% and, 42.6 m respectively.

Finally, HEEDS MDO-Modeler is used to perform the multi-objective optimization process to obtain the optimal results of the pump efficiency and the head. After optimization the efficiency of the centrifugal pump is increased by 3.2% and the optimized head is decreased by 2% as compared to baseline values. In addition, the blade thickness and the volute thickness are decreased by 13.5 % and 10.2 % respectively from the original values.

Table of Contents

List of Tables

List of Figures

Nomenclature

N	Blade count
β_1	Blade angle at the internal of impeller
β_2	Blade angle at exit of impeller
t	Blade thickness
D_1	Inlet diameter of impeller
D_2	Outlet diameter of impeller
Θ	Wrap angle
B_1	Inlet blade width
B_2	Outlet blade width
H	Head
Q	Mass flow rate
η	Efficiency
β_m	The average vane angle
ω	Rotational speed
C_u	Absolute tangential velocity
C_m	Absolute meridional velocity
r	Radius

Abbreviations

CFD	Computational Fluid Dynamics.
GDM 10 x 12 HD	Type of centrifugal pump.
LE	Leading edge
TE	Trailing edge
BL	Boundary layer
K-ε	K-epsilon
RPM	Revolution per minute
R1	Rotor
S1	Stator
Pa	Pascale
atm	Atmosphere
ATM	Topology mesh method
CPD	Centrifugal pump design

Greek Symbols

Ω	Angular velocity	[rad/s]
ρ	Density	[kg/m^3]
g	Gravity acceleration	[m/s^2]

Chapter 1

Background

1.1 Introduction

Centrifugal pumps are widely used in different fields, and found today in almost all industries. In addition, it has many different applications. For example, it is found in municipal works, power plants, agriculture, transport and many other utility services. A centrifugal pump is a mechanical device used to transport the liquid from one place to another. It is also considered a hydraulic machine used to convert mechanical energy into hydraulic energy [1]. The pump contains of a set of rotating blades (vanes), which are surrounded by the volute (casing scroll). Moreover, there are many advantages of a centrifugal pump over a reciprocating pump such as high volume flow rate, easy control, lower manufacturing and maintenance costs.

Furthermore, a centrifugal pump depends on many factors according to its design. For instance, a specialist should test the pump before predicting the performance because there are many extensive patterns, which take time to satisfy the requirements, such as cost, manufacturing, and testing. Computational Fluid Dynamics analysis (CFD) can solve many problems regarding these issues. For example, it is used in the design of centrifugal pumps as an alternative tool. By using computer ability, it becomes possible to increase the performance of the pump using CFD-CFX, which can contribute to helping designers and engineers to overcome many problems in this field [2].

Moreover, when considering a centrifugal pump the impeller is the most important component. It has a significant effect in developing the performance of the pump because energy is generated by fluid flow through the impeller. As a result, experts and engineers in this field must take care of the precise analysis to optimize variables that can impact the performance of the pump. Furthermore, the flow of the pump is sometimes complicated and characterized by diffusion and a strong swirl. Therefore, it is extremely necessary to investigate the internal flow of the centrifugal pump impellers by using some CFD approaches, and optimize its performance using HEEDS MDO-Modeler to increase the efficiency [3].

1.2 Literature Review

Many researchers have conducted CFD (Computational Fluid Dynamic) investigations and design optimizations. Weidong Zhou, Zhimei Zhao and S. H. Winoto, in 2003, conducted a three dimensional simulation of internal flow in three different types of centrifugal pumps. One of the pumps had four straight blades and the other two had six twisted blades. They found that the computational results for compression (from results in both pressure distributions and flow patterns) gave better results for the twisted blade pumps. On the other hand, for the straight blade pump, the computational results from simulation were different from widely published experimental results. It was also found that the predicted results relating twisted-blade pumps were better than those relating to the straight-blade pump, suggesting greater efficiency [4].

J H Kim, K T Oh, et. al, in 2012, used Response Surface Method to optimize the impeller and the design of the volute of a centrifugal pump to increase the efficiency at target head. Their research was carried out in order to improve the performance of the centrifugal pump using the results from a numerical analysis. Moreover, this method of design that they used was suggested by many researchers, like C K Kim, for designing the volute that was suitable for the optimized impeller. It was observed that, by using these methods, the efficiency could be decreased from 98.5 % to 98.2 %, while the head could be increased from 62 m to 64.6 m. The volute was designed using Stepanoff method. The design optimization also incorporated performance evaluations and volute cross sectional area modifications [5].

In 2013, Mehul P and Prajesh conducted CFD analysis of a mixed flow pump impeller. They analyzed the outlet and inlet conditions, such as pressure and velocity, in order to calculate the efficiency of the pump impeller. For the existing impeller, the empirical relations were used to calculate optimum values of the inlet and outlet angles of the impeller. The head of the impeller was improved by changing the outlet angles. The results that were obtained from CFD analysis showed that the head was 7.45 m whereas the experimental results showed that the head was 8.08 m. Moreover, the efficiency was observed 54.27 % in experimental data, but the efficiency of the impeller was as 49.6% using CFD analysis [6].

Ashok Thummar, et. al, in 2012, measured the performance analysis of four various kinds of open well centrifugal pumps. A performance analysis of the centrifugal pump was conducted in order to obtain the best performance point. The four types of pumps chosen for the performance analysis varied in the size of the impeller, which were 165 mm, 210 mm, 170 mm and 123 mm respectively. By conducting a CFD investigation, it was observed that the efficiency of the pump and the flow rate were increased whereas the head was decreased. However, the power input was also increased. It was found that with the test open well-165 mm impeller pump, the head and power increased proportionally with the increase in the flow rate. In addition to that, the efficiency was increased by more than 60 % of the maximum flow rate. Moreover, it was found that the best efficiency point (BEP) was 43 % [7].

T. Prabu, in 2007, improved the design of an impeller and optimized the design parameters, such as vane profile, inlet and outlet vanes angles and the number of vanes in order to improve the performance of the impeller. He also measured and collected flow rate data from experiments. CFD and ANSYS Fluent were used to simulate the performance of the pump. In addition to that, T. Prabu established theoretical and experimental methods for design and testing of a pump as well as studying pressure distribution along two blades with different speeds (2300 rpm, 2500 rpm and 2880 rpm). It found that pressure around the impeller was a non-uniform pressure distribution with 2880 rpm, but it less significant when compared to that of the other speeds (2300 rpm and 2500 rpm) [8].

After an extensive literature review of the past and ongoing research in the field, it was observed that a further investigation of the fluid flow characteristics through the impeller blade, such as pressure, velocity and kinetic energy, and an optimization technique, were required. This paper addresses these issues and explains the details of a research that was carried out for hydrodynamic design optimization.

1.3 History of Centrifugal Pumps

At around 2000 BC, a device called the shadoof was invented by the Egyptians in order to raise water with a bucket at one end a weight at the other. Furthermore, the centrifugal pump was invented by a Greek inventor and mathematician Ctesibius in 200 BC. He invented the water organ, an air pump with valves on the bottom by putting a tank of water between them while putting the rows of pipes at the top. In addition, it is considered a principal design for the reciprocating pump that is used today. In 200 BC, Archimedes designed a screw pump. This invention is considered one of the most important inventions of all time, and it is still utilized today in pumping liquids as well as in agriculture fields, for activities such as irrigation, without depending on electric pumps.

In 1475, a mud lifting machine was invented by Brazilian soldier and scientist, Reti. It could be characterized as a centrifugal pump. In 1588, Italian engineer Agostino Ramelli in his book "The Diverse and Art factitious Machines of Captain Agostino Ramelli," described the sliding vane water pump technology, in addition to other pump technologies. Also, a German engineer, Pappenheim, invented the double deep-toothed rotary in 1636 which is still utilized today to lubricate engines. The gear pump that he made was possible to dispense water with reciprocating slide valves, which were used by Ramelli. In addition to this, Pappenheim drove his machine using an overshot water wheel that was set in motion by stream of water, which was then utilized to feed water fountains.

In 1782, James Watt invented the steam engine, and he connected a rod to the engines in order to convert the motion of the piston's reciprocating motion in to the rotary motion of wheels [9].

Figure 1: Peerless Large Split Case Design from the 1940s.

The first centrifugal pump patent was filed by John Gwynne in 1851. The pumps that he invented were used for land drainage, which are seen nowadays in pump house museums. Gwynnes' steam engines powered those pumps. Then, the pumps of all sizes to cover all industrial applications, from small electric pumps to those rated at 1,000 tons per minute were created by Gwynne by the end of the 19th century. Furthermore, the first German patent for liquid ring vacuum pumps and compressors was filed by Siemens in 1900. In addition, in 1901, the first deep well vertical turbine pump was established by Byron Jackson. The Wood trash pump was invented by Albert Baldwin Wood in 1915.

“Some of Wood's pumps have been in continuous use for more than 80 years without need of repairs. New ones continue to be built from his designs” [9].

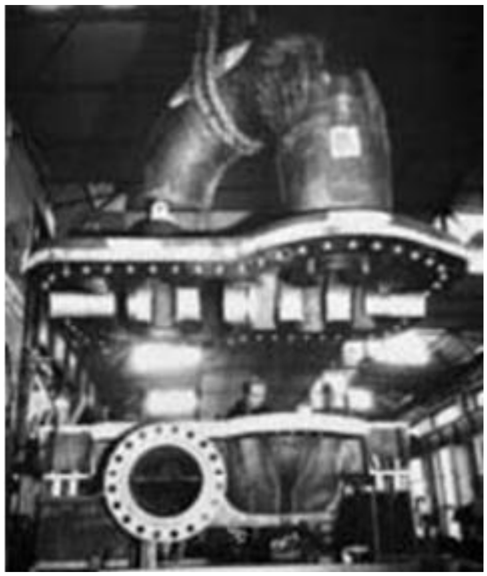


Figure 2: A Single and Two Stage Pipeline Pump Assembly in the 1960s.

1.4 Centrifugal Pumps Development

In the early stages, the research on centrifugal pumps was limited to theoretical study and testing. However, in the past three decades, many advanced methods have been added to develop centrifugal pumps. For example, in many projects, what the engineers and the designers of process plants do is the establish requirements for pumps, which outwits stagnated ranges available from a wide range of pump manufacturing. Because the flow in centrifugal pumps is very complex, technique is required methods to overcome all issues in developing pumps and to increase their performance. For instance, one of these techniques is using a bleeding system with nozzle units. To illustrate, in order to avoid increasing the cavitation in the eye of pump impeller, making the pressure passages of impeller is greater when compared to vapor pressure of the pumped fluid at any of the points of pressure [10].

Due to the development of computer aided design codes, there are many techniques added. A new technique in the modern era is using CFD techniques to develop centrifugal pump to make them consume less power as well as produce less vibrations, and MDO is used to optimize by either reducing the weight or increasing the efficiency. For example, the flow behavior inside the pump and the flow stream flow is analyzed and the optimal design of the centrifugal pump is proposed. In the present days, pumps are more reliable and more powerful. So, they are used in different fields [11].

1.5 Motivation

Centrifugal pumps are common in use, and they can contribute to serving the industry in many things in addition to develop the economies of many countries-for example, agriculture. In the coming years, there is a priority for improving efficiency and for increasing the production and the quality of pumps in terms of the materials.

In order to make the manufacturing of centrifugal pumps more tangible and feasible, studying and conducting investigation of the internal flow through the centrifugal pump especially through the impeller and increasing the efficiency of pumps is more necessary to increase the life expectancy of the centrifugal pumps, by using advanced software codes to perform CFD and MDO. As a result, technology now has improved the performance of the pumps by reducing the cost and producing pumps that have high durability [11].

1.6 Objectives

The objective of this thesis is to conduct an investigation of the hydrodynamic characteristics (pressure and velocity) of flow through the impeller of the centrifugal pump using CFD, and to evaluate a multi-objective optimization process of the centrifugal pump impeller and volute (casing scroll) using HEEDS MDO in order to increase the efficiency and enhance the pump performance.

GDM 10 x 12 HD centrifugal pump is used as a reference pump and the objectives are follows:

- Conduct a CFD investigation of the hydrodynamic characteristics of the fluid flow through the impeller of the pump (pressure and velocity) by performing 2D and 3D on the baseline of GDM 10 x 12 HD pump.
- Define a multi-objective design optimization using HEEDS MDO-Modeler to optimize the efficiency and head.
- Modify the efficiency and head as objectives and relative ratio of inlet and outlet pressure expression as constraints to avoid an increase in head losses.

Chapter 2

Centrifugal Pump Theory

2.1 Impeller and Volute Principle and Theory

A centrifugal pump can be defined as a rotodynamic pump which uses rotating impeller to rise the pressure of a fluid. Fluid enters the pump as a stream into the rotating impeller. The impeller contains a rotating disc with several blades called vanes which are attached to each other, and sometimes recline away from the direction of rotation. Because of suction, flow is seized in the rotating vanes when it comes close to the impeller. As soon as the fluid gets to the impeller's trailing edge it gets accelerated, and the maximum velocity of the fluid gets higher at the impeller's outer diameter. This leaves the impeller into a volute hollow as shown in Figure 3.

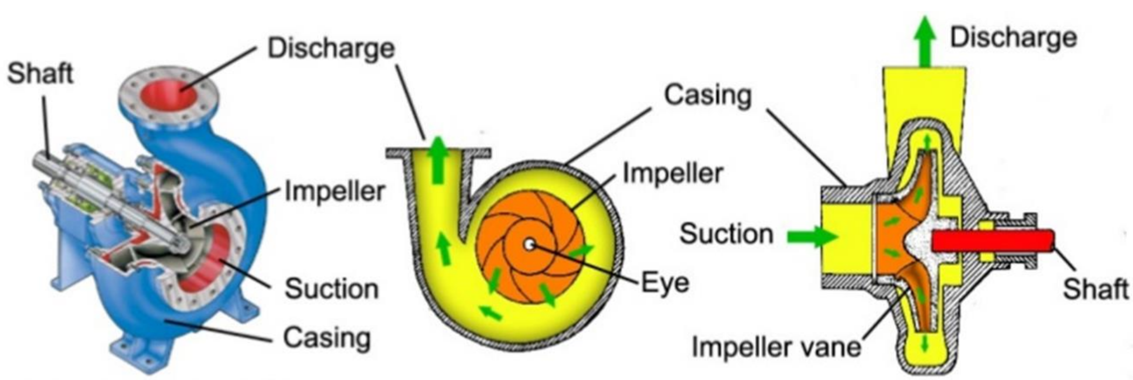


Figure 3: Principle Work and Parts of Centrifugal Pump

The centrifugal force increases the acceleration of the fluid elements due to the difference in radius. Moreover, the process of energy conversation in fluids mechanics follows the Bernoulli's equation, and the total head energy in the pump system is the result of potential head energy, static pressure head energy and velocity head energy.

$$z_1 + \frac{v_1^2}{2*g} + \frac{p_1}{\rho*g} = z_2 + \frac{v_2^2}{2*g} + \frac{p_2}{\rho*g} \qquad (1)$$

Because the centrifugal pump increases the flow speed of a certain fluid, it is fundamentally a velocity machine. After the fluid leaves the impeller, the velocity decreases whereas the pressure increases as it is revealed in Bernoulli's principle. As fluid leaves the pump from its side more fluid is sucked on other side, causing flow.

However, since a volute is a device that gathers outflow and connects to a spiral pipe; it has been widely used in centrifugal pumps. Therefore, the cross sectional area from the inlet to outlet widens proportionality [12].

2.2 Pump Selection

The important part in designing the centrifugal pump are the impeller blade, the choice of twist angle and the material, which allows the customer to purchase only those parts which are necessary for repair. GDM 10 x 12 HD pump was selected for this research, which offers three packing choices. Solid lip seals provide efficient service with a minimum amount of required maintenance. Split compression and V-type packing are available for more severe service conditions, and they can easily be replaced without dismantling the pump.

2.3 GDM 10 x 12 HD Centrifugal Pump

This type of centrifugal pump is more reliable, and has many advantages. For example, it has a heavy steel frame and mounting bracket for horizontal or vertical positioning. In addition, the driving system of GDM 10 x 12 HD is a keyed shaft. Also, it is available for a special coupling and mounting bracket, allowing for a hydraulic motor to be directly mounted to the pump. Further, segmented construction allows the customer to purchase only those parts necessary for repair and maintenance. Change of rotation may be easily accomplished by simply removing. Turning, and repositioning the volute changing the impeller to match the desired direction of rotation is easy. The dynamically closed pattern impellers are designed for moving heavy, abrasive and laden slurries and are available in either clockwise or counterclockwise rotation. The impellers are secured to the shaft by a locking system with a superior design [13].

2.4 Relative Velocities at the Inlet and Outlet of Impeller

The concept of relative velocity in turbo machinery is considered a key idea in turbo machinery. To understand the flow, consider a person standing on the rotating turbo machine as shown in Figure 4 . The person experiences a radial velocity (V) due to the movement of the blades, and tangential velocity (U) due to the rotation of the impeller. The resultant velocity (W) experienced by the person is given by:

$$\vec{W} = \vec{V} - \vec{U} \qquad (2)$$

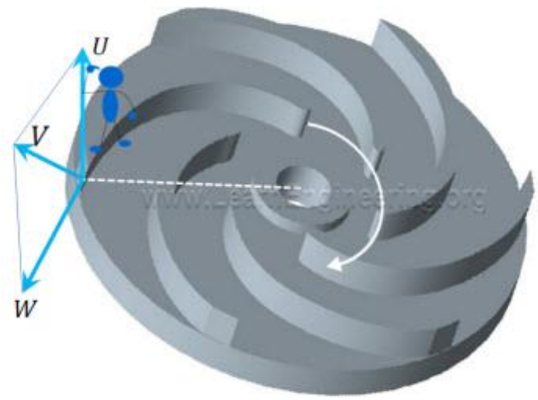


Figure 4: Velocity Triangle in a Turbo Machinery

For a stationary part of a centrifugal pump to have smooth operation, flow should be tangential to the impeller blade. Similarly in a moving device relative velocity should be tangential to the blade profile. With a knowledge of the direction of relative velocity and the vectorial representation of the relative velocity, these three velocities could be drawn as shown in Figure 5. As a result, this is called a velocity triangle [14].

