CFD Simulation of the effect of increasing the number of impeller blades on the performance of a centrifugal pumps

Khalifa Bogarrasa
Department of Mechanical Engineering, Faculty of Technical Sciences, Sebha, Libya
Email: khlifa853000@gmail.com

ABSTRACT

Keywords:
Centrifugal pump impeller, CFD, Head, Efficiency, and Cavitation.

Centrifugal pumps are used in a wide range of industries, including agriculture and domestic applications. Computational Fluid Dynamics (CFD) is the most widely used simulation and analysis instrument. The flow field characteristics inside the turbomachinery are simulated using a 3-D numerical CFD tool. The flow conditions inside a centrifugal pump can be visualized using CFD simulation. The goal of this paper is to model and simulate the flow of water into a centrifugal pump's impeller. Three impellers were modeled using ANSYS CFX-15 techniques, with the only difference being the number of blades on each one (6, 8, and 10). The operating conditions, such as rotational speed and flow rate, were equal for all three impellers. The impeller's characteristics and results were compared for each in terms of head, efficiency, and cavitation phenomenon. It is found that the head and the efficiency are higher for the case of the impeller with 10 blades than for the two cases of 6 and 8 blades. Furthermore, the beginning of cavitation in the blade passage was detected with an increase in flow rate of more than 14 kg/s, and it was approximately equal for all cases. All the results were consistent with the theory of turbomachinery, demonstrating the effectiveness of CFD simulation tools in the design of this type of equipment.

1. INTRODUCTION

Centrifugal pumps are used extensively in both industrial and domestic applications. Without them, the transportation of fluid between different levels and long distances would be much more difficult and expensive. The forces created in the liquid mass inside a centrifugal pump are responsible for its movement and pressure increase [1]. The impeller, which is a place of blades that force the flow passing during them, is the main component of this equipment. Many authors refer to the impeller as the "heart" of a centrifugal pump because it is where the total energy changes. The fluid passing through it is accelerated by its centrifugal action, which increases the pressure energy [2]. The physical properties of the impeller, for instance the diameter, blade angle, and number of blades, have a significant impact on the performance of this apparatus. Changes to these parameters affect the quantity of hydraulic energy available at the impeller outlet, and thus the quantity of work available that the flow can perform [3]. The theory of energy conversion of the pump impeller is based on the angular momentum equation that takes into account one-dimensional flow. The impeller should have an unlimited number of blades under ideal circumstances and with no viscous fluid to appropriately direct the flow and provide the same pressure at each radius of the impeller [4]. As a result, impeller performance is affected by the number of blades. In practice, adding more blades improves flow guidance. However, the loss from viscous friction is greater. Fluid driving with lower energy losses and higher performance is a requirement of great importance in the design of centrifugal pump systems because it leads to lower energy costs and maintenance [5]- [6]. Many modifications in the company are made before the actual implementation of the project, but the cost of prototyping sometimes prevents the company from improving.

In this context, over the past years, computational fluid dynamics (CFD) has emerged as a new approach for problems relating to fluid flow. Before the installation of specific equipment in a given system, numerical simulations have the potential to accurately predict its performance [7]. Therefore, if designers are acquainted with what can be improved before building prototypes, changing the design of a plant is greatly easier and less expensive [8]. Only because greater computational capabilities are becoming accessible, it is possible to build these complicated machines using CFD techniques.

These evolved in tandem with an increase in numerical method accuracy, and CFD transitioned from a purely academic research tool to a competitive industrial company design office [9]- [10]. A general three-dimensional simulation of turbulent fluid flow was presented by Hajari to predict the pressure and velocity fields for a centrifugal pump. The impeller with 7 blades had the highest head coefficient when compared to 5 and 6-blade pumps across the entire range, according to research on the effect of the number of blades on the efficiency head coefficient as a selection criterion. Finally, it was found that the commencement of the separation was significantly influenced by the position of the blades concerning the tongue of the volute [11]. The author of [12] investigated the effects of stream parameters and blade leading edge angle on centrifugal pump performance and cavitation. A pump test rig was built, with a flow rate of $35m^3/h$ and a rotational speed of 1200 rpm. When the influence of the blade leading edge angle on cavitation was studied, it was discovered that as the blade leading edge angle increased, both the total efficiency and the total head increase under the tested operating conditions. It is noticed that for all of the investigated blade leading edge angles, the total head declines slowly as the net positive suction head diminishes.

