# Performance & Cavitation Characterization of Mixed Flow Centrifugal Pump using CFD Simulation

Gaurav Chakraborty
P.G. Scholar, Department of Mechanical Engineering
C.C.E.T
Bhilai, India

R.K. Rathore
Asst. Prof., Department of Mechanical Engineering
C.C.E.T
Bhilai, India

***Abstract*—Centrifugal Pumps are the most common appliances used in various industries, agriculture and domestic application& thus its impeller design thus required a very precise understanding of the internal flow at rated and part load operating conditions. For the cost effective design of pump, it is thus very essential to predict its performance in advance before manufacturing them, which requires understanding of flow behavior in different parts of pump & problems arises due to cavitation. In the present work, firstly performance analysis of centrifugal impeller pump is being carried out by performing analysis on its pressure distribution, velocity distribution from which power and efficiency is being calculated and validated. On the later part, analysis is carried out to perform cavitation2 analysis at various mass flow rate at different rated speed of the pump. Also, the threshold value of mass flow rate is being identified from where a cavitation phenomenon is observed at respective rated speed in order to reduce it.**

***Keywords— Centrifugal Pump [1], Cavitation [2]***

## I. Introduction

Pump is a mechanical device mostly used for raising liquids from a lower level to a higher one. This is done by creating a low pressure at the inlet and high pressure at the outlet of the pump. However, initially work has to be done by a prime mover to enable it to impart mechanical energy to the liquid which ultimately converts into pressure energy. It is mostly in used in industries and residential applications. A centrifugal pump is a non-positive displacement pump that imparts energy to a liquid. Centrifugal pumps are the machines, which utilizes centrifugal force in order to lift fluid from a lower level to a higher level by developing pressure. The centrifugal pump moves liquid by rotating one or more impellers inside a volute casing. The liquid is introduced through the casing inlet from the eye of the impeller where it is picked up by the impeller vanes. In other words, the fluid turbo machinery essentially consists of an impeller rotating in a casing. Fluid enters from the eye of the impeller (at the center of the impellers) and exits though the space between the impeller blades to the space between the impeller and casing walls. The velocity of fluid elements is in both tangential and radial directions, as the impeller start rotating. The velocity as well as the pressure, both increases, relatively, as the fluid flows from the impeller.

### A. *Working Principle*

A centrifugal pump consists of a set of rotating vanes enclosed within a housing or casing that is utilized to impart energy to a fluid through centrifugal force. The vanes are usually slope backwards, away from the direction of rotation. The blades of the rotating impeller transfer energy to the fluid & thus increase velocity. The fluid is sucked into the impeller through impeller eye and flows through the impeller channels formed by the curved blades between the shroud and hub. The fluid is accelerated by pulse transmission while following through the curvature of the impeller vanes from the impeller Centre (eye) outwards. It reaches its maximum velocity at impeller's outer diameter and leaves the impeller into a diffuser or volute chamber.

## II. Literature Review

Centrifugal pumps are mostly used in many industrial, agricultural and household applications, so the pump system may be required to operate over a wide flow range in different applications. The most previous numerical studies were focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps, where the performance of pump deteriorates [4]. With the aid of the CFD approach, the complex internal flows through the different components of pump can be studied at different operating conditions which help in improvement in the performance at off design conditions.

*Mentzoset al.* [1] carried out a numerical simulation of the internal flow in a backward curve vanned centrifugal pump. The MRF approach used to take into account the impeller-volute interaction was completely failed, due to its fixed coupling formulation. However, it is recommended for basic understanding of the flow at various operating points. The transient analysis was suggested as a real tool for understanding of the interaction between impeller and spiral casing.

*Bacharoudis et al.* [2] in 2008 in his research stated various parameters affect the pump performance and energy consumption. In this study, the performance of impellers with the same outlet diameter having different outlet blade angles is thoroughly evaluated. For each impeller, the flow pattern and the pressure distribution in the blade passages are calculated and finally the head-capacity curves are compared with the theoretical one. When the pump operates at off-design conditions, the percentage raise of the head curve, due to the increment of the outlet blade angle, is larger for high flow rates and becomes smaller for flow rates $Q/QN<0.65$. When pump operates at nominal capacity, the gain in the head is more than

6% when the outlet blade angle increases from 20°to 50 °. At a very high flow rate, the increase of the outlet blade angle results in a significant improvement of the hydraulic efficiency of outlet blade angle on the performance of pump.