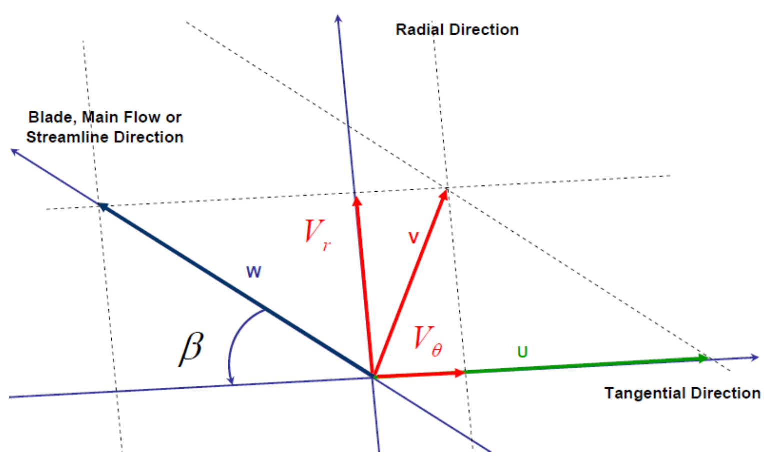


Figure 5: Relative Velocity Direction of Impeller Blade*

***Source of photo: New Mexico Tech.**

2.5 Velocities and Pressures Direction of Volute (Scroll Casing)

The impeller does all the work in a turbo machine. However, the volute collects the flow from the runner and guides it to the discharge. It can be said that the impeller and the volute are operationally dependent on each other, because the pump volute determines the surrounding in which the impeller operates. In addition, the pump volute can have a profound influence on impeller performance and may cause the impeller to work at low efficiency [15].

2.6 Affinity(Similarity) Laws for Centrifugal Pumps

The "Affinity Laws" for centrifugal pumps describe the impact of changes in speed or impeller diameter on pump flow, head, and BHP (brake horsepower). They are useful tools in predicting pump performance changes when speed or impeller diameter are changed under conditions as:

- ✓ variable speed devices are employed,
- ✓ impellers are trimmed, and
- ✓ pump curves, which are plotted at 60 Hz speeds, are to be used across international borders at 50 Hz speeds (and vice versa).

2.7 Affinity(Similarity) Laws

The affinity laws can be defined that, (1) flow will change directly when there is a change in speed or diameter, (2) heads will change as the square of a change as the cube of a change in speed or diameter. Affinity laws are defined in Table 1.

Table 1: Formula Representation of Affinity Laws

$\frac{Q_1}{Q_2} = \frac{D_1}{D_2}$ Or $\frac{Q_1}{Q_2} = \frac{N_1}{N_2}$
$\frac{H_1}{H_2} = (\frac{D_1}{D_2})^2$ Or $\frac{H_1}{H_2} = (\frac{N_1}{N_2})^2$
$\frac{BHP_1}{BHP_2} = (\frac{D_1}{D_2})^3$ Or $\frac{BHP_1}{BHP_2} = (\frac{N_1}{N_2})^3$

where,

Q= Flow Volume Rate

D= Impeller Diameter

N= Speed

H= Head (TDH)

BHP= Brake Horsepower

The subscript 1 indicates "existing condition"; the subscript 2 indicates "new conditions"[16].

2.8 Different Types of Pump Head

- "Total static head – total head when the pump is not running".
- "Total dynamic head (total system head) - total head when the pump is running".
- "Static suction head – head on the suction side, with pump off, if the head is higher than the pump impeller".
- "Static suction lift –head on the suction side, with pump off, if the head is lower than the pump impeller".
- "Static discharge head –head on the discharge side of pump with the pump off".
- "Dynamic suction head/lift –head on suction side of pump with pump on".
- "Dynamic discharge head –head on discharge side of pump with pump on" [17].

$$z_1 + \frac{v_1^2}{2*g} + \frac{p_1}{\rho*g} = z_2 + \frac{v_2^2}{2*g} + \frac{p_2}{\rho*g} + h_2 \qquad (3)$$

$$P_1 + \frac{\rho v_1^2}{2} + z_1 g\rho = P_2 + \frac{\rho v_2^2}{2} + z_2 g\rho + h_2 \qquad (4)$$

Assumptions:
No elongation → $z_1 = z_2 = 0$
Fluid is in steady (V_2=0) $V_1 = V_2 = 0$

$$h = (p_2 - p_1)/(\rho g) + {V_2}^2/(2g) \qquad (5)$$

$$h = \frac{P_{outlet} - P_{inlet}}{\rho*g} \qquad (6)$$

where,

h = total head developed (m)

P_2 = pressure at outlet (N/m^2)

P_1 = pressure at inlet (N/m^2)

ρ = density (kg/m^3)

g = acceleration of gravity (9.81 m/s^2)

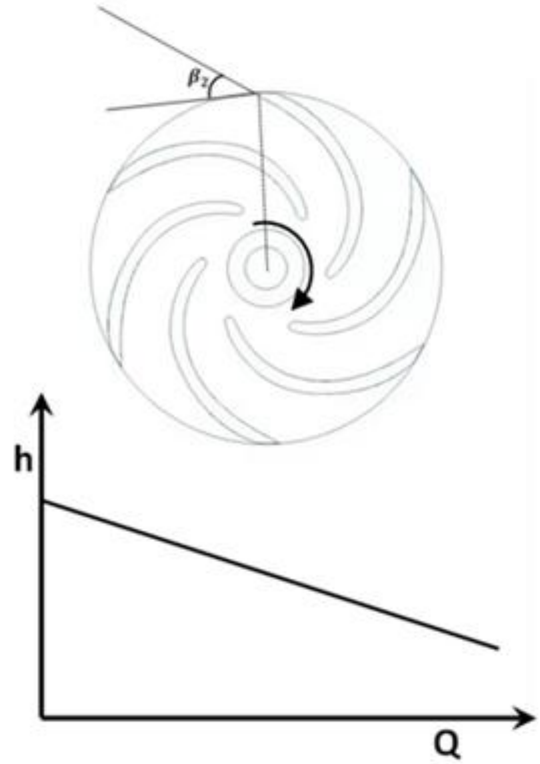


Figure 6: Relationship between Head and Flow for Backward Curved Blade

2.9 Pump Efficiency

Pump efficiency, η (%) is a relationship between the energy generated by the impeller and the energy put into flow. [17]

$$\eta = \frac{P_{out}}{P_{in}} \tag{7}$$

where,

η = efficiency (%)

P_{in} = input power

P_{out} = output power

Total pump efficiency= $\frac{\rho g Q H}{T * 2 * \pi * N}$ (8)

$$\eta = \frac{H * Q * g * \rho}{T * \Omega} \tag{9}$$

$$\eta = \frac{\left[\frac{P_{outlet} - P_{inlet}}{\rho * g}\right] * Q * g * \rho}{T * \Omega} \tag{10}$$

$$\eta = \frac{Q * (P_{outlet} - P_{inlet})}{T * \Omega} \tag{11}$$

ρ: water density = 1000 (kg/m^3)

g: gravitational acceleration = 9.81 (m/s^2)

H: total head (m)

N: Rotation speed (RPM)

T: Torque (N m)

Ω: Angular speed (rad/s)

2.10 Number of Blades

The number of the impeller blades of the centrifugal pump largely effect on the performance. For example, power consumption depends on the number of the blades. It means the more there are, the more consumption of power as well as high cost. In addition, the number of blades plays a major role to influence on the flow turbulence. Therefore, it can design the number of the impeller blades according to the Pfleiderer equation [18].

Number of vanes= $6.5 \frac{D_2 + D_1}{D_2 + D_1} \sin \beta_m$ (12)

First, calculate the average vane angle, and then calculate the number of blades as per Pfleiderer equation:

$$\beta_m \frac{\beta_1+\beta_2}{2} = \frac{20.1+14}{2} = 17°$$

where, $\beta_1 = 20.1°$

$\beta_2 = 14°$

Then,

Number of vanes= $6.5\frac{D_2+D_1}{D_2+D_1}\sin\beta_m = (6.5)\frac{430.6+255.7}{430.6-255.7}\sin 17° = 7.4,$ say 7 blades.

where, $D_1 = 255.7$ mm

$D_2 = 430.6$ mm

2.11 Centrifugal Pump Classification by Flow

Pumps can be classified according the fluid flow through the pump as follows,

- Radial flow pump
- Axial flow pump
- Mixed flow pump

The radial flow pump is a centrifugal pump in which the liquid enters at the center of the impeller and is directed out along the impeller blades in a direction at right angles to the pump shaft [19]. Nowadays, the radial flow pump is centrifugal pumps standard working principle and is required in most applications as shown in Figures 7 and 8.

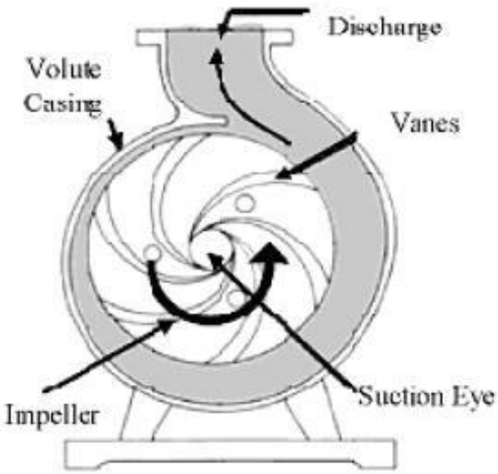


Figure 7: Radial Flow Pump

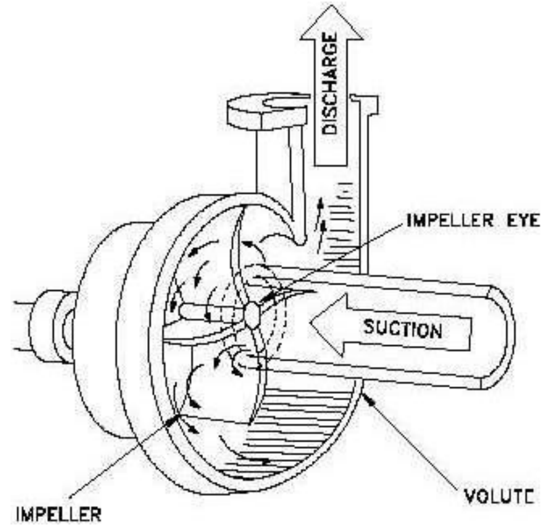


Figure 8: Suction and Discharge in Centrifugal Pump

However, in the axial pumps, in which the liquid exits the pump axial, the flow deflections in the impellers of the radial flow pumps gather higher centrifugal forces. Therefore, this leads to higher pump heads in radial flow pumps, but also to smaller capacity flows [20] as shown in Figure 9.

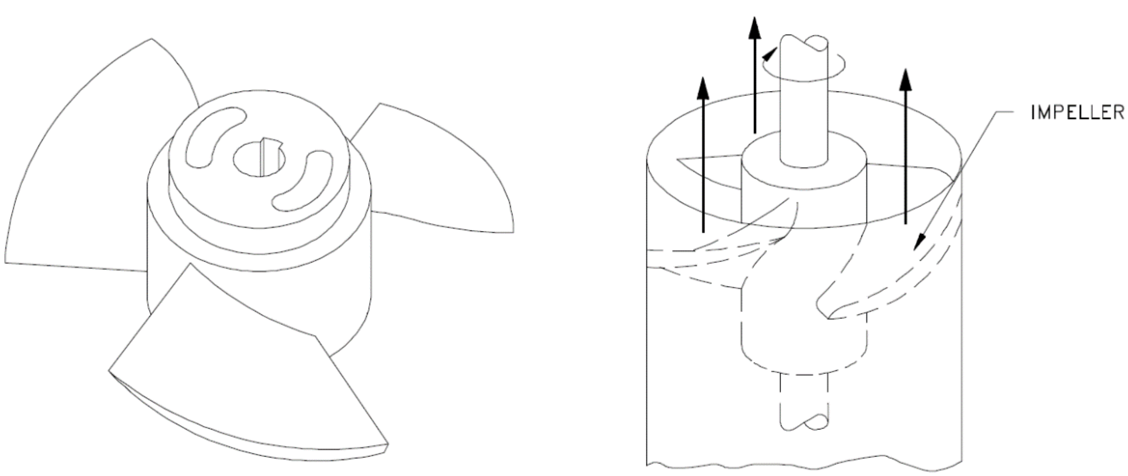


Figure 9: Axial Flow Pump

The third type of the flow pump is mixed flow pump. This flow type collects between the radial and the axil flow pump. As liquid flows through the impeller of the mixed flow pump, the impeller blades push the flow out away from the pump shaft and to the pump suction at the angle is greater than 90 degree [20]. Figure 10 illustrates the impeller of a mixed flow pump and the flow through the mixed flow pump.

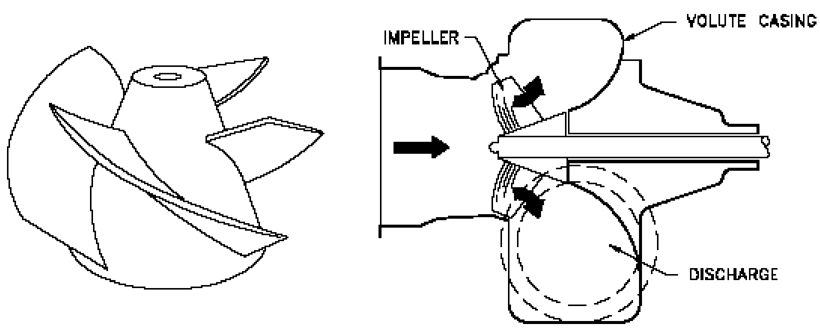


Figure 10: Mixed Flow Pump

2.12 Types of Centrifugal Pumps

Generally, pumps consist of two groups. positive displacement pumps and the dynamic pumps.

2.12.1 Positive Displacement Pump

Fluid flow moves by trapping a fixed amount and forcing (displacing), the trapped volume into the discharge pipe. In addition, an expanding cavity on the suction side and a decreasing cavity on the discharge side are used by some positive displacement pumps. For instance, liquid, which flows into the pump as the cavity on the suction side, expands while the liquid flows out of the discharge as the cavity collapses. However, The volume is constant through each cycle of operation. Figure 11 shows the rotary type of positive displacement of pumps [21].

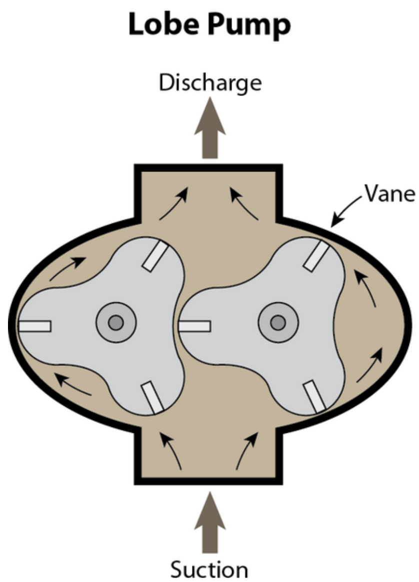


Figure 11: Lop Pump Internals

Another type of positive displacement pumps is progressing cavity sucker-rod pumping (PCP). The pump has a stator, which is typically run into the well on the bottom of the production tubing, whereas the rotor is linked to the bottom of the sucker rod string. Moreover, rotation of the rod string by means of a surface drive system is considered a main part for making the rotor to spin within the fixed stator, creating the pumping action essential to produce fluids to surface as shown in Figure 12 [21].

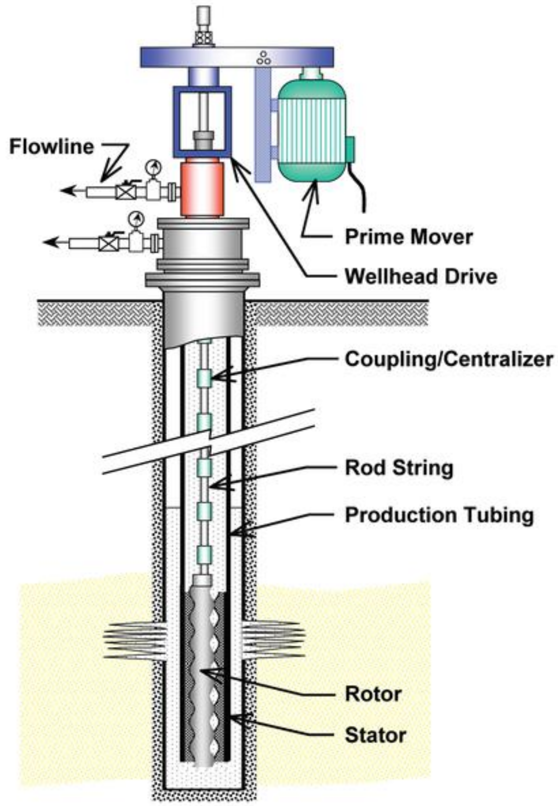


Figure 12: Configuration of a Typical Progressing Cavity Pumping (PCP) System.

2.12.2 Dynamic Pumps

These types of pumps are represented by centrifugal pumps with scroll casing and ESPs systems (Electrical Submersible Pump). Electrical submersible pumps are driven by the cable which is connected to the electric motor. Moreover, the principle work of ESPs is the kinetic energy of the fluid becomes much higher before it is converted to high pressure energy during the through flow. However, the centrifugal pump with scroll casing is usually driven by combustion engine or gas turbine [21].

