International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 10, October - 2013

Investigation of Key Impeller Parameters of Centrifugal Pump Using CFD

Hudson Daniel Raj.E
Asst. Professor
Dept. of Mechanical Engineering
Sri Krishna College of Technology
Coimbatore, India

Kalaimani.T
Asst. Professor
Dept. of Mechanical Engineering
Sri Krishna College of Technology
Coimbatore, India

***Abstract*— The impeller of a centrifugal pump plays an important role in converting the input driver energy into kinetic energy. The impeller is a complex structure and moreover the conventional trial and error method is cost consuming to experiment with. CFD analysis is a solution to this problem.**

This paper presents the numerical investigation of the effects of the key design parameters, including blade number, inlet blade angle, trimmed impeller profile, and impeller diameter, have on the steady state liquid flow in a three-dimensional centrifugal pump. Initially the impeller model is geometrically constructed based on certain performance modified parameters. The model is then meshed and CFD analysis is carried out. The inner flow fields, pressure and velocity distribution are predicted. Results obtained from the analysis are then compared with the actual results and inferences are made.

***Index Terms*—Circumferential velocity, over filing**

I. INTRODUCTION

A centrifugal pump is a non-positive displacement pump that imparts energy to a liquid. A centrifugal pump converts driver energy to kinetic energy in a liquid by accelerating it to the outer rim of a revolving device known as an impeller. The energy created is kinetic energy. The amount of energy given to the liquid corresponds to the velocity at the edge or vane tip of the impeller. Impeller is the rotating part of a centrifugal pump which imparts velocity to the system. The blades of the rotating impeller transfer energy to the fluid thereby increasing velocity. The fluid is sucked into the impeller at the impeller eye and flows through the impeller channels formed by the blades between the shroud and hub. The design of the impeller depends on the requirements for pressure, flow and application. The impeller is the primary component determining the pump performance. Pumps variants are often created only by modifying the impeller. The impeller accounts for nearly 17.5% of losses in a pump.

II. OBJECTIVE

The impeller is the major part of a pump that can be modified to improve its performance. The physical testing of various pump models is a laborious process. The trial and error method is time consuming and has many disadvantages. CAD and CFD analysis are useful tools that reduce considerable time that is usually lost in physical testing. CAD aids in constructing the geometrical profile on a computer and CFD analysis provides the necessary virtual simulation without using any physical effort. Hence an impeller with the performance modified specification is constructed and the flow through it is virtually analyzed using CFD tools to determine the effect of the parameters on the performance

The impeller of a centrifugal pump is the most important component which increases the kinetic energy of the incoming liquid. The impeller is the rotating part which consists of blades which aid in increasing the velocity of the liquid. Impeller is a complex asymmetric component which makes it difficult and expensive to modify and experiment physically.

Computational Fluid Dynamics (CFD) is a tool which aids in conducting or simulating flow through a pump virtually using computer software. An attempt has been made to improve the performance of a typical centrifugal pump obtained from a leading pump manufacturing company. Four parameters namely number of blades, blade inlet angle, outer diameter of impeller and trim profile of blades are identified to be responsible for improving the performance of the centrifugal pump. These four parameters are set in three levels to form a L_9 orthogonal array to design nine experiments by Taguchi method.

Based on the experiment details, the impeller is modelled using CAD software. The modelled impellers are then analyzed using Solidworks Flow Simulation, a CFD software package. Discharge is obtained for these models and optimization is performed by Taguchi method to obtain signal-noise ratio. The best possible combination of parameters is determined and is implemented in an impeller model and CFD analysis is carried out to verify the obtained results.

III. METHODOLOGY

An impeller of a 5 hp centrifugal pump is studied to predict the pressure, velocity distributions and efficiency by altering certain key parameters in the impeller. Initially the standard impeller was modelled by CATIA V5 and analyzed using Flow Simulation software to identify the deviation of CFD results from the standard practical value. The impeller is then studied by modifying certain parameters.