Several studies have used a general three-dimensional simulation of turbulent fluid flow to predict pressure and velocity fields for a centrifugal pump. CFD was utilized to solve the flow field's governing equations. This study explored pump cavitation at the pressure drop zone on the blade using a finite volume method. It was easy determined that the absolute pressure was low enough to create cavitation, there was a significant spike in residuals, and the outlet pressure differential [13]- [14]. Other research has revealed that the number of impeller blades is a significant design element of pumps that has a significant impact on the pump's properties. The model pump has a 5 blade impeller and a design specific speed of 92.7. The blade number is varied to 7, 6, 4 with the casing and other geometric parameters keep constant. The internal flow fields and properties of centrifugal pumps with varying blade numbers are simulated and predicted using the commercial code FLUENT in cavitation and non-cavitation conditions. Rapid prototyping is used to create impellers with varying blade numbers, and their attributes are experienced in an open loop. The comparison of forecast values and experimental findings demonstrates that the predictions are correct. The head of the model pumps grows as the number of blades increases, the changing regulation of cavitation and efficiency characteristics is complicated, but there are optimum values of blade number for each one [15].

Authors in [16] compared the performance of centrifugal pump impellers with identical outlet diameters with varying of blade numbers. Fluent 6.3 software was used to evaluate the efficiency of a centrifugal pump running at 3000 rpm with impeller blades 5, 6, and 7. The numerical study found that as the head and blade number increased, so did the model's head and static pressure, however, the centrifugal pump's efficiency varied with the number of blades.

The present paper is concerned with the head, efficiency and cavitation phenomenon. The CFD techniques were used to model and simulate three impellers of a centrifugal pump with six, eight, and ten blades each. The simulation results were used to create the impeller's characteristic curves, which were then compared for each impeller to evaluate the pump performance.

2. Computational Model

2.1Modeling of impeller

The Ansys Vista CPC and Blade Gen software were used to simulate the inner flow field in the absence of cavitation. To solve the Reynolds-averaged Navier–Stokes equations (RANS) equations, the conventional k-turbulence model was used. To account for the impeller-volute interaction, a steady and moving reference frame is used in the simulation.

ANSYS BladeGen is a tool for creating geometry for turbomachinery blades. BladeGen is an ANSYS Blade Modeler component. The Blade Modeler is a specialized, user-friendly instrument for rapid 3-D modeling of rotating machinery components. The software, which incorporates considerable turbo machinery expertise into a user-friendly graphical interface, may be used to mixed-flow, design axial, and radial blade components for, compressors, fans, pumps, turbines, inducers, turbochargers, expanders, and other applications. The impeller main specifications used in the present study to clarify the effect of changing the number of blades are given in Table 1.

Table 1: Design Specification for Impeller

Parameters	Values
Flow rate	Q= 36 m^3/hr
Head	H = 20 m
Rotation speed	N = 3000 rpm.
Inlet flow angle	90º
Hub blade angle	27º
Mean blade angle	19º
Shroud blade angle	16º
Trailing edge angle	22.5º
Specific speed	Ns =92.7
Blade number,	Z= 6 or 8 or 10
Thickness/ tip diameter	0.03m
Hub inlet draft angle	30º

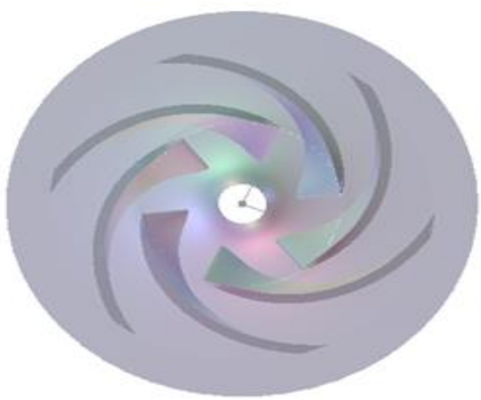

Figure 1 - Impeller with 6 blades.