Figures 13, 14 illustrate centrifugal pump with volute and ESP's system respectively.

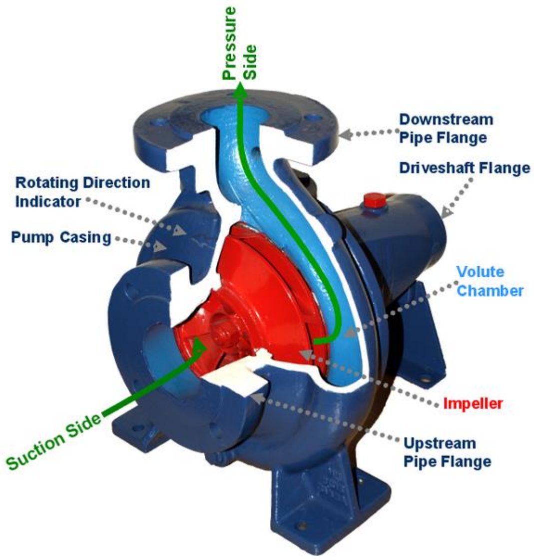


Figure 13: Centrifugal Pump

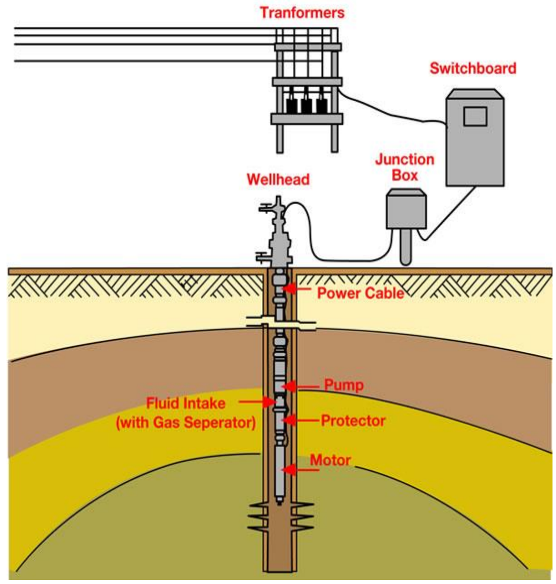


Figure 14: ESP's System*

*Source of photo: New Mexico Tech.

Chapter 3

Software Details

3.1 ANSYS Workbench 14.5

3.1.1 Vista CPD Design

ANSYS CPD Design is an open source centrifugal pump design software, and it can be used to input the values of the pump efficiency and head required to obtain the standard dimensions of the centrifugal pump. Using Vista CPD Design is easy to put the initial design of the pumps in BladeGen modeler. In other words, it makes BladeGen tools easy to use in order to create the proper design of the impeller blade.

3.1.2 BladeGen

BladeGen feature is used to build the impeller blades from 1D to 3D at geometry modeling. BladeGen is also a part of ANSYS Blade Modeler. It is used as a tool for 3D rotor machinery parts. It is utilized to design mixed and radial pumps. Therefore, with this feature, it can be re-design blades for the impeller which is existing to accomplish and create a new design target. BladeGen is extremely important to export files to modeling geometry after creating blade dimensions from 1 D to 3D.

In addition, this feature helps to design different impeller blade models, such as impeller, diffuser of ESP (electrical submersible pump), blade turbine, and compressor blade.

There are two types of the impeller that BladeGen introduces. The first one is standard with initial angle-thickness dialog, and the second is pressure –section type. In this research, the blades are created using angle- thickness, and radial impeller blade of the centrifugal pump. Leading and trailing edges are specified at full diameter simple cutoff and ellipse ratio. As a result, the provision of BladeGen allows the users to do full 3D viscous flow analysis after modeling geometry CAD system [22].

3.1.3 ANSYS TurboGrid Mesh

ANSYS TurboGrid in ANSYS workbench provides a very import mesh feature to turbo machinery system, and it is represent a powerful tool, which allows experts, analysts, and designers of rotating machinery create high-quality hexahedral meshes, whereas conserving the underlying geometry. These meshes are used in the ANSYS workflow to solve complex impeller blade problems. Moreover, ANSYS TurboGrid Standalone has the CFX launcher that makes it easy to run all the modules of CFX without having to use a command line. Therefore, it is used to get fine mesh of the impeller blade of the centrifugal pump using ATM topology method [23].

3.1.4 ANSYS CFD-CFX

ANSYS CFX is used to simulate fluid in different ways, and it has many benefits to allow experts and engineers to test systems in different fields. For example, it is used to simulate water flowing past ship hulls, gas turbine engines, pumps, fans, aircrafts, and compressors [23]. During Computational Fluid Dynamics CFD process, it starts with initial design to geometry generation and mesh. Steady-state preprocessing is used to solve fluid analysis problems depending on Navier-stoke equations. Three parts of CFD as following [24]:

1- CFX-Pre- processer:
   - Geometry domain definition.
   - Grid generation by diving into very small elements.
   - Fluid properties definition.
   - Boundary condition definition.
2- CFX-Flow solver.
   - Solving the equations is done by CFX solver.
3- CFX-Post processor
   - Geometry domain and grid show.
   - Vectors and contours plots.
   - 3D-2D plots

3.2 HEEDS MDO-Modeler

HEEDS MDO-Modeler is used in order to obtain the best and optimum values, or even get design optimum qualities to improve the design criteria. On the other hand, the normal solution approaches do not affect at all time. For instance, sometimes the goals (pressure, mass, efficiency…etc.) get complex and difficult, when experts try to maximize achievability of these goals. In addition, time and cost also play a big role. HEEDS MDO can easily define the parameters, and it is a good option in order to achieve multi-objective design goals. Moreover, using HEEDS MDO-Modeler has significant advantages to choose analysis tools to judge design performance. For example, HEEEDS MDO automatically carries out design iterations while searching for optimum values of the design parameters [25].

3.2.1 SHERPA Method

SHERPA is a hybrid and adaptive algorithm method which is used by HEEDS MDO-Modeler. Moreover, multiple search strategies at once and acclimatizes to the trouble as "learns" about the design space is implemented by SHERPA method. In addition, it utilizes the elements of the multiple search methods at the same time in the proper manner. Besides that, SHERPA has many advantages. For instance, it can find the better solutions at first time, and identifies the better quality of design [26].

Chapter 4

Design of Baseline Model of Pump

4.1 GDM 10 x 12 HD Pump

This type of centrifugal pump is chosen because it has a heavy steel frame and escalating bracket for horizontal or vertical positioning. In addition, special coupling and escalating bracket allows a hydraulic motor to be directly mounted to the pump. Furthermore, segmented construction allows the customer to buy only those parts necessary for required maintenance. Therefore, the selection of this type of the centrifugal pump is more reliable for heavy duty in many different fields [27].

4.2 Baseline Impeller Geometry

Figure 15 shows the meridional view of the impeller, which is created in BladeGen tools in 1D, while Figure 16 illustrates the blade to blade view. A 3D view of the GDM 10 x 12 HD blade impeller is shown in Figure 17. The impeller has seven twist blades, which are created and designed using BladeGen tools in ANSYS workbench.

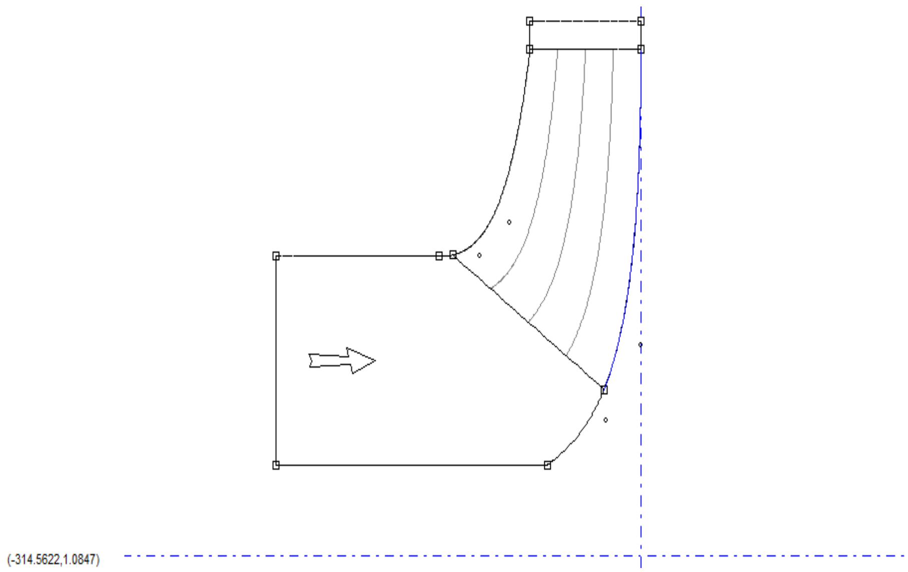


Figure 15: Meridional View of Impeller Blade

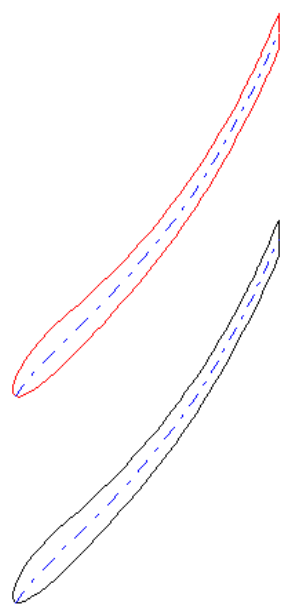


Figure 16: Blade to Blade View

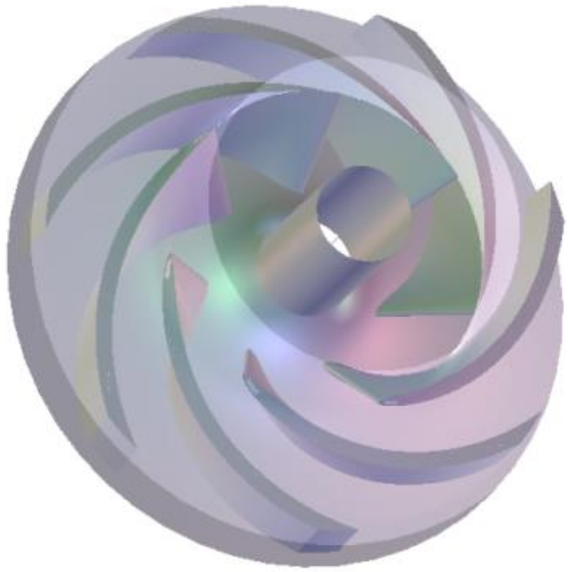


Figure 17: 3D of GDM 10 x 1 HD Blade Impeller

The details of the impeller blade are given in Table 2.

Table 2: Impeller Specifications

Design	Specification
Flow rate	0.251 m3/s
Head	42.6 m
Rotating Speed	1400 rpm
Efficiency	61 %
Inlet Diameter	255.7 mm
Outlet Diameter	430.6 mm
Hub Diameter	77.2 mm
Inlet Angle	20.1°
Outlet Angle	14°
Blade Thickness	12.9 mm
Blade number	7
Outlet width	60.5 mm

4.3 Baseline Volute(Scroll casing) Geometry

Figure 18 shows the geometry of the centrifugal pump volute. Table 3 illustrates the specifications of the scroll casing of GDM 10 x 12 HD centrifugal pump.

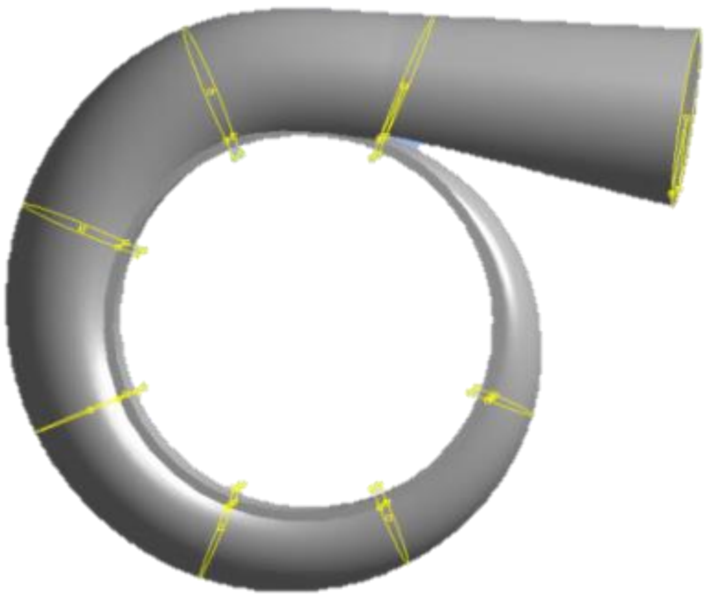


Figure 18: Volute (Scroll Casing) Geometry

Table 3: Section Details of the Scroll Casing

Section Detail	Value
Inlet width	114.3 mm
Cutwater clearance	23.5 mm
Cutwater thickness	9 mm
Exit hydrodynamic diameter	215.1 mm

Table 4 shows the section details of the centrifugal pump scroll casing.

Table 4: Volute Specifications

No.	**Area (mm^2)**	**Centroid radius (mm)**	**Outer radius (mm)**	**Major radius (mm)**	**Minor radius (mm)**	
0	0	238.8	238.8	57.1	0.0	Cutwater
1	2298	249.6	264.4	57.1	25.6	
2	4816	261.5	292.4	57.1	53.6	
3	7509	271.9	314.9	59.5	59.5	
4	10352	281.1	333.8	64.7	64.7	
5	13328	289.6	350.8	70.6	70.6	
6	16428	297.4	366.5	76.6	76.6	
7	19640	304.8	381.1	82.6	82.6	
8	23128	314.0	397.4	88.8	88.8	Throat

Chapter 5

CFD Modeling

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 turbomachinery impeller blades rows. Blade is drawn in a simple 1D meanline method in BladeGen after getting the dimensions from the Vista CPD design tools for GDM 10 x 12 HD 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 [28].

The mesh generation of the impeller blade is done with ANSYS TurboGrid, after that it exported to CFX for physical model definition, solving, and post-processing. Figure 19 shows the ANSYS schematic starting from Blade Design Modeler, CFD analysis, and Multi-Objective Design Optimization using HEEDS MDO.

In this chapter, it will be on the studying of the pre-processing, which contains as following:

- ✓ Geometry definition of the impeller blade.
- ✓ Mesh generation playing a big role in a precise solution.
- ✓ Physical model definition.
- ✓ Achieving fluid parameters.

Figure 19 illustrates the procedure for pump design optimization.

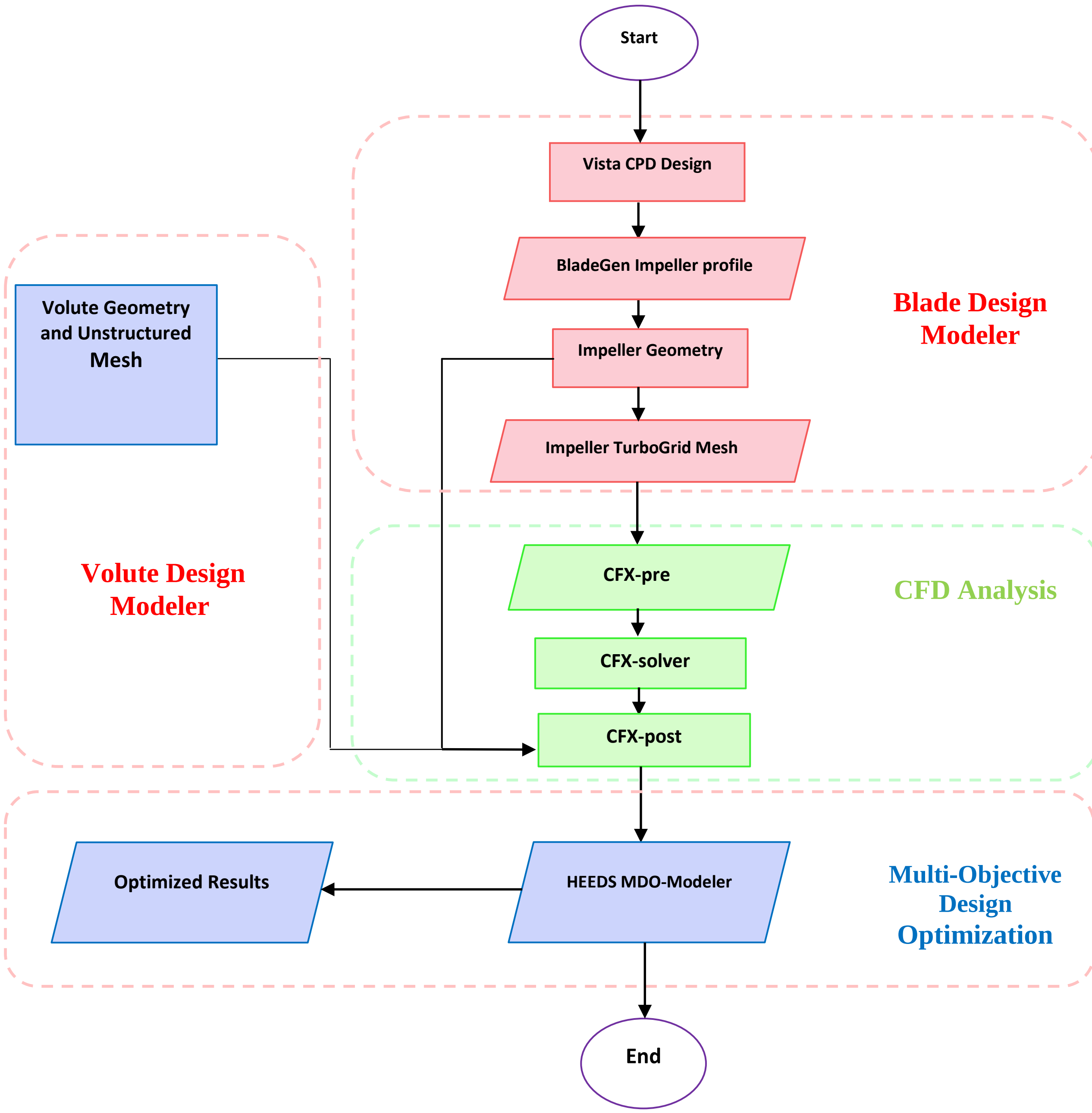


Figure 19: Block Diagram of a Multi-Objective Design Optimization Process

5.1 Geometry of Pump (Impeller and Volute)

In this research, the impeller is considered as a rotor part of the centrifugal pump [R1], while the stationary part [S1] is the volute (scroll casing). The design for two parts is done with ANSYS BladeGen and Geometry Model to develop and determine the hydrodynamic characteristics of fluid flow through the parts especially the rotor part, which is the impeller of the pump. Figures 20 and 21 show the geometry of the pump rotor part and stationary, while Figure 22 shows the two parts of the pump.

