iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 11 **Issue:** VIII **Month of publication:** Aug 2023

DOI: https://doi.org/10.22214/ijraset.2023.55472

Centrifugal Pump Impeller Performance Prediction using Computational Fluid Dynamics

Shalom Nanthagopal[1], R. Rabin[2], Amal David[3], Goutham .J. S[4]
Noorul Islam Centre For Higher Education, India

Abstract: The sewage, drainage, and chemical industries all employ centrifugal pumps, which are a typical type of pump. By adjusting the impeller exit blade width, exit diameter, and trailing edge blade angle, a computational fluid dynamics (CFD) three-dimensional simulation is performed to optimise the pressure head, efficiency, and power needed of centrifugal pumps. The head of the pump is optimised in the CFD study of the impeller to enhance the capabilities of centrifugal pumps. Rotation of the impeller varies with the rotation and mass flow rate.

The geometry of the centrifugal pump impeller in this project is originally built in Solid Works, and CFD analysis is then completed in Ansys CFX. In this study, the initial dimensions of the impeller were taken from the base paper. The performance of the impeller was then assessed using the parameters Head, Power imparted to water by the impeller, and Efficiency. From the analysis, it was discovered that as the number of blades on the impeller increases, the value of head and power imparted as well as the efficiency of the impeller increases. After that, analysis was conducted while maintaining a constant number of blades at various impeller RPMs, and the results showed

Keywords: Centrifugal pump; CFD simulations; reverse running; unsteady flow; radial load.

I. INTRODUCTION

Centrifugal pumps move fluids by converting the kinetic energy of rotation into the hydrodynamic energy of the fluid flow. An engine or electric motor is normally where the rotational energy originates from. They belong to the dynamic axisymmetric work-absorbing turbo machinery subclass. A diffuser or volute chamber (casing) is where the fluid departs the pump after entering the impeller along or close to the rotating axis and being propelled by the impeller.

Pumping for petroleum and petrochemicals, sewage, agriculture, and water are a few common uses. Due to their high flow rates, compatibility with abrasive solutions, mixing potential, and relatively straightforward architecture, centrifugal pumps are frequently used in industrial settings. A centrifugal fan is frequently used to power a hoover cleaner or air handling device

A centrifugal pump, like the majority of pumps, transforms rotational energy, frequently from a motor, into energy in a flowing fluid. A fraction of the energy is converted into fluid kinetic energy. The casing's eye serves as the axial entry point for fluid, which is then captured in the impeller blades and whirling tangentially and radially outward until it exits via all of the impeller's circumferential portions and reaches the diffuser portion of the casing. While flowing through the impeller, the fluid increases in both velocity and pressure. The casing's doughnut-shaped diffuser or scroll portion slows the flow and raises the pressure even further..

II. REVIEW OF LITRATURE

Since centrifugal pumps are often utilised in commercial, agricultural, and domestic settings, various applications may call for the pump system to work across a wide flow range. The design or near-design condition of pumps was the subject of the majority of earlier numerical investigations. Studying the off-design performance of pumps, when the performance of the pump declines, has not received much attention. The complicated internal flows via the various pump components may be investigated under various operating situations with the use of the CFD technique, which aids in improving the performance at off design settings

According to Bacharoudis et al.'s[1] research from 2008, a number of factors influence the performance and energy usage of pumps. This research carefully assesses the performance of impellers with the same outlet diameter but varied outlet blade angles. The head capacity curves are compared with the theoretical one after each impeller's flow pattern and pressure distribution in the blade passageways are computed. When the pump runs under non-design conditions, the head curve percentage increase caused by the increase in output blade angle is greater for high flow rates and decreases for flow rates Q/QN. When a pump runs at full capacity, a rise in head of more than 6% occurs when the output blade angle goes from 20°

According to Kaewnai et al. [2] [2009], the performance of a centrifugal pump's radial flow-type impeller may be analysed and predicted using computational fluid dynamics (CFD) technology. The flow rate, speed, and head at the impeller's duty point are 528 m3/hr, 1450 rpm, and 20 m, respectively. Mesh creation and its refining in various domains were done in the initial stage. The second stage involves the identification of several domains, as well as their material characteristics, border circumstances, and convergence criteria of mesh-equipped module. In the last stage, numerous findings are computed and their effects on the performance of the impeller are analysed. The findings show that the impeller's entire head increase at the design point is around 19.8 m. When 0.6 Q/Qdesign 1.2, the impeller's loss coefficient equals 0.015.

Using the commercial CFD programme FLUENT, Shah et al.[3] [2010] provided a numerical simulation of a centrifugal pump with a 200 m3/hr capacity. To cover the wide range of operation, the simulations were run at six distinct operating points between 30% and 110% discharge. Additionally, it was noted that the pressure rise was non-uniform at partial discharge but relatively progressive and uniform at rated and above rated discharge. Compared to part load and over rated discharge, the fluctuation in velocity at rated discharge is relatively consistent. It was discovered that the k-SST turbulence model performs better than the RNG k-model. Increased discharge results in decreased head, increased power input, and increased pump efficiency. At the time of duty, efficiency is at its peak; thereafter, it is discharged

The flow patterns through the pump, performance results, circumferential area averaged pressure from hub to shroud line, blade loading plot at 50% span, stream wise variation of mass averaged total pressure and static pressure, stream wise variation of area averaged absolute velocity, and variation of mass averaged total IS were all modelled and solved by Rajendran et al. [4] [2012].

