Computer-Aided Model Validation and Performance Improvement of Submersible Pump Through Flow Simulation

Vinoth Rengaraj, Madheswaran Subramaniyan, Omkar Singh,
Navanendra Singh and Abhilasha Singh

April 20, 2023

COMPUTER-AIDED MODEL VALIDATION AND PERFORMANCE IMPROVEMENT OF SUBMERSIBLE PUMP THROUGH FLOW SIMULATION

ABSTRACT

This paper contains validation of actual response values of the pump and simulated experiment values and improves the performance of the submersible pump. The performance of a submersible pump mainly depends upon the pump impeller and casing. The design data and actual functional response values of a submersible pump model were provided by a leading pump manufacturing industry. Based on this, the pump impeller and casing were modeled using the parametric design modeling feature in Solidworks 2013. Computational Fluid Dynamics (CFD) analysis is adopted for predicting pump performance at various mass-flow rates, pressure head, and pump efficiency. The fluid flow simulation of the functional response values of the pump is carried out in the ANSYS-CFX tool. The actual response values of the pump and simulated experiment values are compared and the details are presented. The impeller design parameters impeller diameter, impeller vane width, number of vanes, impeller inlet vane angle and outlet vane angle are modified for the optimum conditions of the pump and the optimum ranges are selected from the literature review. The optimum ranges for various combinations of the pump design parameters are modeled and analyzed. The optimum and better performance of the submersible pump is predicted.

KEYWORDS

Submersible pump, Impeller Design, CFD analysis, Solidworks, ANSYS-CFX

INTRODUCTION

The submersible pump is a device to raise the liquids from a lower level to a higher level due to the pressure created in the pump with the help of centrifugal action. A wide variety of submersible pumps are used in many applications like drain wells, residential and commercial purposes. The pump performance prediction is a complicated task due to the pump the frictional losses. The impeller is the most influential part of the pump performance, and the impeller design parameters are the number of vanes, vane inlet and outlet angles, vane profile direction, impeller outlet radius, and thickness.

The pump's actual response values of pressure head, power, and efficiency are measured by experimentally, but the fluid flow inside the pump cannot be predicted experimentally, so CFD analysis is used to be carried out to predict the pump performance.

S.C. Chaudhari et al. [1] have studied increasing the outlet blade angle and decreasing the inlet blade angle improves the performance of the pump that is simulated in this study. Virajit A.Gundale et al. [2] have carried out the performance of the pump pressure head is improved by increasing the number of blades at a certain level because it will affect the frictional losses also. Igor Tverdokhleb et al. [3] have discussed reducing the external radius of the impeller D_2, and the ratio D_{GV}/D_2 increases the performance of the pump. A.Manivannan [4] has studied the changes in the inlet vane angle did not change the efficiency as much as the changes in the outlet angle. S. Rajendran et al. [5] have discussed increasing the outlet blade angle from 20^0 to 45^0 is an increase of more than 7% of the total head of the pump. S. Chakraborty et al. [6] have studied the impeller outlet diameter, the blade angle, and the blade numbers were found to be the most critical parameters that affected the performance of centrifugal pumps. Massinissa Djerroud et al. [7] performed a numerical analysis of key design parameters, the blade height, the blade number, the outlet blade angle, the blade width, and the impeller diameter, enhancing the centrifugal pump performance. Adnan Ozturk et al. [8] have discussed decreasing the radial gap between impeller blades and diffuser vanes to increase the pressure fluctuations in the vane suction side.

PUMP SPECIFICATIONS

The submersible pump model design data obtained from a leading pump manufacturing company. The submersible pump is analyzed to predict the pressure distributions and velocity distributions of simulated results compared with the actual value given by the company. The following details are used to conduct the analysis. The pump performance details are shown in Table 1.

Table 1: Pump specification details

Rated speed	2800 rpm
Total head	27.85 m
Discharge	4.49 lps
Pump efficiency	43.14 %
Overall efficiency	32.6 %

Impeller design

The impeller of a submersible pump plays an essential role in converting the driver energy into kinetic energy when it rotates inside the submersible pump. The impeller is responsible for directing the flow of water to the stationary casing and forms the central part of the submersible pump.

Table 2: Impeller design parameters values

Impeller design parameters	Values
Impeller outer diameter	163 mm
Impeller inner diameter	41 mm
Eye diameter	35 mm
Shaft diameter	8.93 mm
Inlet vane angle	18.54 ^{0}
Outlet vane angle	24.64 ^{0}
Number of vanes	3
Vane width	5.5 mm

The above-mentioned impeller design parameter values are used to design the Three-Dimensional impeller model with the help of Solidworks 2013 parametric design feature. The three-dimensional impeller model is shown in figure 1 below.