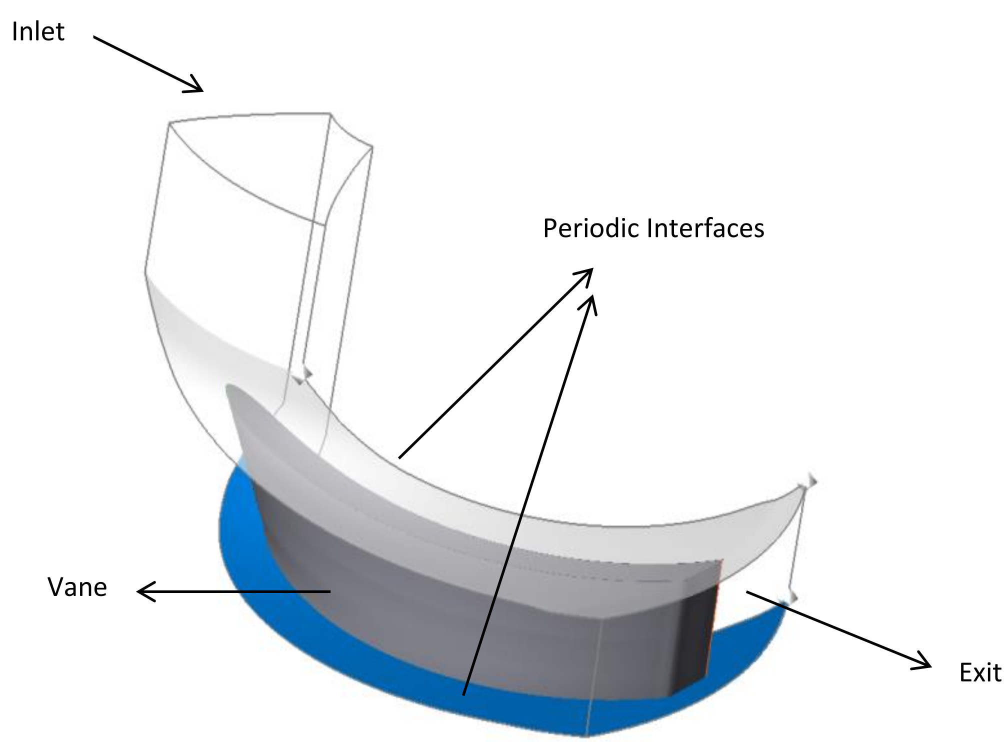


Figure 20: [R1] Rotor Part of Pump

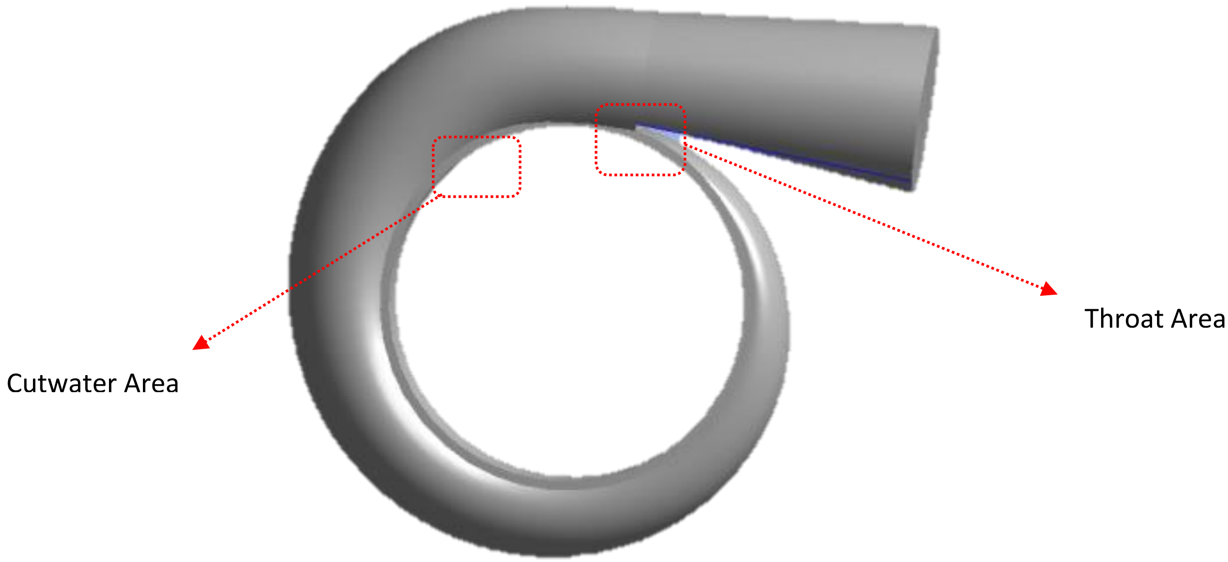


Figure 21: [S1] Stationary Part of Pump

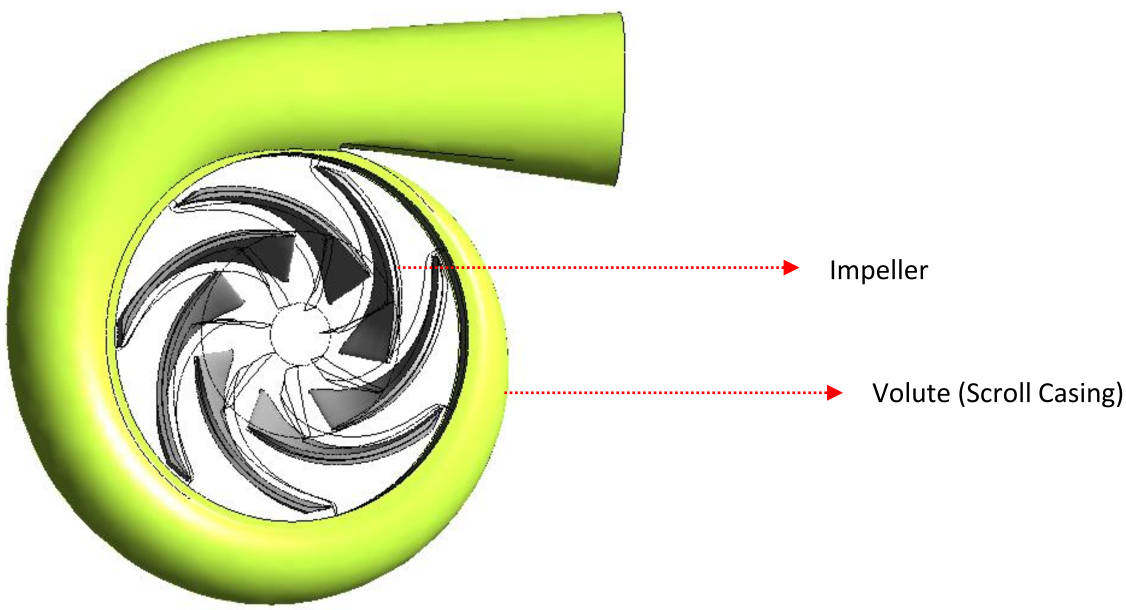


Figure 22: Centrifugal Pump with Impeller and Volute

5.2 Mesh

5.2.1 Impeller Mesh

The blade models were meshed through the use of ANSYS TurboGrid. Therefore, ANSYS CFD tools and TurboGrid structured mesh are used and ATM optimized topology mesh method is selected in the modeler as shown in Figure 23. 25,000 grids were generated to create one passage in the model. The details of the grid system for impeller blade are presented in Table 5. Relatively fine grids were used near the inlet, outlet, and wall surface, while the grids in other areas were relatively coarse [29].

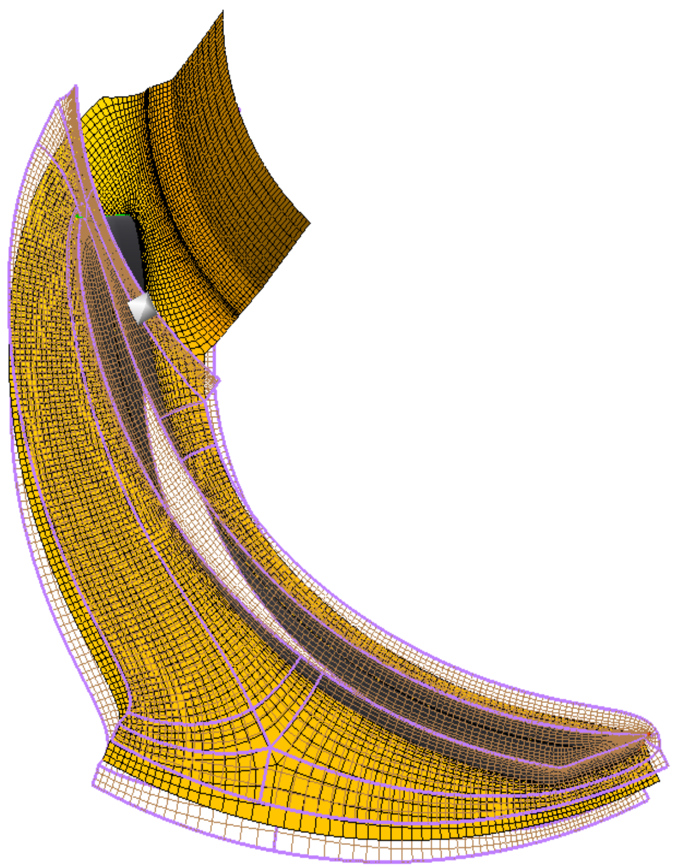


Figure 23: Rotor 1 ATM Topology Structured Mesh

Table 5: Grid Mesh System for Impeller

Number of Nodes	Number of Elements
Fine (250000)	432125

5.2.2 Volute (Scroll Casing) Mesh

The unstructured mesh of the volute was generated by ANSYS workbench with 68322 nodes as shown in Table 6. Unstructured grids used were concentrated near the cutwater areas. Figures 24 and 25 illustrate the details of the grid system of pump volute [4]. Tables 5 and 6 show the grid mesh and the number of nodes and elements for the impeller and the scroll casing.

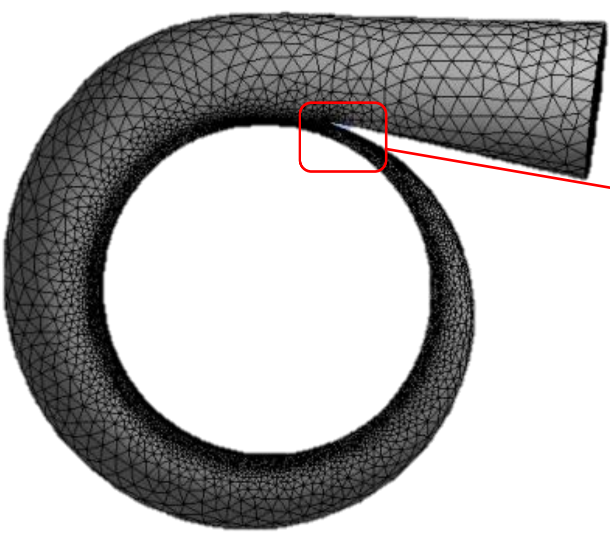


Figure 24: Unstructured Mesh of Volute

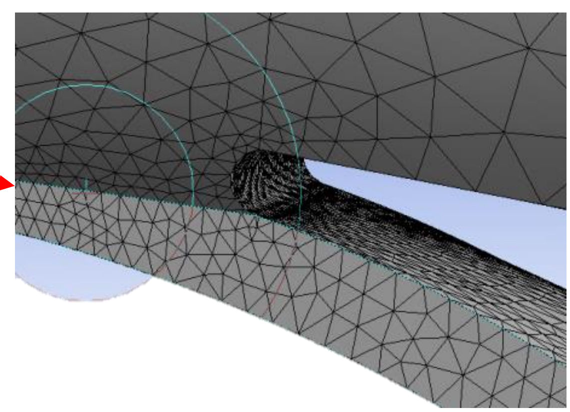


Figure 25: Unstructured Mesh of Volute Throat

Table 6: Grid Mesh System for Volute

Number of Nodes	Number of Elements
63822	188138

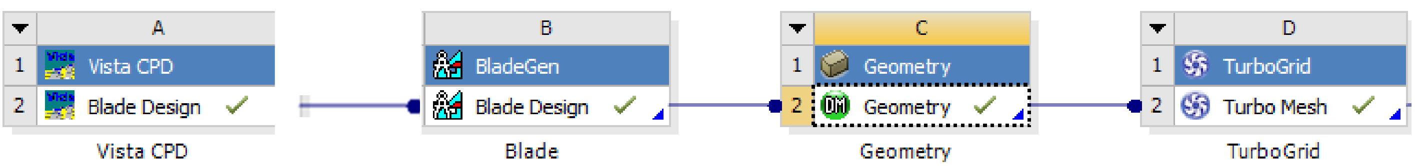


Figure 26: Project Schematic of ANSY Design Modeler and TurboGrid

APPENDEX A details the whole project schematic of Blade Modeler, ANSYS Design Modeler, ANSYS TurbGrid, and ANSYS-CFX.

5.3 Physical Model

Computational Fluid Dynamics (CFD) simulations have been performed with uniform inlet and outlet boundary conditions of the centrifugal pump, which are taken from the meanline analysis. The fluid flow is modeled as incompressible flow using water as a working fluid. K-e turbulence model is used because of its stability, widespread application in commercial software, and it gives good convergence. Moreover, the wall of solid is modeled as no-slip condition. Table 7 provides the specifications of the general CFX pre-processing setup [30].

Table 7: Description of Pre-processor Parameters

Pre-processing Specification	
Analysis type	Steady state
Interference	Rotor frozen
Turbulence model	K-ε
Reference pressure	1 atm
Convergence criteria :	
Residual type	RMS
Residual target	1E-6
Inlet [R1]	
Mass flow rate	0.251 m^3/s
Total temperature	25 $^{\circ}C$
Turbulence intensity	Medium (5 %)
Outlet [S1]	
Static pressure	0 atm
Wall boundaries:	
Mass and momentum	No slip wall
Wall roughness	Smooth

Chapter 6

CFD Simulation and Investigation of the Hydrodynamic Characteristics

6.1 CFD Simulation and Boundary Condition

With the aid of Computational Fluid Dynamics (CFD), the complex internal flow in the pump impeller can be predicted quite well. In this thesis, a steady state solution with k-epsilon turbulence model was used in ANSYS-CFX for both baseline and modified GDM 10 x 12 HD pump models because of its stability, and it gave good convergence. ANSYS-CFX was employed to find velocity distribution, pressure distribution, kinetic energy, and streamlines of the impeller blade as well as setting the efficiency and head as output parameters to make optimization on HEEDS-MDO Modeler [27]. The impeller is modeled in the blade frame, and the volute is in the fixed frame of the reference and both of them are related each other through the “frozen rotor”. Moreover, mass flow rate is applied at the inlet of the pump, and the outlet pressure is used at the outlet of the pump. Furthermore, a smooth nonslip wall is considered over the entire physical surfaces except at the regions between interfaces [5].

6.2 Boundary Condition

At the inlet of computational domain, the mass flow rate is specified (Table 8 and Figure 27). At the outlet, pressure is imposed following the operating condition (Table 8). In addition, a no-slip flow condition is applied on the walls (on the blade, hub and shroud). Because of the change between the reference frame of the static volute and rotating impeller, the interaction of the impeller- volute has been simulated using the Frozen-Rotor interface model as shown in Table 8 [5].

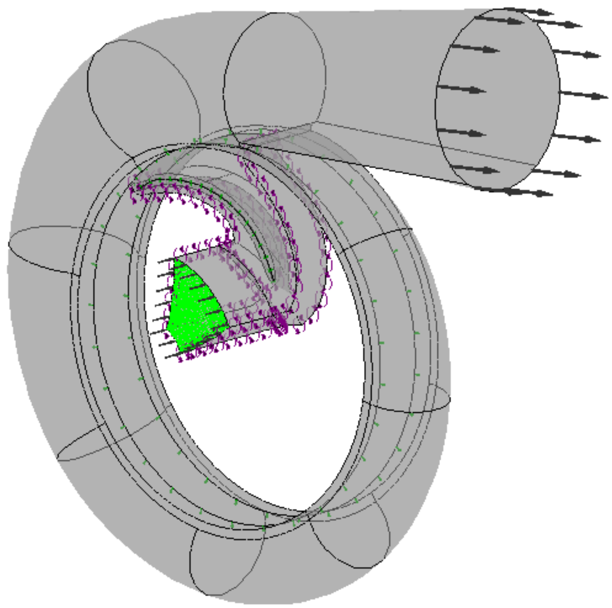


Figure 27: Boundary Conditions applied to the Centrifugal Pump

Table 8: Boundary Conditions of Centrifugal Pump

Flow simulation domain	**Impeller and Volute**
Grid	Impeller structured, volute unstructured
Fluid	Water at standard conditions
Inlet	Mass flow = Q (kg/s)
Outlet	Total pressure = 1 am
Turbulence model	K-ε
Interface impeller-volute	Frozen rotor
RMS (residuals)	10^{-6}

6.3 Pressure and Velocity Distribution

In order to examine pressure and velocity distribution of the impeller blades, the pressure and velocity plots of the impeller blade were generated in post-solver ANSYS-CFX. All simulations are made assuming incompressible flow conditions.