Due to the dynamic head created by the spinning pump impeller, the pressure contours demonstrate a continuous pressure rise from the leading edge to the trailing edge of the impeller. Due to the thickness of the blade, low pressure and high velocities are seen close to the leading edge. Due to the existence of the trailing edge wake, entire pressure loss is seen close to the trailing edge of the blade.

Shah et al.'s[5] [2013] examination of the centrifugal pump demonstrates the use of CFD. The performance prediction at design and off-design settings, parametric research, cavitation analysis, diffuser pump analysis, performance of the pump operating in turbine mode, etc. on centrifugal pumps have all been investigated using the CFD approach. combined equations for the unstable Reynolds average Navier Stokes (URANS)

III. REVIEW OF LITRATURE

Since centrifugal pumps are often utilised in commercial, agricultural, and domestic settings, various applications may call for the pump system to work across a wide flow range. The design or near-design condition of pumps was the subject of the majority of earlier numerical investigations. Studying the off-design performance of pumps, when the performance of the pump declines, has not received much attention. The complicated internal flows via the various pump components may be investigated under various operating situations with the use of the CFD technique, which aids in improving the performance at off design settings.

According to Bacharoudis et al.'s[1] research from 2008, a number of factors influence the performance and energy usage of pumps. This research carefully assesses the performance of impellers with the same outlet diameter but varied outlet blade angles. The head capacity curves are compared with the theoretical one after each impeller's flow pattern and pressure distribution in the blade passageways are computed. as the pump runs under non-design conditions, the head curve percentage increase caused by the increase in outlet blade angle is greater at high flow rates and decreases as more of the energy is converted to fluid kinetic energy. Through the eye of the casing, fluid enters axially and is impeller-caught.

The casing's doughnut-shaped diffuser, also known as the scroll section, slows the flow and raises the pressure even further. From an engineering perspective, the two equation k- turbulence model was shown to be suitable to get a good estimate of the overall performance of the centrifugal pump, with typical errors below 10% compared to experimental data. The performance of pumps has been thoroughly investigated with regard to impeller and diffuser flows, and the study of volute flows has emerged as an intriguing research area. The study of two-phase flow (cavitation and slurry flow), pump handling nonNewtonian fluids, and fluid-structure interaction are the most active fields of research and development. Compared to previous methodologies, the CFD methodology has a number of benefits; yet, because the answer is empirical The design process and parameter optimisation are the primary areas of current research on enhancing the hydraulic performance of centrifugal pumps. The conventional approach to designing centrifugal impellers is more dependent on the expertise of the engineers and normally only addresses the continuity equation of the fluid. In this study, three centrifugal pump impellers are designed by varying blade wrap angles while maintaining other parameters constant. This is done on the basis of the direct and inverse iteration design method, which simultaneously solves the continuity and motion equations of the fluid and shapes the blade geometry by controlling the wrap angle.

Three centrifugal pumps' three-dimensional flow fields are numerically simulated, and the simulation outcomes show that the blade with a larger wrap angle has more potent control ability

In order to accommodate the increased efficiency, needed head, and discharge, V.S. Kadam et al. changed the closer range pump's impeller, enlarging its diameter from 770 mm to 820 mm. The pump is intended to operate more reliably and efficiently. The performance and efficacy of the pump are examined using a CFD study of the pump with adjusted impeller diameter.

Using the groundwork from the earlier study as a base The GSB20-380 hydraulic variant of the ultra-low specific-speed centrifugal pump was created by Jie Jin and colleagues. CFD (numerical simulation and performance testing were employed to examine the model of the centrifugal pumps, to analyse the theoretical advances on the ultra-low specific-speed centrifugal pump,