The model created in the Blade Gen software is shown in figure 1.

Following the simulation, a different geometry was created, changing only the number of impeller blades, to investigate the impact of this physical change on the performance of the pump.

2.2Meshing

The program used to generate the mesh was ANSYS TurboGrid, which was installed on the ANSYS® platform. This software is a powerful tool that allows rotating equipment designers and analyzers to build high-quality hexahedral meshes while keeping the underlying geometry. In the ANSYS workflow, these meshes are utilized to tackle difficult blade passage problems. Ansys TurboGrid generates self-consistent meshes, which are crucial for minimizing mesh dependency when comparing performance forecasts between designs. The BladeGen blade geometry must be exported as TurboGrid input files. Exporting the blade geometry creates the following files: the shroud curve, the hub curve, and the profile curve. These files, in turn, generate a shroud fluid domain, a hub fluid domain, and the blade profile, respectively [17]. Because of periodicity structural features of the geometry of impeller, only single channel (passage) holding only one main blade and one splitter is modeled (Figure 2). To guarantee a good transition between the impeller and the diffuser and to decrease the computation costs. The same periodicity on geometry of diffuser is applied [18]. Figure 3 depicts the resulting mesh of six blades.

After that, An Automatic Topology and Meshing feature (ATM optimized) has been employed inside the impeller, which automates the creation of high quality hexahedral meshes desired for blade passages in the rotating impeller and the diffuser [19]. The obtained result from meshing is shown in table 2.

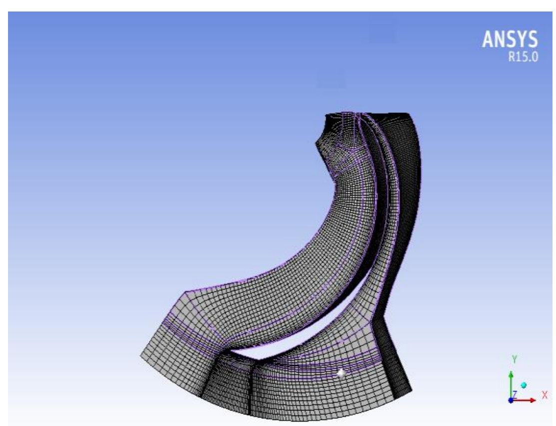

Figure 2 – Mesh in the 6 blades for single channel.

Table 2: Mesh Detail for Impeller

Mesh parameters	6 blades	8blades	10 blades
Number of nodes	144992	134021	135907
Number of Elements	128106	117436	118976

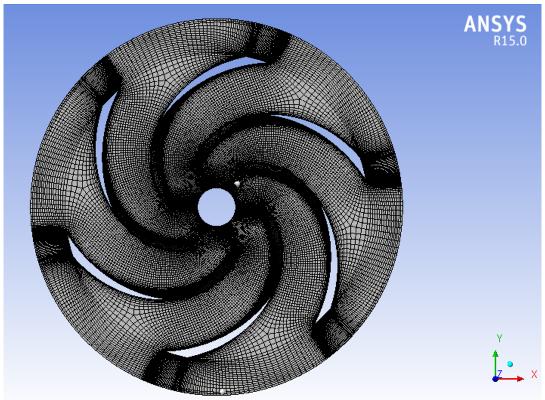

Figure 3 – Mesh in the 6 blades impeller.

