www.ijiarec.com

ISSN:2348-2079

Volume-6 Issue-4

International Journal of Intellectual Advancements and Research in Engineering Computations

Design and analysis of centrifugal pump using computational fluid dynamics

N.Ravi[1], P.Vijayanand[2], Dr.V.Suresh[3]

[1]PG Student, Department of Mechanical Engineering (Machine Design), Holy Mary Institute of Technology and Science, Hyderabad

[2]Associate Professor, Department of Mechanical Engineering, Holy Mary Institute of Technology and Science, Hyderabad

[3]Professor, Department of Mechanical Engineering Holy Mary Institute of Technology and Science, Hyderabad

ABSTRACT

Centrifugal pumps are used to transport liquids/fluids by the conversion of the rotational kinetic energy to the hydro dynamics energy of the liquid flow. The rotational energy typically comes from an engine or electric motor or turbine. In the typical simple case, the fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits. The Contemporary impeller blades in Centrifugal pumps are used in industrial applications are made up of Aluminium or Steel. It is proposed to design a centrifugal pump using Computer Aided Design (CAD) software with various metal alloys and NonMetallic composite materials, analyze its strength and deformation using simulation software. In order to evaluate the effectiveness of Metal Alloys and NonMetallic composites. The present work aim is to change the material and performing the different analysis like Static, Dynamic, Analysis to find the best material to decrease the weight and increase its efficiency by using the software SOLID WORKS (2014 Premium Version). This also involves the method of manufacturing process to realize the Blower using Non-Metallic composite material.

Keywords: *Centrifugal pump, Computer Aided Design (CAD), Metal Alloys, Non-Metallic Composite Materials, SOLIDWORKS, Simulation Analysis.*

INTRODUCTION

The application of fluid machinery has spread its wings in all aspects of human life. The usage of pumps extends to domestic services, commercial and agricultural services, municipal water/wastewater services, and industrial services such as food processing. Pumps are also used in chemical, petrochemical, pharmaceutical and mechanical industries. This chapter deals with the classification of the pumps, and the selection of the pumps based on their applications and their operating principles.

CLASSIFICATION

In general, pumps are classified into Positive Displacement Pumps and Centrifugal Pumps based on their working principles

Positive Displacement Pumps

A positive displacement pump operates by alternately filling a cavity and then displacing a given volume of liquid. The positive displacement pump delivers a constant volume of liquid against varying discharge pressure or head.

Author for correspondence:
PG Student, Department of Mechanical Engineering (Machine Design), Holy Mary Institute of Technology and Science, Hyderabad

Centrifugal Pumps

Centrifugal pumps can be classified based on the manner in which fluid flows through the pump. The manner in which fluid flows through the pump is determined by the design of the pump casing and the impeller. The three types of flow through a centrifugal pump are radial flow, axial flow, and mixed flow

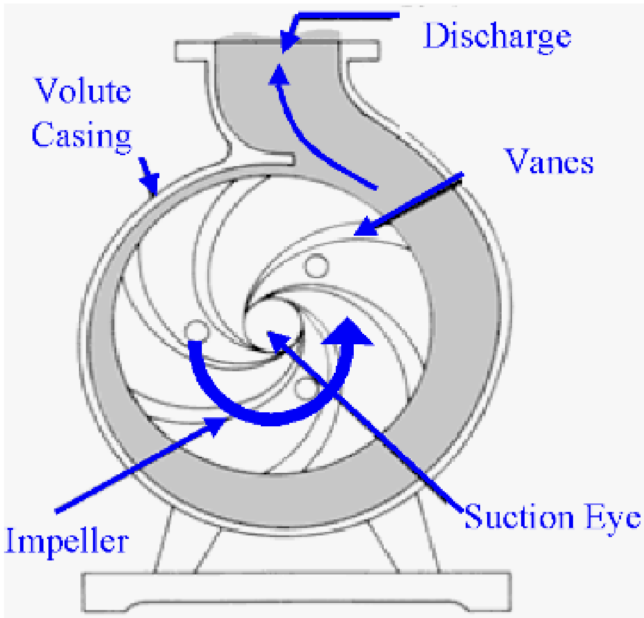


Fig.no 1: Centrifugal Pump

Radial Flow

In a radial flow pump, the liquid enters at the center of the impeller and it is directed out along the impeller blades in the direction at right angles to the shaft of the pump in which the pressure is developed wholly by centrifugal force. When the head requirement is more, radial flow pumps are preferred.