IV. DESIGN CONSIDERATIONS

A. Modelling And Mesh Generation

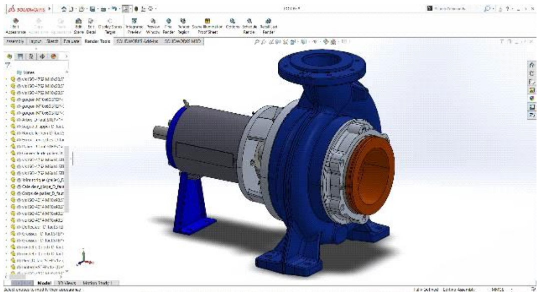


Fig no. 5.1 software image on impeller

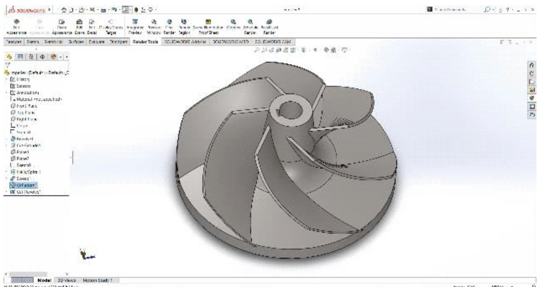


Fig no. 5.1(1) 6 blade impellor

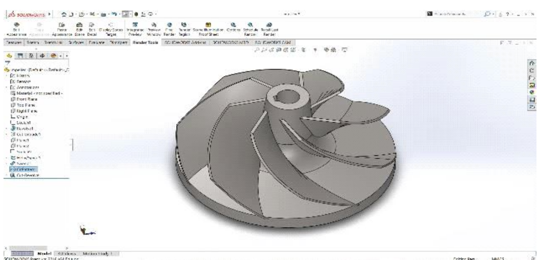


Fig no. 5.1(2) 7 blade impellor

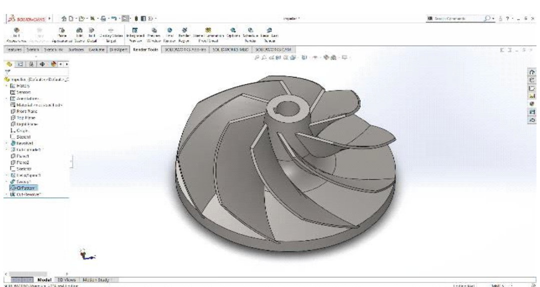


Fig no. 5.1(3) 8 blade impellor

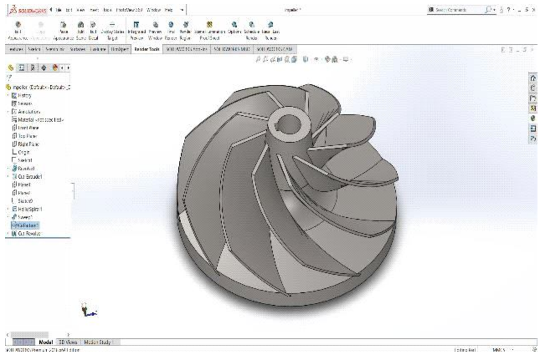


Fig no. 5.1(4) 9 blade impellor

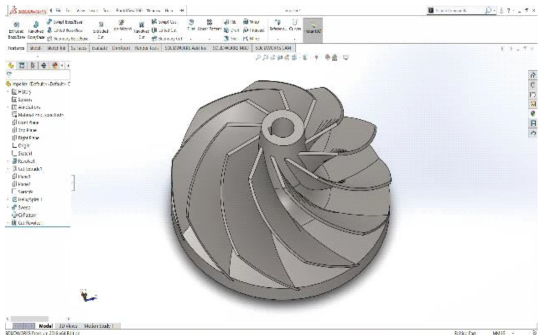


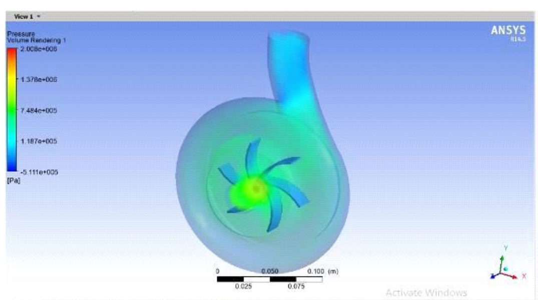
Fig no. 5.1(5) 10 blade impellor

V. RESULT AND DISCUSSION

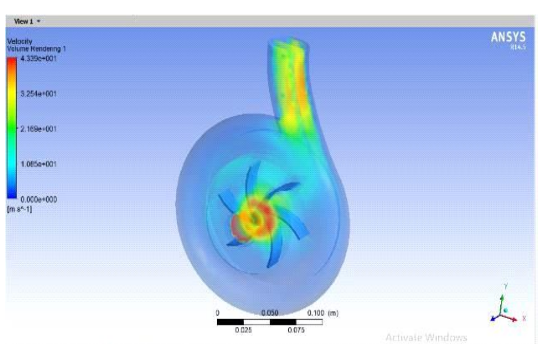
A. Number Of Blades: 6 Blades

1) Speed: 1000rpm

a) Pressure

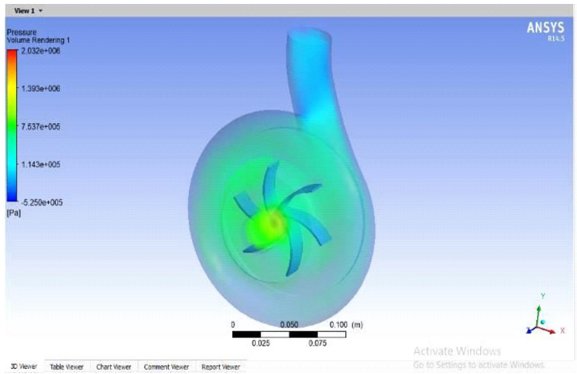


