Validation in the improved performance of Centrifugal pump using CFD

Mr. Nilesh Patil [1], Prof.G.S.Joshi [2], Prof.Dr.V.R.Naik [3]

[1] *Research scholar, Department of Mechanical Engineering D.K.T.E'S Textile and Engineering Institute, Ichalkaranji.*
[2] *Associate Professor, Department of Mechanical Engineering D.K.T.E'S Textile and Engineering Institute, Ichalkaranji.*
[3]Professor, *Department of Mechanical Engineering D.K.T.E'S Textile and Engineering Institute, Ichalkaranji.*

Abstract - *We know that every time it's not possible to manufacture the machine parts and then testing. Instead of that software like Ansys used to test the performance of such machines. Here to find out the improvement in Impeller of centrifugal pumps Computational Fluid dynamics codes are used. After various modifications in geometry of impeller finest geometry is casted and experimentally its improvement is verified.*

Key Words: *Ansys, Centrifugal Pump, Impeller, Computational Fluid Dynamics.*

1. INTRODUCTION

We know that, Centrifugal pumps are the group of turbo machines which are used widely in large scales in industries present theories and practices related to these pumps. So the researchers have put a considerable effort to increase the efficiency in a centrifugal pump . The current scenario in the impeller design interest has considerably increased in stating the pump efficiency, pump performance, impeller design, modification the vane angle through CFD analysis has consideration to elaborate the conceptual frame work of centrifugal pumps.

Nowadays to meet the requirements of are very challenging. So, in order to improve the performance of the centrifugal pumps all the parameters should be taken into consideration. Some researchers explained the design of pump implemented an experimental set up in a laboratory. Nowadays, by using recent theories numerical methodologies of flow field in a centrifugal pump impeller and predicted of the pump performance curves, developed and tested against experimental and statistical data, with encouraging results using Navier-Stokes equations in three dimensions. By using CFD codes shape optimization of impeller blades, validation of Hydraulic design of metallic volute casing and performance are analyzed.

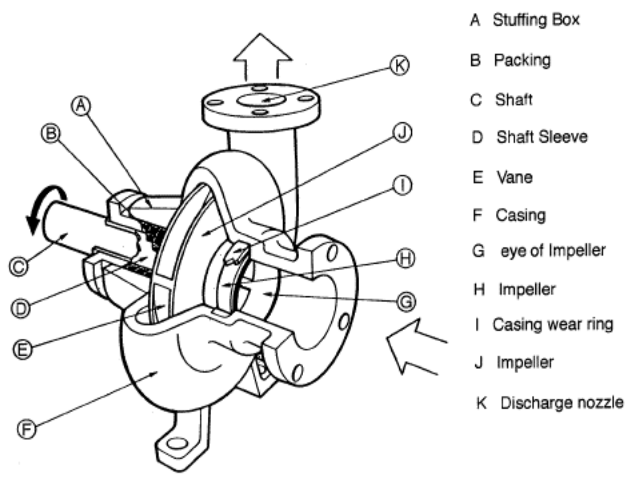


Fig.1geometry of centrifugal pump

2. LITERARURE REVIEW

The impeller of the existing closer range pump has been modified by V.S. Kadam et al. [1] increasing the diameter to 820 mm from 770 mm to suit the higher efficiency, required head and discharge. Pump with higher efficiency and greater stable operating region is designed. The CFD analysis of the pump with modified impeller diameter is carried out to check the performance and efficiency of the pump.

Based on the foundation of the former research Jie Jin et al. [2] have designed and developed GSB20-380 hydraulic model of the ultra-low specific-speed centrifugal pump. In order to rich the achievements on the ultra-low specific-speed centrifugal pump in theory, CFD (numerical simulation and performance tests were adopted to study the model of the centrifugal pumps, to analysis the hydraulic properties of the ultra-low specific-speed centrifugal pump.

A CFD analysis of centrifugal pump to improve discharge by varying blade geometry is done by Anjani Kumar Sinha et al. [3]. The impact of design changes in the water pump is discussed in this project. A CFD module in Ansys software is a tool which is used to find out the improvement in discharge of a water pump. It is observed that the design change in water pump improved the velocity from 1.2m/s to 1.7m/s, so that the discharge is improved by 42% The overall efficiency of the water pump is increased by 25% i.e. its efficiency is improved from 60% to 85%.The design change improves the pressure distribution and peak pressure. The proposed design increases the discharge and velocity of the water. The increase in velocity correspondingly increases the efficiency of the pump.

