International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 1, Issue 2, October 2014

Centrifugal Pump Stress analysis with Optimization

PV Senthiil[1], Mirudhuneka[2], Aakash Shirusshti [3]
Head, Mechanical Engineering, SPIHER, St.Peters University, Chennai-600054, INDIA[1]
SAP Consultant, IBM Ltd, Porur, Chennai[2]
Department of Mechanical, SRM University, Chennai[3]

Abstract: The pump castings are manufactured in isotropic nature materials. In high head, high temperature pumps the pressure acting on the surface is very high. The designer should check whether the materials withstand that much pressure. Design procedures and drawing are done in manually; this is laborious and time consuming process. In competitive environment time factor is important is bring out design to manufacture stage. The only way to overcome the problem is approaching design process by computer aided and computer aided engineering the present work attempts to automate process of hydraulic design of centrifugal casting and generation of 3D geometrical model. CATIA software can be used for generation of geometrical model. The hydraulic design will based mean velocity method. The volute plane profile can be determined at each incremental angle of 45degree; geometrical model of the casing to be generated in CATIA environment using the profile generated. The stress analysis can be carried out in ANSYS platform. Pressure distribution of 20% skew and 50% skew to be analysed for selecting value of thickness. The thickness of the model can be optimized using stress analysis techniques.

Keywords: optimization, suction, pump geometry & process automation

I. Introduction

We introduce the components in the centrifugal pump and a range of the pump types produced by Grundfos. This chapter provides the reader with a basic understanding of the principles of the centrifugal pump and pump terminology. The centrifugal pump is the most used pump type in the world. The principle is simple, well-described and thoroughly tested, and the pump is robust, effective and relatively inexpensive to produce. There is a wide range of variations based on the principle of the centrifugal pump and consisting of the same basic hydraulic parts. The majority of pumps produced by Grundfos are centrifugal pumps in this paper, the gear model is built through Pro/Engineer software. The gear outline with the relation equation forms the gear accurate surface. The parameterization characteristics implement the gear fast modification; thus implement the fast establishment standardization model. The general files style: IGES transfer from Pro/E to finite element analysis software Abaqus 6.10. The static analysis and comparison of performances are carried out. This paper provides the finite element analysis of gear stress influence for the composite material.

II. Principle Of The Centrifugal Pump

An increase in the fluid pressure from the pump inlet to its outlet is created when the pump is in operation. This pressure difference drives the fluid through the system or plant. The centrifugal pump creates an increase in pressure by transferring mechanical energy from the motor to the fluid through the rotating impeller. The fluid flows from the inlet to the impeller centre and out along its blades. The centrifugal force hereby increases the fluid velocity and consequently also the kinetic energy is transformed to pressure. Figure 1.1 shows an example of the fluid path through the centrifugal pump.

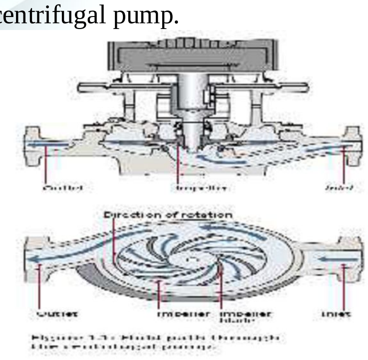

Fig. 1 Model

III. CATIA

Computer Aided Three-dimensional Interactive Application is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systems. Written in the C++programming language, CATIA is the cornerstone of the Dassault Systems product lifecycle management software suite. CATIA competes in the high-end CAD/CAM/CAE market with Creo Elements/Pro and NX (Unigraphics). The competitive imperative to innovate is driving more and more companies to specify, design, manufacture, market and support world-class products via international teams Distributed throughout the world. Increasingly, it is no longer sufficient to simply check if parts fit-together. At first the pump casting components are designed by the CATIA software. The velocity of the water flows to determining by mean velocity for the following method implemented.

IV. ANSYS

A. *Introduction*

ANSYS, Inc. is engineering simulation software (computer-aided engineering, or CAE) developer that is headquartered south of Pittsburgh in the South point business park in Cecil Township, Pennsylvania, and United States. ANSYS as software is made to be user friendly and simplified as much as possible with lots of interface options to keep the user as much as possible from the hectic side of programming and debugging process. ANSYS was listed on the NASDAQ stock exchange in 1996. In late 2011, ANSYS received the highest possible score on its Smart Select Composite Ratings according to Investor's Business Daily. The organization reinvests 15 percent of its revenues each year into research to continually refine the software [1].

B. *What Is Ansys?*

ANSYS is a finite-element analysis package to simulate the response of a physical system to structural loading, and thermal and electromagnetic effects. ANSYS uses the finite-element method to solve the underlying governing equations and the associated problem specific boundary conditions.

C. *Analyzing*

After completed the design a module the pump casting 3D model analyzed by ANSYS to determine the stress induced inside of the pump casting. So the stress created by the pressure of water flows analyzed and optimized. And again designed and analyzed. So the creation of the pressure optimized. The involutes profile not to be affected by a pressure of water flows.

V. CFD Approach

The task of analyzing a turbo-machine using CFD is not straight forward for a number of reasons. Forinstance the complicated geometry requires careful modeling and large grids as well as the interaction of rotational and stationary components requires a liable multiple frame of reference capability to name but two areas. In selecting a CFD package Weir Pumps spent a significant time assessing the commercial packages available.

VI. Grid Generations

Grid generation is a large topic that cannot be covered in detail as part of this thesis. However the main options available for grid generation will be discussed with brief consideration of the merits and drawbacks of each. For further information on this topic. Methods of grid generation are usually arranged into three main categories: structured, un structured and multi-block.

Structured Grid: Structured grids are usually created using hexahedral elements (formed with eight nodes and six quadrilateral faces) or blocks and are built with a repeating geometric and topological structure. The block can be shaped to the modeled domain through stretching and twisting of the block. It is there fore important to gain an understanding of the quality of the grid and this is achieved by monitoring the aspect ratio (amount of stretching) and skew angle (amount of twisting) of the grid elements in the model. Advantages for using structured grid include, a high degree of user control, the fact that hexahedral support a high degree of skewness and stretching before a solution is significantly affected and the solvers generally require the lowest amount of memory for a given mesh size. The main drawback of structured grids is the time and expertise required to create a satisfactory grid for an entire model; in general the time to generate a grid is measured in days. Unstructured Grid: Unstructured grids are usually created using tetrahedral elements arranged in an arbitrary fashion. This method has the ability to be automated to a large extent, making the general time scale for this task to be often measured in minutes and hours [4]. The main advantage of unstructured grid is the ability to generate grids in a short period of time with little user input. This allows the method to be used by in experienced users and also allows very large detailed problems to be tackled. Some drawbacks of un structured grid methods include a lack of user control, a reliance on quality CAD data solver.

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 1, Issue 2, October 2014

It should be noted that one source of error in CFD simulations, related to the choice of structured or unstructured grid, is numerical diffusion (also termed false diffusion). Numerical diffusion a rises from truncation errors that area consequence of representing the fluid flow equations in the discrete form and is minimized when the flow is aligned.

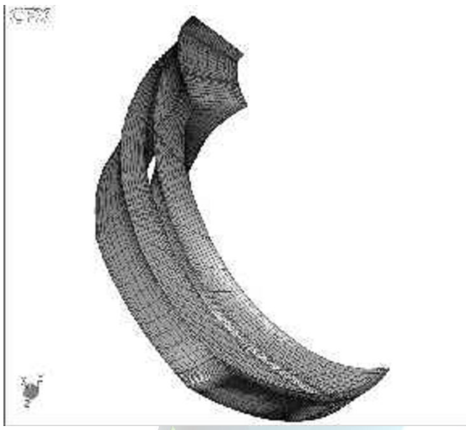

Fig. 2 volute grid

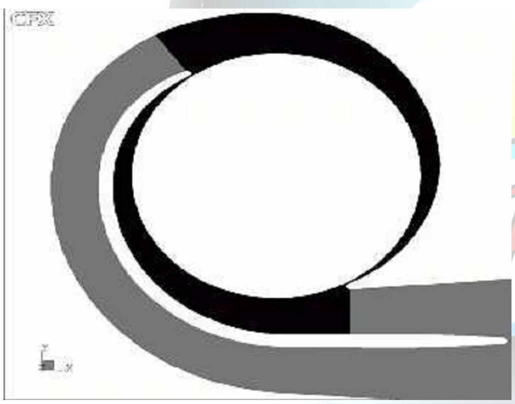

Fig. 3 Volute grid section

The trapezoidal volute design and grid was generated using CFXBuild. The modeling task to create a single continuous grid for the entire volute is extremely complex, especially at the splitter and cutwater regions. This was simplified by splitting the volute into two sections as shown in Figure. The first section, volute' A', contains the interface between the impeller and the volute and the connection between the leakage passage and the volute. The axial grid distribution in the impeller was replicated in the volute to aid the computation off low across the interface. The second section, volute B', contains the flow along the back of the splitter to the outlet and from the cutwater to the outlet. Even with the split model, the generation of the volute proved complex. Unfortunately the diameter given for the oversized impeller was the same as that of the diameter of the volute cutwater. [2]

Fig4. Degree stagger impeller arrangement model

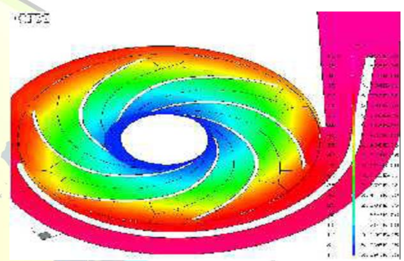

Fig. 5 Pressure plot within the impeller

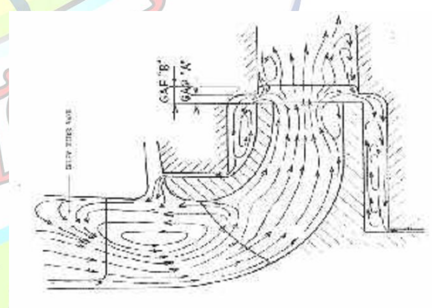

Fig. 6 Flow pattern

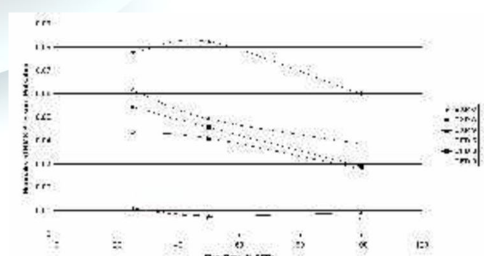

Fig.7 Comparison of experimental test and CFD analysis Results and Optimization

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 1, Issue 2, October 2014

VII. VANE

The vane arrangement can be seen to have an effect on the pressure pulsations at all pump locations. The inline vane arrangement has the strongest pulsations with the 30 degrees vane stagger typically registering lower pulsations; the 15 degree vane stagger pressure pulsation values are between those produced at the two extreme vane positions but are not the average of them.

VIII. CFD APPROACH

The analyses conducted have been arranged so that they can be optimized using a Taguchi process. Thus all results are presented in the Taguchi array format for simplicity and consistency. The core of the CFD analysis results is the output of the twenty-seven transient analyses The size of the cutwater gap exerts a strong influence on the pressure pulsation over a wide range of the locations monitored, with the pressure pulsations reducing as the cutwater gap increases. This influence is strongest at the impeller outlet region (Blade and Shroud positions) and at circumferential locations close to the cutwater/splitter locations in both the volute (C5 and C6) and the leakage flow region (C3 and C4). The influence, although still present, tends to decrease to some degree at regions circumferentially remote from the cutwater/splitter (locations C1, C2, C8, and C9) [3].

The analyses conducted have been arranged so that they can be optimized using a Taguchi process. Thus all results are presented in the Taguchi array format for simplicity and consistency. The core of the CFD analysis results is the output of the twenty-seven transient analyses. Selected results for a few cases have been presented in. To give all of the results in that form would involve many graphs and may be unhelpful. This chapter attempts to summaries key elements of the results, particularly those related to pressure pulsations and performance. CFD results are the simplest analysis that can be conducted. Although limited, simple observation of the results can identify values that appear unusual when compared with the other values and can also indicate the important parameters that effect the pressure pulsation .This data is arranged in the Taguchi array format in with each table containing data for one of the three analyzed flow conditions,1.00Qn, 0.50Qn and 0.25Qnrespectively[5].

IX. CUTWATER GAP

The size of the cutwater gap exerts a strong influence on the pressure pulsation over a wide range of the locations monitored, with the pressure pulsations reducing as the cutwater gap increases. This influence is strongest at the impeller outlet region (Blade and Shroud positions) and at circumferential locations close to the cutwater/splitter locations in both the volute (C5 and C6) and the leakage flow region (C3 and C4). The influence, although still present, tends to decrease to some degree at regions circumferentially remote from the cutwater/splitter (locations C1, C2, C8, and C9). These trends are understandable as the cutwater gap controls the distance between the impeller outlet and the cutwater and it is reasonable that varying the parameter would have the greatest influence in these areas.

TABLE I
RESULTS

Optimization	Cut ware	Va	Snuber	Side wall
Impeller life	6-8%	Staggered vane	1%	100%
Noise and vibration	Close to 6%	30 degrees	---	---

X. CONCLUSION

The final remarks on this project are presented in two sections. This chapter provides a brief description of the salient points regarding the work undertaken along with a summary of the important conclusions. A number of possible avenues of further work have been identified that would enhance, extend and/or refine the information presented here. The main observations and conclusions to emerge from the investigation of pressure pulsation variation with pump geometry are summarized below. Many of these represent unique contributions.

a) A numerical model of the entire pump has been successfully generated and analyzed for the first time. This model consists of a suction inlet, double entry impeller, leakage flow passageways and the volute.

b) The numerical model of the double entry impeller is the first of its type to be successfully modeled for analysis. In total, three different double entry impeller configurations are used in the project. Similarly, the double volute is the first of its type to be successfully modeled for numerical analysis.

c) A total of thirty three transient analyses have been successfully conducted within a parametric analysis, representing 45,000 hours of continual analysis time and consisting of over 550 gigabytes of analysis and result data. This is the first parametric type analysis to be conducted on an entire pump.

d) Velocity flow patterns generated by the numerical analysis identify features of the flow that agree with those identified in published literature. Detailed investigations of the velocity components at the impeller outlet have shown these to be comparable in shape to those published in two independent external experimental tests

e) The pressure levels predicted by the numerical analysis have been successfully generated and give rise to an enormous and interesting data set. The results have been presented by concentrating on fifteen selected locations around the pump. The pressure levels have been converted into RMS pressure pulsations and provide detailed information regarding the pressure pulsation close to the impeller outlet, in the volute and in the leakage flow region.

REFERENCES

[1]. Dalley, S. and Oleson J. (2003). “Sennacherib, rchimedes, and the Water Screw: The Context of Invention in the Ancient World”. Technology and Culture Volume 44,Number 1, January 2003.

[2]. Papin, D. (1689). “Rotailis Suctor et Pressor Hassiacus, in Serenissma Aulo Cassellana demonstratus et detectus,” Actis Eruditorum, p317. Cited in Van Esch, B. P. M., (1997). “Simulation of three-dimensional unsteady flow in hydraulic.

[3]. Elms, K. “The World of Roto dynamic Pumps, chapter 2: The innovators of rotodynamic devices”. (WWW document). http://xstream. fortunecity.com/laras/63/id20.htm (accessed 28th February 2006).

[4]. Gülich, J. (1999). “Impact of Three-Dimensional Phenomena on the Design of Roto dynamic Pumps.” Journal of Mechanical Engineering Science, Proceedings Part C, 1999 Vol. 213, pp. 59-70.

[5]. Hergt, P. (1999). “Pump Research and Development: Past, Present and Future.” ASME Journal of Fluids Engineering, Vol. 121, pg. 248-253.