Development in Performance of Impeller used in Centrifugal Pump by using Computational Fluid Dynamics

Nilesh N Patil
Student Department Mechanical of Engineering
D.K.T.E'S Textile and Engineering Institute, Ichalkaranji.

Prof.G.S.Joshi
Assistant Professor, Department Mechanical of Engineering
D.K.T.E'S Textile and Engineering Institute, Ichalkaranji.

Prof.Dr.V.R.Naik
Professor, Department Mechanical of Engineering
D.K.T.E'S Textile and Engineering Institute, Ichalkaranji.

Abstract- The increased popularity of centrifugal pumps is due to largely to the comparatively recent development of high speed electric motors, steam turbines and internal combustion engines.To improve the performance of centrifugal pumps by studying the key components like impeller to find the optimum conditions using software. A literature review and lots of development in Computational CFD motivated to find out various methods of improving the efficiency of the pump. From this motivation it is proposed to carryout Numerical Static analysis of Impeller of Centrifugal Pump. It is also proposed to carry out the Design Optimization of Impeller for Centrifugal Pump by using CFD

Keywords – Centrifugal Pump, CFD, Impeller, Optimization

I. INTRODUCTION

A centrifugal pump which is widely used throughout industry is a typical turbo-machinery that converts external mechanical energy into pressure and kinetic energy of fluid. It consists of an impeller and volute casing. An impeller is a mechanical device that supplies mechanical energy to fluid and is a key component of any centrifugal pump. Therefore, up to now, many studies have focused intensively on impellers. Fluid that obtains energy from an impeller is discharged through a volute casing, so the characteristics of the volute casing are an important factor if the goal is to discharge fluid with less energy loss.

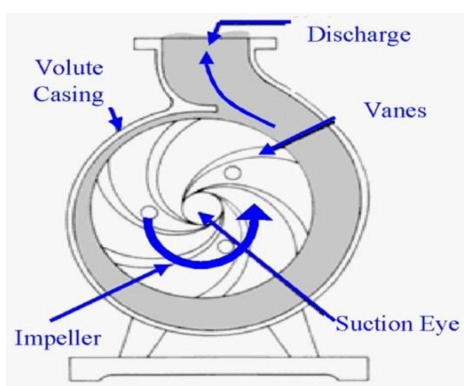

Figure 1 Centrifugal pump

Centrifugal pumps handle high flow rates, provide smooth, non-pulsating delivery, and regulate the flow rate over a wide range without damaging the pump. Centrifugal pumps have few moving parts, and the wear caused by normal operation is minimal. They are also compact and easily disassembled for maintenance. The design of an efficient pumping system depends on relationships between fluid flow rate, piping layout, control methodology, and pump selection. Before a centrifugal pump is selected, its application must be clearly understood.

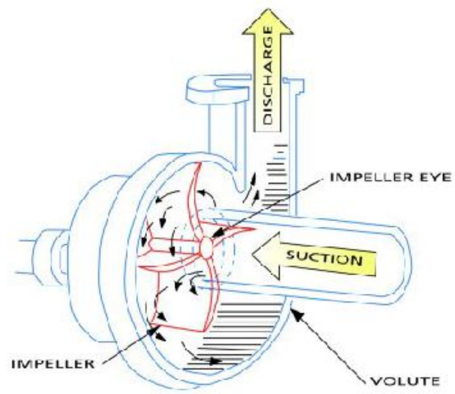

Figure 2 working principle of centrifugal pump

Due to rotation of impeller the fluid from inner radius moves towards the outer radius during this, suction is created at the eye of the impeller. Therefore, continuous lifting of fluid from sump to the pump is carried out and kinetic energy is converted into pressure energy and head is developed from the fluid coming out from delivery pipe.