2.3Boundary conditions:

The impeller domain of a centrifugal pump is seen as a revolving framework of reference with a constant rotational speed of 3000 rpm across the +Z axis. The pump's working fluid is water at 25 degrees Celsius. k- turbulence model is used for turbulence modeling with standard wall functions. The boundary conditions include an inlet static pressure that is exactly equal to the atmosphere (101325 Pa) and an output mass flow rate that varies between 8, 10, 12, 14, 16, 18, 20, 22, 24, and 26 kg/s. Figure 4 depicts the boundary conditions of six blades:

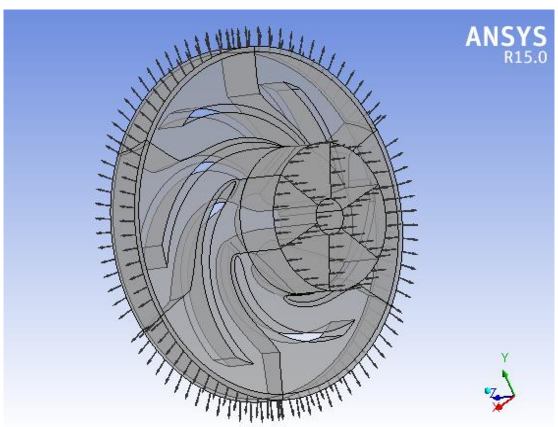

Figure 4-The boundary conditions of six blades.

The Ansys-CFX Solver is utilized to solve three-dimensional viscous fluid Navier–Stokes equations (N-S). The boundary conditions were the same in all three simulations. As a result, any variances in the results are solely due to physical differences between the impellers.

3 Simulation

The models were simulated after the boundary conditions were added, changing the mass flow rate values at the outlet of each impeller. One pressure falling was gained for each mass flow. There were two types of pressure studied: total pressure and static pressure. The static pressure variation does not account for velocity impacts in pressure gain, but the total pressure difference does.

The flow leakage and disk friction losses were not taken into account in the simulations. As a result, the produced hydraulic head indicates the precise work imparted by the impeller blades. Equation 1 is used to calculate this:

$$H = \frac{\Delta P}{\rho g} \qquad (1)$$

Where H is the total head of impeller blade, [m], ΔP is the total pressure increase [Pa], g is the gravity acceleration (9,81m/s^2) and ρ is the density of the water (997 kg/m^3).

4 RESULTS AND DISCUSSIONS

4.1Impellers' Curves

The results of the simulation of each impeller are displayed in Tables 3, 4, and 5 below:

Table 3: Results for the 6 blades impeller.

6 blades			
Mass Flow Rate [kg/s]	Head	Shaft Power	Efficiency
8	19.5	2810.6	55.6
10	18.5	3025.5	60.1
12	17.4	3227.3	63.4
14	16.2	3380.3	65.6
16	14.7	3392.5	67.8
18	13.2	3413.5	68.2
20	11.6	3254.3	69.9
22	9.9	3078.6	69.1
24	7.9	2817.8	66.7
26	6.2	2483.9	63.3

Table 4: Results for the 8 blades impeller.

8 blades			
Mass Flow Rate [kg/s]	Head	Shaft Power	Efficiency
8	19.8	2386.3	61.0
10	18.7	2865.5	64.1
12	17.4	3051.3	67.0
14	16.2	3183.0	69.9
16	15.1	3318.9	71.2
18	13.8	3329.8	72.9
20	12.4	3288.8	74.1
22	11.2	3267.9	73.9
24	9.5	3152.7	71.2
26	7.9	2924.9	68.8

Table 5: Results for the 10 blades impeller.

10 blades			
Mass Flow Rate [kg/s]	Head	Shaft Power	Efficiency
8	21	2505.9	62.9
10	20.3	2869.0	69.3
12	19.0	3102.6	72.1
14	17.8	3251.8	75.3
16	16.4	3309.9	77.8
18	14.9	3342.4	78.8
20	13.4	3273.8	80.0
22	11.6	3095.5	80.5
24	9.6	2870.1	79.1
26	7.5	2557.5	74.9

The impeller curves, illustrated in Figures 4 and 5, can be obtained using these data.

