Design Optimization of Monoblock Centrifugal Pump Impeller using Computational Fluid Dynamics

Dr. K. Ragu
Associate Professor
Dept. of Mechanical
PSG College of Technology Engineering
Coimbatore, India

V. M. Mohamed Ashif
UG Scholar
Dept. of Mechanical Engineering
PSG College of Technology
Coimbatore, India

K. Naveen Kumar
UG Scholar
Dept. of Mechanical Engineering
PSG College of Technology
Coimbatore, India

***Abstract* -The objective of this research work is to improve the efficiency of the existing impeller of a centrifugal pump. The performance curve of the existing pump is drawn by testing it on the experimental test rig. The existing impeller and the casing is modeled using NX CAD software. The CAD model is imported and analyzed using SolidWorks Flow Simulation ,a CFD software package. Suction pressure is given as inlet boundary condition and discharge is given as outlet boundary condition. From the CFD results, the head developed, torque requirement and efficiency are obtained. The existing impeller is modified and CFD analysis is made to determine the efficiency. The number of vanes, inlet blade angle and outlet blade angle of the existing design is modified and variation in the head developed, efficiency and torque are noted. The number of vanes in the present model is 6.The number of vanes is changed to 4,5 and 7 and CFD analyses are made for each model. The outlet blade angle of the existing impeller is 16^0.The outlet blade angle is changed to 14^0 and 18^0 and CFD analyses are made. The inlet blade angle of the existing impeller is 34.8^0.The inlet blade angle is changed to 32.8^0, 36.8^0, 38.8^0 and 40.8^0 and CFD analyses are made for each model. The optimum values of number of vanes, outlet blade angle and inlet blade angle are chosen from these results. The optimized impeller is modeled and CFD analysis is made. It is found that the efficiency is increased by 8.28%.**

Keywords- Centrifugal pump, Computational Fluid Dynamics(CFD), Impeller, Efficiency

I. INTRODUCTION

A wide variety of centrifugal pump types have been constructed and used in many different applications in industry and other technical sectors. However, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters, the effect of which cannot be directly evaluated. For this reason CFD analysis is currently being used in the design and construction stage of various pump types. Computational fluid dynamics (CFD) uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow such as Navier-Stokes equations, for predefined geometries and boundary conditions. CFD analysis begins with a mathematical model of a physical problem. Conservation of mass, momentum and energy must be satisfied throughout the region of interest. CFD applies numerical methods (called discretization) to develop approximations of the governing equations in the fluid region of interest. The result is the predictions for flow velocity, temperature and density for any region where flow occurs.

II. LITERATURE REVIEW

Many researchers have worked in the field of centrifugal pump. Srinivasan[1] had presented design procedure for centrifugal monoblock pump impeller. He has formulated pump efficiency as a result of varying certain impeller parameters-number of blades, inlet blade angle, outlet blade angle, blade width, thickness of the blade. Yunus A.Cengel[2] has provided the guidelines to specify the boundary conditions. CFD analysis of pump is made and efficiency is determined using SWFS by R.Ragoth Singh, M.Nataraj[3]. The step by step procedure for constructing the impeller vane profile by circular arc method is given by Gundale, V.A.,Joshi G.R[4]. Hudson and Daniel Raj.E[5] determined that number of blades and inlet blade angle are the key impeller parameters which affect the performance.The performance of the pump was improved by changing the vane angle of the impeller by Amit H. Bhuptani, Prof. Ravi K. Patel[6]. Sujoy Chakraborty and Bidesh Roya[7] has analyzed the impeller by increasing the number of blades and found that head increases and efficiency varies by increasing the number of blades.

Table 1.Existing impeller parameters

S.No	PARAMETER	SIZE
1	Impeller inlet diameter	32 mm
2	Impeller outlet diameter	130 mm
3	Number of blades	6
4	Inlet blade angle	34.8°
5	Outlet blade angle	16°
6	Blade thickness	4 mm

III. EXPERIMENTAL TEST

The experimental test is done on mono block centrifugal pump. The result is used to compare with CFD result.

