

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

Computational Fluid Dynamic Analysis in De-staging of Centrifugal Pumps

Vishnu R Nair¹, Shinas K V², Souganth Sugathan Manjhiparambil³

Student, Department of Mechanical Engineering, IES College of Engineering, Thrissur, Kerala, India¹

Student, Department of Mechanical Engineering, IES College of Engineering, Thrissur, Kerala, India²

Student, Department of Mechanical Engineering, IES College of Engineering, Thrissur, Kerala, India³

ABSTRACT: Energy requirement increases day by day. Thermal power plants are the measure sources of energy for electricity generation. They are also the measure polluter of the environment. In order to reduce environmental pollution either energy consumption should be reduced or energy should be generated with higher conversion efficiency. In thermal power plants part of energy generated by the plants is being consumed by different auxiliaries. The power consumption by these auxiliaries is very high due to poor operation or bad design of the equipment's. There are two different ways to improve the efficiency either by efficient operation or by facility conversion. The pumps and compressors are the main auxiliaries which consumes sizable power produced by thermal power plants. Here an attempt is made to improve the efficiency of above mention auxiliaries. To improve the performance of the pumps the method of De-staging has been adopted and in-depth study has been done through simulation. Before modifications actual data are collected for pumps. Power and efficiency are calculated and same are compared with ideal values reported in supplier manuals. On the basis of discrepancies in above data, methods for performance improvements are suggested.

KEYWORDS: Centrifugal Pump, CFD, Impeller Design, Ansys Fluent, Solidworks.

I. INTRODUCTION

Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is an important tool for pump designers. Such a CFD tool that is most commonly used, ANSYS Fluent, is being used here. Many tasks can numerically be solved much faster and cheaper than by means of experiments. CFD analysis is very useful for predicting pump performance at various mass-flow rates.

Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analysing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world.

Pump de-staging refers to the removal of one or more impeller from the multistage pump to reduce the energy added to the system fluid. Pump de-staging offers a useful correction to pumps that, through overly conservative design practices or changes in system loads are oversized for their application. Pump de-staging allows the performance curves to be moved upward, achieving roughly the same effects as modification of the diameter. Pump de-staging reduces no of stages, which in turn directly lowers the amount of energy imparted to the system fluid and lowers both the flow and pressure generated by the pump in stages.

A centrifugal pump converts the input power to kinetic energy in the liquid by accelerating the liquid by an impeller. The energy created by the pump is kinetic energy according the Bernoulli Equation. Fluid enters the pump through the eye of the impeller which rotates at high speed. The fluid is accelerated radially outward from the pump casing. A vacuum is created at the impellers eye that continuously draws more fluid into the pump. The pump consists of an

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

impeller rotating inside a pump casing. Water enters at the centre, or eye, of the impeller and is flung out to the periphery (the outer edge of the casing) by centrifugal force as the impeller rotates. The water is collected inside the pump casing and discharged from the pump outlet.

The purpose of the present study is to show a numerical study of a multistage centrifugal pump taking into account the whole 3-D geometry and the unsteadiness of the flow. It has been done with the commercial software package ANSYS Fluent which uses the finite volume method and solves the k- ϵ equation with ability to handle unstructured grids, include relative reference frames and make unsteady calculations with moving meshes.

The study here is carried out on a 146KW 5 stage centrifugal pump that is used in a combined cycle power plant having a power generation capacity of 168MW.

II. CASE STUDY

The pump used in the 168MW combined cycle power plant is providing a discharge of 15.3 kg/cm² with a power rating of 146KW while the required discharge pressure of the pump is 7kg/cm². The aim of this paper is to validate the method of De-staging in bringing down the pressure to the pressure required by the plant for efficient working.

The reduction in pressure will in turn reduce the motor working and the power consumed by it. There by increasing the efficiency of the combined cycle power plant.

The aim is to create a 3D model of the 5 stage multistage pump using three-dimensional modelling software, i.e., Solidworks and study the incompressible flow through the 5 stage centrifugal pump by solving it with a moving frame of reference with constant rotational speed. A set of 3-D incompressible Navier-Stokes equations with the rotational force terms are solved to analyse the flow in centrifugal pump. Turbulence is modelled with k- ϵ turbulence model.

III. MODEL CREATION

The three-dimensional model of the multistage centrifugal pump was constructed using Dassault Systèmes Solidworks. Solidworks is a solid modeller, and utilizes a parametric feature-based approach to create models and assemblies.