The performance of the pump was first determined using the existing vane angles and thickness of the blade then, the inlet vane angle, the outlet vane angle along with the thickness of the blade has been varied to analyze the pump's performance by Krishna Kumar Yadav et al. [4]. The results show that for an initial inlet angle 21.08˚, outlet angle 16.28˚ and blade thickness as 10mm, the efficiency of the pump was 84%. However, the efficiency of pump rises to 89.19% for the optimized angles and blade thickness.

Mohankumar M et al. [5] did the analysis of effects that the pertinent design parameters, including the blade number, the inlet blade angle, trimmed impeller profile and the impeller diameter, have on the steady state liquid flow in three-dimensional centrifugal pump. Geometric modeling, meshing CFD analysis carried out pressure, velocity distribution predicted.

Pranit M Patil et al. [6] analyzed the effect of blade thickness ratio and geometry of the blade at the outlet of impeller on the performance of low specific speed centrifugal pump. In CFD, This model was then simulated for operating conditions of a low specific speed centrifugal pump. The first parameter blade thickness ratio was varied and its effect on head and efficiency were studied independently. The optimum blade thickness ratio for the given application was determined .The geometry of the blade at the outlet of the blade was varied and change in efficiency was observed. The effect of both these parameters was studied by keeping overall efficiency of the pump as optimization parameter. The results obtained indicated the increase in the efficiency. The results obtained indicated overall efficiency rise of 4.743% from the standard design of impeller.

3. METHODOLOGY

The proposed work is to carry out the modification in the design of impeller, its manufacturing and testing of centrifugal pump which will result into the improved efficiency. The methodology used is shown as below.

I. Study of design of impeller for current performance to determine any faults that may be arising.

II. Modification in design of impeller till the requirements of head, discharge by the impeller will be fulfilled to qualify requirement of Pump Testing Standard (IS 6595 Part-1).

Modification in outlet angles of existing impeller

↓

Geometrical modeling of impeller model

↓

Static Analysis impeller model

↓

CFD Analysis of impeller models-Base model, Model no.1, 2, 3 & 4

↓

Casting of best impeller model from CFD Analysis

↓

Performance testing & Validation of improved results

Fig2.flow chart of methodology

3.1 Modification in outlet angles of impeller:

Here the objective is:

a) To perform the CFD analysis of Centrifugal Pump of existing design to get the pressure drop and velocity across the impeller outlet.

b) Design optimization of the impeller geometry to increase the velocity at the outlet of the impeller.

Table1.modifications in blade angles of impeller

Description	Base Model	Model No 1	Model No 2	Model No 3	Model No 4
Blade Angle	20^0	19.36^0	25.11^0	26.88^0	20.01^0
Mass flow rate	11 lit/sec				

3.2 Geometrical modeling and meshing of modified impeller model:

Now to go for static analysis or CFD analysis it's required to convert the 2D model into solid model. It is done by using CATIA.