Table 2. Experimental Test Data

Suc tion Head	Delive ry Head	Total Head	Discha rge	Watt Meter Reading	Out put Power	Overall Efficien cy
mm of Hg	m of H_2O	m of H_2O	LPS	kW	kW	%
440	1	6.98	1.5	0.746	0.102	13.76
440	6	11.9	1.3	0.735	0.153	20.78
440	10	15.9	1.07	0.714	0.167	23.38
440	12	17.9	0.49	0.644	0.086	13.42
0	22	22	0	0.595	0	0

The performance curve of the existing pump is shown in the figure1. It is observed that the maximum efficiency obtained is 23.38% as shown in table2.The total head is 16 m and the discharge is 1 LPS at the best efficiency point.

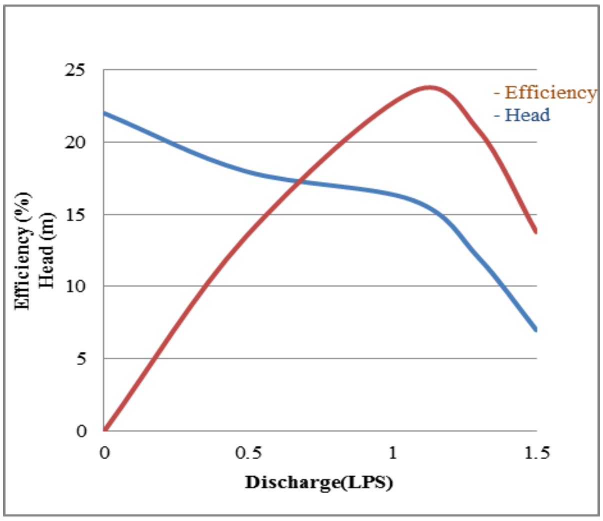

Fig 1.Performance curve of existing pump

IV. CFD ANALYSIS

The existing impeller parameters are shown in table1.The centrifugal pump is modeled using NX CAD software. The Centrifugal pump assembly file is imported using IGES format into Solidworks Flow Simulation Package. The impeller CAD model and imported pump model are shown in figures 2 and 3. Rotation feature is applied in the Z-axis global co-ordinate system. Water is given as the rotating fluid. CFD analysis is carried out for the best efficiency point. The boundary conditions given are:1.Static pressure at inlet: 42693.12 Pa 2.Outlet volume flow: 1 LPS and 3.Real Wall (Stator).

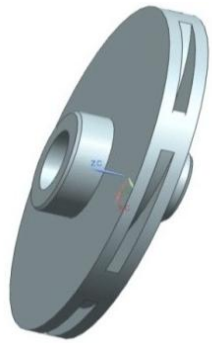

Fig 2. Impeller CAD Model

Surface goals are provided at appropriate faces to determine the mass flow rate, head developed, torque requirement and efficiency. The efficiency is specified as a equation goal shown below in equation (1).

$$\eta = \frac{(Poutlet - Pinlet) * Q}{T * \omega} \quad (1)$$

Where, *Pinlet* is the static pressure at the pump's inlet (Pa), *Poutlet* is the static pressure at the impeller's outlet (Pa), Q is the volume flow rate (m^3/s), ω is the impeller rotation angular velocity (rad/s), and T is the impeller torque (N·m).

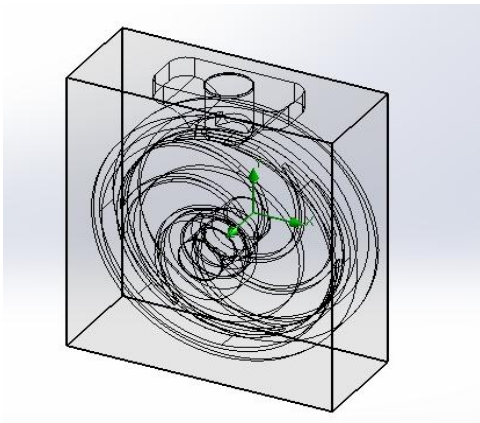

Fig 3.Imported pump model

Table 3.CFD Solver Results

Goal Name	Unit	Value
Mass Flow Rate at inlet	[kg/s]	1.086
Mass Flow Rate at outlet	[kg/s]	-1.086
Pressure at inlet	[Pa]	46499.346
Pressure at impeller outlet	[Pa]	159401.976
Torque (Z) on impeller	[N*m]	1.525
Efficiency	[%]	24.536

The efficiency obtained from CFD analysis is 24.53% as shown in the table 3.The deviation from the experimental value is 1.15%.CFD always predicts higher since it does not include the volumetric efficiency. The pressure and velocity distribution of the existing impeller are shown in figures 4 and 5.