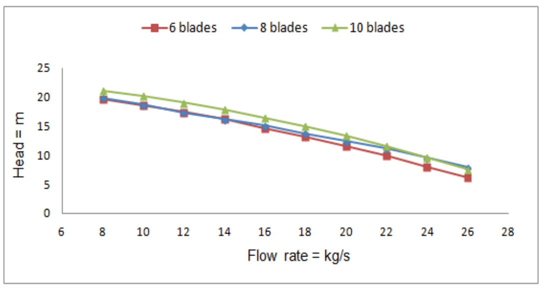

Figure 5. Shows the curve Result of Head vs Discharge.

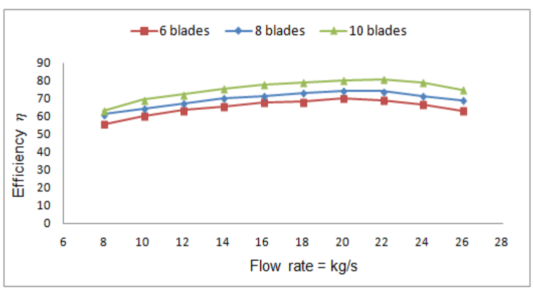

Figure 6. Shows the curve Result of pump efficiency vs Discharge

Figure 5 depicts a head characteristics curve obtained by varying the number of blades as the discharge increases. Because the pump's speed was kept constant, it is possible to notice that as discharge increases, the head drops. The highest value of head obtained was at 8 kg/s with ten blades, and the minimum head was at 26 kg/s with six blades. The form of the curve varies with the capacity of the pump. It is obvious from tables (3, 4, and 5) and the graph that the head is a function of flow rate. With lower flow rates, a high head can be obtained. The head versus discharge curve has a similar shape to a normal pump curve. As a result, it is feasible to conclude that the model with ten blades performs better in terms of the impeller head. This occurs across the entire volume flow rate range. By evaluating the curves individually, it is possible to see that their shapes are consistent with the curves contained in the manufacturer's catalog. These findings support those obtained by [20] and are consistent, in terms of trend, with Affinity Laws [21]. Under various operating conditions, an increase in the head results from the pressure differential between the suction and discharge regions, causing an increase in the head. The suction pressure rises as the pump capacity rises, but the discharge pressure falls. As a result, when a pump runs at a high flow rate, the inlet suction pressure drops faster than when it runs at a low flow rate, causing cavitation to occur more quickly [22] .

Figure 6 depicts the fluctuation of pump efficiency with increasing discharge at a 3000 rpm rotational speed. As the pump's speed was kept constant, it can be observed that efficiency increases with increasing discharge, achieves a maximum at rated circumstances, and then declines when discharge exceeds rated conditions. The efficiency versus discharge curve resembles a typical pump curve in form.

4.2 Pressure contours

Figures 7, 8, and 9 show the pressure distribution on the mid-span of the impeller at a flow rate of 10 kg/s (nominal flow) for the three simulated impellers (6, 8, and 10 blades). The pressure contours indicate a continuous pressure rise from the leading edge to the trailing edge of the impeller due to the dynamic head created by the rotating pump impeller. The total pressure on the pressure side of the blade is found to be larger than that on the suction side. The pressure difference between the suction side and the pressure side of the impeller blade grows from the leading edge to the trailing edge of the blade. The suction side's leading edge of the blades has the minimum static pressure value within the impeller. The total pressure patterns fluctuate along the impeller's span. Low total pressures are recorded at the impeller's hub. Total pressures rise as the span increases due to the high dynamic head near the blade's tip. Because of the vane thickness, High velocity and low total pressure are noticed towards the leading edge on the suction side of the blade. It can be observed that the pressure distribution is more regular with

10 blades than with 8 and 6 blades. It is the outcome of improved flow guiding that leads to improved velocity distribution and, as a result, improved pressure distribution.