*Medvitz et al.* [1] in his findings used multi-phase CFD method to analyze centrifugal pump performance under cavitating conditions. The homogeneous two phase RANS equations were used and places where mixture momentum and volume continuity equations were used, they are solved along with vapor volume fraction. Performance trends of partial discharge and blade cavitation, including breakdown, were precisely analyzed and compared qualitatively with experimental measurements.

## III. Problem Identification

Head, power, efficiency and cavitation of the centrifugal pump are considered as the main area of this paper. This maximum head is mainly determined by the outside diameter of the centrifugal pumps impeller and the speed of the rotating shaft. The shaft power of a pump is nothing but the mechanical power transmitted to it by the motor shaft, while fluid power is the energy per second carried in the fluid in the form of pressure and kinetic energy. The efficiency is the ratio of output power to input power. Cavitation is the hydraulic phenomena of formation and collapse or implosion of vapor bubbles in a pump. It occurs when the vapor pressure is higher than the suction pressure. If the vapor pressure is on higher side bubbles collapse with high energy, and can remove metal from the internal casing wall, and leave indent marks. Due to increase in internal recirculation of the pump, fluid will gets heated and its vapour pressure increases. In this condition, if the vapour pressure exceeds the suction static pressure, it will result into cavitation. On the other hand, increase in mass flow rate will increase the pressure loss in the suction line and this will result in decrease in suction pressure. In this condition, if suction pressure fall below the vapor pressure of fluid, pump will result into cavitation. So there must be a threshold value of mass flow rate is need to be identified for safe operating condition of the pump, and to avoid cavitation.

## IV. Objectives

The purpose of the present study is to analyze the performance of pump's various parameters like pressure, head, velocity flow distribution, cavitation and internal bleeding and leakage; rotational periodicity of the impeller model.

The objective of this research is "to investigate the effect of cavitation on the centrifugal pump model at various mass flow rate and different speed and to find out the threshold value of the mass flow rate from where the cavitation phenomena will gets started."

## V. Research Methodology

A 3D CAD model of industry specific design specification have been taken up from reference paper [1], [2]. The modeling is done in Solidworks®2016 Software and design specification for impeller are as follows.

TABLE I. Design Specification for Impeller

| Parameters | Values |
|---|---|
| Inlet Blade Angle | 16 ° |
| Outlet Blade Angle | 27 ° |
| Blade Height | 47 mm |

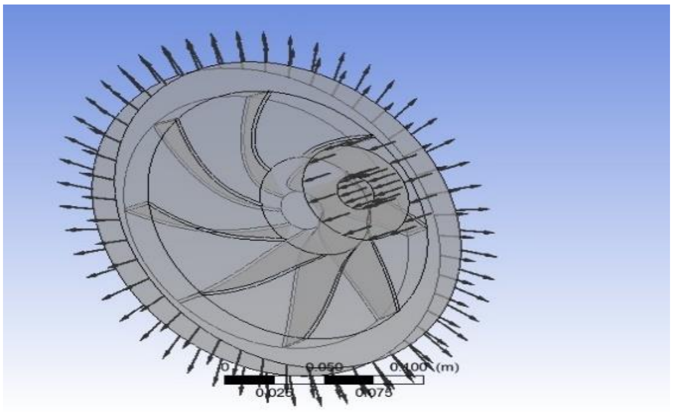


Fig. 1. 3D- CAD Model of Impeller.

### A. Assumptions

Following Assumptions were made for analysis.

- Steady state condition is observed.
- Constant flow Properties is assumed.
- Flow is considered as Incompressible Flow
- The walls were assumed to be smooth hence any disturbances in flow due to surface roughness were not taken in consideration.
- Rotating faces of impeller were taken as wall and no slip condition is assumed i.e. smooth wall is assumed.
- Water is considered as working fluid.
- Standard K-Ɛ model is used for turbulence modeling with standard wall functions. The standard K-Ɛ model is a semi –empirical model usually used to solve 2D Problems based on model transport equations for the turbulence Kinetic Energy (K) and its dissipation rate (Ɛ).