6.3.1 Pressure Distribution

Pressure distribution plots of the impeller blade used in baseline pump are shown in Figures 28, 29 (a, b). It can be clearly seen that pressure is different over the impeller blade surfaces. It is much higher at the trailing edge and less toward the leading edge of the blade, which is normal at the cutwater area between the impeller and the volute.

6.3.2 Velocity Distribution

For the velocity distribution, it can be seen that the velocity flow is higher at the trailing edge where the turbulence occurs at cutwater region between the impeller and the volute. On the other hand, the velocity at the inlet of the impeller blade (suction) is smaller, and increases drastically toward the outlet of the blade. The velocity distribution of the impeller blade is shown in Figures 30, 31 (a, b)

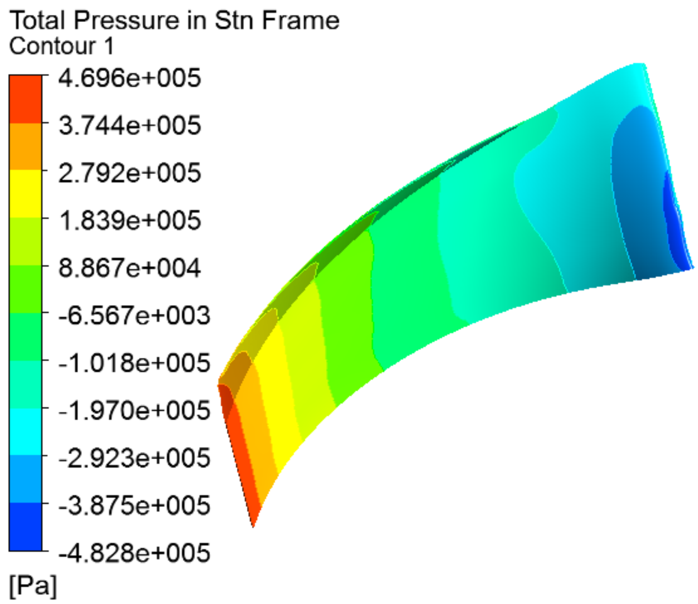


Figure 28: Pressure Distribution through the Impeller Blade (a)

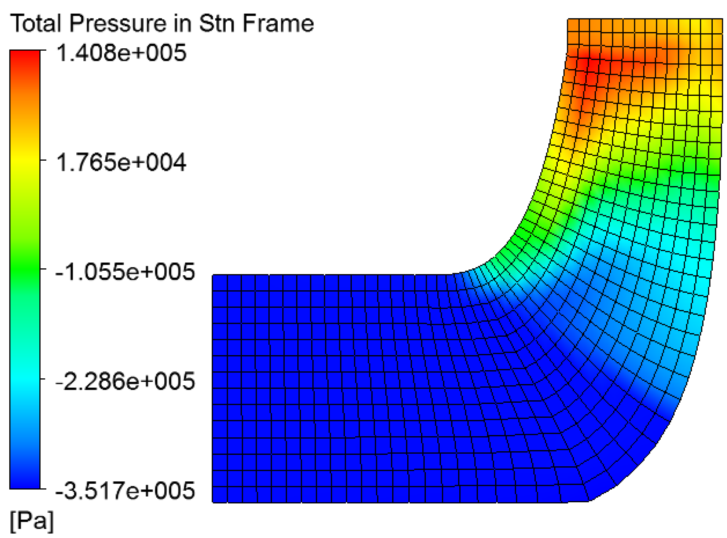


Figure 29: Pressure Distribution through the Impeller Blade (b)

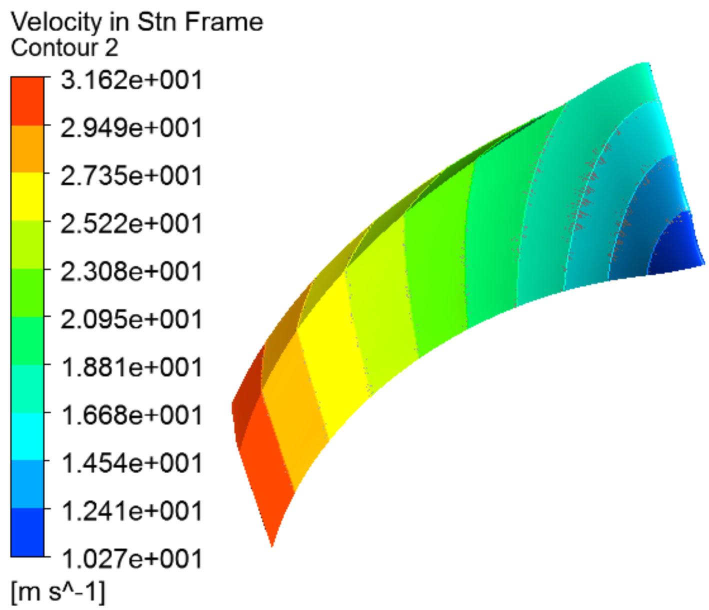


Figure 30: Velocity Distribution through the Impeller Blade (a)

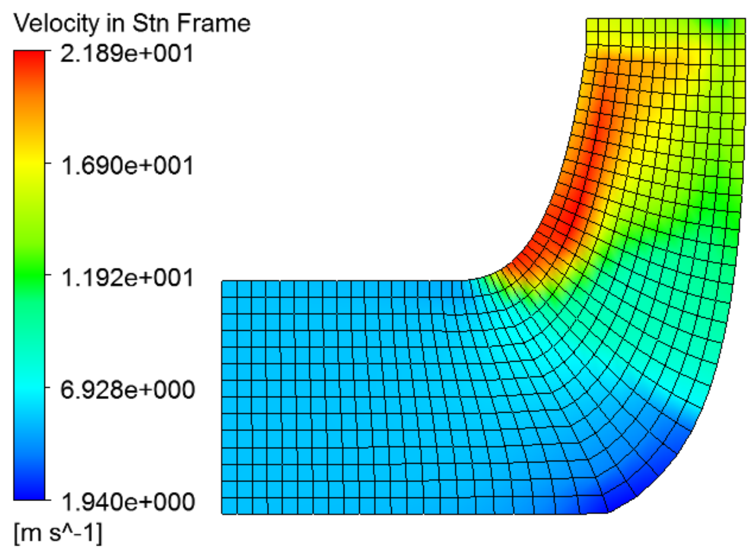


Figure 31: Velocity Distribution through the Impeller Blade (b)

Figure 32 (a, b) shows that the streamlines of the centrifugal pump. It can be observed that the vortices are generated due to turbulence at the trailing edge. This causes head losses as a result of acceleration of water flow in this region, and increases torque and power consumption unless the relative ratio of outlet pressure and inlet pressure is kept constant.

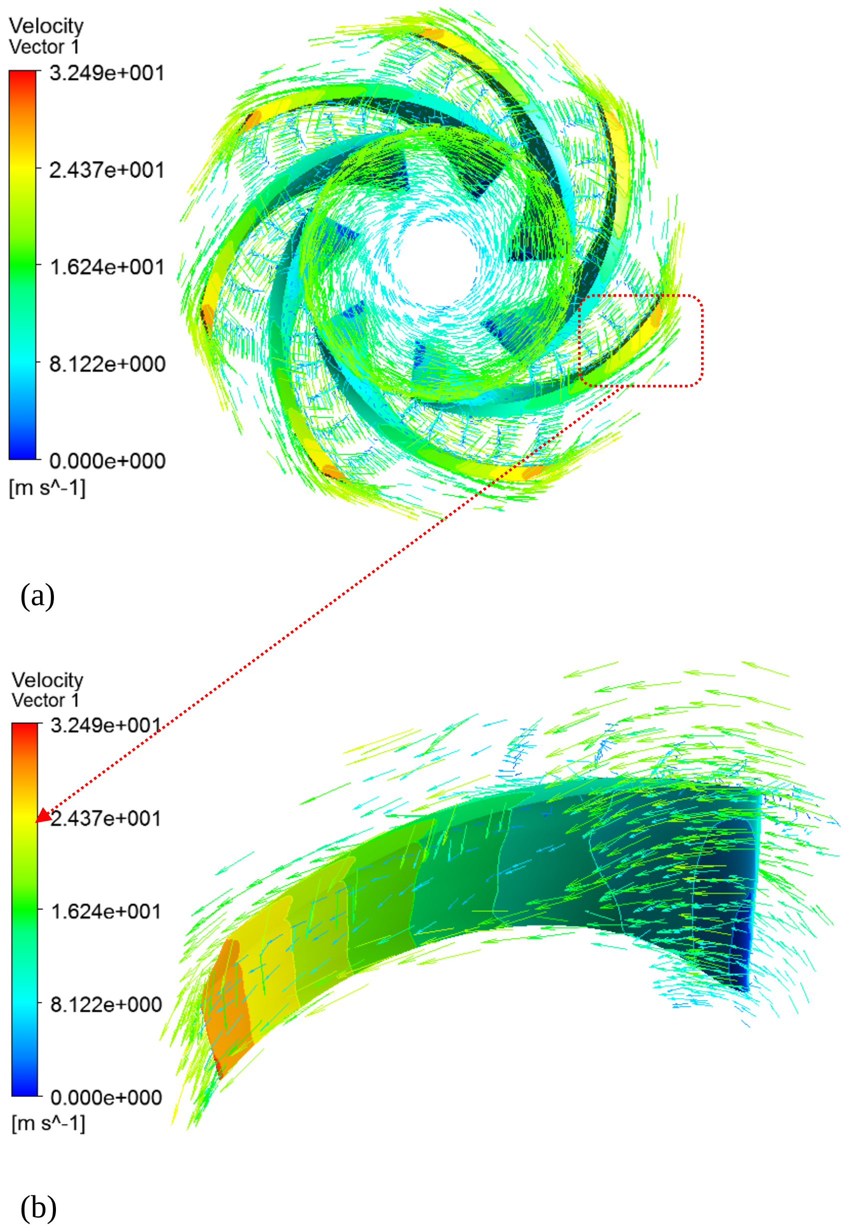


Figure 32: Velocity Vector Distribution through the Impeller Blade (a, b)

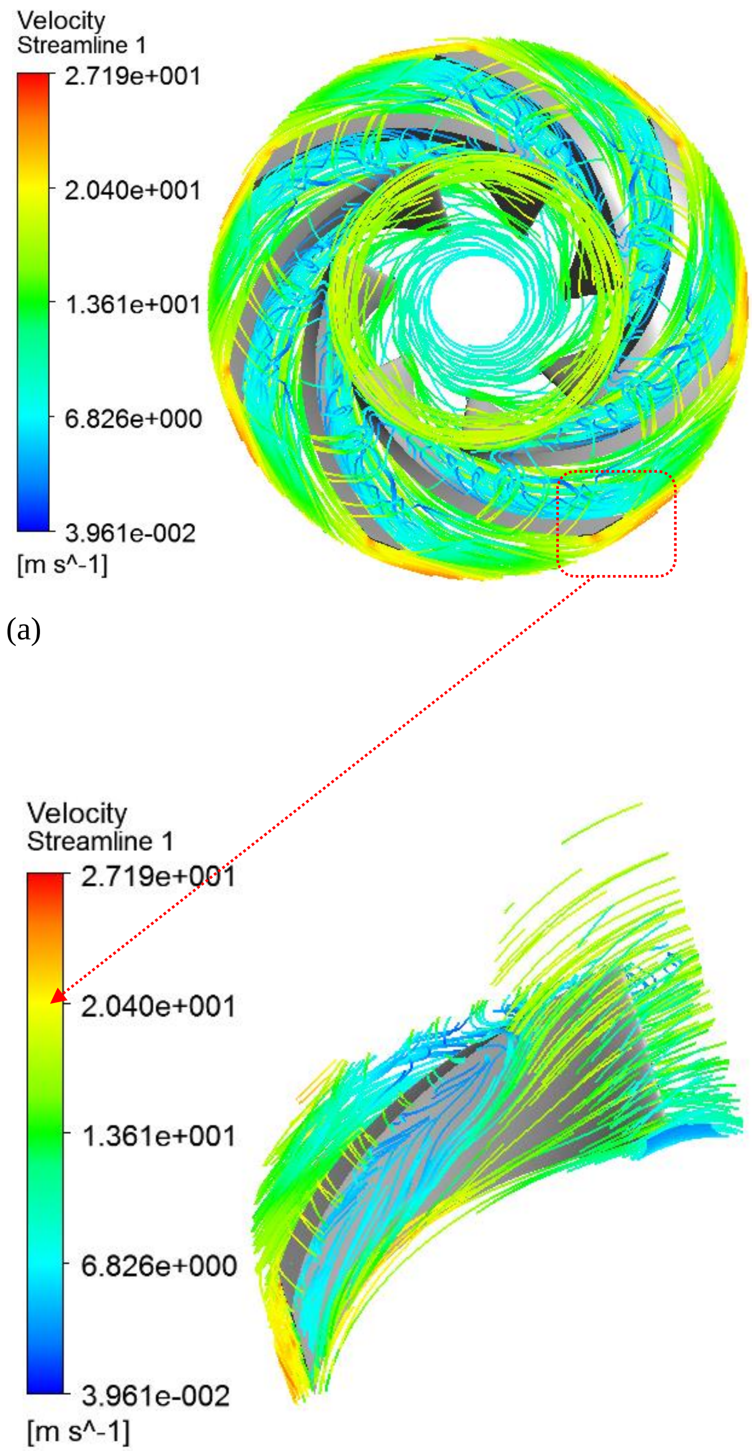


Figure 33: Streamline Distribution through the Impeller Blade (a, b)