II. LITERATURE REVIEW:

In the present study by H.Safikhani et al [1], multi-objective optimization of centrifugal pumps is performed in three steps. In the first step, efficiency and NPSHr in a set of centrifugal pumps are numerically investigated using commercial software NUMECA. In second step modeling of above both parameters with respect to geometrical design variables is done. Finally using obtained polynomial neural networks, multi-objective genetic algorithms are used for Pareto based optimization of centrifugal pumps considering two conflicting objectives, efficiency and NPSHr.Such combining application of GMDH type neural network is very promising in discovering useful and interesting design relationships.

The operation of a centrifugal pump working in direct mode and as a centripetal turbine was investigated by John S. Anagnostopoulos et al. [2]using a commercial CFD code. The predictions of the radial load showed a minimum radial thrust near design conditions in pump mode and an increasing magnitude with flow rate in reverse mode of operation. The magnitude of the total radial load resulted lower than the maximum total load in pump mode for operating below turbine rated conditions. Thus, it can be concluded that the mechanical design of the machine and of the shaft bearings must be carefully undertaken if a usual operation in reverse mode is to be expected.

Advanced numerical techniques for adaptive grid refinement and for treatment of grid cells that do not fit the irregular boundaries are implemented by E.C.Bacharoudis et al.[3] in order to achieve a fully automated grid construction for any impeller design. The overall efficiency curve of the pump was found to agree very well with the corresponding experimental data. Finally, a numerical algorithm based on the unconstrained gradient approach is developed and combined with the evaluation software in order to find impeller geometry that maximizes the pump efficiency, using as free design variables the blade angles at the leading and the trailing edge. The results verified that the optimization process can converge very fast and to reasonable optimal values.

The influence of the outlet blade angle on the performance is verified with the CFD simulation by Zhou et al. [4]. As the outlet blade angle increases the performance curve becomes smoother and flatter for the whole range of the flow rates. When pump operates at nominal capacity, the gain in the head is more than 6% when the outlet blade angle increases from 20 deg to 50 deg. However, the above increment of the head is recompensed with 4,5% decrease of the hydraulic efficiency. Moreover, at high flow rates, the increase of the outlet blade angle causes a significant improvement of the hydraulic efficiency.

The commercially available 3D Navier-Stokes code called CFX, which has a standard k-€ two-equation turbulence model, was chosen by Erik dick et al. [5] to simulate the internal flow of various types of centrifugal pumps-M1, M2, M3. The predicted results of the head-flow curves are presented over the entire flow range. It was found that the predicted results for pumps M2 and M3 were better than those for pump M1, which suggests that the efficiency of pumps M2 and M3 will also be higher than that of pump M1. Thus, future work will be focused on improving the design of pump M1. It was found that when the flow rate decreased below a certain value of the design flow rate, backflow occurred near the pressure surface of the pump impeller. That might occur because when the flow rate through the impeller decreases, the impeller passage correspondingly “narrows” itself so that continuity theory can be satisfied.

Fluent provides three calculation methods for analysis of turbo machinery flows: the Multiple Reference Frame method (MRF), the Mixing Plane method (MP) and the Sliding Mesh method (SM) by Selvarasu A. et al. [6]. In all three methods, the flow in the rotor is calculated in a rotating reference frame, while the flow in the stator is calculated in an absolute reference frame. In the MRF and MP methods steady flow equations are solved, while in the SM method, unsteady flow equations are solved. The SM method does not introduce physical approximations.

From above methods it is concluded that, Steady calculation methods like the Frozen Rotor method and the Mixing Plane method cannot be used with confidence to analyze the performance of volute centrifugal pumps. Only a truly unsteady method like the Sliding Mesh technique is able to correctly reproduce this flow behavior. The predicted performance by the Sliding Mesh method can be used with confidence.

The impeller of the existing closer range pump has been modified by V.S. Kadam et al. [7] increasing the diameter to 820 mm from 770 mm to suit the higher efficiency, required head and discharge. Considering economic incentive for operating range and efficiency is gained by better understanding of the influence of the tongue. 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. Efficiency of the pump from CFD results is coming 82 % and by actual performance test efficiency is coming 81.37%, by which it is confirmed that CFD analysis is clearly validated.