A. EXISTING PUMP DATA

An impeller of a commercial 5 hp domestic submersible centrifugal pump obtained from a leading pump manufacturing company is studied by varying its parameters. The specifications of the centrifugal pump considered is shown in Table 1 below

Table 1 Existing Pump Data

Pump Std	IS:14220
Motor Rating	3.7kw / 5 hp
Speed	2780 rpm
Total head	24 m
Discharge	8 lps
Delivery size	50 mm
Voltage	380
Overall η	43.5%

The existing specifications of the impeller under study is given in Table 2 below

Table 2 Impeller dimensions

Number of blades	6
Eye Diameter	58 mm
Type	Shrouded (enclosed)
Outer Diameter	170 mm
Thickness	5 mm

B. PARAMETERS

There are many parameters that influence the efficient functioning of a centrifugal pump. The parameters are generally modified to improve the performance. By varying the parameters based on proven research findings, the performance of the system can be improved significantly. The key parameters that influence the impeller are the number of blades, inlet angle, outer diameter and trimmed exit vanes. [3]

- **OUTER DIAMETER**

As per BEE recommendations, impeller diameter reductions greater than 5% to 10% of the maximum will increase the NPSHR (net positive suction head required). The outer diameter of the impeller determines the circumferential velocity of the exit vanes.

The outer diameter is reduced by 5%, 7.5% and 10% of the original dimension to form three levels in L9 orthogonal array

- **INLET VANE ANGLE**

The initial kinetic energy developed by the fluid depends on the inlet vane angle of the impeller. The velocity triangles are used to calculate the inlet vane angle.

From inlet velocity triangle,

$$\tan \theta = v_{f1} / u_1$$

Inlet angles of 25º,30º and 35º are set in three levels of the orthogonal L9 array in this work.

- **NUMBER OF BLADES**

The selection of number of blades is crucial because, selecting a higher or lower number might result in considerable performance reduction. The number of blades is given by

$$Z = [\ 6.5 * (D_2+D_1 / D_2 - D_1) * \sin(\beta_1 + \beta_2)\ ] / 2$$

In this work, the impeller is modelled with 6, 7 and 8 blades in three levels in L9 orthogonal array. The blade number should not be increased or decreased drastically as it will have an adverse effect on the performance of the impeller

- **TRIMMED EXIT VANES**

The concept of underfiling the exit vanes at the outlet is extensively used in the industrial field to improve the efficiency. Under filing will increase the pump capacity, especially for large circulating pumps.

The exit angle of the fluid will change resulting in a higher head at design flow, but no change will occur in shut off head. Due to the reductions in the region of the fluid exiting the vanes the efficiency of the pump should improve slightly. The underfiling process should be performed accurately. The smaller the size of the pump, the larger will be the effect. The technique of under filing is critical. Sharp corners, where the vane joins the shroud, can initiate cracks and eventually lead to impeller failure. At least 0.0125 inches (3 mm) of vane tip thickness must remain after the under filing.

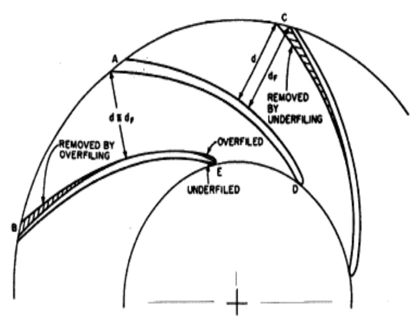


Fig 1. Impeller trimming.

Over filing is the process of trimming the leading edge of the impeller blades. The net area and head remains unchanged which results in smoother blades. Over filing increases the efficiency of the pump.[9]

In this work, three profiles namely underfiled, overfiled and normal blades are employed to improve the performance of the pump.

Based on the modifications on impeller, 4 parameters with 3 levels are used to frame a L9 orthogonal array based on Taguchi method.

Table 3. L_9 ORTHOGONAL ARRAY EXPERIMENTS

Expt. No	No. of blades	Outer dia D2(mm)	Inlet blade angle(deg)	Impeller trim
1	6	163.4	25	Over filing
2	6	159.1	30	Normal
3	6	154.8	35	Underfiling
4	7	163.4	30	Underfiling
5	7	159.1	35	Over filing
6	7	154.8	25	Normal
7	8	163.4	35	Normal
8	8	159.1	25	Underfiling
9	8	154.8	30	Over filing

C. MODEL CONSTRUCTION

Computer Aided Drawing is a valuable tool which aids in the construction of a geometric model in a computer. The impeller model for the actual specification and the nine models based on L9 orthogonal array were created in CATIA V5 software.