Figure 7-: Pressure Distribution of Centrifugal Pump Impeller with 6 blades

Figure 8-: Pressure Distribution of Centrifugal Pump Impeller with 8 blades

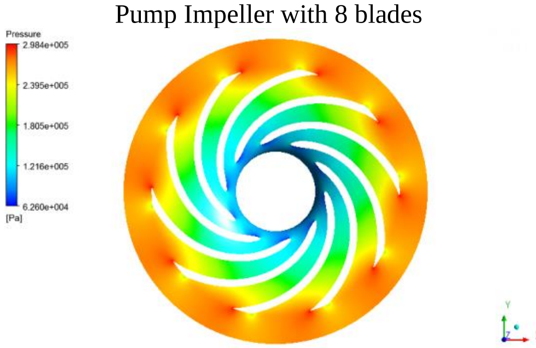

Figure 9-: Pressure Distribution of Centrifugal Pump Impeller with 10 blades

4.3 Cavitation analysis

Figures 10 and 11 depict the cavitation study of an impeller model with six blades operating at a constant 3000rpm with mass flow rates of 16 and 26 kg/s. The plot's red area indicates that the absolute pressure is less than or equal to 3170 Pa (pressure saturation). That means a pressure decrease and attaining the vapor pressure on the impeller blades. This causes cavitation and has an impact on the centrifugal pump's performance. At flow rates from 16 to 26 kg/s, the phenomenon of cavitation can be noticed on the front of the blades at all of type of impellers. However, cavitation-free functioning can be detected at the flow rates from 8 to 14 kg/s. According to the above analysis, cavitation becomes more intense with increasing of flow rate at the leading edge of the blade.

Figure 10-: Cavitation Model on six blades with 16 kg/s mass flow rate.

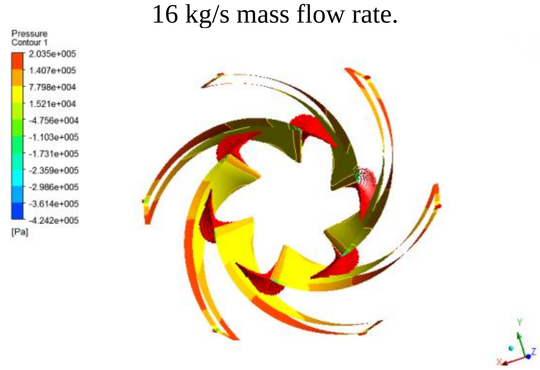

Figure 11-: Cavitation Model on six blades with 26 kg/s mass flow rate.

5. Conclusions

The effects of blade number on the inner flow of centrifugal pump characteristics were investigated using computational fluid dynamics in this study. Based on the above results, several following conclusions were drawn concerning the impact of blade numbers and varying flow rates on centrifugal pump performance.

- Due to the dynamic head produced by the moving pump impeller, the pressure contours show a continuous increase of pressure from the leading edge to the trailing edge of the impeller.
- Because of the thickness of the blade, high velocities and low pressure are observed towards the leading edge of the blade.
- Total pressure loss is detected along the trailing edge of the blade Because of the existence of the trailing edge wake.
- The overall head of the pump decreases as the design flow rate increases.

- Low pressure is noticed towards the leading edge of the blade.
- The simulation results revealed that the impeller blade number of 10 generated the maximum efficiency.
- The case of using 10 blades was determined to be the best case for the analyzed instances, followed by the case of using 8 blades, and finally the case of using 6 blades.

References

[1] Robert, R. Centrifugal Pumpes Design and Application, United States: Gulf publishing Company, Houston, 1992.

[2] Bogarrasa, K. "The effect of rotational speed variation on the performance in the centrifugal pump," *Journal of Research and Applications in Mechanical Engineering,* vol. 10, no. 1, pp. 1-2, 2022.