Impeller: The impeller is measured using digital vernier gauge and comparing with CAD drawings the shroud as shown is drawn in the 3D window and revolved and a series of 8 vanes are extruded from the surface.

Table 1: Design Specification of Impeller

Outer diameter of Impeller	350mm
Inlet diameter of Impeller	180 mm
Inlet blade angle	16°
Outlet blade angle	32.25°
Thickness of blade	5mm
Number of blades	8
Shaft diameter	62.7mm
Key	12 by 6mm

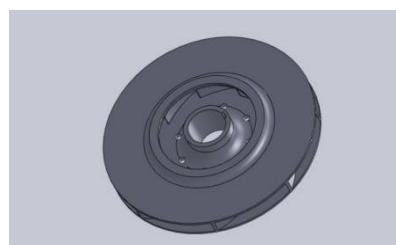


Fig. 1 Solidworks Model of Impeller

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

Diffuser: The diffuser is generated by extruding a series of 12 vanes are Boss-Extruded from the surface. The diameter of the diffuser is 624mm, blade height is 23.6mm and the central bore is 200mm in diameter.

Table 2: Design Specification of Diffuser

Outer diameter of diffuser	624
Inlet diameter of diffuser	180 mm
Inlet blade angle	16°
Outlet blade angle	32.25°
Thickness of blade	5mm
Number of blades	12

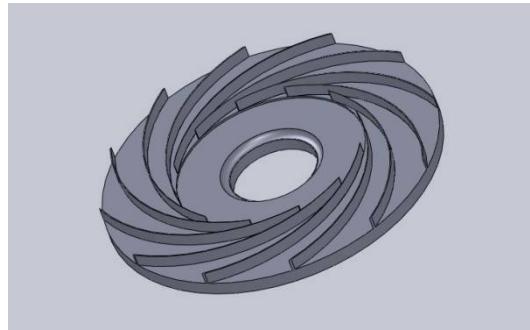


Fig. 2 Solidworks Model of Diffuser

Volute/Casing: The volute was designed by revolving the sketch around an axis. The casing of the pump, as per the design charts has a diameter of 635mm and a length of 788mm. The inlet section has a diameter of 450mm and the outlet section has a diameter of 200mm. The outlet pipe extends from the pump for about 3m.



Fig. 3 Solidworks Model of Volute/Casing

Shaft: The shaft of the multistage centrifugal pump was extruded from a circle having diameter of 62.7mm.

IV. ANALYSIS

The analysis of the 5 stage centrifugal pump was done using Fluent module in ANSYS workbench. ANSYS is a comprehensive suite that spans the entire range of physics, providing access to virtually any field of engineering simulation that a design process requires.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

The incompressible flow through the pump was analysed considering it a pressure based problem as pressure is the phenomena that varies noticeably with the implementation of De-staging. Each stage of the pump impeller was individually analysed for saving time and better accuracy.

The model generated in the Solidworks work phase is imported in to the fluent window the import was then meshed using a body sizing of 0.01 element size.