Initially the actual model of impeller is constructed virtually using CATIA V5 software. The actual model is tested to find out any deviations in the actual output with the simulated output. The deviation is an indication of the accuracy of the CFD results. If the deviation is less, the results obtained by CFD simulation are accurate and the corresponding results of nine experiment models can be validated.

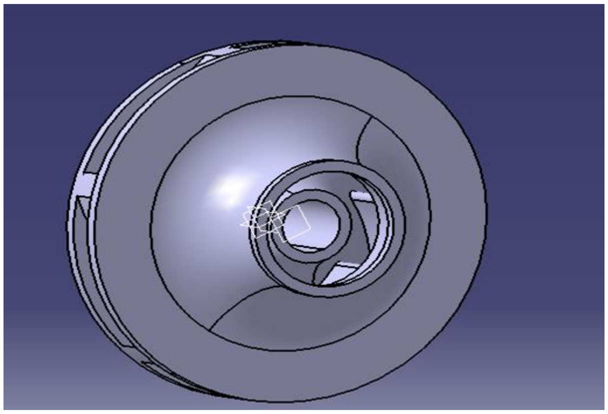


Fig 2. Impeller model in CATIA V5.

D. CFD ANALYSIS

The geometric model and the nine experiment models based on L9 orthogonal array created with the help of CATIA V5 software were exported to the SolidWorks Flow Simulation software for carrying out the CFD analysis. CFD analysis was carried out by specifying the input data

The program enables goal-based analysis, so before running the analysis, the goals should be chosen, which, include pressure drop between the inlet and outlet to determine head, volume flow rate at the inlet to determine the volume rate, and torque on the impeller to determine the hydraulic efficiency.

SolidWorks Flow Simulation automatically creates the mesh and iterates to a converged solution. The analysis provides information on the head and volume flow rate generated by the pump at 2780 rpm, and determines the discharge of the pump. From that information and the needs of anticipated pump users, the user can conduct as many “what if” studies as desired, analyze them, redesign the impeller blades, and select the best impeller blade design for maximum efficiency. The program enables goal-based analysis for the user to choose the required goals before running the analysis.

IV. RESULTS AND DISCUSSION

The results of the actual standard impeller model and the nine models designed by L9 orthogonal array were analyzed virtually by Solidworks Flow Simulation, a CFD Software package. The optimized combination of parameters was obtained and the combination was implemented in a model and analyzed virtually to validate the results. The results obtained are discussed in this chapter.

In Solidworks Flow Simulation software, results are obtained in the Goal Plot dialog box. All added goals together with their current values are listed at the top part of the box, as well as the current progress towards completion given as a percentage. The progress value is only an estimate and generally (but not necessarily) increases with time.

Convergence is an iterative process. The discretization of the flow field imposes conditions on each parameter and each parameter cannot reach an absolutely stable value but will oscillate near this value from iteration to iteration. When SolidWorks Flow Simulation analyzes the goal's convergence, it calculates the goal's dispersion defined as the difference between the goal's maximum and minimum values over the analysis interval reckoned from the last iteration and compares this dispersion with the goal's convergence criterion dispersion, either specified by you or automatically determined by SolidWorks Flow Simulation. Once the oscillations are less than the convergence criterion the goal becomes converged.

ISSN: 2278-0181
Vol. 2 Issue 10, October - 2013

The impeller modelled with the standard dimensions was analyzed using the Flow Simulation software to find the relative error percentage and deviation of the CFD results from the standard impeller. Numerical simulations were then carried out on 9 models based on L9 orthogonal array to predict the performance of the impeller and to determine the flow rate for the given input data. Successive iterations were performed by the Flow simulation solver to obtain the flow rate, pressure distribution, peripheral velocity distribution and torque of the impeller.

