ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 2, February 2018

Finite Element Analysis of Centrifugal Pump Impeller

[1] Krishan Kumar Singh, [2] S.V. Nimje
[1] PG Student, [2] Assistant Professor,
Department of Mechanical Engineering, Defence Institute of Advanced Technology (Deemed University)
Girinagar, Pune, India

Abstract: **-- A centrifugal Pump is hydraulic machines used in the process of transferring fluids from one place to other and these pump have a vibrant role in the domestic, industrial and marine application. The present work deals with the analysis of impeller with two different materials. The detail three dimensional FE analyses have been performed for impeller of the centrifugal pump. The analysis of stress and deformation under loading environments such as pressure and rotational velocity have been evaluated for the present structure and critical location has been identified. An attempt has been made to investigate the effect of pressure and rotational velocity on the impeller with the purpose of estimating the magnitude of stress and deformation on the impeller with different materials. The study is also made to suggest the suitable material for an impeller of a centrifugal pump by comparing the results obtained for two different materials Stainless Steel (Grade 316 UNSS31600) and Nickel Aluminum Bronze (UNS C95800). Accordingly, the suitable recommendation has been made for impeller of the centrifugal pump.**

Key words**: Impeller, Stress Analysis, Deformation Analysis.**

I. INTRODUCTION

A pump is a hydraulic machine which converts the mechanical energy into hydraulic energy in the form of pressure energy. The energy transfer is generated mainly by the centrifugal force in the impeller (due to pressure and rotational velocity fields act as coupled force) [1]. The centrifugal pumps are widely used in the domestic, industrial, marine application and other technical sectors. However, their design and performance prediction process is still a difficult task. Generally, the geometry is complex, asymmetric and great number of free geometric parameters, the effect of which cannot be directly evaluated [2]. For this reason Finite Element & CFD analysis is currently being used in the design and construction stage of various pump types. Using the finite element analysis the effect of pressure on impeller blades can be predicted and analysed. The finite element (Structural) analysis helps us to identify the deformation, stress and strain concentration at critical location of the pump impeller subjected to different loading environment (pressure and rotational velocity) conditions. The purpose of the present study is to investigate the effect of pressure and rotational velocity to analysis strength of a centrifugal pump impeller, by using the commercial softwares like SOLIDWORKS and ANSYS Workbench.

II. FINITE ELEMENT ANALYSIS OF IMPELLER

Finite element method is the most versatile of all numerical techniques available. Accurate analysis of centrifugal pump impeller can be performed by FEA and this is a tool that can provide physical insight and accurate results. In the present research, strength analyses of different material impeller under pressure and rotational velocity fields in a coupling have been studied using three-dimensional geometrically finite element (FE) analyses.

A. Modelling and Meshing of Pump

To study finite element analysis on the impeller of pump, three-dimensional model of an impeller is required to generate in the software. The modelling was done in two different softwares. The impeller design as shown in Figure 1 & 2 is considered for the present analysis. The main dimensions of impeller, the inner radius $r1 = 42.5$ mm, the outer radius $r2 = 94.5$ mm, the plate thickness of impeller $t = 4$ mm, the outer angle of blade $\beta 2 = 26.63°$ the shafting radius $r3 = 12$ mm, the blade high trailing $h1 = 5.5$ mm, the blade high leading $h2 = 13.15$ mm, the thickness of blade $t1 = 4.04$ mm and six number of long space twisted blades are arrange alternately for the structural analysis of impeller.

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 2, February 2018

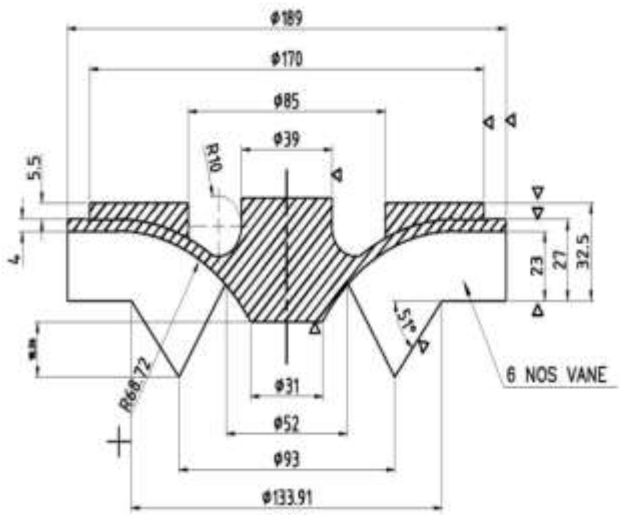

Figure 1. Geometry and configuration of impeller.