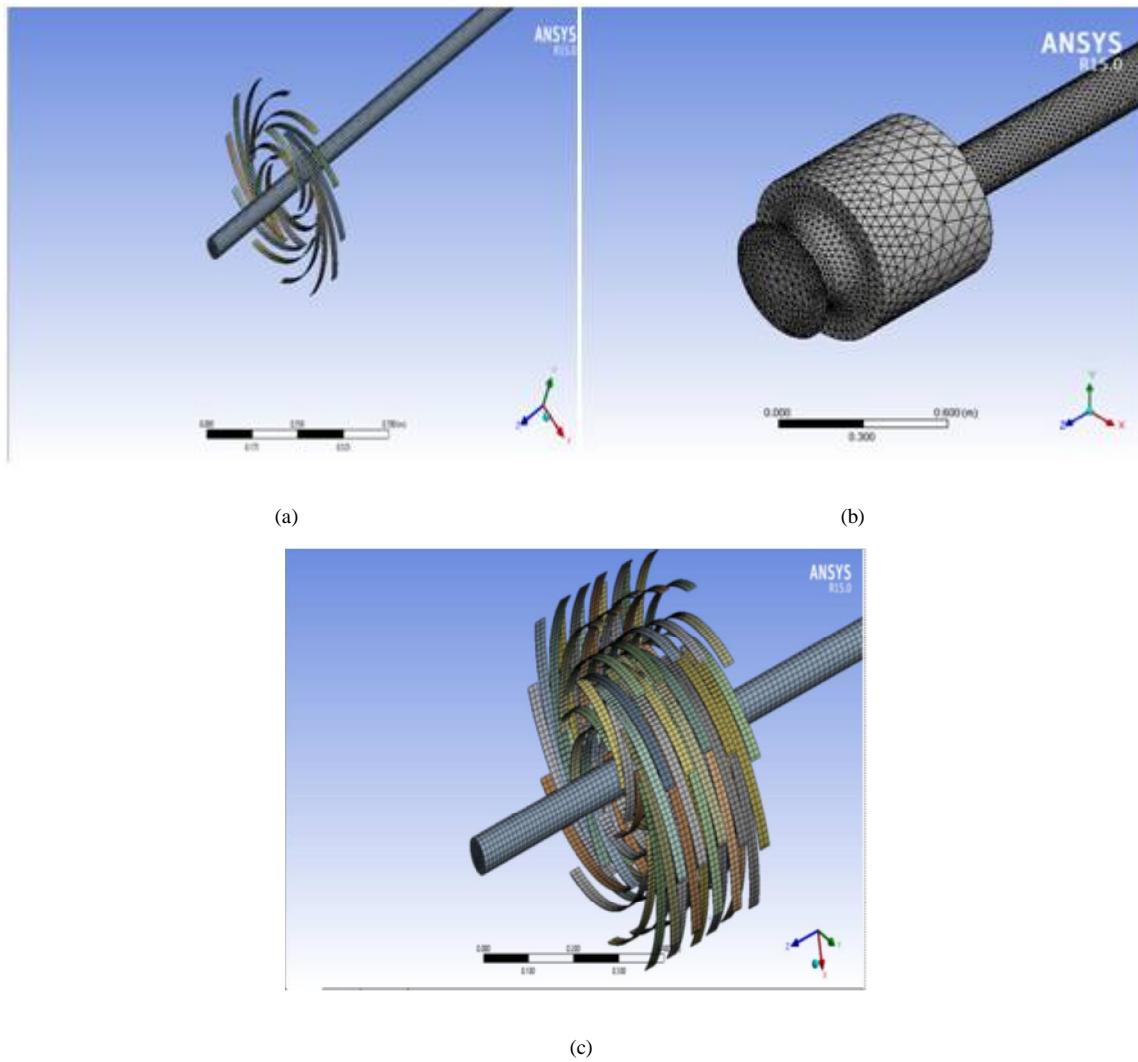


Fig. 4 Meshing (a) single impeller stage (b) volute casing with volumetric meshing (c) meshing of all the five stages

The turbulence model was set to k- ϵ model and the inlet pressure was defined in the inlet face and the outlet was measured. This outlet pressure obtained was then given to the next stage and the process continued till the final stage until the rated pressure was reached for cross-reference.

Boundary conditions: Centrifugal pump impeller, diffuser and the shaft domain is considered as rotating frame of reference with different rotational speed of 1485 RPM. The working fluid through the pump is water. k- ϵ turbulence model is used for calculation. Inlet static pressure of 0.05055 kg/cm^2 for the first stage, 1.23 kg/cm^2 for the second stage, 2.12 kg/cm^2 for the third stage, 6.75 kg/cm^2 for the fourth stage, 11 kg/cm^2 for the fifth stage and the

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

corresponding outlet pressure was calculated with respect to each inlet pressure. Three dimensional incompressible N-S equations are solved with ANSYS-CFX Solver.

V. OBSERVATIONS

The results obtained from the Ansys Fluent analysis is shown in Fig 5. The various contours obtained show the pressure variation along the length of the pump.