[3] Purushothaman,K. Rajendran, S. "Analysis of a centrifugal pump impeller using ANSYS-CFX, " *International Journal of Engineering Research & Technology (IJERT),* vol. 3, 2012.

[4] Yang, C. Wang, F. "CFD Analysis and Optimum Design for a Centrifugal Pump Using an Effectively Artificial Intelligent Algorithm," *Micromachines*, vol. 13, no. 8, pp. 2-20, 2022.

[5] Seppo, K. Principles of turbomachinery. 2nd ed, Hoboken, New Jersey: Wiley, 2019.

[6] Jianfang, C. Weidong, S. "Influence of blade inlet angle on the performance of a single blade centrifugal pump," *Engineering Applications of Computational Fluid Mechanics* , vol. 15, no. 1, pp. 462-475, 2021.

[7] Yedidiah, S . "A Study in The Use of CFD In The Design of Centrifugal Pumps," *Engineering Applications of Computational Fluid Mechanics,* vol. 2, pp. 331-343, 2008.

[8] Ragoth, R. Nataraj, M. "Analyzing pump impeller for performance evaluation using RSM and CFD," *Desalination and Water Treatment,* vol. 52, pp. 6822-6831, 2014.

[9] Gato, L. Pascoa, J. Mendes, A. "A fast iterative inverse method for turbomachinery blade design," *Mechanics Reserch Communications,* vol. 36, pp. 630-637, 2009.

[10] Govendan, N. Siva, P. Anbarasu, T. “Influence of number of impeller blades on the performance of centrifugal pumps by using ansys software,” *International Journal of Research and Development,* vol. 4, no. 10, pp. 164- 187, 2019.

[11] Hajari, A. Jafarzadeh, B. Alishahi, M. "The flow simulation of a low specific speed high speed centrifugal pump," *Appl. Math. Modell*, vol. 35, p. 242–249, 2011.

[12] Spyridon, D. "Experimental Investigation and Passive Flow Control of a Cavitating Centrifugal Pump," *Int. J. Rot. Mach,* vol. 2012, pp. 1-8, 2012.

[13] Pande, R. "Computational Fluid Dynamics (CFD) of Centrifugal Pump to Study the Cavitation Effect," *Int. J.Theor. Appl. Res. Mech. Eng,* vol. 4, pp. 60-64, 2015.

[14] Chao,W. Shi, B. Ruan, H. "The Cavitation Characteristics of High Speed Centrifugal Pumps With Different Impeller Types," *Front. Energy Res,* vol. 10, pp. 1- 9, 2022.

[15] Gamal, R. Magdy, A. Khalil, Y. "Effect of impeller blades number on the performance of a centrifugal pump," vol. 58, no. 1, 2019.

[16] Chakraborty, S. Datta, P. Choudhary, K. "Performance prediction of centrifugal pumps with variations of blade number, " *J. Sci. Ind. Res,* vol. 3, pp. 373-378, 2011.

[17] ANSYS, CFX Solver Theory Guide, ANSYS Inc, 2016.

[18] Tamer, S. Aly, M. Mohamed, K. "Micro TJE centrifugal compressor performance prediction," *Engineering of Science and Military Technologies*, vol. 2, no. 4, pp. 185-203, 2018.

[19] Marchionni, M. Sayad, S. " Modelling and performance analysis of a supercritical CO2 system for high temperature industrial heat to power conversion at off-design conditions," in *3 rd European supercritical CO2 Conference*, Paris, France, 2019.

[20] Sang, S. Rakibuzzaman, H. "A Study on Multistage Centrifugal Pump Performance Characteristics for Variable Speed Drive System," *Procedia Engineering*, vol. 105, pp. 270-275, 2015.

[21] Robert, L. Pumping Station Design. 3rd edition, Butterworth, 2008.

[22] Shuai, H. Song, W. "The effect of Rotation Speed on Cavitation of Small Flow Micro-High Speed Centrifugal Pump," in *2nd International Conference on Materials Science, Machinery and Energy Engineering*, 2017.