### B. Meshing

Meshing of the impeller is done in Ansys Workbench 15.0®. The obtained result from meshing is shown in table below.

TABLE II. Meshing Detail for Impeller

| Mesh Parameters | Values |
|---|---|
| Number of nodes | 54318 |
| Number of Elements | 266729 |

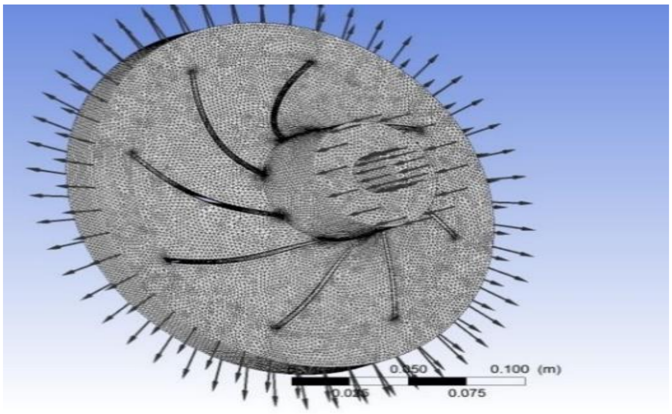


Fig. 2. Meshed Model of Impeller.

### C. *General Boundary Conditions*

- The pump inlet was defined as total pressure boundary conditions and mass flow rate was given at the pump outlet.
- Rotating faces of impeller were considered as wall and no slip condition is applied i.e. smooth wall is assumed.
- Water is taken as working fluid.
- Standard K- ἐ model is used for turbulence modeling with standard wall functions. The standard K-ἐ model is a semi –empirical model based on model transport equations for the turbulence Kinetic Energy (K) and its dissipation rate (ἐ).
- Default value for number of iteration to be performed for the simulation of centrifugal pump analysis in software is set at 10000.

### D. *Boundary Condition for Solving Cavtitation Problem*

TABLE III. BOUNDARY CONDITION FOR CAVITATION

| Parameters | Values |
|---|---|
| Rotational Speed (In rpm) | 7000,7700,8100,8400 |
| Mass Flow Rate (In lpm) | 40,60,80,120 |

## VI. ANALYSIS

### A. *Velocity Flow Distribution Plot at Speed (a)7000,(b)7700;(c)8100;(d)8400 rpm*

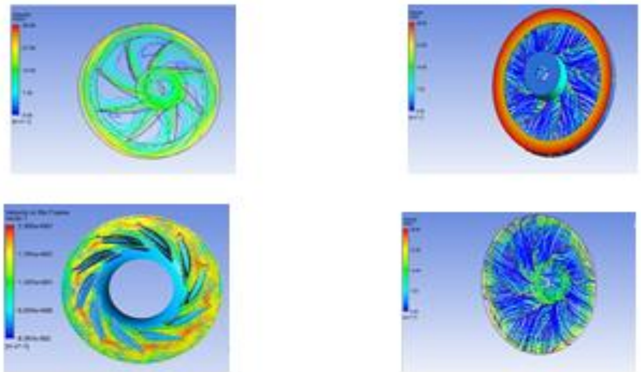

Fig. 3. Velocity Contour Plot at rated speeds.

### B. *Pressure Distribution Plot at Speed (a)7000,(b)7700;(c)8100;(d)8400 rpm*

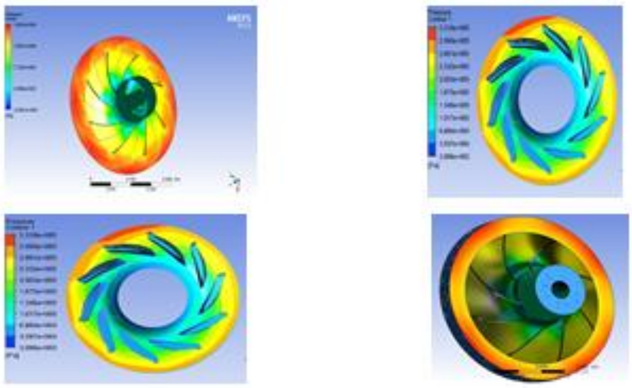

Fig. 4. Velocity Distribution Plot at rated speeds.