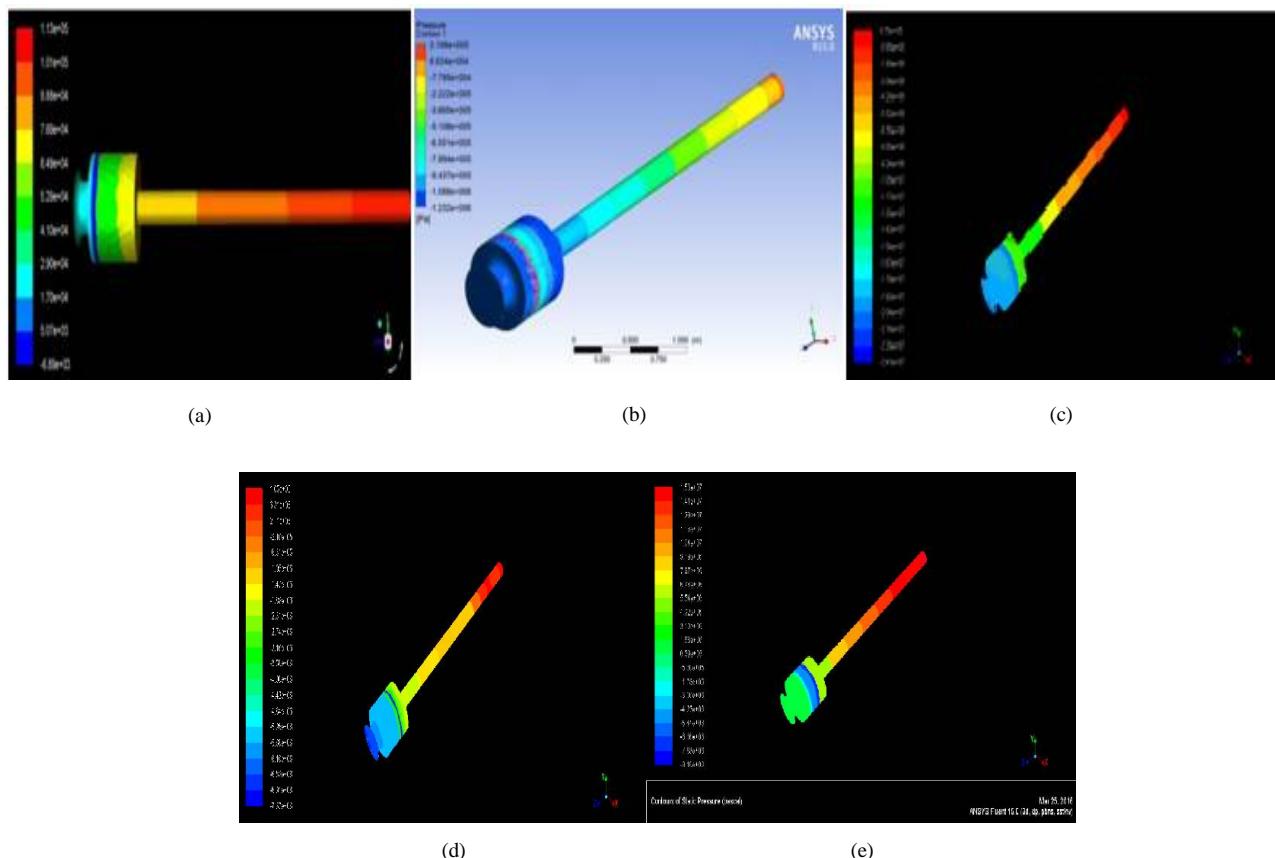


Fig. 5 Observations of pressure contours of different stages (a) First stage single impeller (b) Second stage single impeller (c) Third Stage single impeller (d) Fourth Stage single impeller (e) Fifth Stage single impeller.

Fig 5(a) shows the pressure contour of the first stage impeller. As can be observed from the figure, pressure at outlet is more than pressure at the inlet.

Fig 5(b) gives the pressure contour through the second stage. The inlet condition for this stage is the output of the first stage. Outlet pressure is more than the inlet pressure.

Fig 5(c) is the pressure contour through the third stage. Inlet condition for this stage is taken from that of the output of the first two stages combined. Here also, outlet pressure observed is more than the inlet pressure.

Fig 5(d) shows the pressure contour through the fourth stage impeller. The combined results of the first three stages are given as inlet. We obtain an outlet pressure which is greater than inlet.

Fig 5(e) shows the pressure contour through the fifth stage. The outlet pressure is at maximum in this case.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

The plots obtained from the ANSYS Fluent are depicted in Fig 6.