III. NUMERICAL APPROACH FOR DEVELOPMENT OF IMPELLER FOR CENTRIFUGAL PUMP BY USING CFD

Finite element procedures at present very widely used in engineering analysis. The procedures are employed extensively in the analysis of solid and structures and of heat transfer and fluids and indeed, finite element methods are useful in virtually every field of engineering analysis

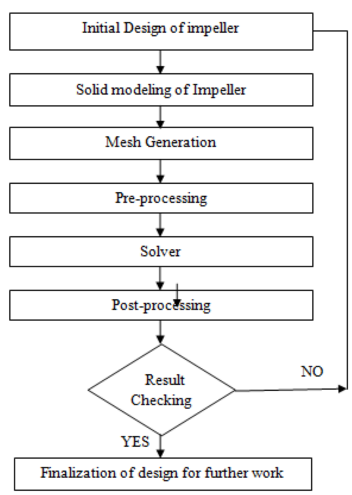

Figure 3. Flowchart to carry out better results using CFD

Basic Objective of the development of the Impeller of Centrifugal Pump.

- To perform the CFD analysis of Centrifugal Pump of existing design to get the pressure drop and velocity across the impeller outlet.
- Development of the impeller geometry to increase the velocity at the outlet of the impeller.

3.3 .Numerical Simulation:

ANSYS CFD module is used to carryout CFD analysis of optimized designs of the impellers of Centrifugal pump.

Figure below shows Geometrical model considered for analysis.

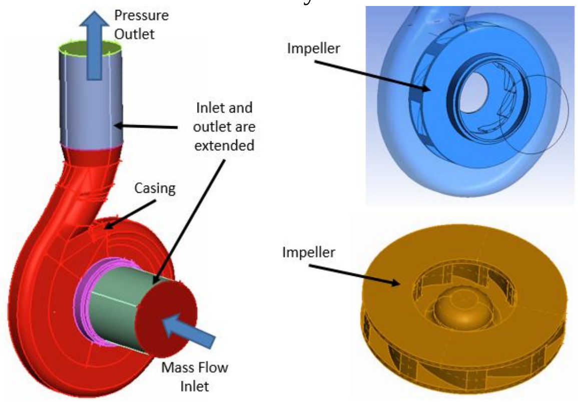

Figure 4 Geometry of Impeller of Centrifugal Pump

Following is the table with the details of Impeller models

Description	The outlet angles at Impeller
Base Model	20^0
Model 1	19.36^0
Model 2	25.11^0
Model 3	26.88^0
Model 4	20.1^0

Table.1 Impeller models with outlet angles.

Mesh Details:

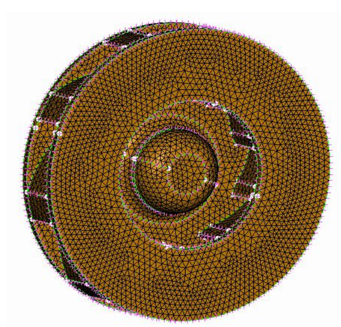

Figure 5: Meshed model of impeller

Table below shows no. of elements and node count details

Model	No. of Nodes	No. of Elements
Model 2	**68803**	**321001**

Boundary Conditions

Following boundary conditions are considered for analysis of centrifugal pump.

Inlet:

Table below gives inlet mass flow:

Mass Flow Inlet		
Cases	LPS	Kg/s
From Base to Model 4	11	10.99

Fluid Zones:

Zones	Motion
Impeller	Rotational
Casing	Stationary
Suction Pipe	Stationary
Discharge Pipe	Stationary

Average Velocity (m/s)		
Cases	Impeller Outlet (m/s)	Pump Outlet (m/s)
Base Model	12.15	3.85
Model 1	11.75	3.79
Model 2	12.97	3.64
Model 3	12.72	3.79
Model 4	12.23	3.86