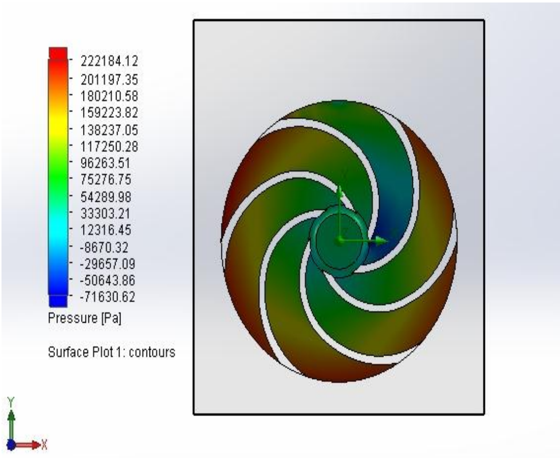

Fig 4.Pressure distribution in existing impeller

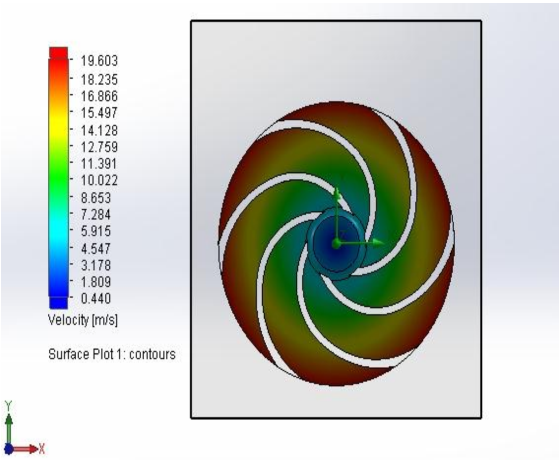

Fig 5.Velocity distribution in existing impeller

V. IMPELLER DESIGN MODIFICATIONS

The existing impeller is modified and CFD analysis is made to determine the efficiency. The number of vanes, inlet blade angle and outlet blade angle of the existing design shown in table 4 are modified and variation of head developed, efficiency and torque requirement are noted.

Table 4. Impeller Parameters Considered For Modification

S.No	PARAMETER	SIZE
1	Number of blades ,Z	6
2	Inlet blade angle , β1	34.8°
3	Outlet blade angle, β2	16°

The number of vanes in the present model is 6.The number of vanes is changed to 4, 5 and 7. The outlet blade angle of the existing impeller is 16^0.The outlet blade angle is changed to 14^0 and 18^0 using circular arc method. The inlet blade angle of the existing impeller is 34.8^0.The inlet blade angle is changed to 32.8^0, 36.8^0, 38.8^0 and 40.8^0 using circular arc method. The performance of the modified impeller is obtained for three different boundary conditions. Static pressure of 42693.12 Pa is given as boundary condition at inlet. Volume flow is given as boundary condition at outlet. CFD analysis is carried out for three different volume flow rates 0.5 LPS,1 LPS and 1.3 LPS. CFD analysis is made for each model to determine the optimized parameters.

VI. RESULTS AND DISCUSSIONS

In this chapter, the variation of head, torque and efficiency for various impeller modifications are discussed.

A. CFD Results Obtained by Varying Number of Blades of the Impeller

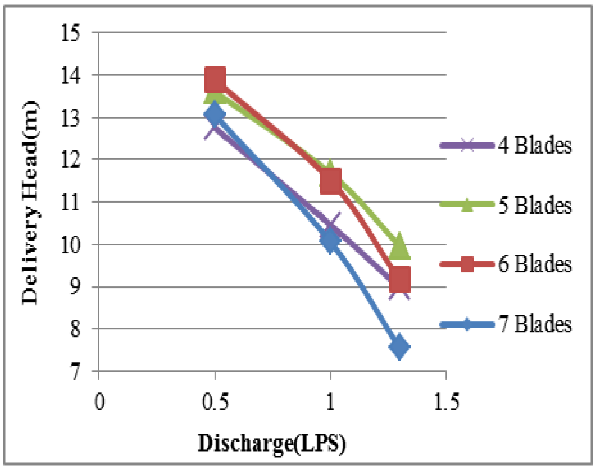

Fig 6. Discharge vs Head

Selection of number of blades in impellers is very important. As the number of blades decreases, angle of divergence increases and secondary losses increases. For 4 blades, head reduces due to flow separation and circulatory

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 04, April-2015

losses .As the number of blades increases, frictional losses increases. The power rating also increases due to increased blade loading. For 7 blades, the head reduces due to frictional losses .From the CFD result, the optimum number of blades is 5. The performance curves are shown in figures 6, 7 and 8.