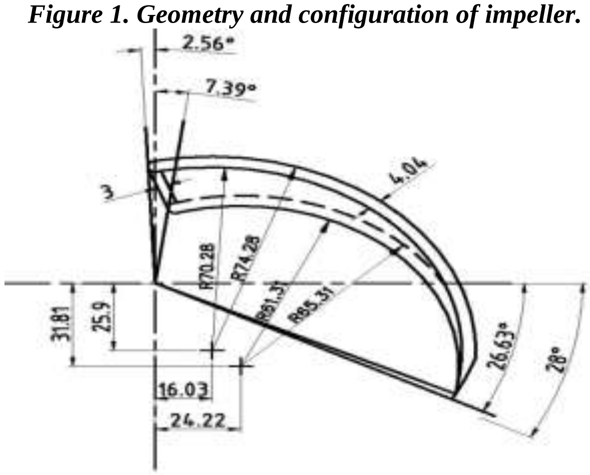

Figure 2. Geometry of impeller blade.

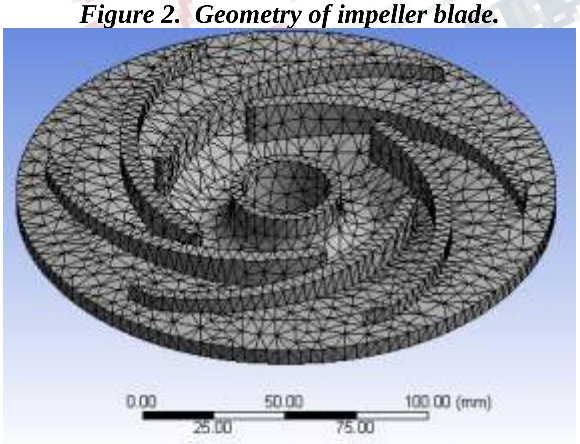

Figure 3. Meshing of pump impeller (tetrahedral)

Meshing of the pump assembly is done by using ANSYS workbench. Meshing of the impeller consist of tetrahedral elements. Meshing consists of following numeric values:-

- Number of Elements = 165801
- Number of Nodes = 281564

III. ANALYSIS OF CENTRIFUGAL PUMP IMPELLER

A. Structural Simulation of Pump Impeller

The finite element analysis includes the structural testing of the impeller of centrifugal pump. The analysis on the impeller is done by applying 6 bar pressure on the surface of blades and 314 rad/s of the rotational velocity on the impeller, set as global rotating reference frame. The pressure distributions on the blade surfaces and rotational velocity in the y-axis of rotation, (x-axis and z-axis are fixed) are transferred to the surface of the solid impeller mesh, applied loading environment help us to get the critical locations. In order to check the contour of total deformation, equivalent stress and equivalent strain in impeller blade and surface plate. Analysis is done for the material Stainless Steel (Grade 316 UNSS31600) and Nickel Aluminum Bronze (UNS C95800) respectively.