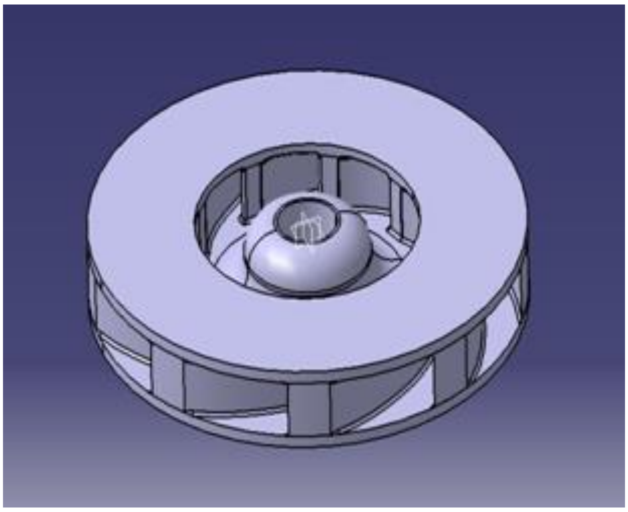


Fig3.geometric modeling of impeller

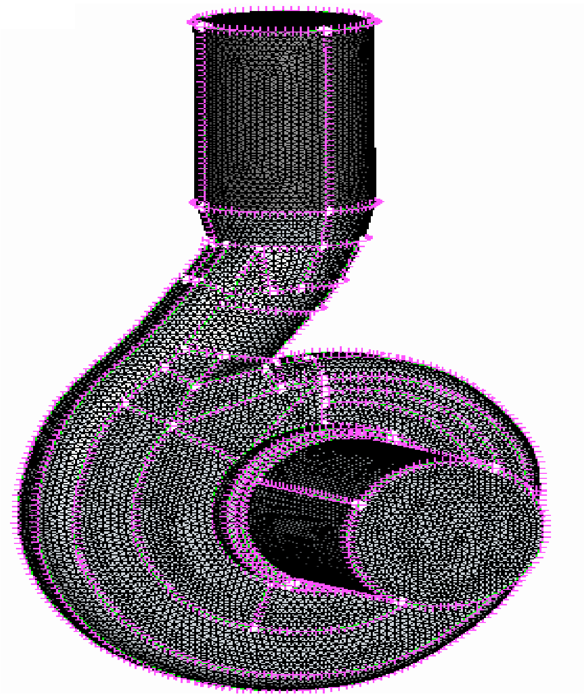


Fig.4 Meshing of impeller Model

Table .2.Meshing details

Description	No. of Nodes	No. of Elements	Quality (Skewness)
Model No 2	68803	321001	0.85

3.3 Static Analysis

ANSYS workbench static structural module is used to carry out comparative numerical static analysis of impeller of the centrifugal pump. The comparison made between basic model of impeller and modified impeller

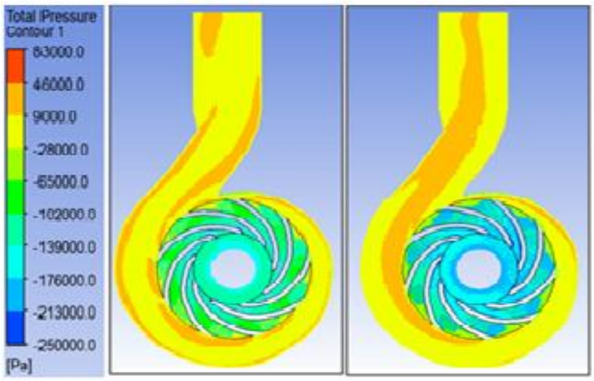
Table2.Comparison of static analysis

Quantity	Base Design	Modified Design (Rev_02)
Vector Displacement	0.0023 mm	0.0035 mm
Tangential Displacement	0.0023416 mm	0.003364 mm
Radial Displacement	0.000482 mm	0.0011165 mm
Equivalent Von Mises Stress.	30.84 MPa	8.98 MPa

3.4 CFD Analysis

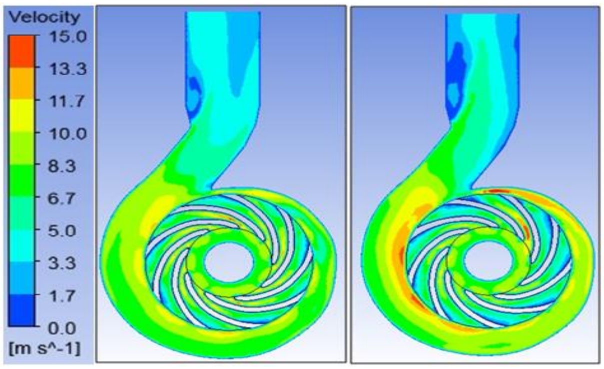
The analysis of the proposed design of the water pump is carried out using analysis software Ansys. Since this project deals with flow of fluids, and involved with variables like velocity and pressure, the CFD module of the Ansys software is selected for the study.