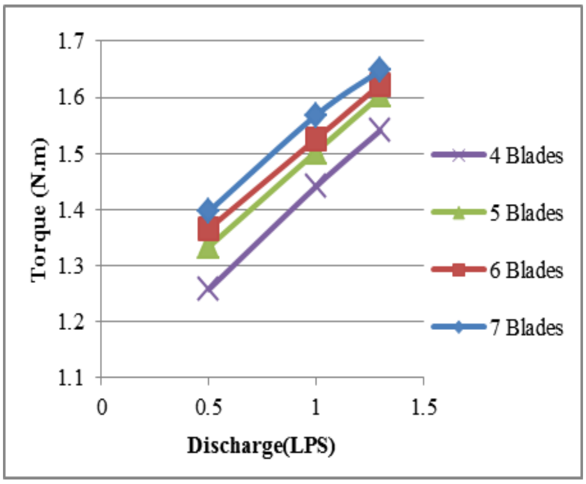

Fig 7. Discharge vs Torque

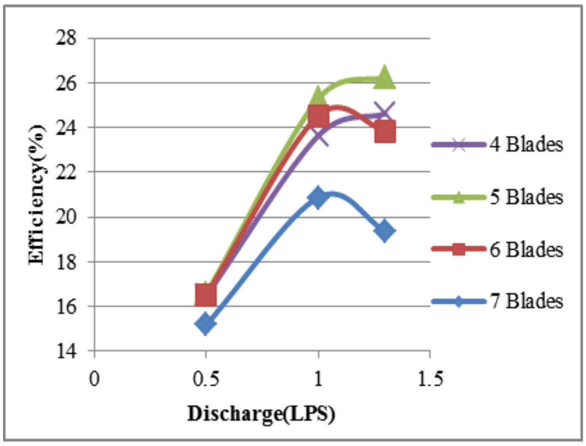

Fig 8. Discharge vs Efficiency

B. CFD Results Obtained by Varying Outlet Blade Angle of the Impeller

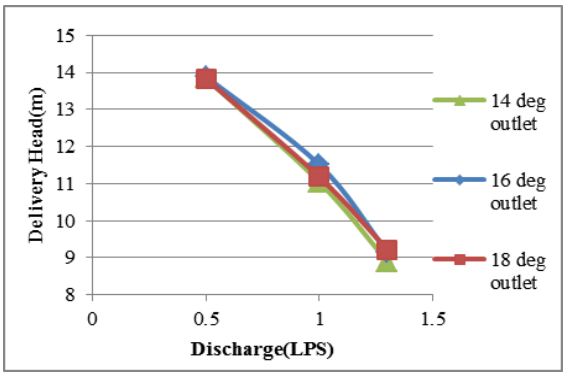

Fig 9. Discharge vs Head

The head developed depends on β_2.For 14^o outlet angle , head reduces due to decrease of β_2. As β_2 increases, the angle of divergence increases which induces flow separation losses. The power rating also increases with increase of β_2.For 18^{o}outlet angle, the torque requirement increases. From the CFD result, the optimum blade angle at outlet is 16^o. The performance curves are shown in figures 9, 10 and 11.