Figure 1: Model of Radial flow Impeller

Casing design

The casing is used to transfer the liquids with the conversion of kinetic energy into pressure energy after the fluid has left the impeller. It provides a more controlled flow and a more efficient conversion of velocity head into pressure head. The standard casing model construction is generated in Solidworks 2013, as shown in figure 2 below.

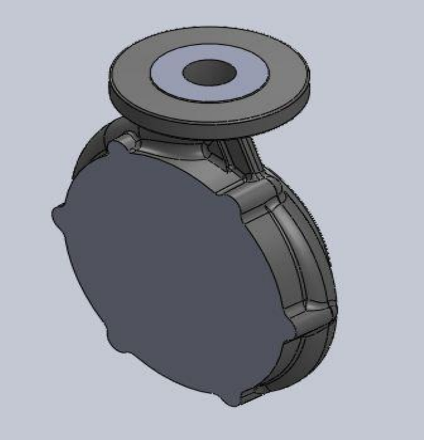

Figure 2: Model of the Pump casing

Fluid domain method

The fluid domain method is an additional FSI option within ANSYS CFX that allows simulation of the unlimited motion of fluid regions. The fluid domain model construction is generated, as shown in Figures 3 and 4. This model represents the cavity in the impeller and casing part where the fluid is going to travel in the submersible pump.

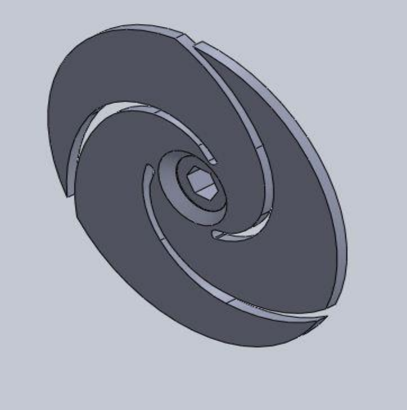

Figure 3: Impeller fluid domain part

Figure 4: Casing fluid domain part

ANSYS-CFX ANALYSIS

The entire analysis is carried out in ANSYS 13.0 software and the fluid flow simulation is carried out in the ANSYS-CFX tool. ANSYS-CFX is a commercial CFD program used to simulate fluid flow in a variety of applications. ANSYS CFX is a general-purpose Computational Fluid Dynamics (CFD) software suite that combines an advanced solver with powerful pre-processing and post-processing capabilities.

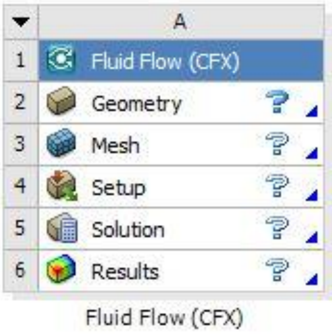

Figure 5: ANSYS-CFX tool layout

Geometry model

The pump impeller fluid domain model, casing fluid domain model, and inlet pipe are designed using Solidworks design parametric feature and saved in IGES format. The pump modeling parts of IGES files are imported into the Design explorer tool. The assembled geometry model is shown in figure 6, given below.

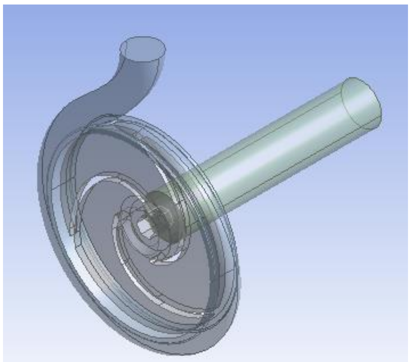

Figure 6: Geometry model

Mesh

The pump geometry model has meshed with an option of relevance center, smoothing, and transition are fine, slow, and fast selected. The grid sensitivity test is analyzed for better accuracy of the result. The meshed model is shown in figure 7, given below.

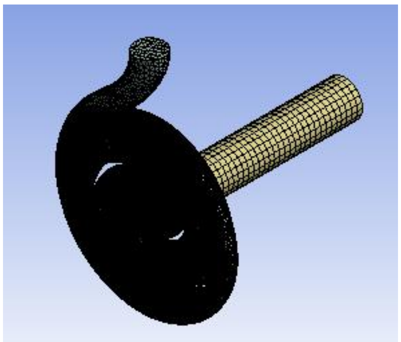

Figure 7: Meshed model

The automatically generated mesh model has the number of nodes and elements are 8,24,528 and 52,68,218.