The modified geometries have been incorporated in ANSYS. The CFD analysis is done to find the considerable improved velocity, head etc. The results after CFD analysis are shown below:



Base Model Model No2

Fig.5.Pressure contors of impeller models



Base Model Model No2

Fig.6.Velocity contors of impeller Model in CFD

Table 3. Impeller velocities

Average Velocity (m/s)		
Cases	Impeller Outlet (m/s)	Pump Outlet (m/s)
Base	**12.15**	3.85
Model No 1	11.75	3.79
Model No 2	12.97	3.64
Model No 3	12.72	3.79
Model No 4	12.23	3.86

Table 4. CFD analysis of impeller models

Description	Angle	Pressure in (Pa)	Impeller Outlet (m/s)
Base	20^0	165587.4	12.15
Model no 1	19.36^0	158907.9	11.75
Model no 2	25.11^0	204563.3	12.97
Model no 3	26.88^0	168234.1	12.72
Model no 4	20.01^0	185132.3	12.23

3.5 Performance Testing (Readings)

After CFD analysis, the casting of modified impeller (Impeller No 2) is done. Then the casting of impeller is incorporated in the casing and it is tested. The results after performance testing are mentioned as below.

Table 5.Experimentation results

Case	Base Impeller Model	Modified Impeller (Model No 2)
Discharge (lit/sec)	11.20	12.02
Pump Input (kw)	3.181	2.349
Pump Output (kw)	2.098	3.272
Efficiency in %	66.0	71.79

Graph

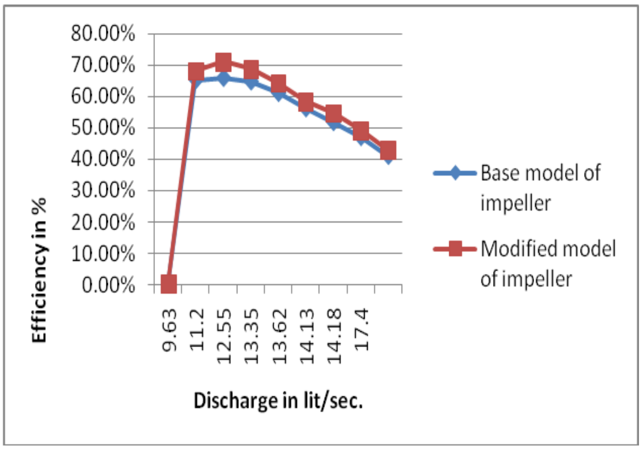


Fig.7.Graphical representation of results

4. CONCLUSION

1. Impeller model No 2 shows considerable improvement over the pressure drop. This help in improving the pump head.
2. Power consumption in Modified impeller is less as compared to base model.
3. Experimental results from the table no 5 shows that after performance of impeller 2 with modified geometry is improved. The efficiency of pump is increased from 66% to 71.79%. Hence the validation of improvement in the of Centrifugal Pump is done.

REFERENCES

[1] V.S.Kadam,S.S.Gawade,H.H.Mohite,N.K.Chapkhane, "Design and Development of Split Case Pump Using Computational Fluid Dynamics", Institute of Technology, Nirma University,Ahmedabad,08-10 December,(2011),pp-1-4.

[2] Jie Jina Ying Fana,Wei Hana Jiaxin Hub, "Design and Analysis on Hydraulic Model of the Ultra-low Specific-Speed Centrifugal Pump" aSchool of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou 730050, China; bChina Fristheavy Industries, Qiqihaer 161042, China, International Conference on Advances in Computational Modeling and Simulation, Procedia Engineering Volume 31, (2012),pp-110 – 114.

[3] Anjani Kumar Sinha, John Rajan, Eriki Ananda Kumar, "A CFD Analysis of Centrifugal Pump to Improve Discharge By Varying Blade Geometry" International Journal of Mechanical And Production Engineering, ISSN:2320-2092,Vol-2,Isuue-11,(2014),pp-15-19.

[4] Krishna Kumar Yadav, Karun Mendiratta, V.K. Gahlot, "Optimization of Design of Mixed Flow Centrifugal pump Impeller Using CFD", International Journal of Research in Engineering and Technology, ISSN: 2319-1163,pp- 365-370.

[5] Mohankumar M, Hudson E Daniel Raj, M. Vratharaj, "Analysis of Effect of Impeller Parameters on

Performance of Centrifugal Pump Using CFD" International Journal of Research in Mechanical Engineering, ISSN:2349-3860, (2014), pp-1-5.

[6] Pranit M. Patil, Shrikant B. Gawas, Priyanka P. Pawaskar, Dr. R. G. Todkar, "Effect of Geometrical Changes of Impeller On Centrifugal Pump Performance", International Research Journal of Engineering and Technology,(2015),pp-220-224.