### C. *Cavitation Analysis at Speed 7000rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 rpm*

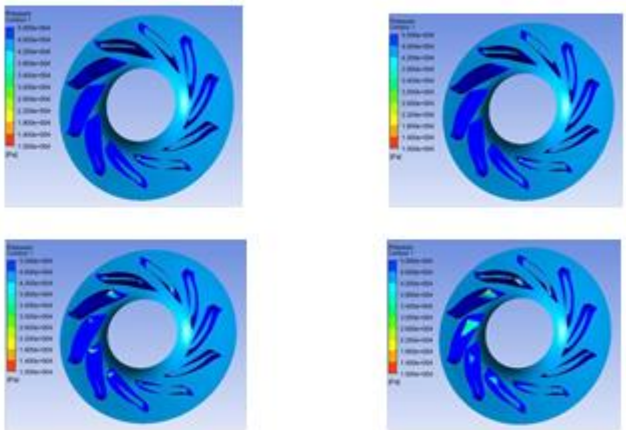

Fig. 5. Cavitation Analysis at Speed 7000rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 lpm.

Cavitation results in loss of performance and degradation of life in the centrifugal pumps. Hence, the analysis of cavitation is a very important aspect of any centrifugal pump. Following points have been concluded after performing cavitation analysis.

- In Fig 5, shows the cavitation analysis of an impeller model at constant 7000rpm with the mass flow rate of (a) 40, (b) 80, (c) 120 and (d)150 lpm. We observed cavitation free operation at 40, 80 and 120 lpm. The formation of cavitation on the blades can be seen after 150 lpm running at 7000rpm.To investigate more precise value of mass flow rate from where cavitation phenomena starts we will do successive iterations in order to get threshold value of mass flow rate from where cavitation phenomena will occur at speed 7000rpm.

TABLE IV. CAVITATION ANALYSIS AT 7000RPM WITH ITERATED VALUE OF MASS FLOW RATE

| Mas Flow Rate (In lpm) | Comments |
|---|---|
| 150 | Cavitation Not Observed |
| 151 | Cavitation Not Observed |
| 151.50 | Cavitation Observed |
| 152 | Cavitation Observed |

### D. *Cavitation Analysis at Speed 7700rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 rpm*

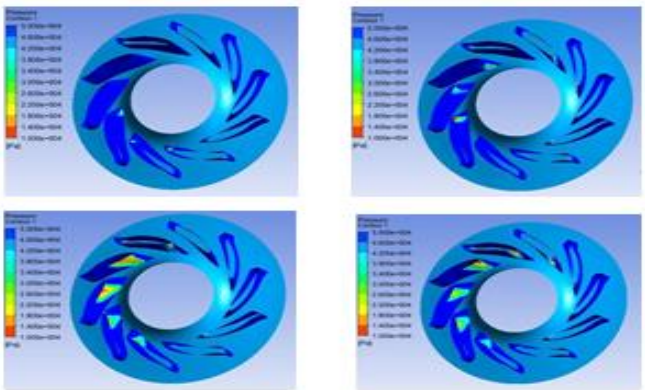

Fig. 6. Cavitation Analysis at Speed 7700rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 lpm.

In Fig 6, the cavitation analysis of an impeller model at constant speed 7700rpm with the mass flow rate of (a) 40, (b) 80, (c) 120 and (d)150 lpm shown . We observed cavitation free operation at 40, 80 & 120 lpm. The formation of cavitation on the blades can be seen after 120 lpm running at 7700rpm. To investigate more precise value of mass flow rate from where cavitation phenomena starts we will do successive iterations in order to get threshold value of mass flow rate from where cavitation phenomena will occur at speed 7700rpm.