b) Velocity

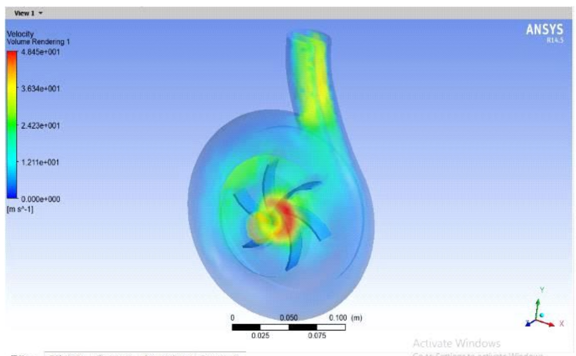


2) Speed: 2000rpm

a) Pressure

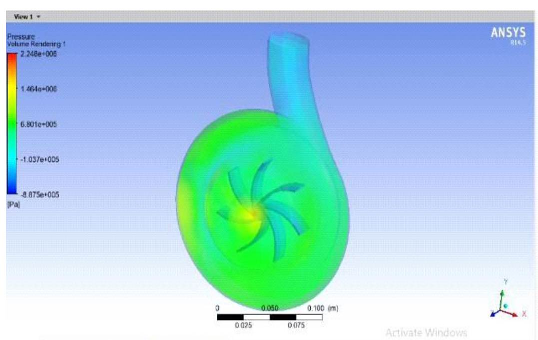


b) Velocity

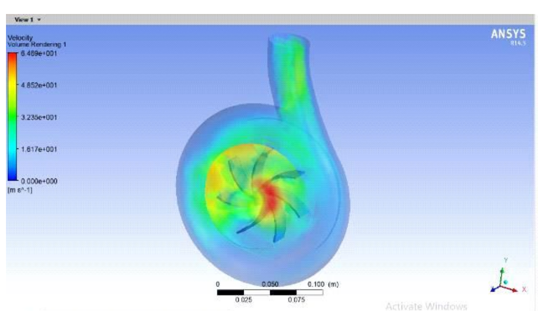


3) Speed: 4000RPM

a) Pressure



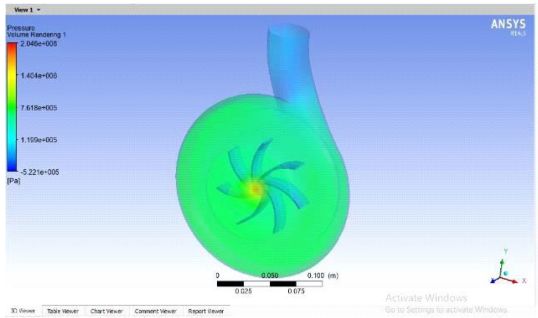
b) Velocity



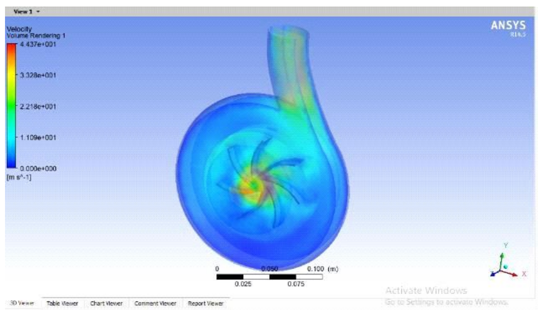
B. Number Of Blades: 7 Blades

1) Speed: 1000RPM

a) Pressure

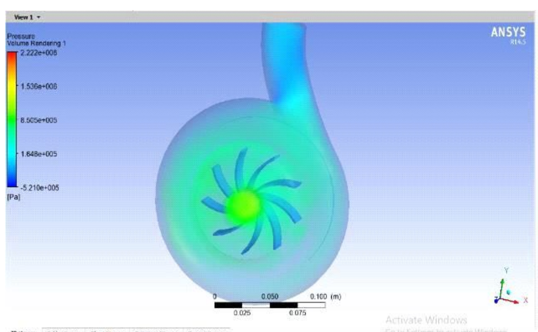


b) Velocity

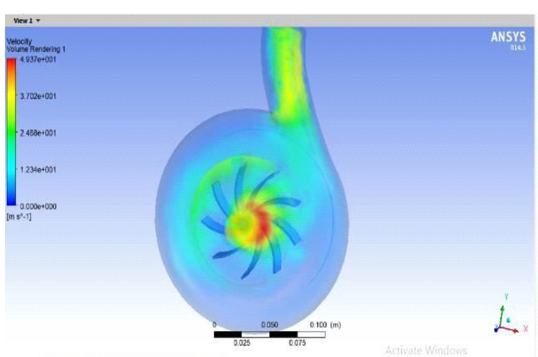


2) Speed: 2000rpm

a) Pressure



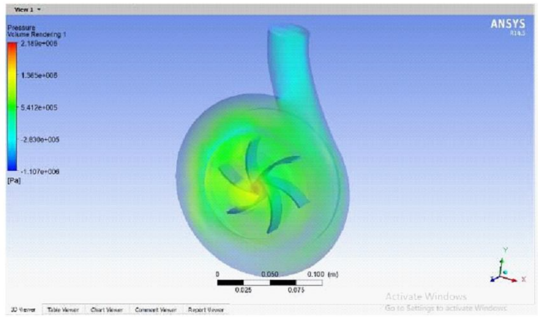
b) Velocity



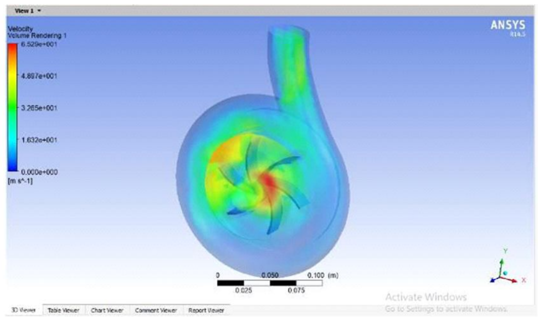
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue VIII Aug 2023- Available at www.ijraset.com