Boundary conditions

The steady flow model is selected to improve the convergence rate and stability of results. The pump model inlet and outlet boundary conditions are set as pressure and mass flow rate. Also, the rough wall condition is applied all over the region, and the k-έ turbulence model condition is selected for better turbulence calculation results. The number of iterations is chosen as 1000, and the converged result is between 400-600. The pressure distribution and velocity distribution in the pump fluid flow are shown in Figures 8 and 9 below.

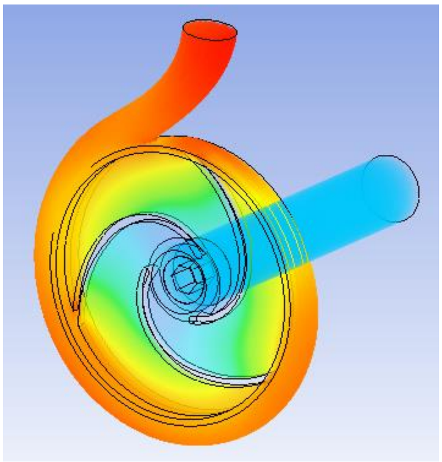

Figure 7: Pressure Distribution

The above pressure distribution diagram clearly denotes that the static pressure gradually increases from the impeller outlet to the casing discharge level.

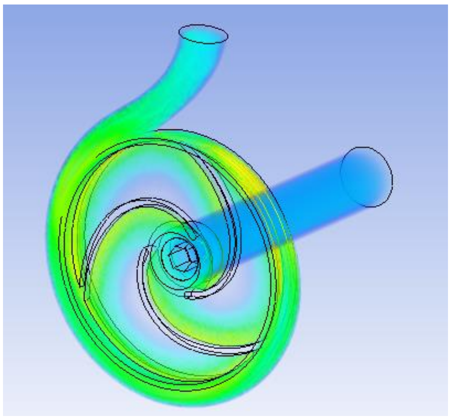

Figure 8: Velocity Distribution

From the velocity distribution diagram, it can be clearly denoted the velocity gradually increases from the impeller inlet to the outlet.

The inlet velocity, outlet velocity, static head, velocity head, pressure head, total head, and torque are found in the expression tool.

Total head = 26.82 m

Discharge = 4.49 m

The different discharge values are given as boundary conditions at the pump outlet, and the same inlet boundary conditions are selected. The analysis is repeated for the different rates of discharge, and the pressure distribution and velocity distribution diagrams note down. From this diagram, the pump total head and torque are calculated through the expression tool in the ANSYS-CFX. From this, submersible pump performance values are calculated.

VALIDATION

The submersible pump's actual performance of various discharge vs. total head curves is given by the company. From simulation result values, the discharge vs. total head curve is plotted. The Simulated result value and actual response value of the Discharge vs. Total head pump performance curve were compared. The Discharge vs. Total head graph is shown in figure 10.

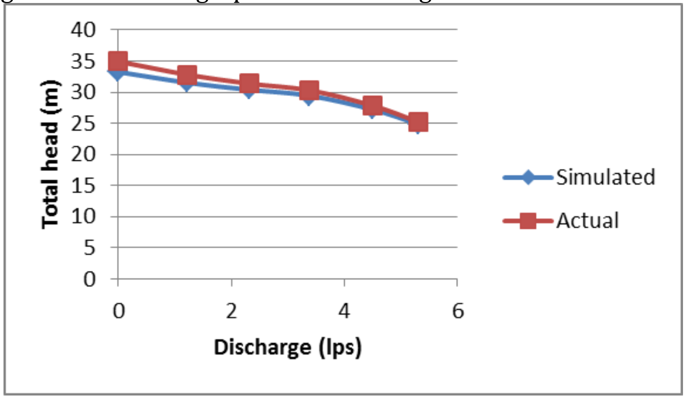

Figure 10: Total head vs. Discharge graph

From this graph, the simulation values are mostly matched with the actual response pump values so that the simulation procedure and result are accurate.

MODIFIED DESIGN

The pump impeller outlet diameter, Vane width, Number of vanes, Impeller inlet vane angle and outlet vane angles are modified with the optimum range.

Combinations-I

The following three parameters are considered for the first-level combination.