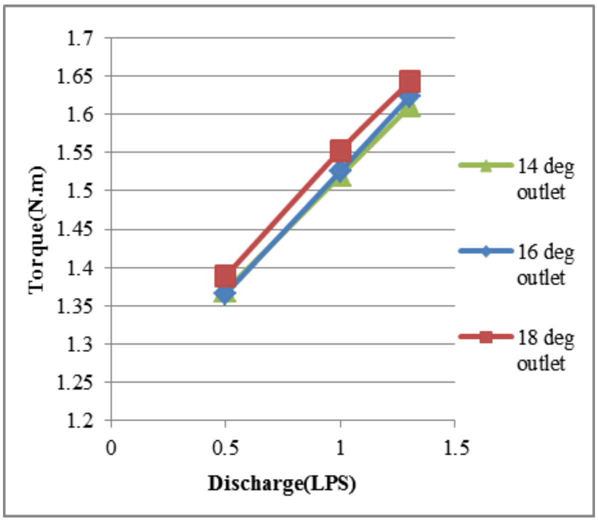

Fig 10. Discharge vs Torque

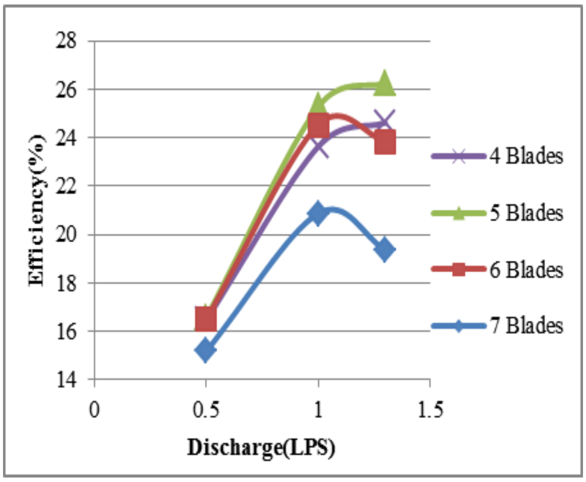

Fig 11. Discharge vs Efficiency

C. CFD Results Obtained by Varying Inlet Blade Angle of the Impeller

The inlet blade angle β_1 is determined as $\beta_1 = \beta + \delta$ where $\beta = \tanh\left(\frac{C_{m1}}{U_1}\right)$ and δ-angle of attack. The inlet blade angle is increased by '$\delta = 3^0$ to 10^0' in order to reduce shock losses at inlet. For the existing design, β is 31.8^0 and δ is 3^0.The angle of attack is changed to 1^0,3^0,5^0,7^0 and 9^0.The head developed increases with increase of angle of attack upto 38.8^0.Beyond that the head developed reduces due to inlet whirl. It is found that higher efficiency is attained when α_1 is slightly lower than 90^0 i.e., $85^0 < \alpha < 90^0$. For 38.8^0 inlet blade angle, absolute angle α_1 is 85^0.From the CFD result, optimum blade angle at inlet is 38.8^0. The performance curves are shown in figures 12, 13 and 14.