Axial Flow

In an axial flow pump, the impeller pushes the liquid in the direction parallel to the pump shaft in which the pressure is developed by the propelling or lifting action of the vanes of the impeller on the liquid. Axial flow pumps are sometimes called propeller pumps because they operate essentially as the propeller of a boat does. When more flow rate is required, axial flow type pumps are preferred.

Mixed Flow

In a mixed flow pump, the flow direction is mixed. In this the pressure is developed partly by centrifugal force and partly by the lift of the vanes of the impeller on the liquid. These pumps are used for medium head and medium flow applications

LITERATURE REVIEW

Lazarkiewicz Stepen (1965) has explained the procedure to calculate the dimensions of the impeller and has also discussed various methods for determining the shape of the impeller blades, which are referred to by the designers. The effects of the individual parameters that influence the performance of the impeller were studied by the authors Lobanoff (1985) and Stepanoff (1948)

Van Esch (1997) in his thesis has shown that the flow in hydraulic pumps of the radial and mixed flow type, operating at conditions not too far from design point, can be considered as an incompressible potential flow, where the influence of viscosity is restricted to thin boundary layers, wakes and mixing areas. He also concluded that the design of hydraulic turbo machines has reached the stage where improvements can be achieved only through a detailed understanding of the internal flow. The internal flow structure in a centrifugal pump impeller is complex, involving streamline curvature, system rotation, separation and turbulence effects.

Ravi N et al., Inter. J. Int. Adv. & Res. In Engg. Comp., Vol.–06(04) 2018 [2592-2597]

Pedersen et al (2003) has done an exhaustive study to visualize the flow in the centrifugal pump impeller and his research outcome provides a detailed instantaneous data of the impeller flow field in the rotating passage of a centrifugal pump impeller.

Lazarkiewicz Stepen (1965) has explained the procedure to calculate the dimensions of the impeller and has also discussed various methods for determining the shape of the impeller blades, which are referred to by the designers. The effects of the individual parameters that influence the performance of the impeller were studied by the authors Lobanoff (1985) and Stepanoff (1948).

METHODOLOGY

Calculate pump specific speed

N_s □ $\frac{1450* \sqrt{1000/3600}}{40^{3/4}}$ □□□□□□□□□□□□□□
48.07460RPM

Calculate the input and output power

P_o □□1000* 9.806* 40 * 0.2778 □109kW

P_s □109/ 0.86 □126.7kW say 127kW

Calculate input torque to pump

T=127000*60/2*3.14*1450 =837 N.m

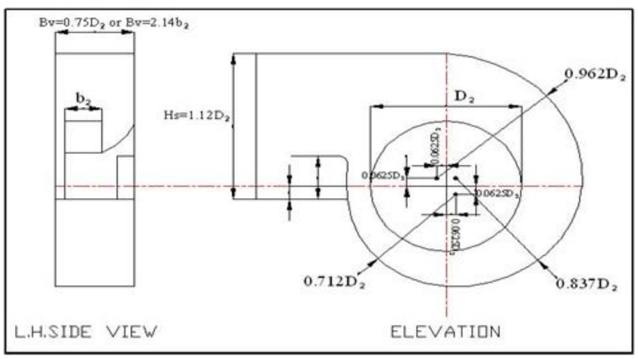
Calculate the shaft diameter

T □

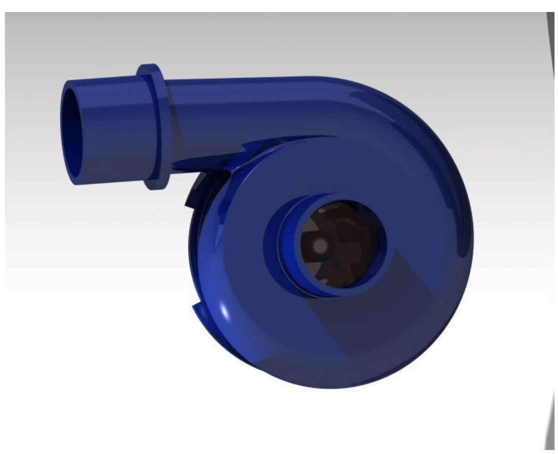
=8343$kg.cm$

d □ 16 * 818.16 □ 47.05mm say 50mm

□□□□□□□□□□ 40000000



2d drawing of centrifugal pimp



Centrifugal pump SOLIDWORKS CAD MODEL

Materials

Aluminium alloy 6061

Density	2.7 g/cm³
Tensile strength	124–290 MPa
Melting Temperature	585 °C
Thermal conductivity	151–202 W/(m·K)