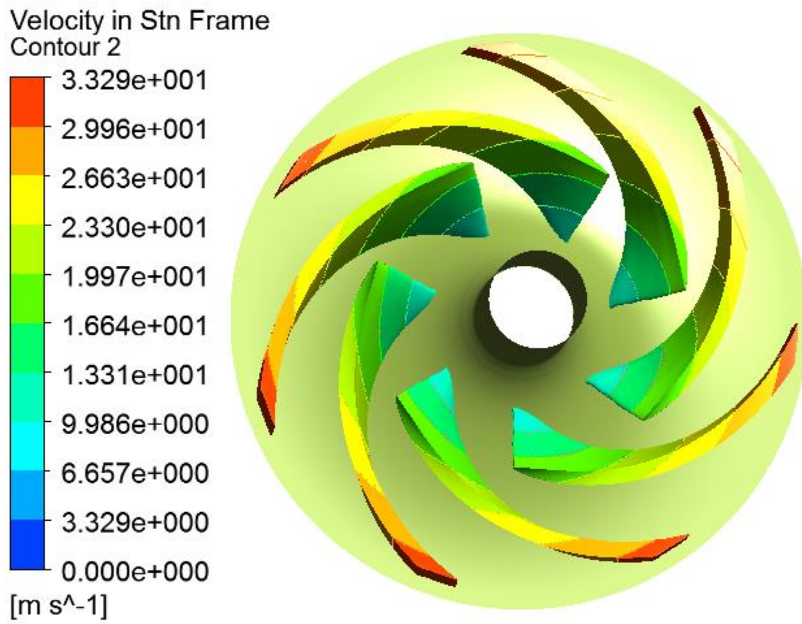


Figure 34: Velocity Contour in Stn Frame through the Impeller Blades

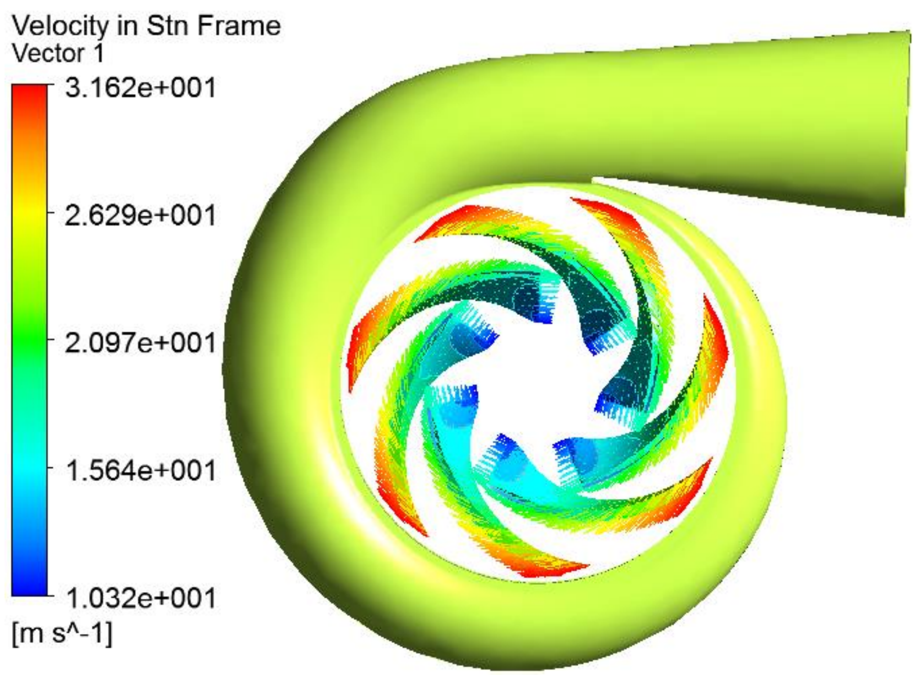


Figure 35: Velocity Vector in Stn Frame through the Impeller Blades

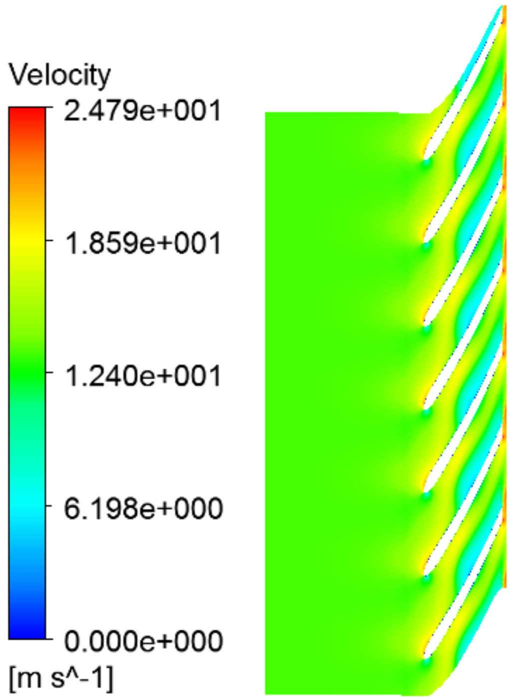


Figure 36: Velocity Contour View Blade to Blade

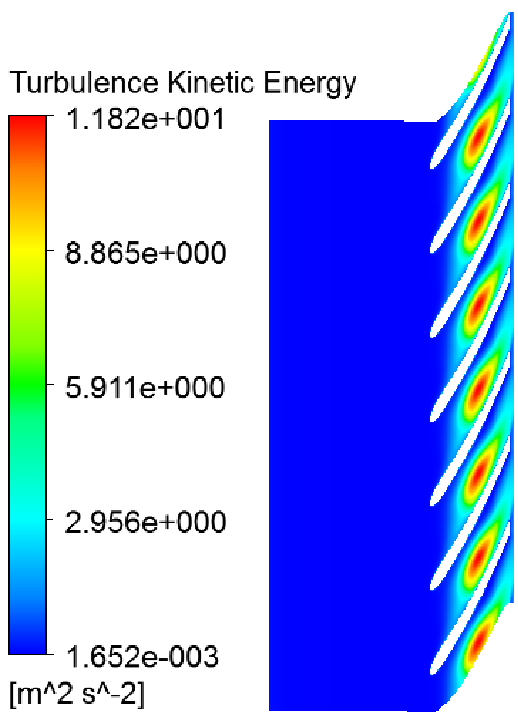


Figure 37: Turbulence Kinetic Energy Contour View Blade to Blade

Figure 33 (a, b) shows the streamline distribution through the impeller blade. It can be clearly seen that the fluid flow stream is much higher at the trailing edge as compared with the leading edge, while Figures 34, 35 illustrate the velocity of fluid flow through the impeller blade. Figures 36, 37 show the velocity and turbulence kinetic energy view blade to blade respectively. Also, Tables 9, 10 illustrate minimize and maximize velocity and pressure fluid flow through the impeller blade respectively. Figure 38 shows the water flow through the pump and both velocity and vector contours.

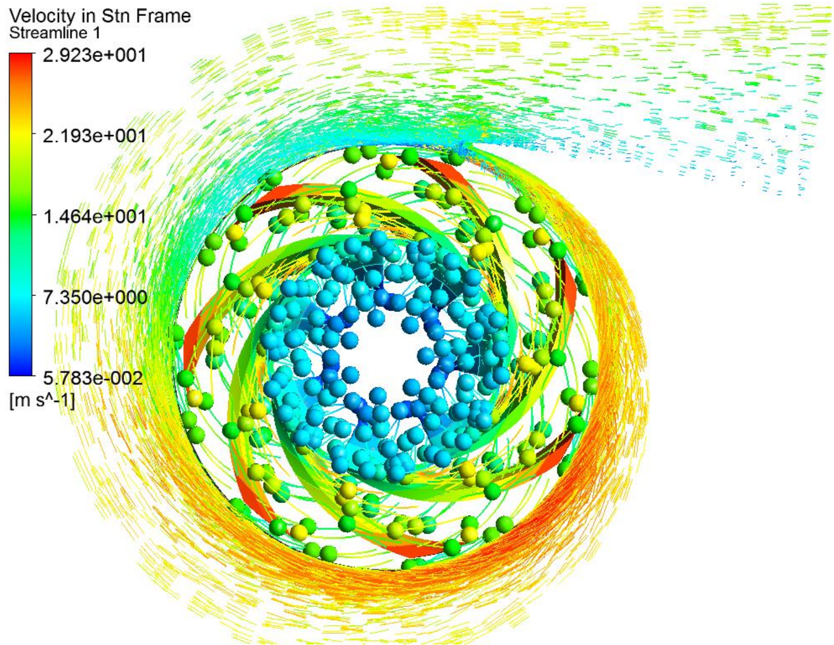
Table 9: Minimum and Maximum Fluid Flow Velocity through the Impeller Blade

Minimum Velocity (m/s)	**Maximum Velocity (m/s)**
1.027e + 001	3.162e + 001

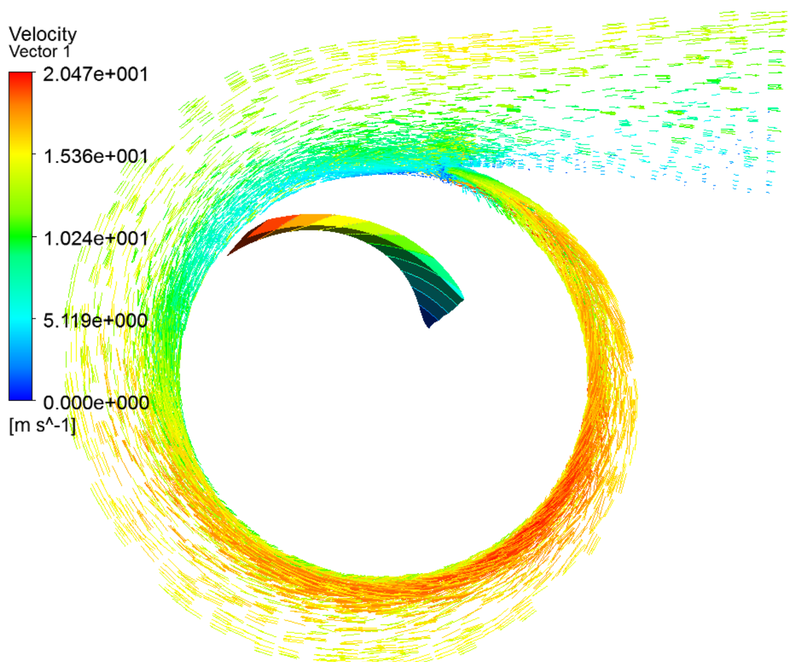
Table 10: Minimum and Maximum Fluid Flow Pressure through the Impeller Blade

Minimum Pressure (Pa)	**Maximum Pressure (Pa)**
-4.828e + 005	4.696e + 005

Figures 39 and 40 show the pressure and velocity change along the blade and the volute of the centrifugal pump. After the fluid leaves the impeller, the velocity increases whereas the pressure decreases. In addition, the load distribution along the blade is observed to be smooth.



(a)



(b)

Figure 38 (a, b): Fluid Velocity Contour and Vector through the Impeller Blades and Volute

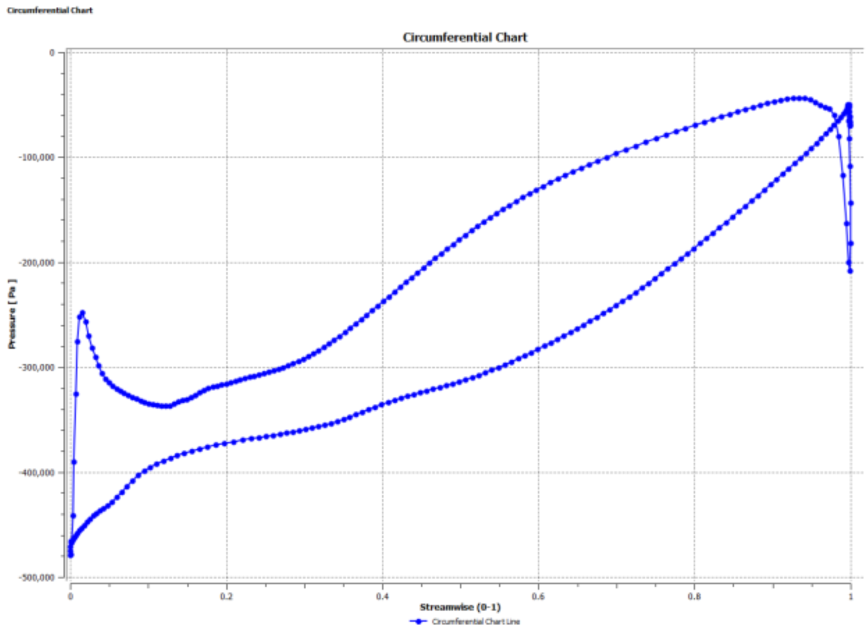


Figure 39: Pressure along the Centrifugal Pump

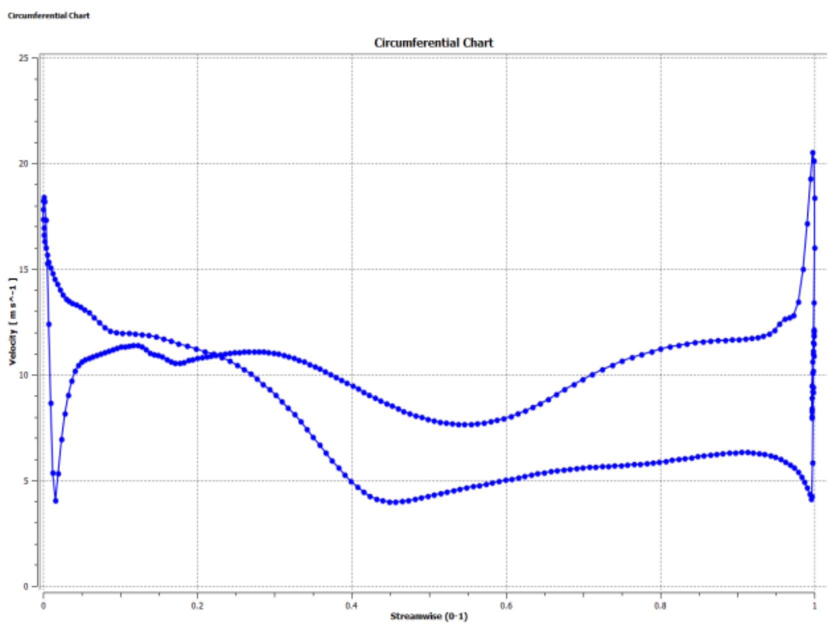


Figure 40: Velocity along the Centrifugal Pump

Chapter 7

Multi-Objective Hydrodynamic Design Optimization

7.1 HEEDS-MDO Modeler

HEEDS MDO-Modeler is used to find the optimum values of centrifugal pump impeller and volute. The hybrid and adaptive algorithm (SHERPA) optimization method is used by HEEDS MDO-Modeler. It uses the elements of the multiple search methods at the same time in the proper manner [31].

7.1.1 Advantages of SHERPA

SHERPA method has advantages for optimization values as following:

- ✓ In Sherpa method, the performance improves with every several iterations, whereas the performance with Algorithm method does not.
- ✓ Sherpa method can be used for removing user specified tuning parameters because it acclimatizes with each problem.
- ✓ In the first time of simulation, it can find the better solutions by using this method of optimization successfully.
- ✓ It helps many non-experts in different fields to apply optimization successfully.
- ✓ Global and local research can be performed at the same time by this method [31].

7.1.2 Multi-Objective Design Optimization

The objective of this design study is to maximize efficiency and head of GDM 10 x 12 HD centrifugal pump. The relative ratio of outlet pressure at the exit of the pump to the inlet pressure at the entrance should not increase to more than 0.15166, which is set as a constraint because the more the relative pressure ratio increases, the more head loses will occur. The optimization will be accomplished using 23 variables, related to the shape of the impeller and the volute of the centrifugal pump. SHERPA method is used to perform the optimization study. Table 11 shows the constraint and the multi-objective of the MDO process.

Table 11: Constrain and Multi-Objective for MDO Process

Objectives	Efficiency	0.61
	Head	42.6 m
Constraint	Ratio of head losses $(\frac{P_2}{P_1})$	0.15166

7.2 Centrifugal Pump Optimization

The optimization is accomplished using 23 variables relating to the shape of the impeller and the volute of the centrifugal pump. Table 12 illustrates the baseline design of the impeller and the volute.

Table 12: Baseline Design

	Parameter Name	value
Impeller Variables Geometry	Beta-1	34.316
	Beta-2	22.5
	Beta-3	20.683
	Beta-4	22.5
	Blade-thickness	12.919
	Blade-number	7
Volute Model Geometry	r3	238.78
	b3	114.3
	b2	60.49
	Clear side	23.469
	Volute thickness	9.0429
	theta2	0
	thetaCR	14
	minor 1	25.602
	minor 2	53.645
	minor 3	59.506
	minor 4	64.7
	minor 5	70.605
	minor 6	76.64
	minor 7	82.62
	minor 8	88.773
	diffLength	355.36
	Ex Hyd-diffDiam	215.07
All dimensions in (mm) and angles in degree		

7.3 Optimum Results

For the new GDM 10 x 12 HD centrifugal pump optimum design, HEEDS MDO was set and optimum results were generated. 100 iterations were used to get the feasible and non-feasible results (maximum and minimum values) of the baseline variable evaluations. Figure 41 shows the various design evaluations of the variables through the iterations carried out to arrive at the final optimized design.

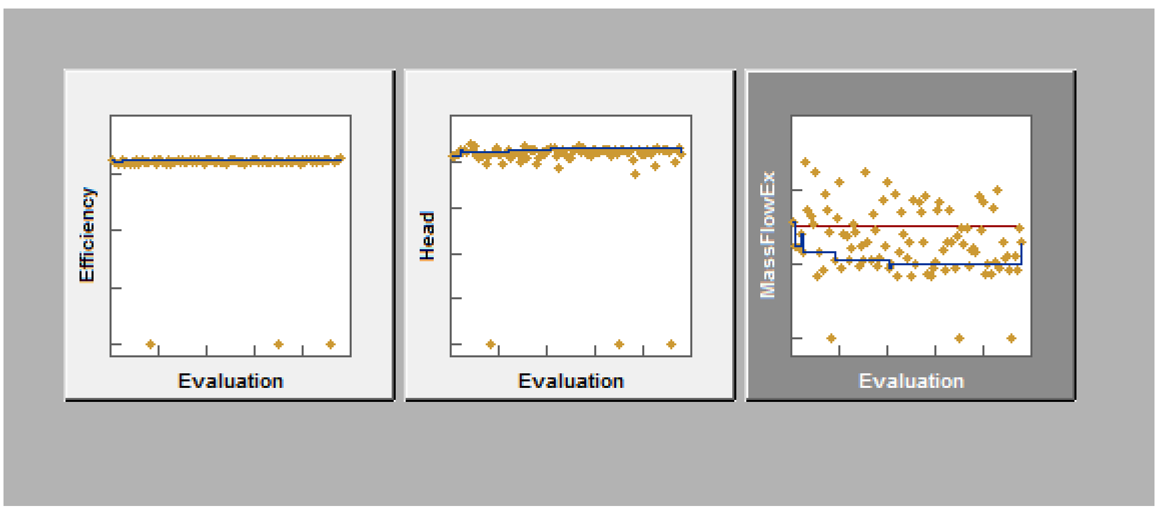
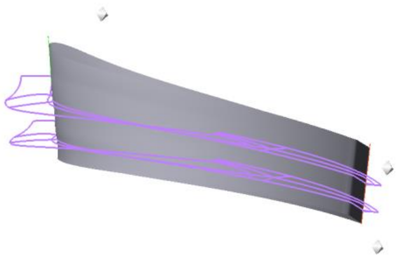


Figure 41: Objectives and Constraint

Table 13 shows the optimized design of the impeller and the volute getting from HEEDS MDO-Modeler process.



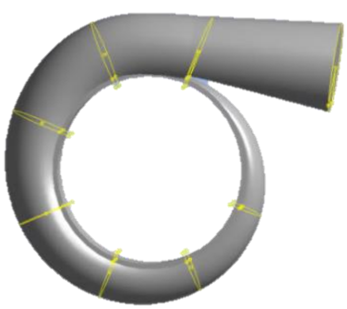


Table 13: Optimized Design

	Parameter Name	value
Impeller Variables Geometry	Beta-1	33.15
	Beta-2	20.4
	Beta-3	21.97
	Beta-4	24.75
	Blade-thickness	11.18
	Blade-number	7
Volute Model Geometry	r3	239.68
	b3	114.26
	b2	59.1
	Clear side	25.2
	Volute thickness	8.12
	theta2	-2.64
	thetaCR	15.68
	minor 1	24.92
	minor 2	52.35
	minor 3	60.3
	minor 4	66.05
	minor 5	69.7
	minor 6	75.4
	minor 7	80.5
	minor 8	88.7
	diffLength	356.18
	Ex Hyd-diffDiam	213
All dimensions in (mm) and angles in degree		

Table 14 shows a comparison of the baseline design and the optimized design in terms of efficiency and head. The efficiency was increased by about 3.2 % and the head was decreased by about 2 % from baseline values. Tables 15 and 16 illustrate the optimized minimum and maximum velocities and pressure through the impeller blade respectively. The vortex of turbulence of fluid flow in optimized values is much less than baseline values, as shown in Figure 43.