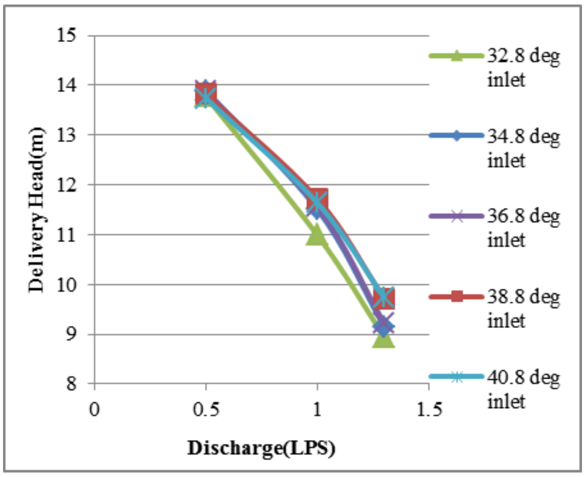

Fig 12. Discharge vs Head

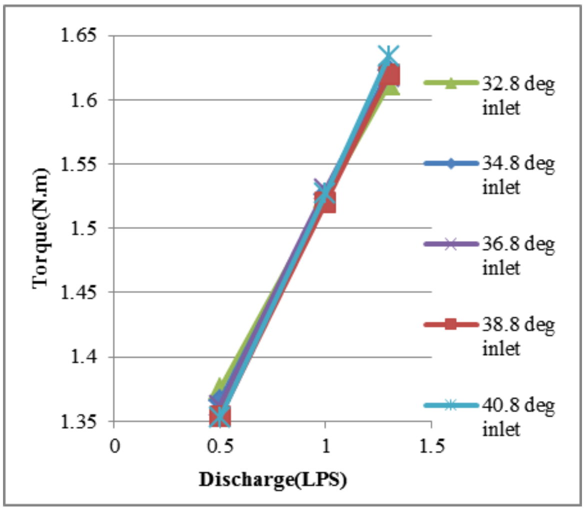

Fig 13. Discharge vs Torque

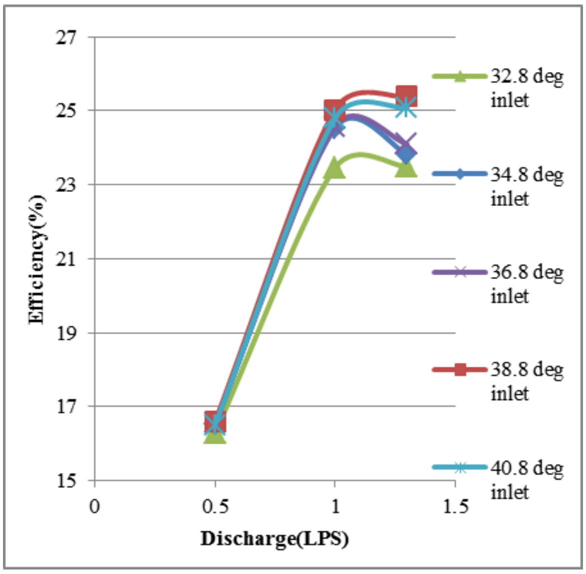

Fig 14. Discharge vs Efficiency

VII. CONCLUSIONS

Based on the CFD results and theoretical justifications, the optimum design parameters of the existing impeller are 5 blades , 16^0 outlet blade angle and 38.8^0 inlet blade angle. The optimized impeller section view is shown in figure 15.