3) Speed: 4000rpm

a) Pressure



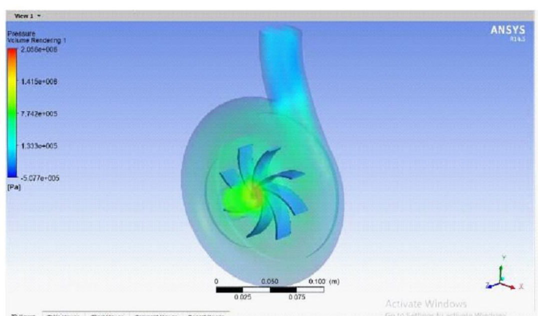
b) Velocity



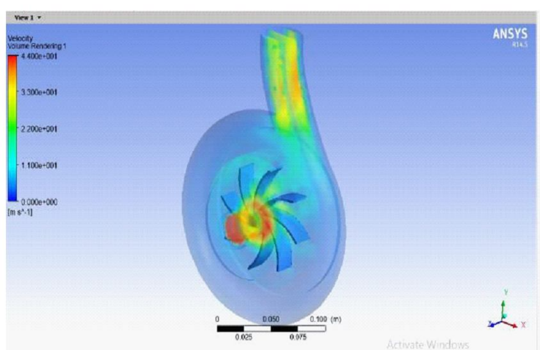
C. Number Of BLADES: 8 Blades

1) Speed: 1000RPM

a) Pressure

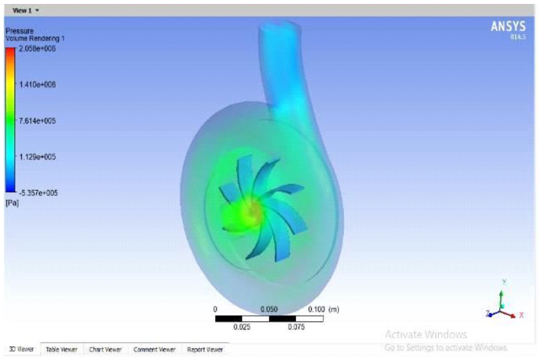


b) Velocity

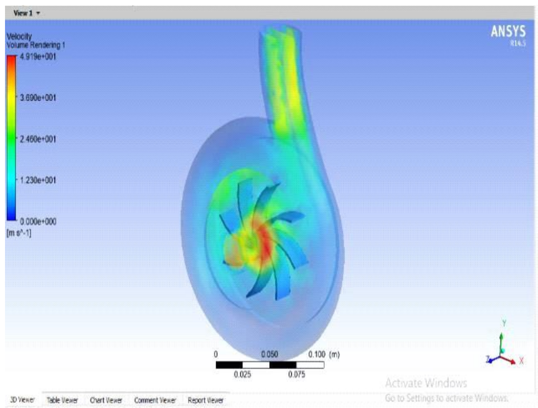


2) Speed: 2000RPM

a) Pressure

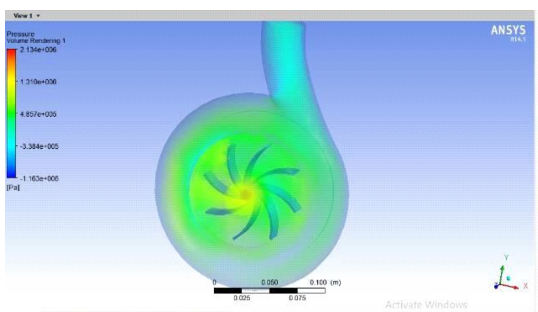


b) Velocity

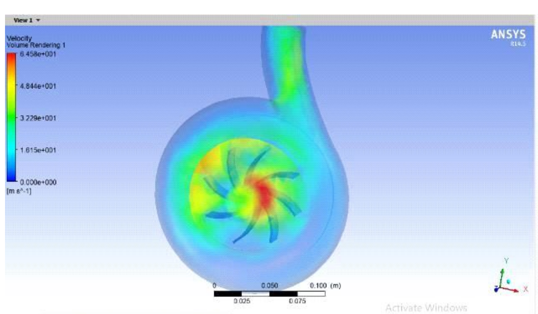


3) Speed: 4000RPM

a) Pressure



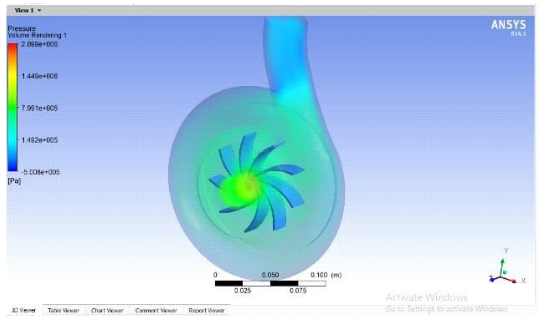