TABLE V. CAVITATION ANALYSIS AT 7700RPM WITH ITERATED VALUE OF MASS FLOW RATE

| Mas Flow Rate (In lpm) | Comments |
|---|---|
| 120 | Cavitation Not Observed |
| 124 | Cavitation Not Observed |
| 124.50 | Cavitation Observed |
| 125 | Cavitation Observed |

### E. *Cavitation Analysis at Speed 8100rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 rpm*

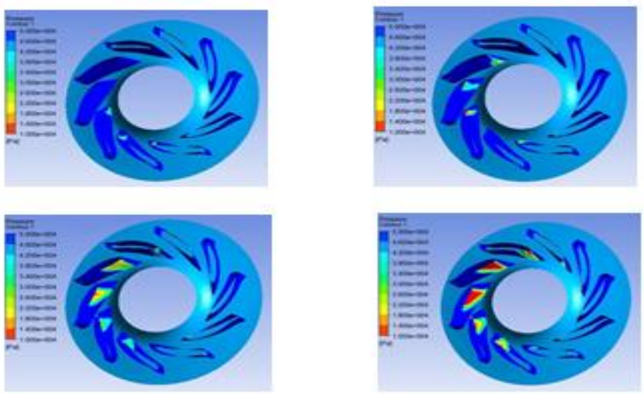

Fig. 7. Cavitation Analysis at Speed 8100rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 lpm.

In Fig 7, the cavitation analysis of an impeller model at constant speed 8100rpm with the mass flow rate of (a) 40, (b) 80, (c) 120 and (d)150 lpm shown . We observed cavitation free operation at 40, 80 lpm. The formation of cavitation on the blades can be seen after 80 lpm running at 8100rpm. To investigate more precise value of mass flow rate from where cavitation phenomena starts we will do successive iterations in order to get threshold value of mass flow rate from where cavitation phenomena will occur at speed 8100rpm.

TABLE VI. CAVITATION ANALYSIS AT 8100RPM WITH ITERATED VALUE OF MASS FLOW RATE

| Mas Flow Rate (In lpm) | Comments |
|---|---|
| 80 | Cavitation Not Observed |
| 81 | Cavitation Not Observed |
| 81.40 | Cavitation Observed |
| 81.50 | Cavitation Observed |

### F. *Cavitation Analysis at Speed 8400rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 rpm*

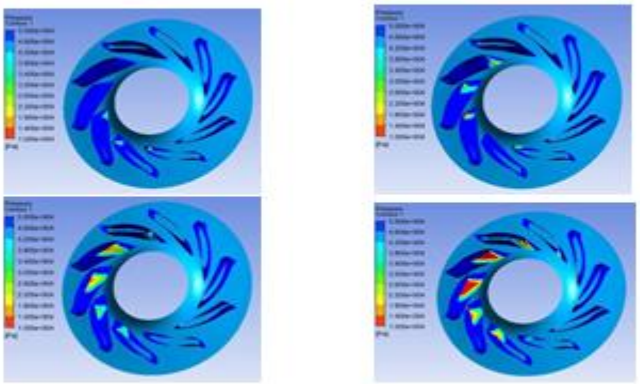

Fig. 8. Cavitation Analysis at Speed 8400rpm & mass flow rate (a)40,(b)80;(c)120;(d)150 lpm.

In Fig 8, the cavitation analysis of an impeller model at constant speed 8100rpm with the mass flow rate of (a) 40, (b) 80, (c) 120 and (d)150 lpm shown . We observed cavitation free operation at 40 lpm. The formation of cavitation on the blades can be seen after 40 lpm running at 8400rpm. To investigate more precise value of mass flow rate from where cavitation phenomena starts we will do successive iterations in order to get threshold value of mass flow rate from where cavitation phenomena will occur at speed 8400rpm.

TABLE VII. CAVITATION ANALYSIS AT 8400RPM WITH ITERATED VALUE OF MASS FLOW RATE

| Mas Flow Rate (In lpm) | Comments |
|---|---|
| 40 | Cavitation Not Observed |
| 40.50 | Cavitation Not Observed |
| 41 | Cavitation Observed |
| 41.50 | Cavitation Observed |

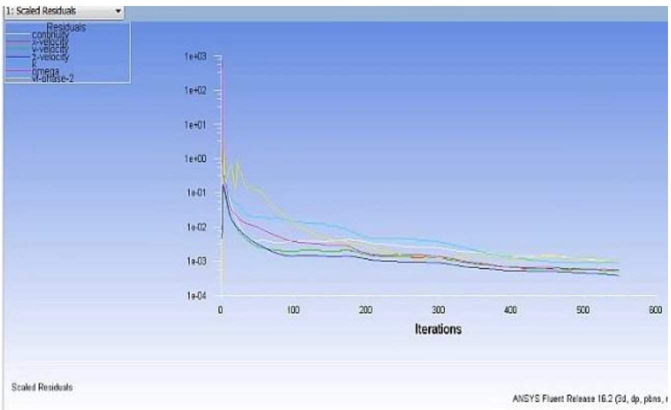


Fig. 9. Convergence Report Obtained after successive Iterations.