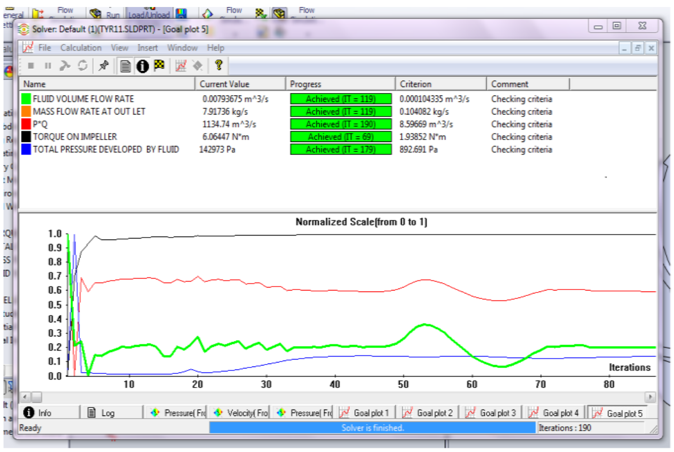


Fig 3. CFD Solver results for impeller model

The volume flow rate of the actual model of the impeller obtained from the CFD analysis is 0.00793 m^3/s or 7.93 lps. The volume flow rate of the impeller as given by the pump manufacturing company is 0.008 m3/s or 8 litres/sec. The relative error percentage was found to be 0.875%. This result indicates that the output obtained from the CFD analysis is quite closer to the results obtained from the pump manufacturing company.

The CFD analysis carried out on the 9 experiment models resulted in better volume flow rate output for all the experiment models. The results for the L9 orthogonal array are given in Table 4.

Table 4. L9 orthogonal array results

Expt . No	No. of blades	Outer dia D_2(mm)	Inlet blade angle(deg)	Impeller trim	VOLUME FLOW RATEm^3/s
1	6	163.4	25	Over filing	0.008842
2	6	159.1	30	Normal	0.008512
3	6	154.8	35	Underfiling	0.008311
4	7	163.4	30	Underfiling	0.008362
5	7	159.1	35	Over filing	0.008841
6	7	154.8	25	Normal	0.008411
7	8	163.4	35	Normal	0.01005
8	8	159.1	25	Underfiling	0.0080
9	8	154.8	30	Over filing	0.01004

Minitab software was used to obtain the rank of the parameters. The Larger the better S-N ratio was used. The trim profile and number of blades proved to be the major parameters as shown in Figure 4

Session

Larger is better

Level	blade number	OUTER DIA	INLET ANGLE	TRIM PROFILE
1	-41.36	-41.03	-41.50	-40.70
2	-41.38	-41.47	-40.97	-40.95
3	-40.62	-40.86	-40.88	-41.70
Delta	0.76	0.61	0.63	1.00
Rank	2	4	3	1

Fig 4. Rank of parameters in Minitab software

The signal-noise ratios for each for the individual experiment models are shown in Figure 5

Worksheet 2 ***

↓	C1	C2	C3	C4	C5	C6	C7	C8
	blade number	OUTER DIA	INLET ANGLE	TRIM PROFILE	DISCHARGE	SNRA1		
1	6	163.4	25	1	0.008842	-41.0690		
2	6	159.1	30	2	0.008512	-41.3994		
3	6	154.8	35	3	0.008311	-41.6069		
4	7	163.4	30	3	0.008362	-41.5538		
5	7	159.1	35	1	0.008841	-41.0700		
6	7	154.8	25	2	0.008411	-41.5030		
7	8	163.4	35	2	0.010050	-39.9567		
8	8	159.1	25	3	0.008000	-41.9382		
9	8	154.8	30	1	0.010040	-39.9653		
10								
11								

Fig 5. Signal-noise ratios for discharge

The main-effects plot for Signal-noise ratios was performed to obtain the most significant level for each of the 4 parameters of the impeller.

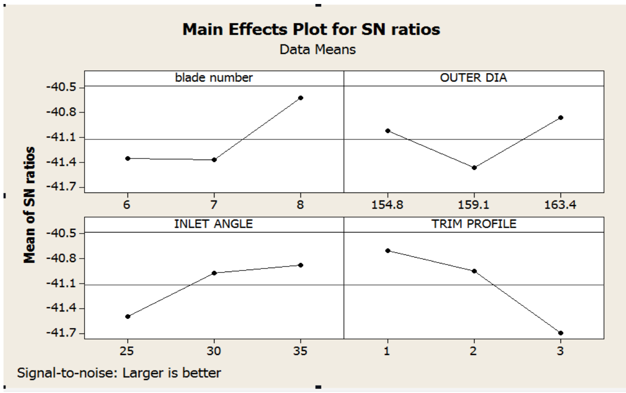


Fig 6. Main-effects plot for SN ratios

The ANOVA for discharge using Sequential SS for tests was performed. The value of R^2 was found to be 92.72% validating the chosen parameters are significant parameters