b) Velocity



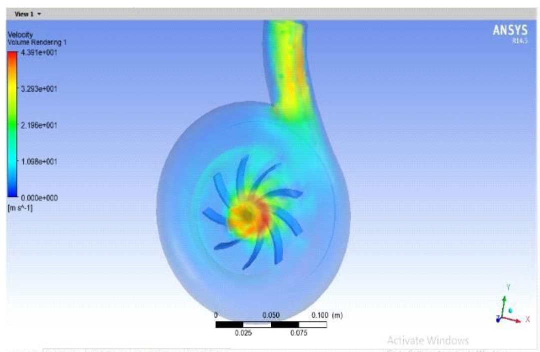
D. Number Of Blades: 9 Blades

1) Speed: 1000RPM

a) Pressure

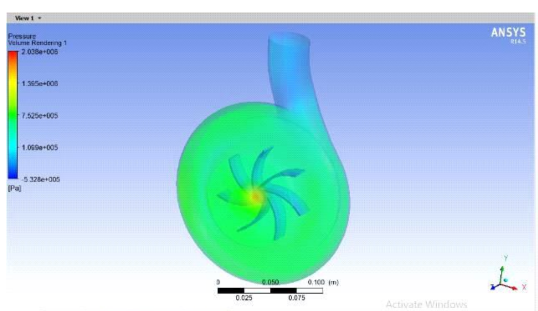


b) Velocity

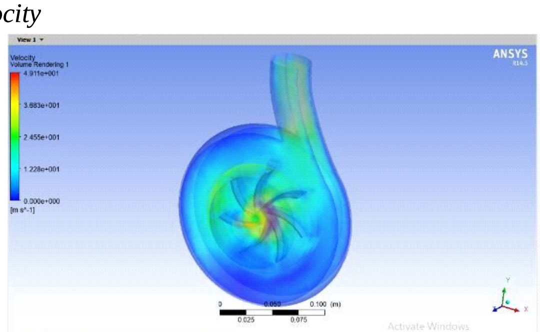


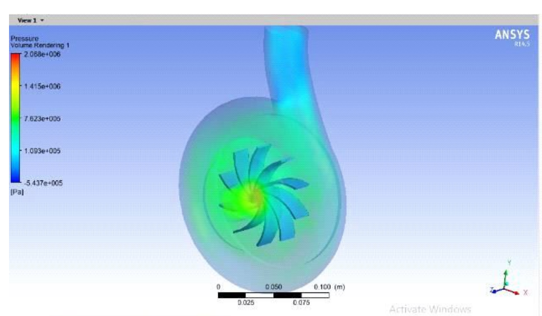
2) Speed: 2000RPM

a) Pressure



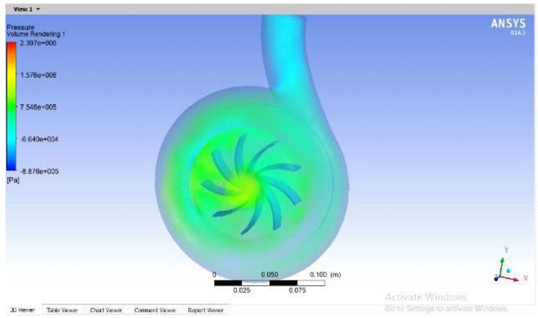
b) Velocity



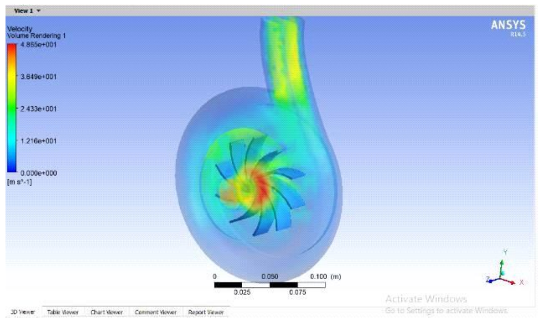


International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue VIII Aug 2023- Available at www.ijraset.com