Impeller outlet diameter: 161 mm, 162 mm, 163 mm
Vane width: 6 mm, 8 mm, 10 mm
Number of vanes: 3, 4, 5

S. No	Impeller diameter (mm)	Vane width (mm)	Number of vanes
1	162	6	3
2	162	6	4
3	162	6	5
4	162	8	3
5	162	8	4
6	162	8	5
7	162	10	3
8	162	10	4
9	162	10	5
10	163	6	3

11	163	6	4
12	163	6	5
13	163	8	3
14	163	8	4
15	163	8	5
16	163	10	3
17	163	10	4
18	163	10	5
19	164	6	3
20	164	6	4
21	164	6	5
22	164	8	3
23	164	8	4
24	164	8	5
25	164	10	3
26	164	10	4
27	164	10	5

Keeping the design impeller parameters other than the above-mentioned parameters constant and the analyses are carried out for the above combinations. The pump performance values are noted down, and the optimum impeller design parameters value is selected from the combinations-I.

Impeller outer diameter = 162 mm
Impeller width = 6 mm
Number of vanes = 3

Keeping these impeller design parameters constant, the impeller inlet vane angle and outlet vane angles are varied.

Combinations-II

Inlet vane angle = 16^0, 17^0, 18^0, 19^0
Outlet vane angle = 22^0, 23^0, 24^0, 25^0, 26^0

S. No	Inlet vane angle	Outlet vane angle
1	16^0	22^0
2	16^0	23^0
3	16^0	24^0
4	16^0	25^0
5	16^0	26^0
6	17^0	22^0
7	17^0	23^0
8	17^0	24^0
9	17^0	25^0

10	17^0	26^0
11	18^0	22^0
12	18^0	23^0
13	18^0	24^0
14	18^0	25^0
15	18^0	26^0
16	19^0	22^0
17	19^0	23^0
18	19^0	24^0
19	19^0	25^0
20	19^0	26^0

Above mentioned combinations are analyzed, and the optimum design parameters values of inlet vane angle = 18^0 and outlet vane angle = 26^0 are selected from the analyzed result values.

Total head = 28.848 m
Discharge = 4.49 m
Pump efficiency = 45.208 %
Overall efficiency = 36.01 %

RESULT

The existing and modified design of the submersible pump's overall efficiency are 32.6 % and 36.01 %. The submersible pump's overall efficiency is a 3.41 % increase from the existing design to the modified design based on the optimum design parameter conditions.

CONCLUSIONS

In this work, an impeller model of a submersible pump was created using Solidworks 2013 software from the data provided by a prominent pump manufacturer and then analyzed in ANSYS - CFX.

An attempt has been made to validate the performance of the submersible pump by the total head and mass flow rate. The standard impeller model designed in this work is a reasonable representation of the actual pump.

The Actual response values of the pump and simulated values are validated. The design parameter values are modified based on the optimum condition, and the analysis is conducted.

From the analysis best combination of pump design parameters is selected for the optimum design. For the optimum design parameters, the pump performances are noted, and overall efficiency is calculated and increased to the existing design

REFERENCES

1. S.C.Chaudhari, C.O.Yadav, and A.B.Damor**,** Aug 2013"*A comparative study of mixed flow impeller CFD analysis and experimental data of submersible pump*" IJRET ISSN 2321-8843, Vol.1, Issue 3, pp 57-64.
2. Virajit A.Gundale and S.A.Patil, May 2012 "*Improvement in the design of a radial type vertical submersible open well pump impeller using CFD*" IJERIA, ISSN 0974-1518, Vol.5, No.II, pp 99-108.
3. Igor Tverdokhleb, Elena Knyazeva, and Aleksander Birukov, "*About designing the flow part of a multi-stage pump with a minimum radial dimension*."
4. A.Manivannan, "*Computational fluid dynamics analysis of a mixed flow pump impeller*" IJES Vol 2, No 6, pp.200-206.
5. S. Rajendran and Dr.K.Purushothaman, , May 2012" *Analysis of a centrifugal pump impeller using ANSYS-CFX*" IJERT, ISSN 2278-0181, Vol. 1, Issue 3.

6. S. Chakraborty and K.M. Pandey," *Numerical studies on effects of blade number variations on the performance of centrifugal pumps at 4000 RPM*."
7. Massinissa Djerroud, Guyh Dituba Ngoma, and Walid Ghie," *Numerical identification of key design parameters enhancing the centrifugal pump performance: Impeller, Impeller-Volute and Impeller-Diffuser*" ISRN, Vol.2011
8. Adnan Ozturk, Kadir Aydin, Besir Sahin, and Ali Pinarbasi, March 2009, "*Effect of impeller-diffuser radial gap ratio in a centrifugal pump*" JSIR, Vol.68, pp.203-213.