Analysis of Variance for DISCHARGE, using Sequential SS for Tests

Source	DF	Seq SS	Adj SS	Seq MS	F	P
blade number	1	0.0000010	0.0000010	0.0000010	3.26	0.145
OUTER DIA	1	0.0000005	0.0000005	0.0000005	0.13	0.733
INLET ANGLE	1	0.0000006	0.0000006	0.0000006	2.11	0.220
TRIM PROFILE	1	0.0000016	0.0000016	0.0000016	5.16	0.086
Error	4	0.0000007	0.0000012	0.0000003		
Total	8	0.0000044				

S = 0.000548096   R-Sq = 92.72%   R-Sq(adj) = 85.45%

Fig 7. ANOVA for discharge of models

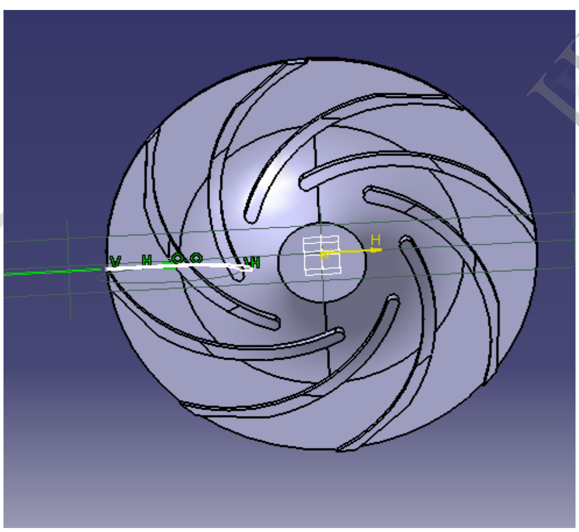


Fig 8. Optimized model in CATIA V5 software

The optimized combination of parameters obtained is used in modeling an impeller using CATIA V5 software. The model is then analyzed using Solidworks Flow Simulation software to obtain the flow rate to validate the results of L9 Orthogonal array by Taguchi method. The optimized model constructed in CATIA V5 software is shown in Fig 4.34.

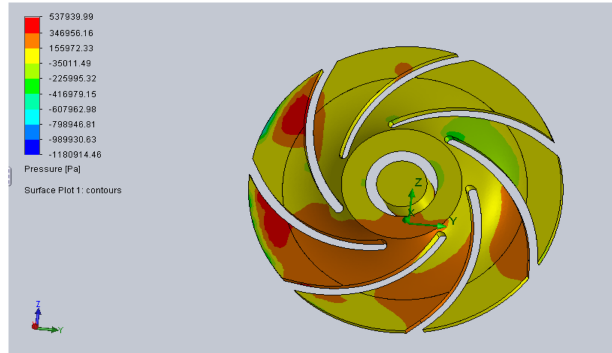


Fig 9. Pressure distribution of optimized model

The pressure contours and velocity distribution contours for the optimized combination model are shown in Figures 9&10

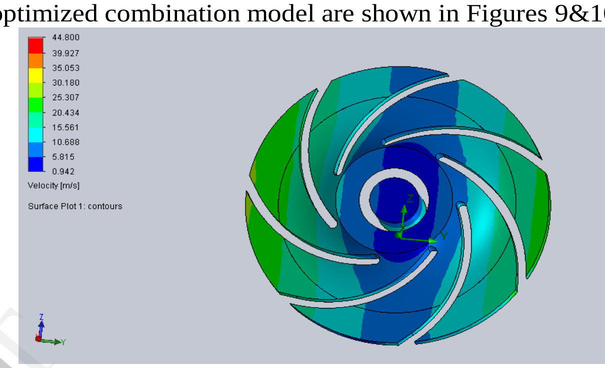
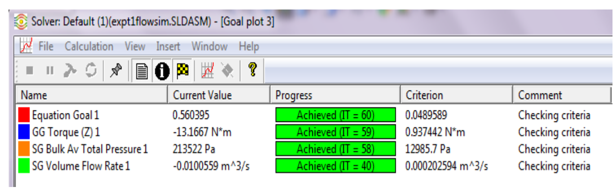