## VII. RESULTS AND DISCUSSIONS

After performing successive iterations at various rated speed with different mass flow rate, the entire result can be summarized as below.

TABLE VIII. SUMMARIZED RESULT OF THRESHOLD MASS FLOW RATE AT DIFFERENT RPM

| Speed (In lpm) | Comments |
|---|---|
| 8400 | 41 |
| 8100 | 81.40 |
| 7700 | 124.50 |
| 7000 | 151.50 |

## CONCLUSION

- CFD analysis was carried out at design and off design condition and results obtained are satisfactory. The Simulation was performed by using turbulent modeling tool standard k-Epsilon. Performance charts, cavitation analysis, pressure contours and velocity vector contour are obtained after analysis.
- The mesh is generated successfully using ANSYS-CFX. The performance results are satisfactorily matching with test data, hence mesh quality is good.
- After analysis the performance results shows that total static head is the function of the mass flow rate with constant operating speed.
- From analysis it was observed that the formation of cavitation on the blade is increasing with the increase of mass flow rate and rotating speed.
- Additionally, from Table VIII; for every rated speed threshold mass flow rate is being identified which will help the designers to design the pump to run in a safe operating condition and to avoid cavitation.

## REFERENCES

[1] S.Rajendran and Dr.K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX," International Journal of Engineering Research & Technology, Vol. 1, Issue 3,pp.-67-71 2012.

[2] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, "Parametric Study of a Centrifugal Pump Impeller by Varying th e Outlet Blade Angle," The Open Mechanical Engineering Journal, Vol. 2, pp.-75-83, 2008.

[3] Mohammed Khudhair Abbas, "Cavitation in centrifugal pumps," Diyala Journal of Engineering Sciences,Vol.1, pp. 170-180, 2010.

[4] Erik Dick, Jan Vierendeels, Sven Serbruyns and John Vande Voorde, "Performance prediction of centrifugal pumps with cfd -tools," Task Quarterly 5, Vol. 4, pp.-579–594, 2001.

[5] Liu Houlin, Wang Yong, Yuan Shouqi, Tan Minggao and Wang Kai, "Effects of Blade Number on Characteristics of Centrifugal Pumps,"Chinese journal of mechanical engineering, Vol. 23,pp.-22-28; 2010.

[6] Manivannan.A "Computational Fluid Dynamics of a Mixed Flow Pump Impeller", International Journal of Engineering, Science and Technology, Vol. 3, pp.- 200-208;2010.

[7] S R Shah, S V Jain and V J Lakhera, "CFD based flow analysis of centrifugal pump," Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai,pp.201-209; 2010.

[8] P.Usha Shri ans C.Syamsundar, "Computational analysis on performance of a centrifugal pump impeller," Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, pp.-167-169; 2010.

[9] Marco Antonio Rodrigues Cunh and Helcio Francisco Villa Nova, "Cavitation modelling of a centrifugal pump impeller," 22nd International Conference of Mechanical Engineering, Ribeirao Petro, Sao Paulo, Brazil, pp.-157-159;2013.

[10] Myung Jin Kim, Hyun Bae Jin, and Wui Jun Chung, "A Study on Prediction of Cavitation for Centrifugal Pump," World Academy of Science, Engineering and Technology, Ph.D. Thesis, University of Toronto, 1999.

[11] Christian Allerstorfer, "Bachelor Thesis on Centrifugal Pumps", Department of design and Conveying Technology, MU Leoben.

[12] Amit H. Bhuptani, Prof. Ravi K. Patel, K.M. Bhuptani, "Design and analysis of centriugal pump", Department Of Mechanical Engineering, Ph.D. Thesis; U.V. Patel College of Engineering And Technology, Kherva, Gujarat.

[13] Bureau of Energy Efficiency, "Pumps and Pumping System", Pp. - 17-103; First Edition 2002.

[14] Computer aided manufacturing technologies for centrifugal compressor impellers." Journal of Materials Processing Technology ;Pp-701-753; Second Edition, 2002.