E-Glass / Epoxy

Density	1.90 g/cm³
Tensile strength	490 MPa
Melting Temperature	177 °C
Thermal conductivity	0.15-0.25 W/(m·K)

Testing the performance of a pump using conventional and experimental method is time consuming and costly. It takes a long time to approve a design, if we make multiple proto type and then test each of them. Instead, simulation is used to find the performance of an impeller and casing using Computational Fluid Dynamics software. Once the flow pattern and performance are satisfactory, then we can go for one or two types to validate the simulation results.

Initially it deals with the modelling approach of the pump. The simulation is carried out for different speeds with varying operating conditions. Then the simulation results are compared with the experimental results for the same operating conditions to confirm the suitability of this approach.

ANALYSIS OF FLUID FLOW DOMAIN

The prediction of flow is very complicated due to the rotation and curved three-dimensional shape of impellers. In order to know more about the flow pattern, simulation of flow is essential. The simulation can be carried out by commercially available computational fluid dynamic packages.

The objective of the simulation is to study the detailed distribution of the flow parameters. The simulation procedure should be established with the experimental results for an available pump, and can be extended to the new design modifications. The simulation results provide a better insight to the designer by doing parametric studies. The experiment compliments the simulation results. This is helpful for fixing the simulation parameters and based on these parameters, the modified designs are validated. This reduces the number of trials, which ultimately reduces the time and cost for the development.

Introduction to CFD

Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics, from aerospace propulsion to weather prediction. CFD is commonly accepted as referring to the broad topic encompassing the numerical solution, by computational methods. These governing equations, which describe fluid flow, are the set of Navier-Stokes equation, continuity equation and any additional conservation equations, for example, energy or species concentrations.

Since the advent of the digital computer, CFD, as a developing science, has received extensive attention throughout the international community. The attraction of the subject is two fold. Firstly, there is the desire to be able to model physical fluid phenomena that cannot be easily simulated or measured with a physical experiment, for example, weather systems. Secondly, there is desire to be able to investigate physical fluid systems more cost effectively and more rapidly than with experimental procedures.

Traditional restrictions in flow analysis and design limit the accuracy in solving and visualization of the fluid-flow problems. This applies to both single and multi-phase flows, and is

Ravi N et al., Inter. J. Int. Adv. & Res. In Engg. Comp., Vol.–06(04) 2018 [2592-2597]

particularly true of problems that are three dimensional in nature and involve turbulence, chemical reactions, and/or heat and mass transfer. All these can be considered together in the application of CFD, a powerful technique that can help to overcome many restrictions inherent in traditional analysis.

CFD is a method for solving complex fluid flow and heat transfer problems on a computer. CFD allows the study of problems that are too difficult to solve using classical techniques. The flow path inside the impeller of the centrifugal pump is intricate and this can be analyzed using CFD tool, which provides an insight into the complex flow behavior.

The process of performing CFD simulations is split into three components: Setting up the simulation: Pre - processing (interactive) Solving for the flow field: Solver (non - interactive / batch process)

RESULTS AND DISCUSSION

Simulation Results

The results of the flow analysis can be viewed using plots or through surface integrals as shown in Figures 4.1 to 4.6. Surface integrals provides the required value at the given boundary condition. Plots give a graphical view of the results and they are of two types, namely, contour and vector plots. Contour plots show the constant magnitude for a selected variable (isotherms, isobars etc.). Vector plot is used to give the direction in which the flow occurs.

From Figures, it is inferred that there is a uniform distribution of pressure and velocity around the impeller in the volute casing for the maximum efficiency condition over the full open condition. The difference in the flow pattern is the cause of the radial and axial imbalances.rom, it is inferred that a highly non-uniform distribution of pressure is observed around the impeller in the volute casing, but it is less when compared to that of 2300 rpm. As a result, the impeller is subjected to some static radial thrust.