Table 14: Baseline and Optimized Model Values

	Head (m)	**Efficiency [%]**
Baseline Model	42.626	61
Optimized Model	41.62	63

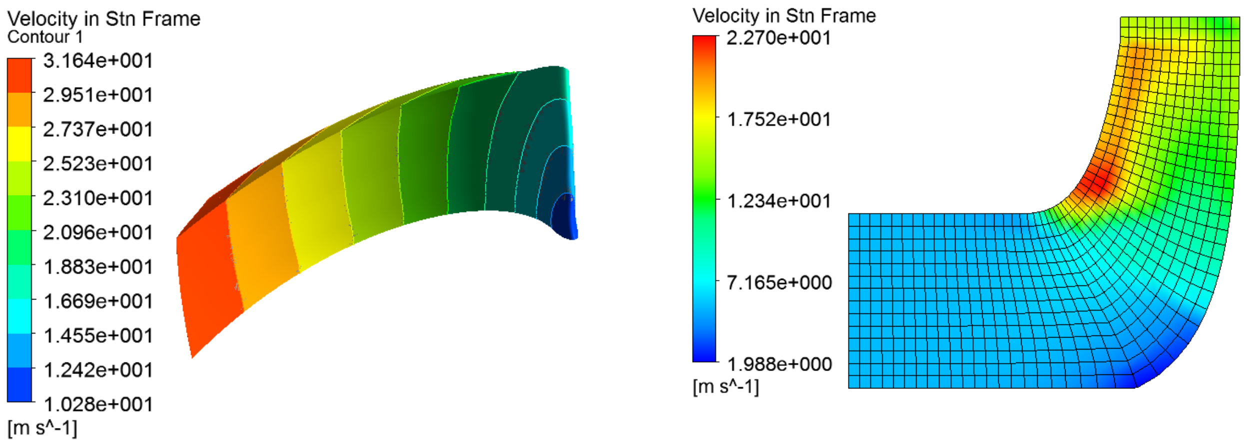
Table 15: Optimized Minimum and Maximum Fluid Flow Pressure through the Impeller Blade

Minimum Velocity (m/s)	**Maximum Velocity (m/s)**
1.028e + 001	3.164e + 001

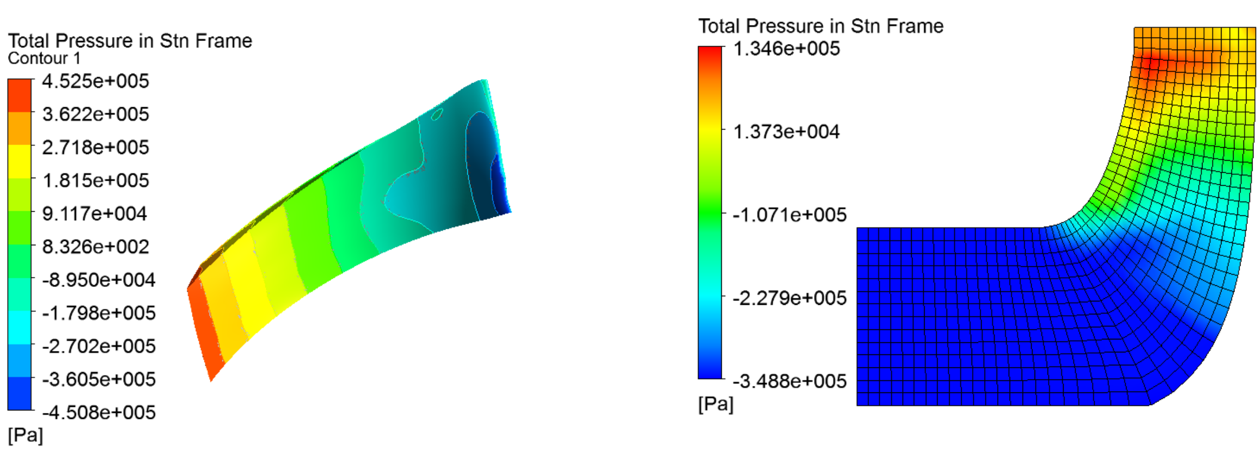
Table 16: Optimized Minimum and Maximum Fluid Flow Pressure through the Impeller Blade

Minimum Pressure (Pa)	**Maximum Pressure (Pa)**
-4.508e + 005	4.525e + 005

Figures 42 and 43 show pressures , velocities and the turbulence concentration after optimization. It can be clearly seen that the turbulence is reduced after optimization.



(a)



(b)

Figure 42: Optimized Min and Max Velocity and Pressure through the Impeller Blade (a, b)

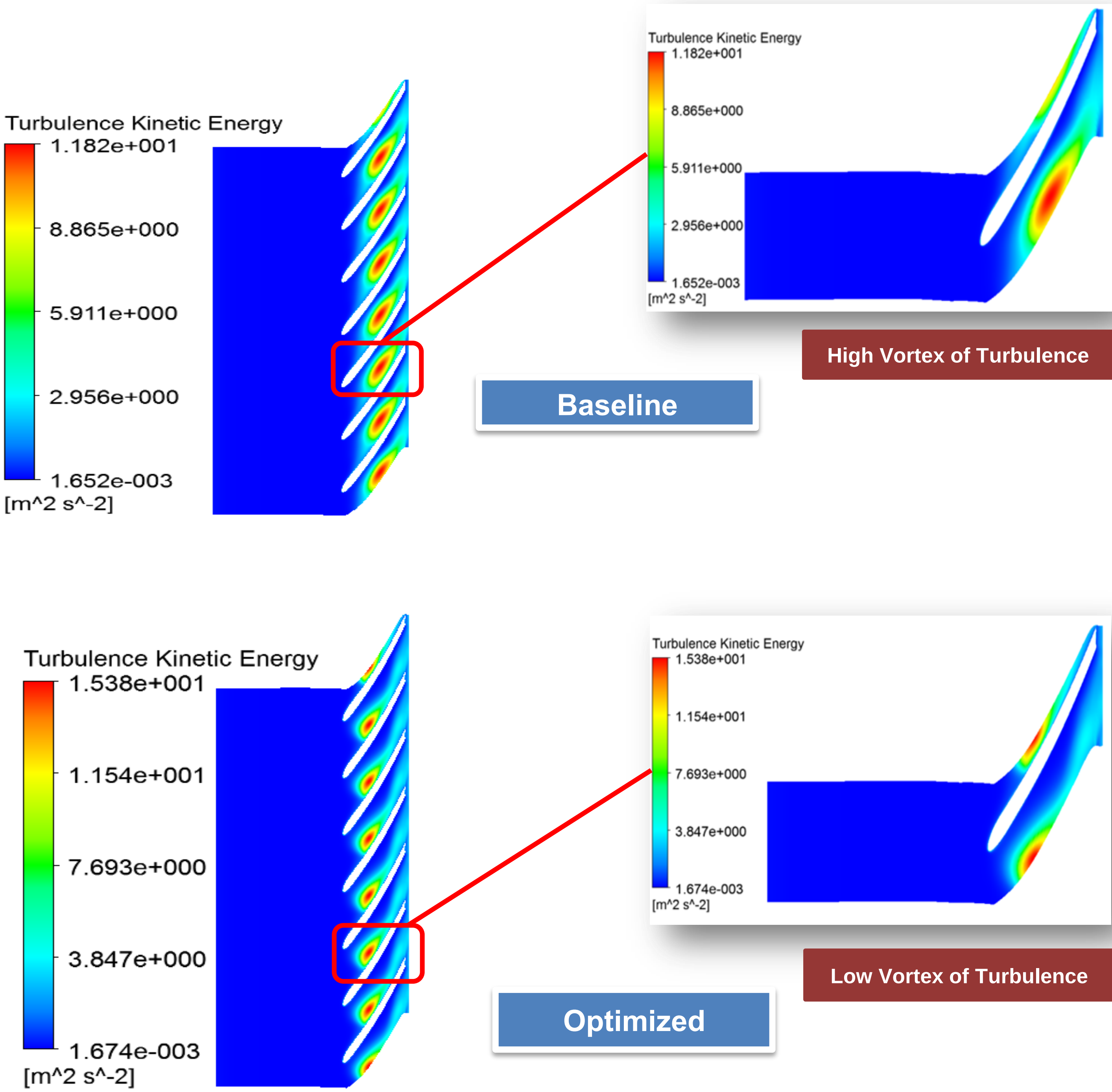


Figure 43: Turbulence Flow over Blade to Blade before and after Optimization

7.4 Performance In SHERPA Method Optimization

It can be clearly observed that the performance is much better with SHERPA method, as shown in Figure 44.

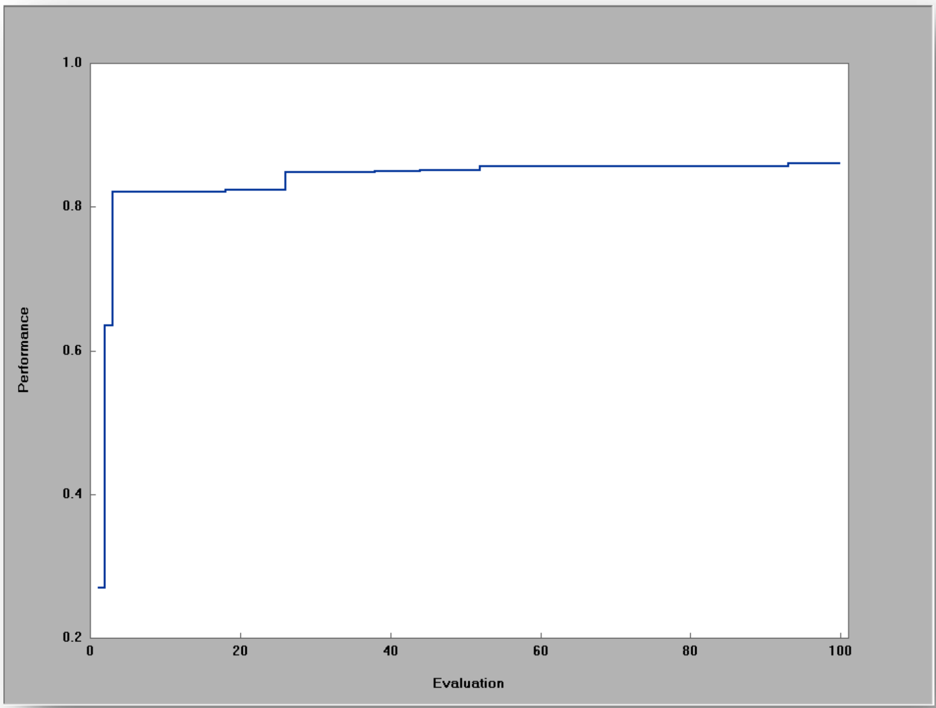


Figure 44: SHERPA Optimization Method

Chapter 8

Conclusion

A CFD investigation of the hydrodynamic characteristics of fluid flow through the impeller blade of the centrifugal pump was conducted, and was represented in terms of pressure and velocity. A pressure analysis was conducted, and it was found that the pressure is higher at the trailing edge than the leading edge of the impeller blade, decreasing rapidly at the outlet. The velocity of fluid flow through the blade was higher at the trailing edge as compared to the leading edge, which was determined to be normal because the kinetic energy was increased in this area of the blade. A multi-objective design optimization process was carried out for the GDM 10 x 12 HD centrifugal pump using 2D BladeGen and ANSYS for CFD analysis, and HEEDS MDO-Modeler for optimization. It could be clearly seen that every single objective could not simultaneously reach the optimum value in the multi-objective optimization, but a compromise among the objectives was needed. The hydrodynamic performance of the optimized pump efficiency was improved as compared to the baseline design. However, the head was decreased from the baseline design to prevent the increase of head losses.

The following were observed in the optimized design:

Efficiency was increased by 3.2 %

Head was decreased by 2 %

Blade thickness was decreased by 13.5 %

Volute thickness was decreased by 10.2 %

The vortex of the turbulence after optimization is improved as compared with the baseline. It is decreased drastically through the blade area.

Future Considerations

- ✓ More realistic design could be produced.
- ✓ Considering more design variables.
- ✓ Considering a more complex design.
- ✓ Including more stages, hub, casing, and more adding layers between shroud and the hub of the impeller blade.
- ✓ Performing acoustic and thermodynamic analyses.
- ✓ Reduce the number of blades.
- ✓ Understanding HEEDS MDO methods.
- ✓ Solve discrepancy between HEEDS MDO and ANSYS workbench solution.

BIBLIOGRAPHY

[1] Vasjaliya, N., Gangadharan, S. N., "Aero-Structural Design Optimization of Composite Wind Turbine Blade.", MSc Thesis, Embry-Riddle Aeronautical University, Daytona Beach, FL, 2013.

[2] ANSYS CFX Introduction (.pdf), V12.0.1 and V13.0, ANSYS Inc.

[3] Weidong, Z., "Investigation of Flow Through Centrifugal Pump Impellers.", PhD Thesis, Natoinal University of Singapore, Singapore, 2004.

[4] Zhou, W., and Zhao, Z., "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamic.", International Journal of Rotating Machinery: P. 49-61, 2003.

[5] Kim, J H., "Design Optimization of a Centrifugal Pump Impeller and Volute using Computational Dynamics", Department of Mechanical Engineering, Hanyang University 1271 Sa-3-dong Sangnok-gu, Ansan, 425-791, Republic of Korea." IOP : 1-9., 2012

[6] Mehta, M., Patel, P., "CFD Analysis of Mixed Flow Pump Impeller", International Journal of Advanced Engineering Research and Studies, College of Engineering, Dist. Gandhinagar, St. Gujarat, India, 2013

[7] Thummar, A., et al, "Experimental Investigation of Open Well Centrifugal Pump Performance" International Journal of Advanced Engineering Research and Science, Engineering and Technology, R. K. Technical Campus, Gujarat Technological University, Rajkot-360002, Gujarat, India, 2012.

[8] Prabu, T., "Hydrodynamic Design of a Centrifugal Pump Impeller", Phd Thesis, Anna University, Channel- 600 025, 2007.

[9] "Pumps and Systems." The History of Pumps: Through the Years. N.p., 01 Jan. 2012. Web.

[10] Zaher, M.A., "New Development in Centrifugal Pumps for Optimum Cavitation Performance.", Vo1.16., Unitec Institute of Technology, Auckland, New Zealand, 1971.

[11] Vasjaliya, N.G., “Fluid-Stracture Interaction and Multidisciplinary Design Analysis Optimization of Composite Wind Turbine Blade”, MSc Thesis. ERAU-Daytona Beach, 2013.

[12] Allerstorfer, C., "Centrifugal Pump." Thesis Becheler , Vol. 40, Leoban.

[13] Gardner Denver Centrifugal Pump. Vol.1., Houston, Richmond, 2007.

[14] "Fundamentals of Turbo Machinery." ~ Learn Engineering. N.p.,Web. 23 May 2014.

[15] Pavesi, G., "Impeller Volute and Diffuser Interaction." NATO OTAN, Vol.27, 2006.

[16] Wilson, S., "PUMP AFFINITY LAWS FOR CENTRIFUGAL PUMPS.", Vol. 3., GRUNDFOS, 2000.

[17] Morvey, Z. K. and Gvozdenac, D. D, "Centrifugal Pumps.", " Applied Industrial Energy and Environmental Management", Engineering ToolBox. Vol. 25.

[18] "Log Mean Temperature Diff And Arthematic Temperature Difference - Student.", Cheresources.com Community. N.p., n.d. Web. 22 May 2014. <http://www.cheresources.com/invision/topic/17746-log-mean-temperature-diff-and-arthematic-temperature-difference/>.

[19] "Centrifugal Pump Classification by Flow." Centrifugal Pump Classification by Flow., 02 June 2014.

[20] "Radial Flow Pump." Radial Flow Pump. N.p., n.d. Web. 02 June 2014.

[21] "Mixed Flow Centrifugal Pump - Engineers Edge." Mixed Flow Centrifugal Pump - Engineers Edge. N.p., n.d. Web. 02 June 2014.

[22] "Pump." Wikipedia. Wikimedia Foundation, 23 May 2014. Web. 24 May 2014.

[23] "Progressing Cavity Pump (PCP) Systems." -. N.p., n.d. Web. 24 May 2014.

[24] Takacs, G., "Electrical Submersible Pump Manual, Design, Operation, and Maintenance". Oxford, UK, Apr, 04, 2012.

[25] Anan, V., "Design Methodology for Aerodynamic Design of Centrifugal compressor". M Sc. Thesis, VIT University, India, 2007.

[26] ANSYS-14-TurboGrid-Introduction, (Http://www.scribd.com/doc/95124517), 2012.

[27] "Design Optimization with HEEDS MDO." Discover Better Designs, Faster. Web. 17 May 2014., 2014

[28] "Centrifugal Pump." Gardner Denver: ,Vol. 1, 2007

[29] Shulman, V. , "Performance Improvement through Velocity Triangle Optimization-Drive Redesign for Mitigating the Horseshoe Vortex in Axial Turbines", . M.Sc. thesis ERAU. Daytona Beach, Florida, 2014.

[30] Mehta, M., and Patel, P., "CFD Analysis of Mixed Flow Pump Impeller." International Journal of Advanced Engineering Research and Studies, Collage of Engineering, Dist. Gandhinagar, St. Gujarat, India. (n.d.): 1-5., 1997.

[31] Hedi, M. L, Kanfoudi, H., and Zgolli R., "Numerical Analysis of the Flow Through in Centrifugal Pumps." International Journal of Thermal Technologies, B.P. 37. Le Belvedere, 1002 Tunisia department, University, Tunis. 216-221, 2012.