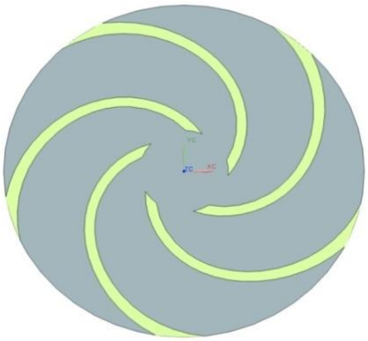

Fig 15. Optimized Impeller Section View

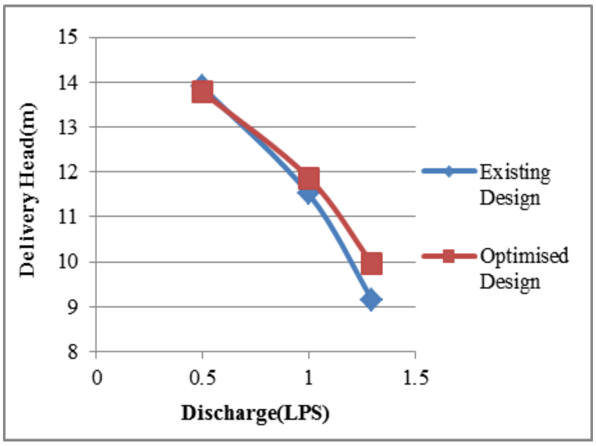

Fig 16.Discharge vs Head

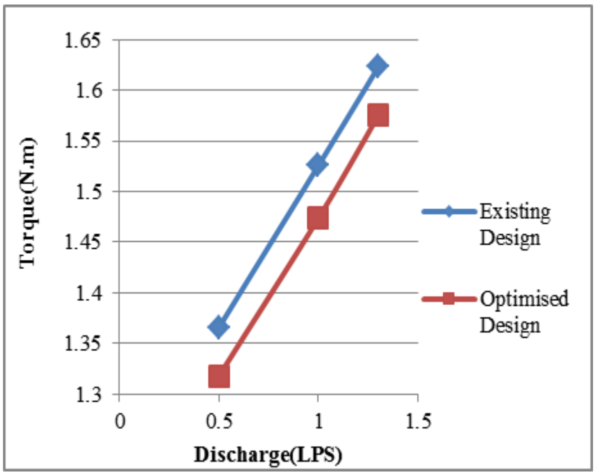

Fig 17.Discharge vs Torque

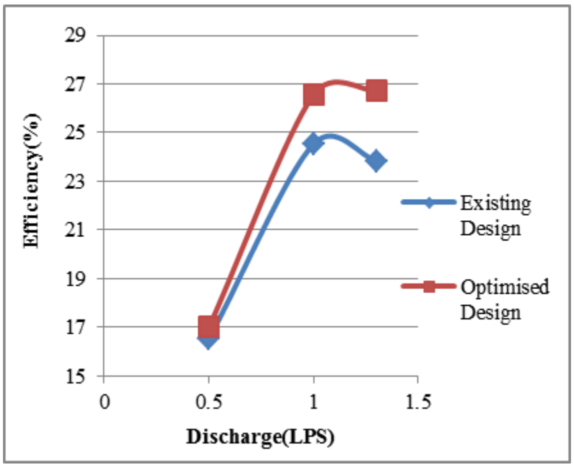

Fig 18.Discharge vs Efficiency

The impeller with optimum design parameters is modeled and CFD analysis is carried out for three different volume flow rates 0.5 LPS, 1 LPS and 1.3 LPS and the performance is obtained. The performance curves and pressure distribution for optimized impeller are shown in figures 16, 17, 18 and 19.For the optimized impeller design, the torque requirement is decreased by 3.4%. The head developed is increased by 3.22% and the efficiency is increased by 8.28% at the best efficiency point.

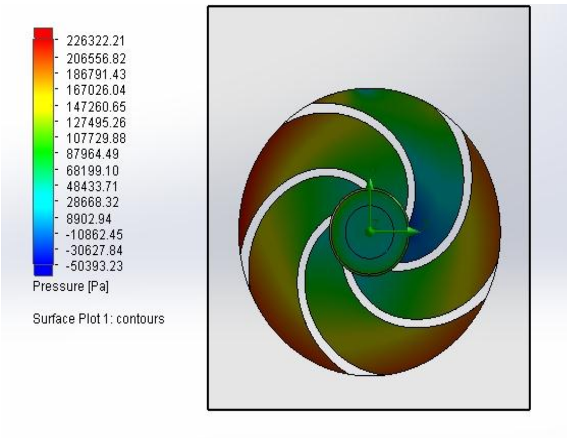

Fig 19.Pressure distribution in the optimized impeller

Thus CFD analysis is an effective tool to calculate quickly and inexpensively the effect of design and operating parameter of pump. By properly designing pump impeller, the efficiency of pump can be improved.

ACKNOWLEDGMENT

The Authors would like to thank SKS Accessories, Coimbatore and PSG College of Technology for successful completion of the project work.

REFERENCES

[1] Dr.K.M.Srinivasan," Rotodynamic Pumps (Centrifugal And Axial)",New Age International Publishers,2008.
[2] Val S.Lobanoff, Robert R.Ross,"Centrifugal Pumps-Design And Application", Second Edition, Gulf Publishing Company, Texas, USA.
[3] Yunus A.Cengel, John M.Cimbala,"Fluid Mechanics-Fundamentals And Applications", Mcgraw Hill Higher Education,2006.
[4] C.P. Kothandaraman, R.Rudramoorthy, "Fluid Mechanics And Machinery", New Age International Publishers,2007.
[5] R.Ragoth Singh, M. Nataraj," Design And Analysis Of Pump Impeller Using SWFS", ISSN 1 746-7233, England, UK World Journal Of Modelling And Simulation Vol. 10 (2014) No. 2, Pp. 152-160P.
[6] Gundale V.A., Joshi G.R. "A Simplified 3d Model Approach In Constructing The Plain Vane Profile Of A Radial Type Submersible Pump Impeller", Research Journal Of Engineering Sciences ISSN 2278 – 9472 Vol. 2(7), 33-37, July (2013).
[7] Manivannan," Computational Fluid Dynamics Analysis Of A Mixed Flow Pump Impeller", International Journal Of Engineering, Science And Technology, Vol.2,No.6,2010,Pp.200-206 .
[8] Hudson Daniel Raj.E Kalaimani.T "Investigation Of Key Impeller Parameters Of Centrifugal Pump Using CFD " International Journal Of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 2 Issue 10, October – 2013.
[9] Amit H. Bhuptani, Prof. Ravi K. Patel, K.M. Bhuptani," Design And Analysis of Centriugal Pump", Journal of Information, Knowledge And Research In Mechanical Engineering.
[10] Sujoy Chakraborty A, K.M. Pandey , A.Bidesh Roya ,"Numerical Analysis On Effects Of Blade Number Variation On Performance of Centrifugal Pumps With Various Rotational Speeds", International Journal Of Current Engineering And Technology ISSN 2277 – 4106.