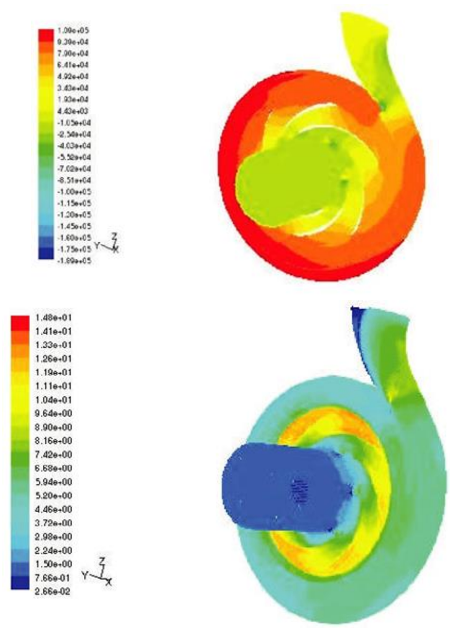


Figure 4.3.Static pressure and velocity distribution for 2500 rpm for Aluminium alloy 6061

CONCLUSION

Modelling and simulation of centrifugal pump impeller has done using Solid Works software. After observing the CFD analysis values we can conclude that e-epoxy has the better performance is given compared to the Aluminium alloy E-glass/Epoxy material is non metallic component so, the Epoxy/E-glass material because it has high performance and ot will give good efficient capa and reasonable manufacturing cost chattering noise will be low compared to other materials during the functioning process.

- For manufacturing the centrifugal pump impeller we can proceed

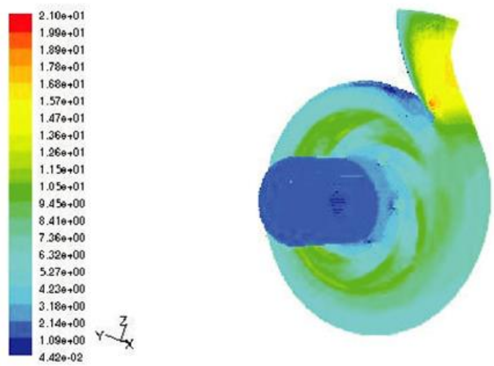


Figure 4.2 Static pressure and velocity distribution for 2300 rpm for E-Glass / Epox

REFERENCES

[1]. E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris (2008),"Parametric study of centrifugal pump by varying the outlet blade angle", The open mechanical engineering journal, 2008, 75-83.
[2]. LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai (2010),"Effects of blade number of characteristic of centrifugal pump ",Chinese journal of mechanical engineering, 2010.
[3]. B.Mohan, B.E. Kumar, "Structural Analysis on Impeller of an Axial Flow Compressor using Fem", International Journal of Engineering Research & Technology ISSN: 2278-0181 3(10), 2014.
[4]. S.Rajendran and Dr.K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX", International Journal of Engineering Research & Technology, 1(3), 2012.
[5]. A Syam Prasad, BVVV Lakshmipathi Rao, A Babji, Dr.P Kumar Babu, "Static and Dynamic Analysis of a Centrifugal Pump Impeller", International Journal of Scientific & Engineering Research, 4(10), 2013.
[6]. Neelambika, Veerbhadrappa (2014) "CFD analysis of mixed flow impeller", International journal of research engineering and technology.
[7]. Sambhrant Srivastavaa, Apurba Kumar Roy and Kaushik Kumar, "Design of a mixed flow pump impeller and its validation using FEM analysis", Science Direct, Procedia Technology 14, 2014, PN 181 – 187.
[8]. Santosh Shukla, Apurba Kumar Roy and Kaushik Kumar, "Material Selection for blades of Mixed Flow Pump Impeller Using ANSYS", Science Direct, Materials Today: Proceedings 2, 2015, 2022 – 2029.
[9]. Basawaraj, H.Hasu (2016), "A study analysis on centrifugal pump impeller guide vane with fem approach by using different material", IJES 8, 2016.
[10]. S.Rajendran and Dr.K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX," International Journal of Engineering Research & Technology, 1(3), 2012.
[11]. 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, 2010.
[12]. P.UshaShriansC.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, 2010.
[13]. E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, "Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle," The Open Mechanical Engineering Journal, no 2, 2008, 75- 83. [5] Marco Antonio Rodrigues Cunh and Helcio Francisco Villa Nova.