International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249–6890; ISSN(E): 2249–8001
Vol. 10, Issue 2, Apr 2020, 955–964

EXPERIMENTAL TEST AND VALIDATION WITH SIMULATION RESULTS FOR IMPELLER OF CENTRIFUGAL PUMP USING CFD ANALYSIS

KAPIL PANDYA & NITIN JAYAKUMAR

Department of Mechanical Engineering, School of Diploma Studies, RK University, Rajkot, Gujarat, India

ABSTRACT

The main objective of this research is to increase the head and efficiency of the pump so that the cooling system of a tractor can be improved. Centrifugal pumps are the most commonly used turbomachinery devices. They are utilized to raise the pressure or induce flow in a manipulate volume. The experimental setup was established at the industry to calculate the head and efficiency of the pump at various speeds and discharges. Later the CAD models of the mixed flow impeller using CAD modelling software Solidworks were developed and simulation were carried out with specified boundary conditions using ANSYS CFX software. The velocity and pressure in the outlet of the impeller could be predicted. All the experimental and simulation data are validated as the results of the head and efficiency were closer to each other. Hence, it could be predicted that what parameters are selected for improvement in the head and efficiency of the pump.

KEYWORDS: Tractor, ANSYS CFX, Solidworks, Mixed Flow Impeller, CFD Analysis & Meshing

Received: Feb 15, 2019; **Accepted**: Mar 05, 2020; **Published**: Apr 02, 2020; **Paper Id.**: IJMPERDAPR202093

Original Article

INTRODUCTION

Computational Fluid Dynamics (CFD) analysis is applied worldwide for the design of aeromodelling applications, pumps, turbo machines, etc. In many area of pumping systems, engineer's choice always point to Centrifugal pumps. Earlier because of its simple design, the use of centrifugal pump led to its success, but now designs have become complex due to its specific needs as per the requirement. With the competitive/trending market and technological advancement many parameters are taken to improve the overall working efficiency of the pump so that it could be used in various industrial sectors with minimum maintenance. [9] The basic operating conditions like features of fluid, suction head parameters and other technical data helped the engineers to work on predicting the critical parameters like efficiency, discharge, etc. It is easy to operate and maintain because of its simple design unless the design changes as per the requirement. Solidworks and ANSYS are mostly used for the modelling and smiluation of any mechanical component to identify/predict the optimum working conditions where efficiency can be stablised or improved. And hence these datas can be verified in concern with the experimental results.

LITERATURE REVIEW

Mr. Jekim J. Damor et.al, has done experimental analysis on centrifugal water pump with keeping fixed parameters of head, discharge and impeller diameter to evaluate the performance of the pump. The validation was performed with the reference of model developed in Solidworks software and CFD analysis in ANSYS CFX software. [1]

Sujoy Chakraborty et.al, has performed 2D numerical investigation of static pressure and incompressible flow by varying the number of blades of centrifugal pump. The objective of the work was to improve efficiency of the pump. The evaluation was performed on FLUENT 6.3 software for 5, 6 and 7 impeller blades. [2]

S. C. Chaudhari et.al, has used CFD analysis to forecast the velocity and pressure in the outlet of impeller. The empirical equations were used to calculate the optimum vane angles and with the help of that model was developed in Solidworks software. Hence, they concluded that head of the pump is improved by changing the inlet, outlet vane angles and no. of blades. [3]

Lamloumi Hedi et.al, has done numerical simulation for 3D fluid flow in centrifugal pump. This helped in understanding the complex internal flows in water pump. Navier-Stokes equations were used for the simulation of flow inside the vane less impeller. [4]

Rakesh Joshi has used Fluent to investigate the pump performance to evaluate the pressure and velocity distribution inside the pump passage. [11]

By reviewing previous works, it is stated that there are two methods to optimize the Flow rate, Head and Efficiency of centrifugal pump by means of experiment and simulation procedure. Experiments were performed considering various working conditions which was later verified in reference with the simulation data and hence that results in closer range of the values.