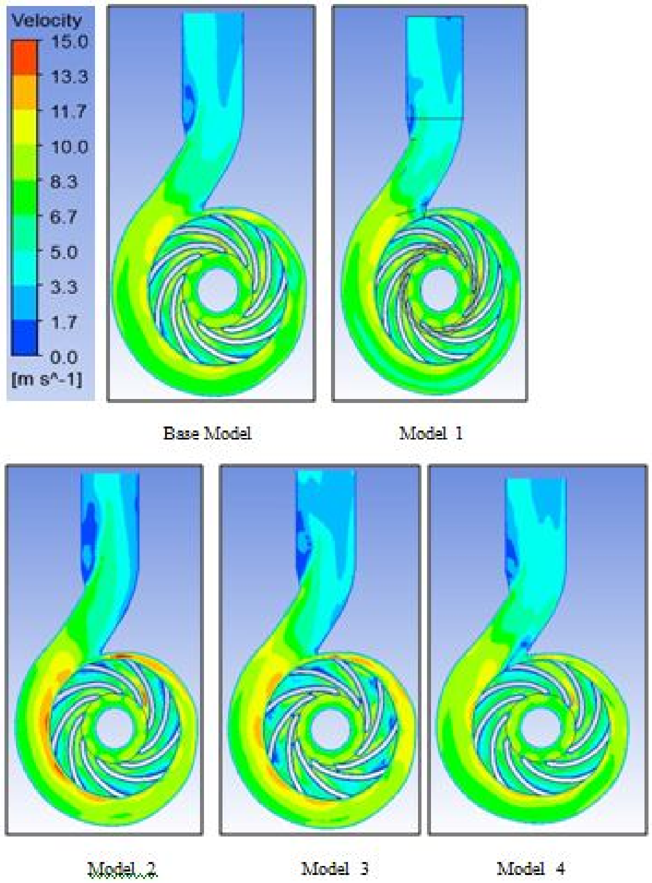

Figure.6 Velocity Vectros of impeller models

IV. RESULT DISCUSSION

The results have been incorporated geometries of the total 4 impellers excluding basic (existing) impeller model in Ansys with various outlet angles and the Impeller model number 2 shows considerable improvement that is 21 meters of water. This may help in improving the pump head.

Average Velocities at the impeller outlet are increased in the Impeller model number 2 (~13 m/s). This will help to improve the flow characteristics along with the increase in discharge. Model number 2 with 25.11^0 blade angle gives considerable improvement in head and impeller velocity as compared to Base case. Hence this can be considered for the further studies.

V. CONCLUSION

In this paper various modifications in impeller geometries are studied using ANSYS workbench. From the counterplots it has been observed that, flow characteristics for the impeller improved with increase in outlet velocity which can be further used for experimentation.

REFERENCES

[1] H. Safikhani, A. Khalkhali and M. Farajpoor, “Pareto Based Multi-Objective Optimization of Centrifugal Pumps using CFD, Neural Networks and Genetic Algorithms”, Engineering Applications of Computational Fluid Mechanics Volume-5, No.1,Pages 37-48 (2011).

[2] John S. Anagnostopoulos , “A fast numerical method for flow analysis and blade design in centrifugal pump impellers”, Computers & Fluids Volume- 38 (2009), Pages 284–289.

[3] 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, 2008, Volume-2, Pages 75-83.

[4] Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H.Winoto, “Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics”, International Journal of Rotating Machinery, Volume-9(1): Pages 49–61, 2003.

[5] Erik dick, jan vierendeels, sven serbruyns and john Vande Voorde, “Performance Prediction of Centrifugal Pumps with CFD-Tools”, Task Quarterly 5, Volume-4(2001), Pages 579-594.

[6] Selvarasu.A, Kanthavel. K, Krishnaraj.R, Ravi.G, Ramesh.R ,“ Conceptual Framework of Centrifugal Pumps towards Industrial Revolution”, International journal of Scientific & Research, Volume 3, Issue 2, ISSN2229-5518, February-2012, Pages1-7.

[7] 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, Pages 1-4