B. Structural analysis of Stainless Steel (Grade 316UNSS31600) pump impeller

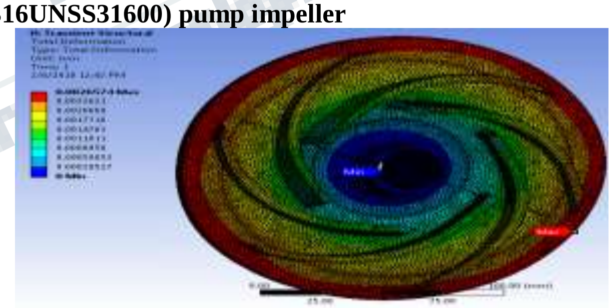

Figure 4. Total deformation in pump impeller

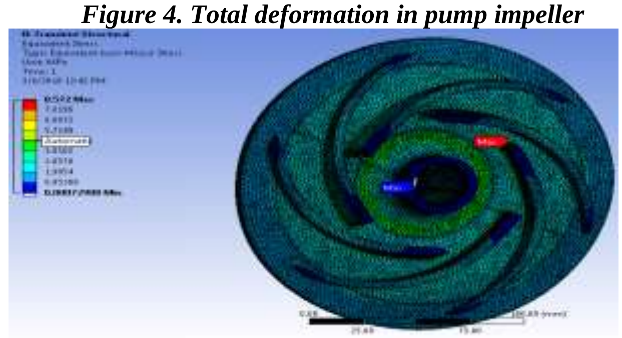

Figure 5. Maximum Stress induced in pump impeller

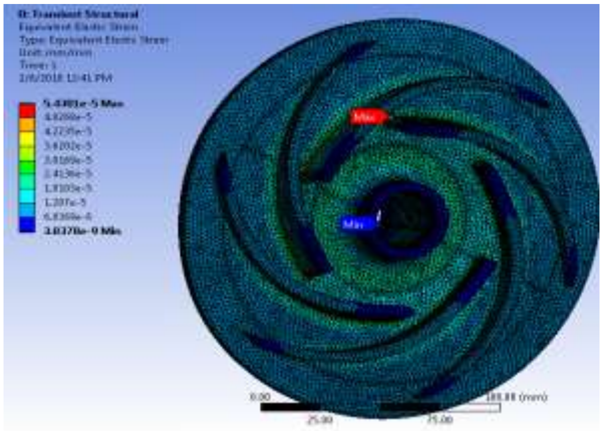

Figure 6. Maximum strain induced in pump impeller

C. Structural analysis of Nickel Aluminum Bronze (UNS C95800) pump impeller

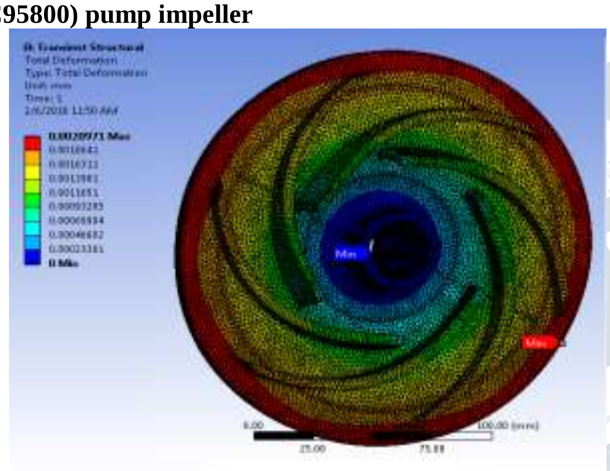

Figure 7. Total deformation in pump impeller

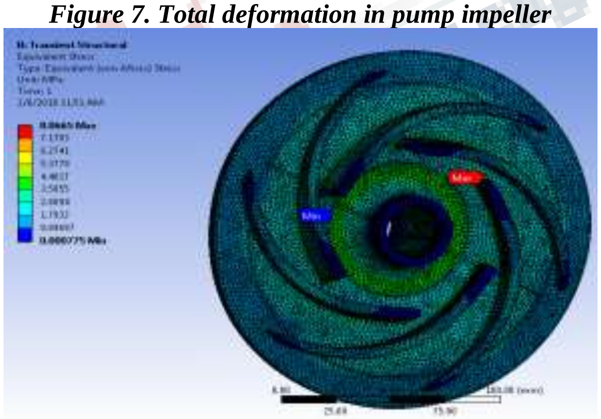

Figure 8. Maximum Stress induced in pump impeller

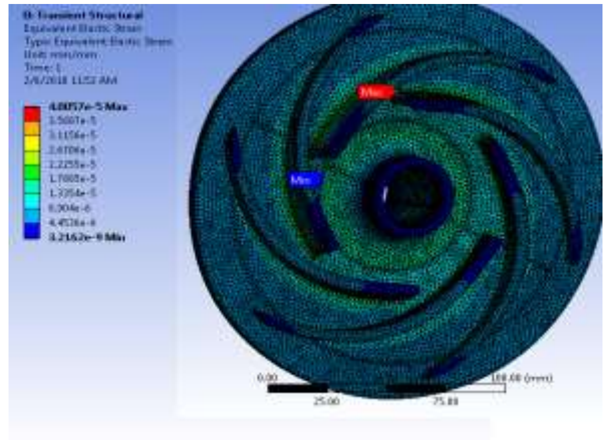

Figure 9. Maximum strain induced in pump impeller

IV. RESULTS AND ANALYSES

Sr.No	Material	Deformation (mm)	Stress (MPa)	Strain
I	*Stainless Steel (Grade 316 UNSS31600)*	.0026514	8.572	5.430E-5
II	*Nickel Aluminium Bronze (UNS C95800)*	.0020975	8.0667	4.0057E-5

Table1- Stress and Deformation of Impeller

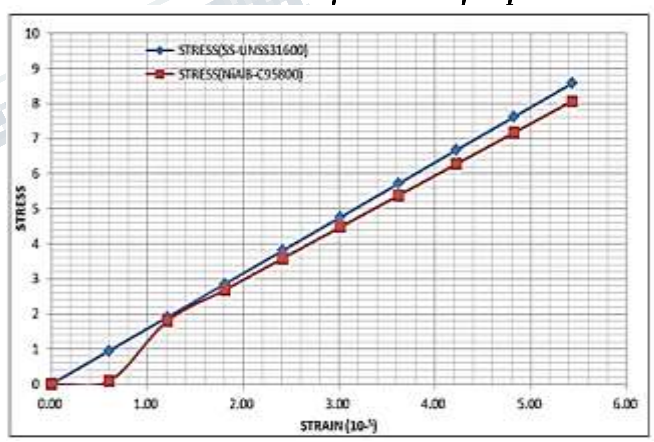

Figure 9. Maximum strain induced in pump impeller

The pressure and rotational velocity fields in a coupling are applied during simulation. Table-1 shows the maximum stress and deflection in two different materials used in impeller for analysis. The static pressure increases continuously from the inner to the outer region of the impeller due to the impulse of the blades. The high deformation regions are seen close to the outside edge of the

blades in both materials. Figure 4 & 8 shows the equivalent stress distribution of the impeller caused by the flow water pressure (6 bars) and rotation velocity of the impeller. It can be seen that there are certain stress concentrations at the root of the impeller blades.
The maximum deflection induced in metallic pump impeller i.e. Stainless Steel (Grade 316 UNSS31600) material is .002657mm, which is within safe limits. Hence based on rigidity, design is safe. The maximum induced stress for the same material is 8.572 MPa which is less than the allowable stress. Hence design is safe based on strength. If we compare corresponding deformation of the material Nickel Aluminum Bronze (UNS C95800) to the above result, Ni-Al-B having minimum deformation and stress concentration at the root of blades .Hence there is less chance of failure of pump impeller in a given loading environment in case of Nickel Aluminum Bronze material.

V. CONCLUSION

The strength of the impeller is analyzed using finite element method. The result indicates that the largest stress concentration occurred around impeller blades root surface and deflection occur at edge of blade, it indicates that the Nickel Aluminium Bronze (UNS C95800) material impeller having minimum deflection and stress concentration at the root of impeller blades. Hence Nickel Aluminium Bronze is more suitable material for impeller.

REFERENCES

[1] JinbaoLin, YanjuanJin, Zhu Zhang, and Xiaochao Cui, "Strength Analysis of the Carbon-Fiber Reinforced Polymer Impeller Based on Fluid Solid Coupling Method," Hindawi Publishing Corporation Engineering,vol.17,no.3,pp.133–136,1981.

[2] Stickland, M.T. and Scanlon, T.J. and Blanco-Marigorta, E. and Fernandez-Francos, J. and Gonzalez-Perez, J. and Santolaria-Morros, C. (2000) Numerical flow simulation in a centrifugal pump with impeller-volute interaction. In: Proceedings of the ASME 2000 Fluids Engineering Division Summer Meeting. ASME. ISBN 0791819795.

[3] E. Cezmi Nursen and Erkan Ayder, "Numerical Calculation of the Three-Dimensional Swirling Flow inside the Centrifugal Pump Volutes", International Journal of Rotating Machinery, vol.-9, pp. 247–253, 2003.

[4] Zhang Shujia, Zhu Baolin, Hu Qingbo and Li Xianhua, "Virtual performance experiment of a centrifugal pump", 16th International Conference on Artificial Reality and Telexistence--Workshops (ICAT'O6) 2006.

[5] M. H. ShojaeeFard, F. A. Boyaghchi and M. B. Ehghaghi, "Experimental Study and Three Dimensional Numerical Flow Simulation in a Centrifugal Pump when Handling Viscous Fluids", IUST International Journal of Engineering Science, vol.-17, no.-3-4, pp. 53-60, 2006.

[6] LI Xian-hua, ZHANG Shu-jia*, ZHU Bao-lin and HU Qing-bo, "The Study of the k-ε Turbulence Model for Numerical Simulation of Centrifugal Pump", Key Laboratory of Mechanical manufacture and Automation Zhejiang University of Technology, July 2006.

[7] K.W. Cheah, T. S. Lee, S. H.Winoto, and Z.M. Zhao, "Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions", International Journal of Rotating Machinery, vol.- 2007, Article ID 83641, 8 pages, April