PROBLEM STATEMENT & METHODOLOGY

To improve the efficiency and head of impeller used in centrifugal pump so that cooling efficiency system of tractor engine is improved and hence to reduce the effect of heat when tractor is in working condition. To carry out experiments and simulation analysis, following method is adopted:

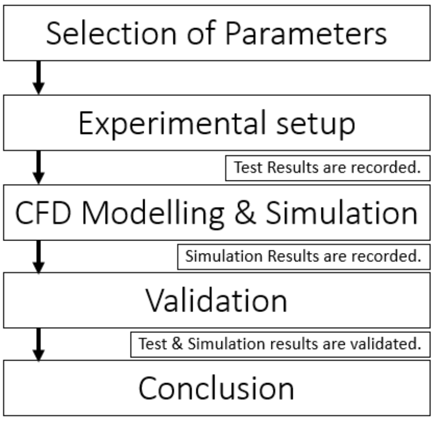

Figure 1: Work Methodology.

EXPERIMENTAL SETUP

The pump that is designed in this study is tested in the test stand of TECHNO INDUSTRIES PVT. LTD. There are four wells in the test stand. Those wells are opened to a pool. The valves are used to discharge and control the flow for pumps to be tested in each well. The crane is used to carry the pumps inside the test stand and position them to the suitable well. The valves are connected to the control panel which is in the control room of the test stand. The control panel is also connected to the main electric panel to start and stop the electric motor that drives the pump. There are manometers and pressure transducers which are capable of measuring pressures. The flow rate of the pump to be tested is measured by flow meter that is installed on discharge pipes. The pumps are tested by using cold clean water as working fluid.

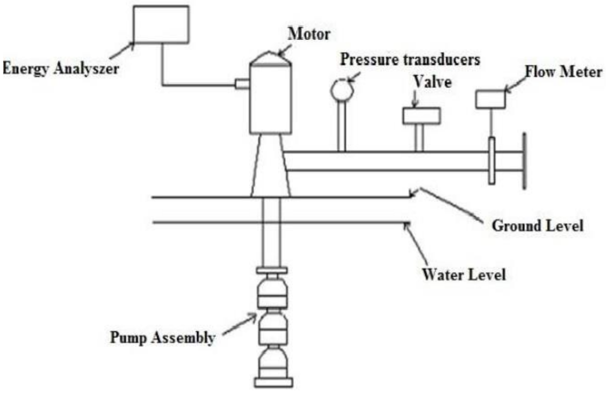

Figure 2: The Pump that is Designed in this Study is Tested in the Test Stand of TECHNO INDUSTRIES PVT. LTD.

Table 1: Design specification of Impeller

Parameters	Values
Outer diameter of Impeller	220mm
Inlet diameter of Impeller	105mm
Inlet blade angle	24mm
Outlet blade angle	45mm
Thickness of blade	6mm
Number of blades	8
Shaft diameter	22mm
Key	7 by 7mm

EXPERIMENTAL TEST PROCEDURE & RESULTS

The test procedure has been carried out at TECHNO INDUSTRIES PVT. LTD. which has received ISO 9001-2008 accreditation by **ISOQAR LTD**.

Figure 3: Experimental set-up. [1]

Table 2 - Experimental Test results

Sr. No	Pump Speed (RPM)	Discharge (LPM)	Head (m)	Efficiency (%)
1	1216	7.0	1.1	20.17
2	1702	12.0	2.32	36.87
3	2797	30.0	4.8	72.27

SIMULATION PROCEDURE & RESULTS

Impeller Geometry: The 3 D model of impeller geometry which basically consists of inlet and outlet for entry and exit of fluid. The impeller geometry consists of 8 blades.

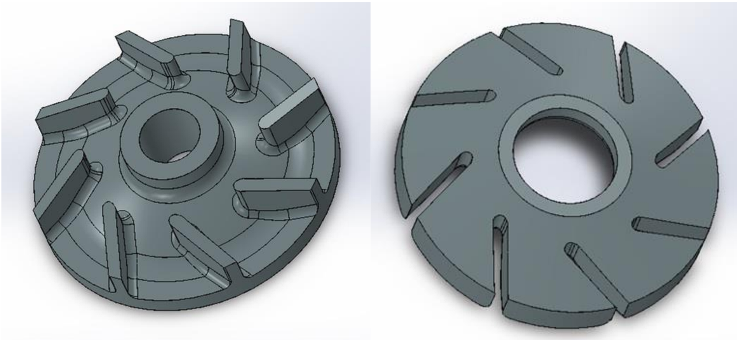

Figure 4: Model of Impeller & Cavity Model in Solidworks.

Cavity Model in Solidworks

The cavity model of impeller is produced using solid works so that the area is provided for the flow of fluid. A cavity method is mathematical model to solve some mean field types of model in statistical physics, specially adopted to disordered systems. It has been used to compute properties to ground states in many condense matter and optimization problem. The cavity method plays important role in optimization process.

Meshing of Impeller

The Meshing of impeller is done to find the number of nodes, number of elements, type of meshing is 3D and type of element is tetrahedral. Parameters selected for meshing are No. of Nodes as 440549, No. of elements as 2219378, Meshing Type as 3D & Type of Element as Tetrahedral

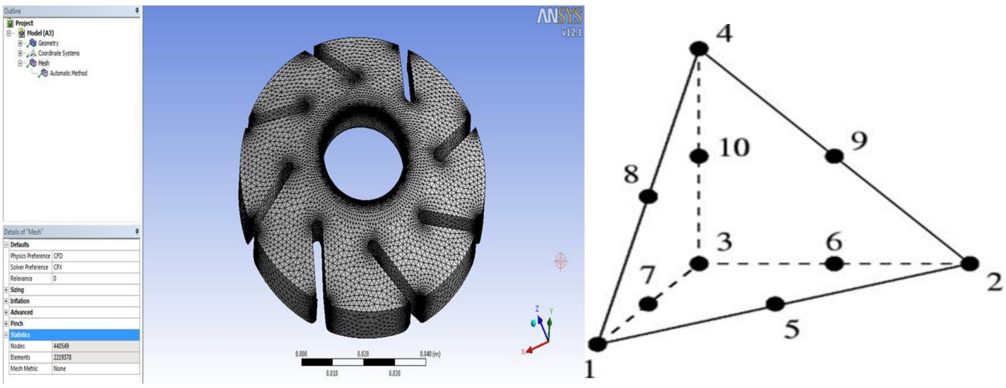

Figure 5: Meshing of Impeller and Type of Meshing Element [6].

Domain motion: - Rotating & Rotating about: - Y Axis for the above figure.

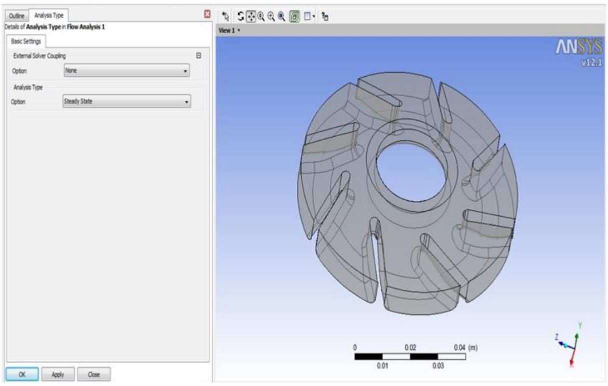

Figure 6: Analysis of Impeller.

Define Hub& Shroud as a Rotating Wall

The wall used for hub & shroud is rotating as there is no slip in the wall, wall roughness is smooth and heat transfer is adiabatic as there is no heat transfer.

Define Inlet for Impeller

The inlet mass ow rate is as per requirement and normal to boundary condition. Static frame total temperature is 300K.

Define Outlet Pressure

The outlet pressure is 1.01325bar which is static pressure.

Define Solver Control Criteria

The number of outer look iteration is 100. The convergence criteria are used for CFD analysis. Convergence criteria is 1e-4. Allow the analysis to get run for some time.

For above mentioned pump speed, inlet pressure contour, outlet pressure contour and velocity stream line are shown below:

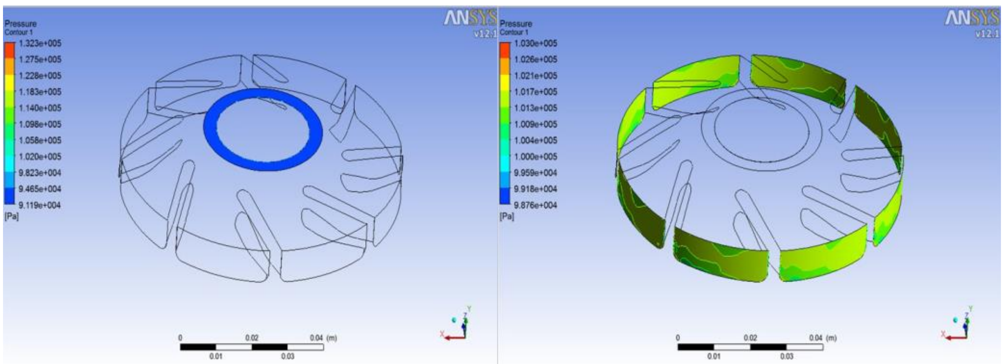

Figure 7: Inlet& Outlet Pressure Contour for 1216 RPM.

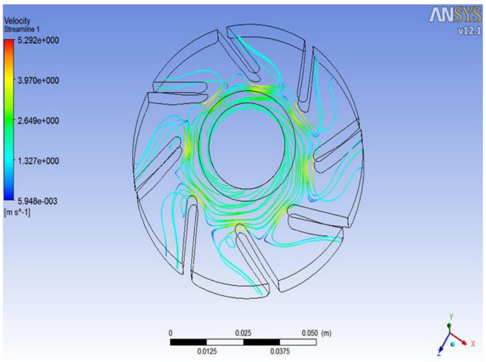

Figure 8: Velocity Stream line for 1216 RPM.

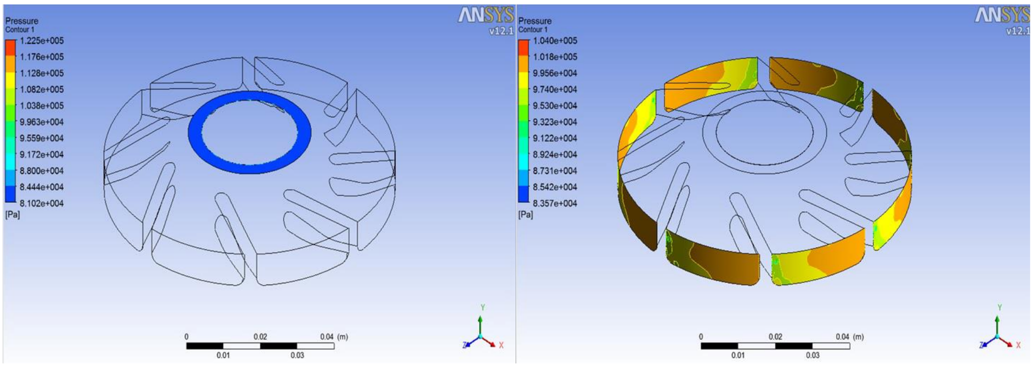

Figure 9: Inlet& Outlet Pressure contour for 1702 RPM.

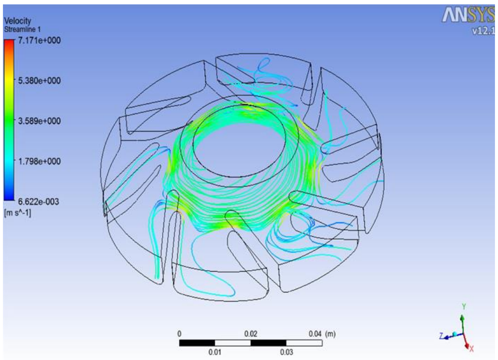

Figure 10: Velocity Stream line for 1702 RPM.

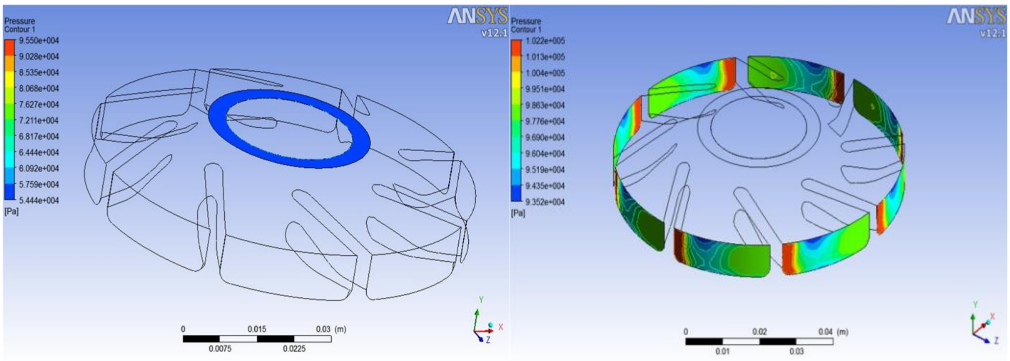

Figure 11: Inlet& Outlet Pressure Contour for 2797 RPM.

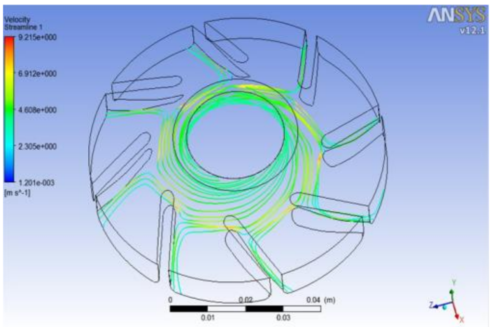

Figure 12: Velocity Stream line for 2797 RPM.

Head for the following data can be calculated as Head = (Outlet Pressure - inlet Pressure)/(Density x 9.81)

From the above simulation results for various speed, we conclude that when the RPM is increased along with discharge, the head and efficiency increase, so the maximum head & efficiency are obtained at 2797 RPM.

Table 3 - Simulation Results for Various Speed

Sr. No	Pump Speed (RPM)	Discharge (LPM)	Head (m)	Efficiency (%)
1	1216	7	1.2	20.34
2	1702	12	2.34	36.94
3	2797	30	4.86	72.48

VALIDATION

The validation between the Experimental and Simulation result are compared for Discharge v/s head in case of Experiment and discharge v/s head in case of CFD. The validation between the Experimental and Simulation results are compared for Discharge v/s efficiency in case of Experiment and discharge v/s efficiency in case of CFD.

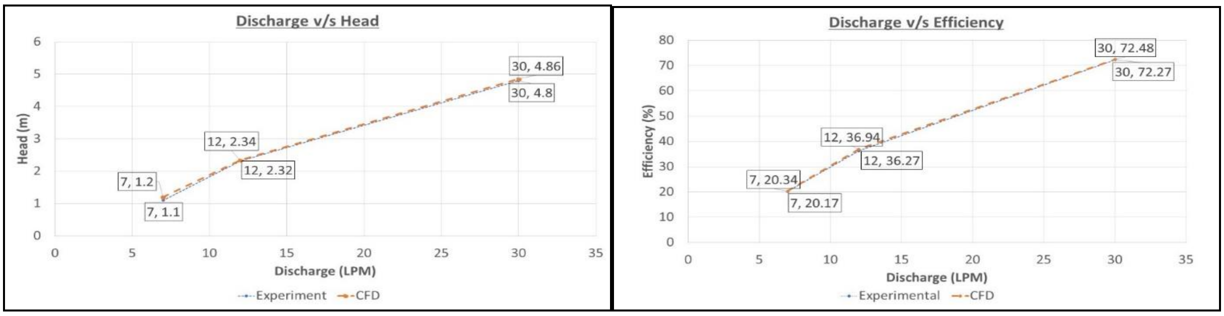

Figure 13: Discharge Vs. Head and Discharge v/s Efficiency.

From the validation, we can conclude that 2797 RPM gives maximum Head and Efficiency. 2797 RPM gives 4.86 m Head and 72.48(%) Efficiency, which is experimental result. 2797 RPM gives 4.8 m Head and 72.27 (%) efficiency, which is simulation result. The Head and Efficiency value are more for 2797 RPM as compared to 1216 RPM and 1702 RPM.

Table 4: Validation of Experimental and CFD Results

Sr. No	Pump speed (RPM)	Discharge (LPM)	Head (m)			Efficiency (%)		
			Exp	CFD	Diff	Exp	CFD	Diff
1	1216	7	1.1	1.2	0.1	20.17	20.34	0.17
2	1702	12	2.32	2.34	0.02	36.27	36.94	0.67
3	2797	30	4.8	4.86	0.06	72.27	72.48	0.21

CONCLUSIONS

Following points are noted:

- Experiments were carried out at various operating conditions and same cases were analysed using software.
- After comparing the results of CFD analysis and Experimental setups, there are minute differences as shown in the previous graphs and are validated.
- The best operating condition was obtained when ow rate was 30 LPM, Head = 4.86 m, efficiency = 72.48 % for the selected pump.
- To calculate head and efficiency rotational speed and pressure plays a vital role.

ACKNOWLEDGEMENTS

We, the authors would like to extend our gratitude towards TECHNO INDUSTRIES PVT. LTD. Ahmedabad & Panchhnath Auto PVT. LTD. Rajkot for allowing us to perform the experiments at their premises, Prof. Chetankumar Patel for the support and guidance in preparing this paper and staff members of the Department of Mechanical Engineering for motivating us in this process. We are also thankful to Dr. Nilesh B Kalani and RK University for allowing us the platform for carrying out research & publishing paper.

REFERENCES

1. *Jekim J. Damor et.al, Experimental and CFD Analysis of Centrifugal Pump Impeller- A Case Study, IJERT*
2. *Sujoy chakraborty et al. (2013) Performance prediction of centrifugal pumps with variation of blade number, Journal of scientific and industrial research, Vol. 72, pp. 373-378*
3. *S. C. Chaudhari etal. (2013) A comparative study of mix flow pump impeller CFD analysis and experimental data of submersible pump, IMPACT: IJRET, ISSN 2321-8843, Vol. 1, Issue 3, pp. 57-64*
4. *Lamloumi Hedi etal. (2012) Simulation study and Three-Dimensional Numerical Flow in a Centrifugal Pump, International Journal of Thermal Technologies, ISSN 2277 - 4114, Vol.2, No.4, pp. 209-215*
5. *Mehta P. Mehul. (2013) Performance analysis of mixed flow pump impeller using CFD, International Journal of Emerging Trends in Engineering and Development, Vol. 1, Issue 3*
6. *Ashish J. Patel, Bhaumik B. Patel. (2014)Design And Flow Through CFD Analysis Of Enclosed Impeller, International Journal of Engineering Research & Technology (IJERT), Vol. 3 Issue 7, 1366-73*
7. *V. S. Kadam etal. (2011) Design and Development of Split Case Pump Using Computational Fluid Dynamics, Institute of technology, Nirma university, Ahmedabad-382 481.*
8. *Kapil Pandya, Chetankumar M. Patel. (2014) A Critical review on CFD Analysis of centrifugal pump impeller, InternationalJournal of Advance Engineering and Research Development (IJAERD), Volume 1, Issue 6, 1-7*
9. *E. C. Bacharoudis etal. (2008) Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle", The Open Mechanical Engineering Journal, Vol. 2, pp. 75-83*
10. *S R Shah etal. (2012)CFD for centrifugal pumps: A review of the state-of-the-art, Chemical, Civil and Mechanical Engineering Tracks of 3rd Nirma University International Conference, NUiCONE 2012, Procedia Engineering 51, pp. 715-720*

11. *Manivannan. (2010) Computational fluid dynamics analysis of a mixed flow pump impeller International journal of Engineering, science and Technology Vol. 2, No. 6, pp. 200-206*
12. *Rakesh Joshi: Computational investigation of flow field in a centrifugal slurry pump, Mechanical department, Thapar University, Patiala*
13. *" Pre-Stress Modal Analysis of A Centrifugal Pump 509 Impeller forDifferent Blade Thicknesses ", International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), Vol. 7, Issue 6, pp. 507-516*
14. *" Design of Centrifugal Pump Impeller and its Effects on Cavitation ", International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), Vol. 9, Issue 4, pp. 753-760*
15. *" Computational Fluid Dynamics Analysis of Impellr Design for a Pump ", International Journal of Mechanical Engineering (IJME), Vol. 5, Issue 4, pp. 63-74*
16. *" Advanced Synchronized Manufacturing System ", IMPACT: International Journal of Research in Engineering & Technology (IMPACT: IJRET), Vol. 2, Issue 3, pp. 193-208*

17.