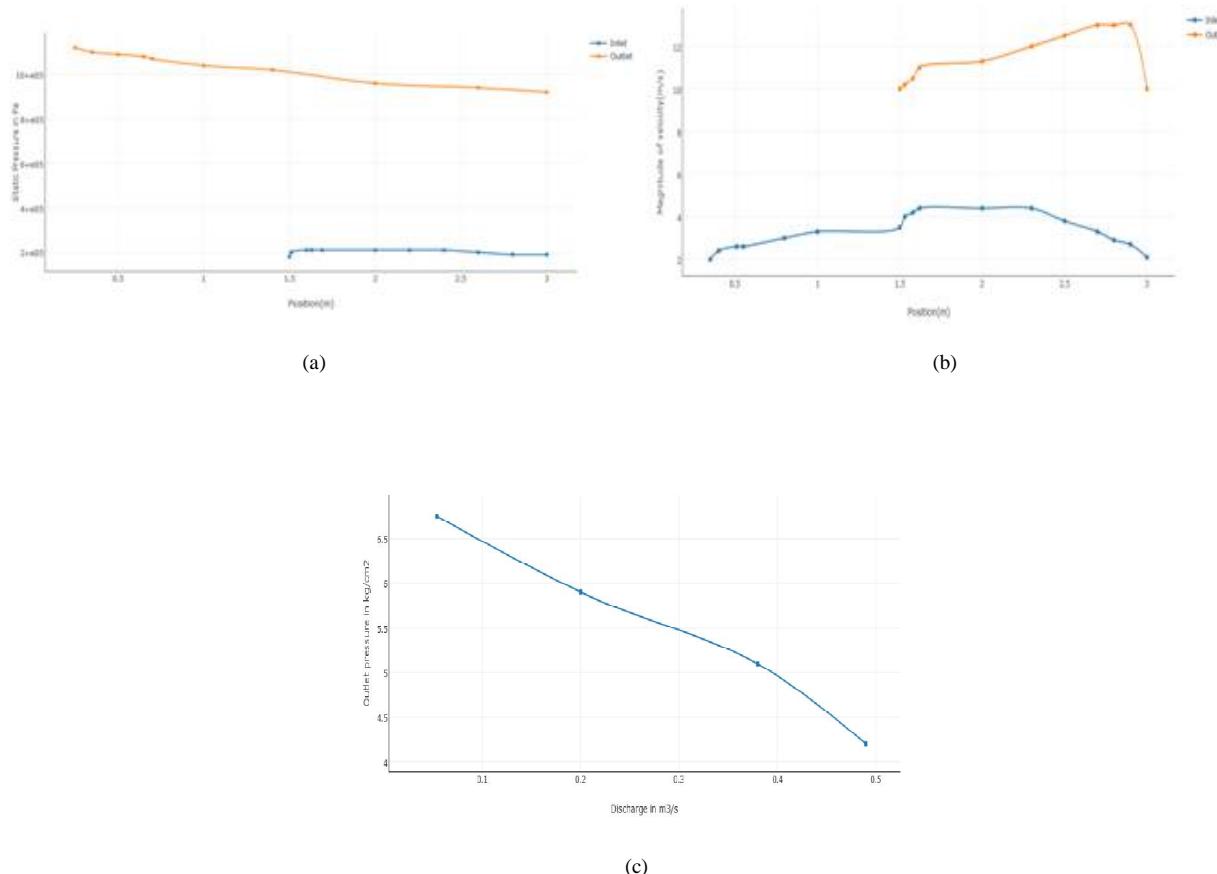


Fig.6 Result and Plots (a) Static Pressure v/s Position (b) Magnitude of velocity v/s Position (c) Pressure v/s Discharge

Fig 6 shows a simplified plot from the data obtained from the results of the numerical model in Ansys fluent. Fig 6(a) shows the distribution of pressure at inlet and outlet. The pressure at outlet is considerably higher than that of the inlet. Fig 6(b) shows the velocity distribution at inlet and outlet. Fig 6(c) shows the pressure v/s discharge curve. This curve follows the general characteristic behaviour of centrifugal pumps in that the pressure decreases as discharge from the pump increases.