Fig 10. Velocity distribution of optimized model


Solver: Default (1)(expt1flowsim.SLDASM) - [Goal plot 3]

File Calculation View Insert Window Help

Name	Current Value	Progress	Criterion	Comment
Equation Goal 1	0.560395	Achieved (IT = 60)	0.0489589	Checking criteria
GG Torque (Z) 1	-13.1667 N*m	Achieved (IT = 59)	0.937442 N*m	Checking criteria
SG Bulk Av Total Pressure 1	213522 Pa	Achieved (IT = 58)	12985.7 Pa	Checking criteria
SG Volume Flow Rate 1	-0.0100559 m^3/s	Achieved (IT = 40)	0.000202594 m^3/s	Checking criteria

Fig 11. CFD Solver results of optimized model

The CFD solver consumed 57 iterations for the goals to be converged. The volume flow rate of the modelled impeller for the best optimized combination is obtained as 0.010599 m^3/s from the CFD analysis. The volume flow rate obtained from the analysis is higher than the value of the flow rate of the standard impeller given by the pump company. The torque obtained is very high at 20.7687 Nm. The total pressure of the impeller was found to be at 282919 Pa. The pressure distribution contours indicate that the pressure is unevenly distributed throughout the cross section of the impeller. High pressure regions are found near the circumference of the impeller. The velocity contours indicate an increase in velocity from the eye to the exit of the impeller. The velocities near the exit area of the vanes are greater than 25 m/s.

V. CONCLUSION

In this work, an impeller model of a centrifugal pump was created using CATIA V5 software from the data provided by a leading pump manufacturer and then analyzed in Solidworks Flow Simulation software. The relative error percentage was determined to find the deviation of the CFD results from the actual standard results.

An attempt was made to improve the performance of the impeller by modifying certain key parameters like blade number, outer diameter of impeller, inlet blade angle and trim profile. The experiments were designed based on L9 orthogonal array with three levels of four parameters. The experiment models were constructed using CATIA V5 software and the models were analyzed in Solidworks Flow Simulation, a CFD software. The results obtained from the experiments showed a reasonable increase in volume flow rate output. The results obtained were optimized by Taguchi method to identify a suitable combination of parameters.

Another model was constructed in CATIA V5 software based on the best combination of optimized parameters. The model was then analyzed virtually to validate the results obtained by optimization and they showed good conformity.

VI. REFERENCES

[1] Khalid. S. Rababa (2011), "The Effect of Blades number and shape on the Operating Characteristics of Groundwater Centrifugal Pumps", European Journal of Scientific Research, vol 2, No.6, pp 243-251.

[2] Ll Gang Wen(2011), "Inverse Design of Impeller Blade of Centrifugal Pump with a Singularity method", Jordan Journal of Mechanical and Industrial Engineering, vol 9, No.6, pp 119-128.

[3] Manivannan.A(2010), "Computational Fluid Dynamics of a Mixed Flow Pump Impeller", International Journal of Engineering, Science and Technology, vol 3, No.2, pp 200-208.

[4] Ton Shigemitsu, Fukitomi Junichiro, Kensuke Kaji and Wada Takashi (2012), "Unsteady Internal Flow Conditions of Mini Centrifugal Pump with Splitter Blades", Proceedings of 4th Asian Joint Workshop on Thermophysics and Fluid Science.

[5] Subramaniam.L and Sendilvelan.S (2012), "Modal Analysis of a Centrifugal Pump Impeller", International journal of Engineering, Science and Technology, vol 9, No.2, pp5-14.

[6] BPMA, "The European System Guide", First Edition, 2011.

[7] Jacobsen Brix Christian, "The Centrifugal Pump", Grundfos, First Edition.

[8] Bureau of Energy Efficiency, "Pumps and Pumping System", First Edition 2002.

[9] Lev Nelik, "Centrifugal and Rotary Pumps-Fundamentals with Applications", CRC Press.

[10] Val.S.lobanoff, Robert.R.Ross, "Centrifugal Pumps,-Designs and Applications", Jaico Publishing House, 2nd Edition.

[11] Igor.J.Karassik and Roy Carter, "Centrifugal Pumps-Selection, Operation and Maintenance",F.W.Dodge Corporation.