Appendix A (Project Schematic Process)

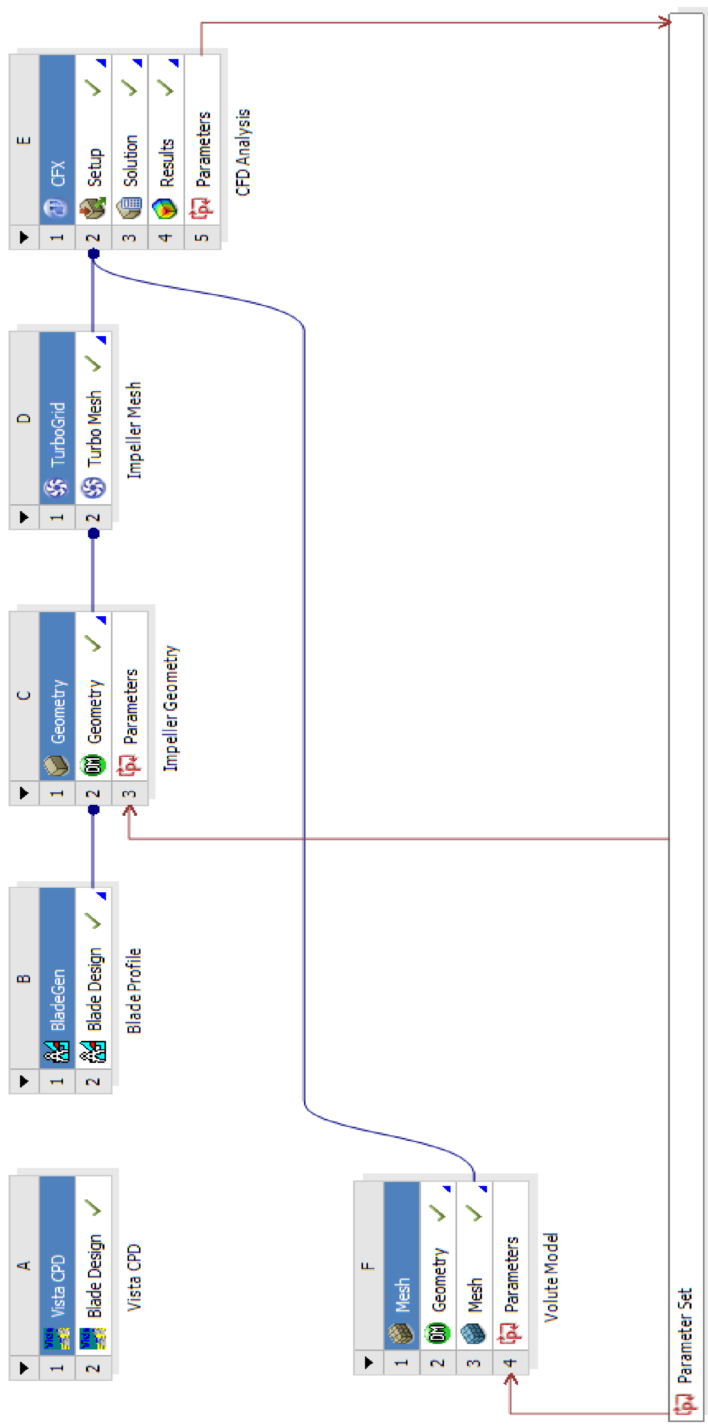


Figure 45: Project Schematic Process in ANSYS Workbench

Appendix B (Number of Iterations)

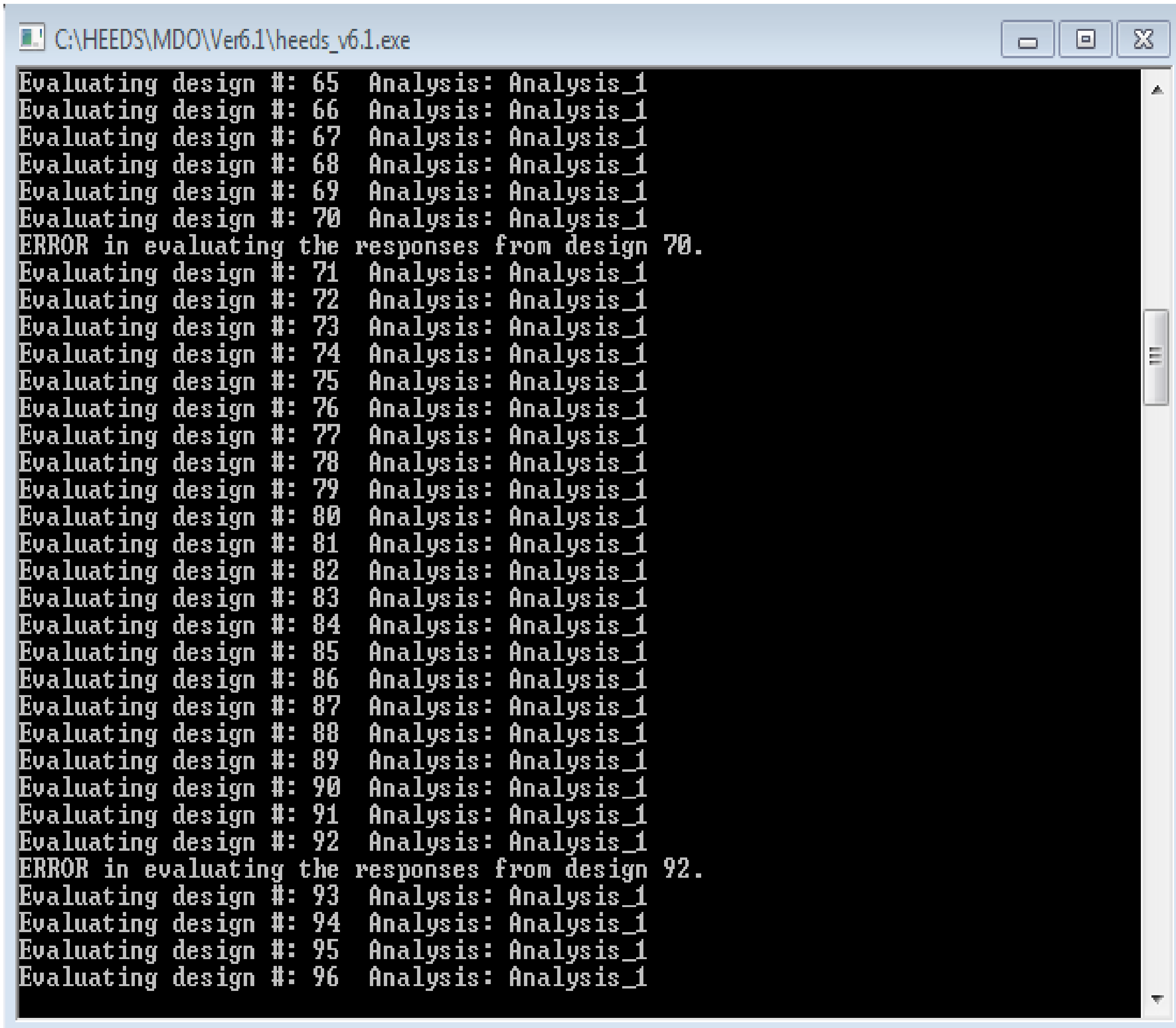


Figure 46: Number of Iterations in HEEDS MDO-Modeler

Appendix C (Efficiency and Head Process)

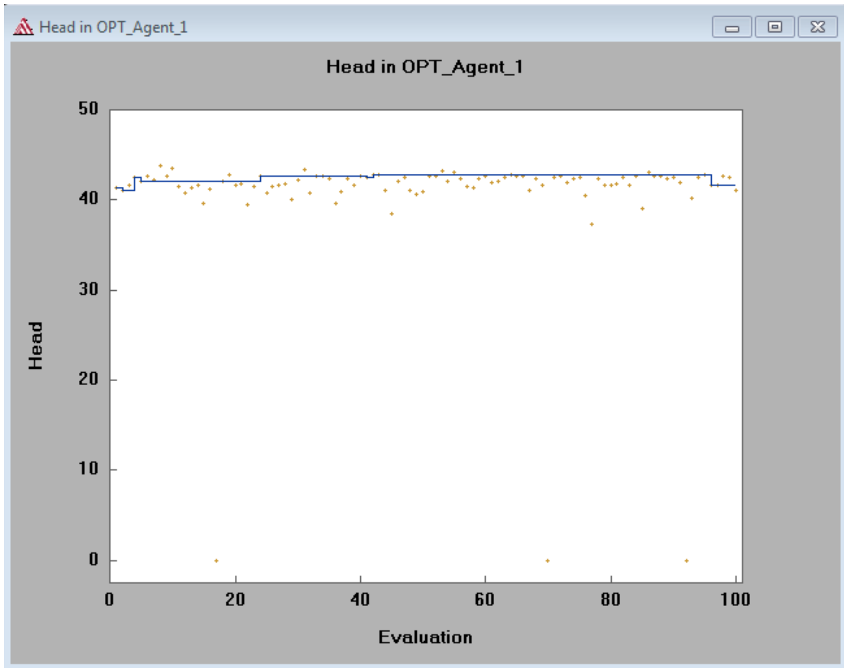


Figure 47: Head Objective Process

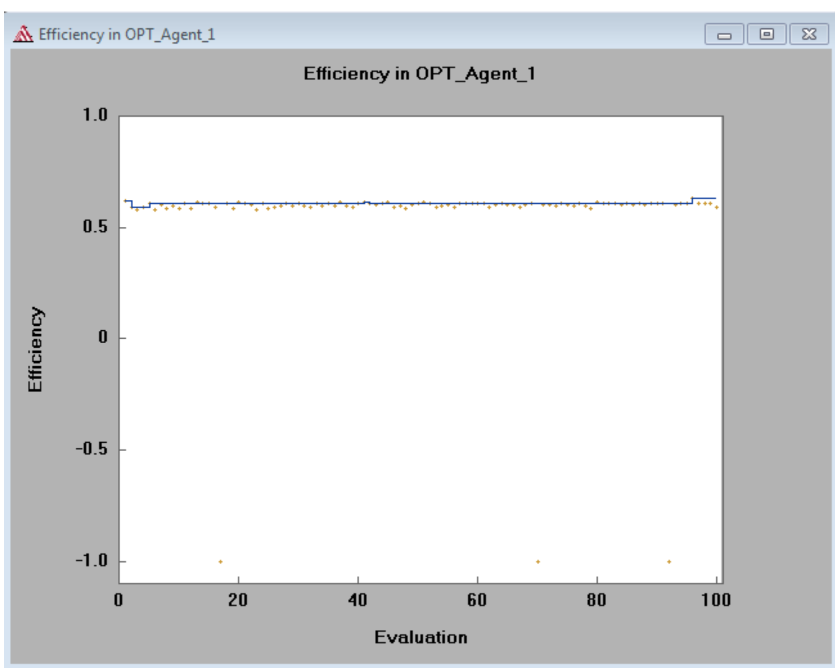


Figure 48: Efficiency Objective Process

Appendix D (Variables Evaluations and Constraints)

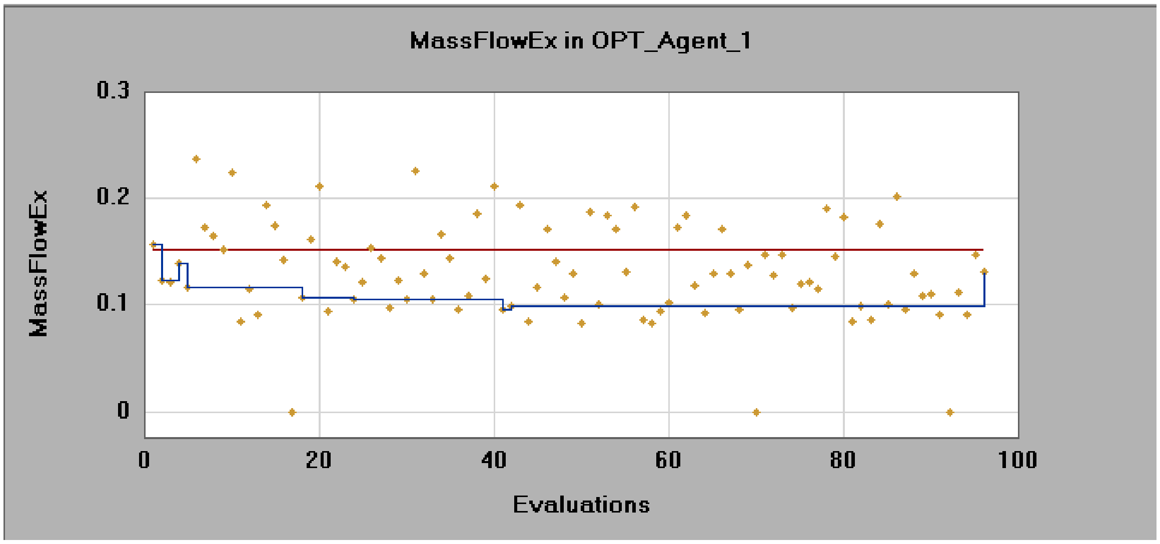


Figure 49: MassFlow (Head losses) Constraints Process (P_2/P_1)

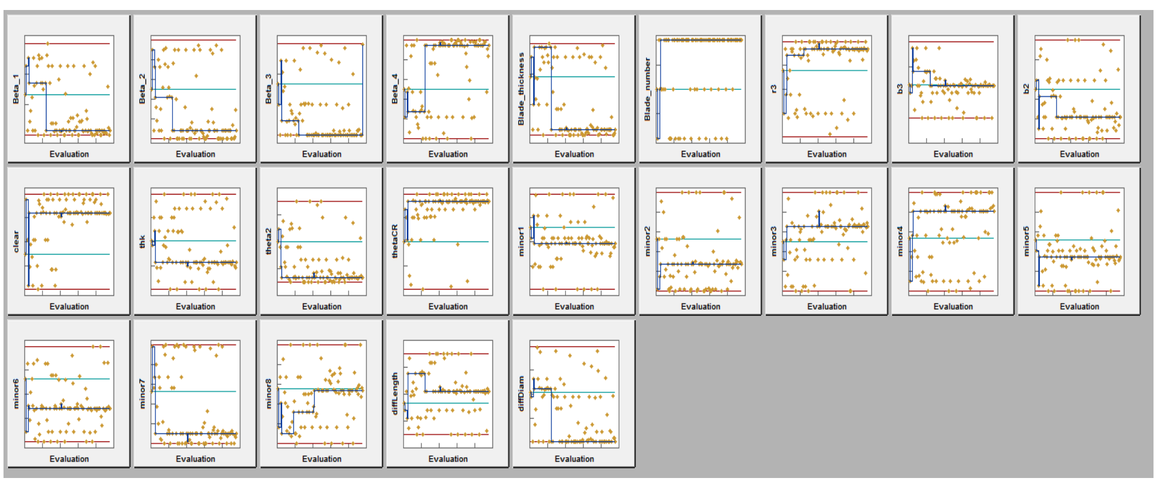


Figure 50: Variables Evaluation

Appendix E (Design Variables)

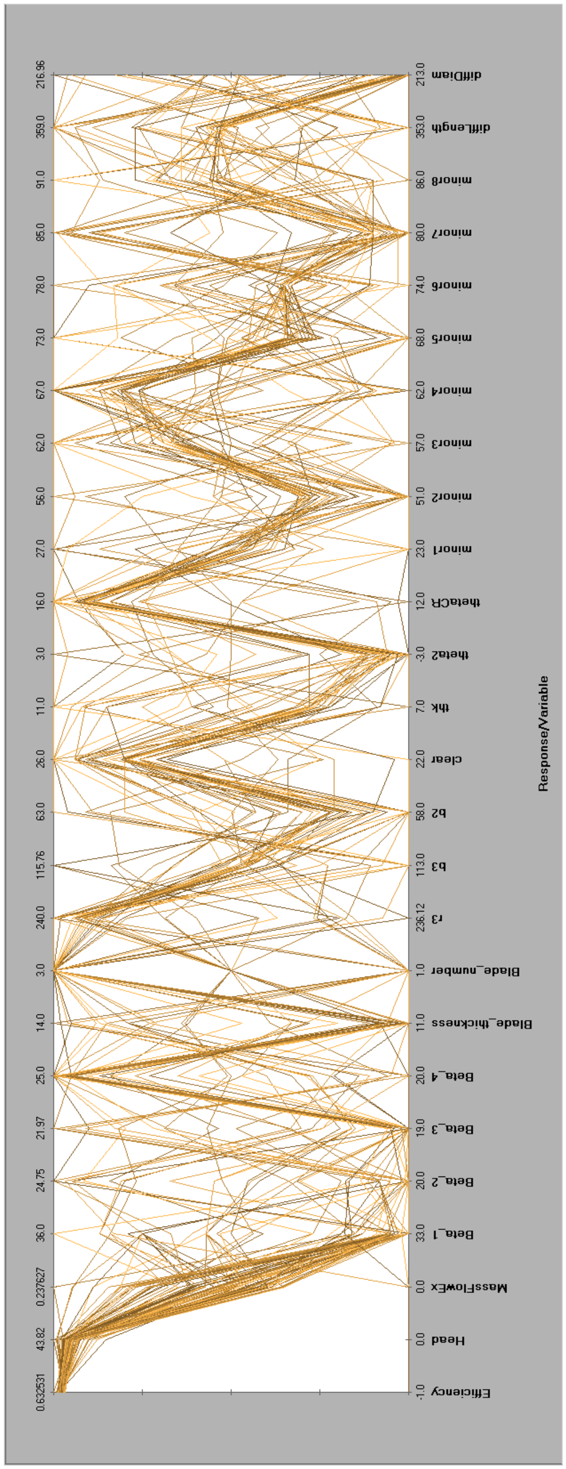


Figure 51: Responses of Design Variables

This page is intentionally left blank.