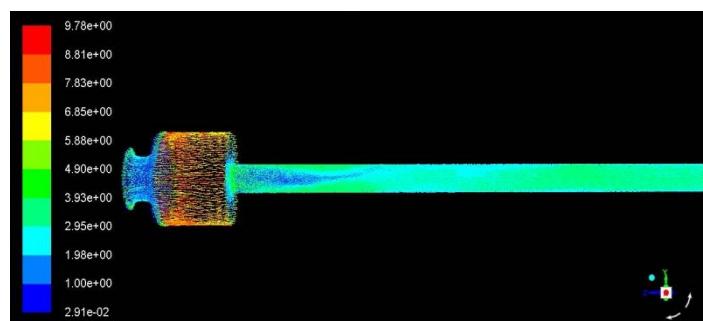


Fig. 7 Velocity Contour of the fluid passing through the pump

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 4, April 2016

The velocity contour of the 5 stage centrifugal pump shows a high increase in velocity near the impeller stages and a decrease in velocity as it moves toward the discharge which indicates an increase in pressure, as shown in Fig 7.

VI. SUGGESTIONS AND IMPROVEMENTS

The Simple Payback Period (SPP) of de-staging is compared with impeller trimming and variable frequency drive in the following table. Impeller trimming involves reducing the impeller diameter by machining its outer edges whereas, variable frequency drives employs a variable speed motor to control the speed of the pump shaft.

Table 3: Simple Payback Period

Sl. NO	Suggested Method	Total Investment	Annual Savings	Simple Payback Period
1	De-staging	Rs 1,59,000	Rs 16,57,000	34 days
2	Impeller Trimming	Rs 1,14,046	Rs 6,70,690	62 days
3	Variable Frequency Drive	Rs 22,58,809	Rs 18,66,240	435 days

From the data in Table 3, it is clear that de-staging has the least simple payback period and will be the most economical for the industry.

VII. CONCLUSION

A Computational investigation of pressure variation within the given Condensate Extraction Pump was conducted successfully. The pressure and velocity contours were obtained which define the distribution of pressure and velocity at different regions within the flow area of the pump. The major conclusions drawn are:

- 1) The simple payback period for De-staging, Impeller trimming and VFD were calculated and found to be 34, 62 and 435 days respectively. Although the VFD is very efficient and smooth to operate, it would be in excess to the needs of the industry. Impeller trimming is irreversible. De-staging, on the other hand, is simple to implement and satisfactorily fulfills the requirements.
- 2) Solidworks models of impeller, diffuser, shaft and volute casing were constructed on the basis of known specifications and calculations.
- 3) The CFD model of the pump was created which consists of impeller, diffuser and the volute casing.
- 4) The pressure levels predicted for different stages have been found to be consistent with data obtained from de-staging conducted on identical pump.
- 5) While doing the CFD analysis of de-staging of the CEP, it was found that the pressure drops to about 6.75 kg/cm^2 , when two impellers are removed. The total head with five impellers were 15.3 kg/cm^2 . This will considerably reduce the power consumed by the CEP. Hence this condition with de-staging of two impeller stages is the most preferable method of optimizing the flow through the CEP.

REFERENCES

- [1] R. Ragot Singh, M. Natraj, "Design and analysis of pump impeller using SWFS", World Journal of Modelling and Simulation, Vol. 10, pp. 152-160, 2014.
- [2] Tejas N. Raval, Dr. R. N. Patel, "Optimization of Auxiliary Power Consumption of Combined Cycle Power Plant", Sciverse Sciedirect, Vol. 51, pp. 751-757, 2013.
- [3] Harisha S, Dr. Y. J. Suresh, "Rotor Dynamics Analysis of a Multistage Centrifugal Pump", International Journal of Innovative Research in Advanced Engineering, Vol.3, Issue 9, pp. 16055-16063, 2014.
- [4] Timothy J. Cookson, Nicholas G. Lang, Edward J. Thornton, "Adjustable Speed Drives Applied To Large AC Induction Motor And Pump Systems", Proceedings Of The Twenty Fourth International Pump Users Symposium, pp. 75-80, 2008.
- [5] Mr. Jekim J. Damodar, Prof. Dilip S. Patel, "Experimental and CFD Analysis of Centrifugal Pump Impeller – A Case Study", International Journal Of Engineering Research And Technology, Vol. 2, Issue 6, 2013.
- [6] Gurvinder Singh, John W Mitchell, "Energy Savings From Pump Impeller Trimming", ASRAE, Vol. 40, pp. 60-65, 1998.
- [7] Gopikrishna Balakrishnan, Rohan Mathew Philip, Aravind Sudev, Ruben Bijy Thomas, "Comparative Research of Methods for Performance Improvement of High Pressure Boiler Feed Pumps in Power Plants", International Journal of Engineering Research & Technology, Vol. 4, pp. 566-570, 2015.