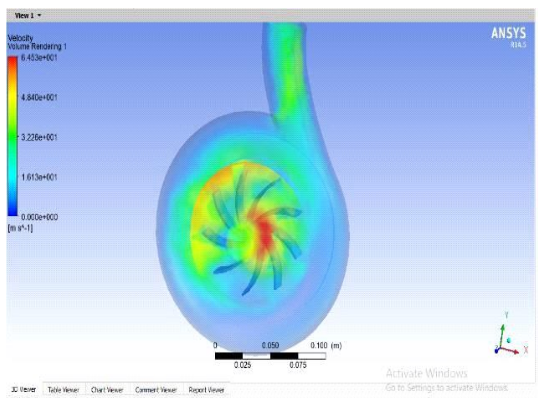
3) *Speed: 4000RPM*
a) *Pressure*



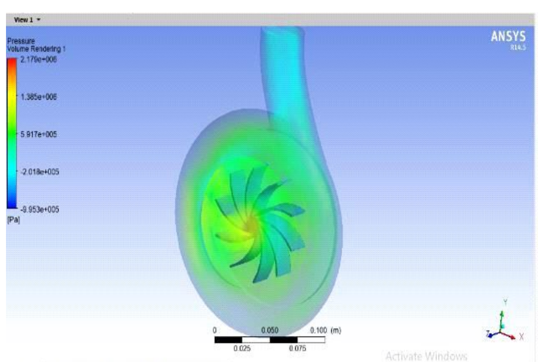
b) *Velocity*



4) *Speed: 4000RPM*
a) *Pressure*



b) *Velocity*

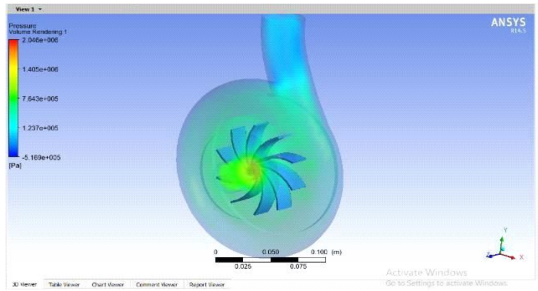


International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue VIII Aug 2023- Available at www.ijraset.com

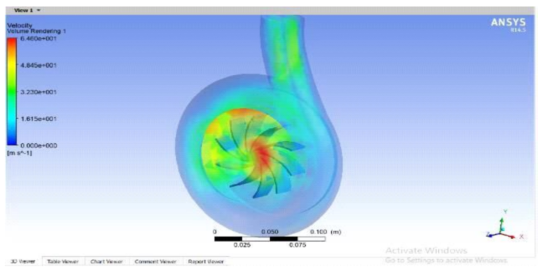
E. Number Of Blades: 10 Blades

1) *Speed: 1000RPM*

a) *Pressure*

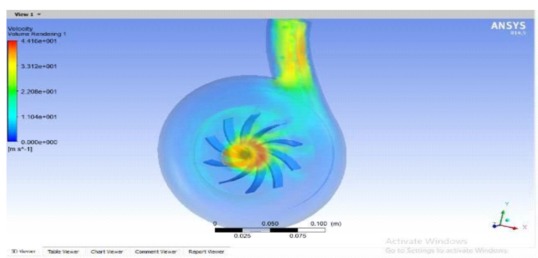


b) *Velocity*



2) *Speed: 2000RPM*

a) *Pressure*



F. Comparision Of Results

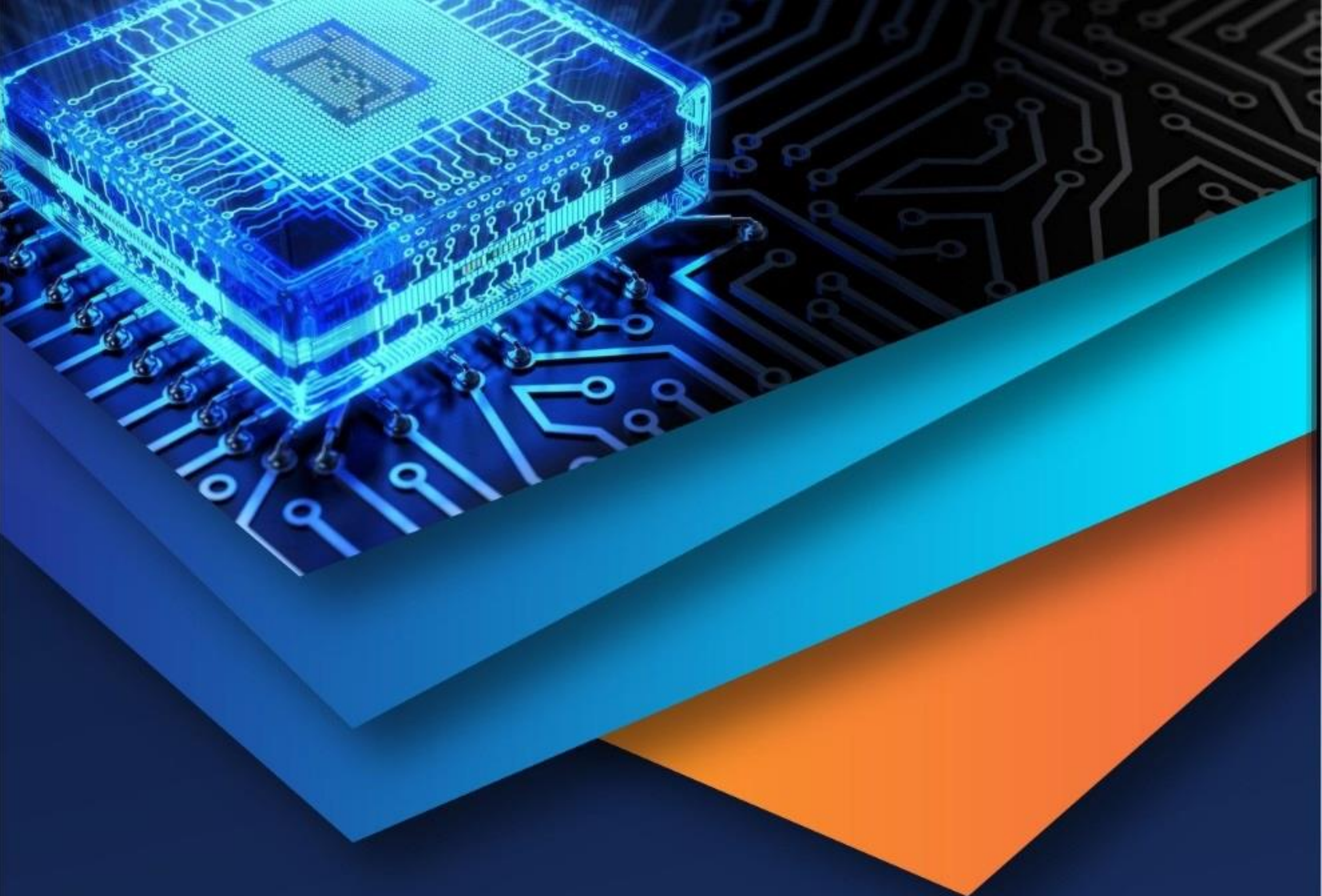
CFD results		Pressure	Velocity
Type of blades		(Pascal)	(ms-1)
6 Blades	1000rpm	2.008e+006	4.339e+00
	2000rpm	2.032e+006	4.845e+00
	4000rpm	2.248e+006	6.469e+00
7 Blades	1000rpm	2.046e+006	4.437e+00
	2000rpm	2.222e+006	4.937e+00
	4000rpm	2.189e+006	6.529e+00
8 Blades	1000rpm	2.056e+006	4.400e+00
	2000rpm	2.058e+006	4.919e+00
	4000rpm	2.134e+006	6.458e+00
9 Blades	1000rpm	2.009e+006	4.391e+00
	2000rpm	2.038e+006	4.911e+00
	4000rpm	2.397e+006	6.453e+00
10 Blades	1000rpm	2.046e+006	4.416e+00
	2000rpm	2.068e+006	4.865e+00
	4000rpm	2.179e+006	6.460e+00

VI. CONCLUSION

The specified parameters are used to produce the pump impellers, and a numerical analysis is carried out using a fluid flow simulation programme. The solid works design programme is used to simulate the pump impellor using a range of instructions. The solid works component file is changed into an IGS file before being imported into ANSYS Workbench. The comparison of analytical and theoretical data shows that the simulation can predict the features of the pump with accuracy. The number of blades increased from 6 to 10 at a certain rotation speed based on the numerical results. The tables indicate that the seven-blade pump impellor is operating efficiently. As a result, in compared to blades with various rotational speeds, the velocity rating is high.

REFERENCES

[1] C.P. Kittredge, "Centrifugal pumps used as hydraulic turbines", J. Eng. for Power, 1961, series A, pp. 74-77.
[2] B. Neumann, The interaction between geometry and performance of a centrifugal pump, Mechanical Engineering Publications Limited, London (1996).
[3] J. Fernández, E. Blanco, J. Parrondo, M.T. Stickland, and T.J. Scalon, "Performance of a centrifugal pump running in inverse mode", J. Power and Energy, 2004, 218(4), pp. 265- 271.
[4] K. Majidi, "Numerical Study of Unsteady Flow in a Centrifugal Pump", J. Turbomach., 2005, 127(2), pp. 363-371.
[5] R. Byskov, C.B. Jacobsen, N. Pedersen, "Flow in a Centrifugal Pump Impeller at Design and Off-Design Conditions—Part II: Large Eddy Simulations", J. Fluids Eng., 2003, 125 (1), pp. 73-83.
[6] M. Zhang and H. Tsukamoto, "Unsteady Hydrodynamic Forces due to Rotor-Stator Interaction on a Diffuser Pump With Identical Number of Vanes on the Impeller and Diffuser", J. Fluids Eng., 2005, 127(4), pp. 743-751.
[7] R. Dong, S. Chu, J. Katz, "Effect of Modification to Tongue and Impeller Geometry on Unsteady Flow, Pressure Fluctuations and Noise in a Centrifugal Pump", J. Turbomach., 1997, 119 (3), pp. 506-515



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY