Improving the Efficiency and Performance of Centrifugal Pump through Model Development and Numerical Analysis for the Pump Impeller

Malik N. Hawas, Akeel Abbas Mohammed, and Audai Hussein Al-Abbas
Al Mussaib Technical College, Al Furat Al Awsat Technical University, Babil , Iraq
Email: com.mlk@atu.edu.iq, com.a.abbas@atu.edu.iq, aalabbas@atu.edu.iq

***Abstract*— In centrifugal pumps, the flow physics and dynamic performances are generally affected by any modification in blades shape and design layout. The investigations of estimating the optimum impeller geometry and the appropriate number of blades with a fitting angle need to find a better insight. At the present time, the dominant verification method used for investigation these characteristics are numerical simulation. Commercial code Fluent (CFD) under ANSYS software has been used for investigation the working characteristics of pump impeller under different conditions by using two different geometrical models. For these purposes, two different impellers with different blades number in 3D configuration are designed by using single arc curved blades design method and submitted for analysis and simulation to determine the best characteristics through comparison procedure. Conventional impeller case used as a base for comparison purposes during recording any changes associated with each individual case such as heads, flow rates and efficiency. The solution of dynamic analysis is carried out to approve that impeller structure can resist and withstand many variable loads and turbulent conditions. Results approved that, the static pressure, total head and efficiency are proportional with blades numbers and blades geometry. Even more, it is found that there are some important parameters have some effects on centrifugal pump performance such as inlet diameter of impeller and blades angle. It can be concluded from this works that suitable predicted results are estimated, and these analyzed results can used and adopted for this type of centrifugal pumps.**

***Index Terms*— centrifugal pump, impeller, blades, CFD, efficiency**

I. INTRODUCTION

Nowadays many research applications in hydraulics and fluid dynamics fields are focusing on decrease the losses, power consumption and increase the efficiency of the specific equipment like pumps. Centrifugal pump is an important type which used to converts the mechanical energy to kinetic energy due to force generation which imparts on fluid through the impeller blades [1]. Rotation vanes in centrifugal pump which enclosed inside the pump housing is used to transmit and impart the kinetic energy to the fluid by centrifugal force, and then the liquid will forced by pressure to move through the impeller vanes [2].

Design parameters like blades number has highly effect on pump efficiency. Blades number also has some influences on the total pump characteristics like efficiency and head. When the blade number increase, the flow velocity will increase due to the crowding near diffusion zone, and this flow will decrease when the impeller blades is few due to increase in diffusion losses [3]. In this time being; the revolution in computing technology and the rapid development in many analysis software like (ANSYS), makes the numerical simulation and computational fluid dynamics (CFD) among the better choices as a tool to study and estimate the optimum characteristic of pumps. This type of simulation is very useful in predicting and estimating many characteristics of pumps performance and gives many solutions before any further steps [4].

The physical situation of the working pump is including the most factors governing the performance which can predicted and the actual values will be visible and observed through computational method. Simulation analysis by (CFD) is a powerful design and estimating tolls to reduce the time, cost and enhance the results. It can reduce the error and offering in a very wide range by giving alternative choices with the possibility of running the experiments many times with difference parameters [5].

Optimization process needs high talent and skills from designers, but the incorporation of these skills with (CAD) system will fast up the generation of design process and give reasonable solutions for many design parameters [6]. Researches approved that, any compound between short and long blades in pump impeller will leads to improve in pump efficiency, because it can strongly prevent any development in which it called (wake flow) due to uniform distribution in velocity near the inlet suction [7]. Compound of short and long blades in same impellers casing will have a good impact on pump efficiency due to variation in setting places. The main characteristics and performance of the pump can predicted by right numerical and simulations procedure. Many simulation results approved that, when the angle of blades increase the flow rate efficiency of the pump will increase [8].

Manuscript received May 5, 2018; revised July 14, 2019.

doi: 10.18178/ijmerr.9.1.60-65

It's found that the pump with high blades thickness and little tip clearance can provide suitable results and promising better hydro-dynamic performance. The main purpose in small tip is to minimize the dead area between the impeller and the case surface [9]. The difference in pressure between suction side and diffusion side at the blade edge will decrease by using of (bladelet) technique. This difference in pressure will decreases the fluid circulation in the outlet zone of impeller and directed the fluid toward suction side [10, 11].

II. Design Process and Model Development

In this paper; backward impeller blades type are adopted, and curved blades single arc design method is used to for layout the impeller blades. Backward blades are more efficient in maximize power and impart high rotational force to the fluid. "Fig. 1," and "Fig. 2," illustrates two dimensional and three dimensional impeller for the two types of impeller used in this research.

The case of centrifugal pump which containing impeller disc are designed according to same features and configuration of impeller to fit exactly which the other dimensions and tolerances.

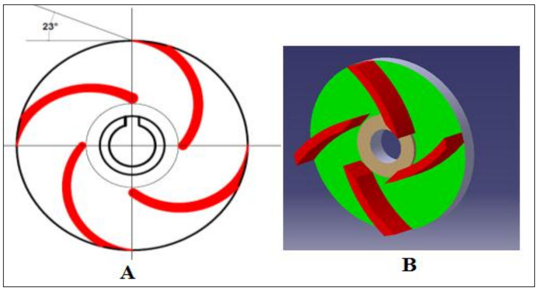

Figure 1. (2D) and (3D) for first impeller model.

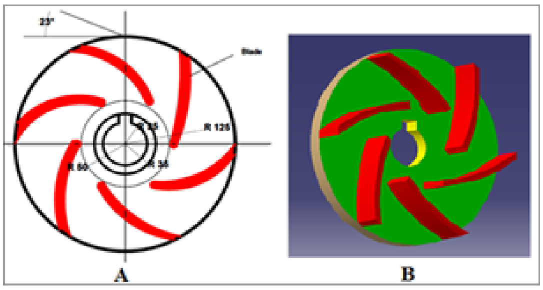

Figure 2. (2D) and (3D) for second impeller model.

"Fig. 3, A" and "Fig. 3, B" shows (3D) assembly model containing impeller, rotating shaft, casing and discharge side for both models.

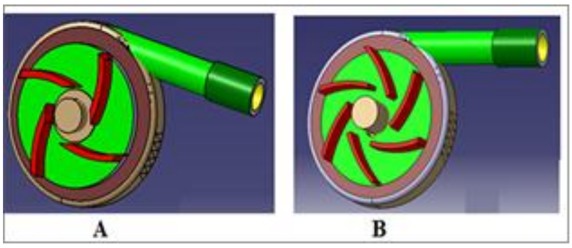

Figure 3. (A) Assembly of four blades, (B) Assembly of six blades model.

The main design parameters of impeller that used for investigation and implementation the characteristics of centrifugal pump and find out the calculations and results are listed in Table I.

TABLE I. Main Design Parameters Used in Calculations

No.	Designation	Values
1	Impeller Outlet Diameter	250mm
2	Impeller inlet Diameter	75 mm
3	Blades Number	(4) and (6)
4	Angular Speed (N)	1000 RPM
5	Head (H)	28 m
6	Shaft diameter	50 mm
7	blade outlet angle (β)	23 °

Water is the circulating fluid used in this work, and the main physical properties of water are listed in Table II.

TABLE II. Main Design Parameters Used in Calculations

No	Designation	Values
1	Density (ρ)	1000 kg/m^3
2	Viscosity (μ)	8 x 10^{-4} Pa–s
3	Conductivity(k)	0.677 W/m–K
4	Specific Heat (C_p)	4216 J/kg–K

Some important calculations must be implemented like the hydraulic diameter, available net positive suction head (NPSHa) and suction specific speed.

$$D_h = 4A / P_w \tag{1}$$

where (D_h) is hydraulic diameter, (A) is the cross-sectional area of section side and (P_w) is the wetted perimeter.

$$NPSH_a = P \pm H + H_f - H_{vp} \tag{2}$$

where (P) is the absolute pressure on the surface of the liquid, (H) is the elevation distance from the surface of the liquid, (H_f) is the friction loss, (H_{vp}) is the vapour pressure of the liquid.

It's important to know that (NPSHa) should be > (NPSHr). The specific speed at section is:

$$\text{Suction Specific speed (Nss)} = N\sqrt{Q} / (NPSHr)^{0.75} \tag{3}$$

where (Q) is the discharge and (N) is impeller speed in (r.p.m).

III. Simulation and Numerical Analysis

Computational fluid dynamics code (CFD) by ANSYS software has been used for prediction and capture the important characteristics of speeds and pressures. The specific design speed in this model was (1000) rpm, with two different impeller blades numbers as (4, 6). The others geometrical parameters are kept constant.

Different boundary conditions are used in this analysis depending on each individual case. For inlet velocity zone (suction zone); Cartesian component system are used, the inlet velocity is (0.5) and fluid temperature is (293) K. Back flow intensity ratio (4) % is used in pressure outlet zone conditions and (No slip) condition is used in wall fluid zone. “Fig. 4,” illustrate some boundary conditions in blades impeller.

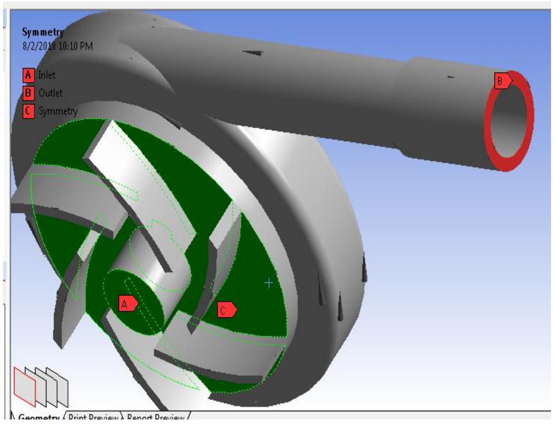

Figure 4. Boundary condition in blade impeller.

Program controlled mesh type are used for element order in both (4) blades and (6) blades impeller. Statistical of element mesh for the (4) blades impeller model show that there are (196008) elements and (289556) nodes. These big numbers of elements and nodes refers to mesh accuracy and valuable predicted results. Smooth transition also used in inflation option which gives high accuracy and better element distribution. “Fig. 5,” and “Fig. 6,” shows the mesh distribution and display properties with statistical.

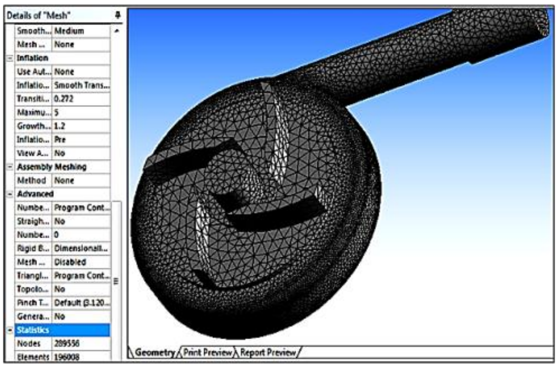

Figure 5. Mesh (4) blades impeller

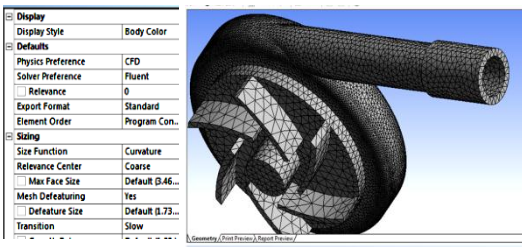

Figure 6. Mesh (6) blades impeller.

Smoothing the mesh is an important facility which can generate elements and increase the mesh accuracy. “Fig. 7,” and “Fig. 8,”shows impellers after mesh smoothing.

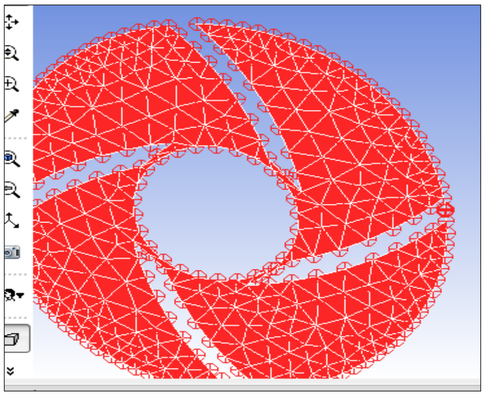

Figure 7. Four Impellers after mesh smoothing.

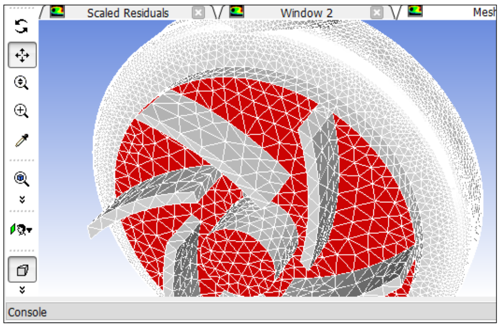

Figure 8. Six Impellers after mesh smoothing.

For more evaluation to the method above; some experiments have been carried out. The reference (Shojaeefard et al 2012), can be used as a guide for authors in explanations and as a tests programs. System of steel pipes with diameter of (60) mm connected with axial suction centrifugal pump, tank (1500) litters and the flow is controlled by gate valves on both suction and discharge pipe with pressure gages as a completed rig for this experiment. “Fig. 9,”illustrate completed Assembly of experimental rig.

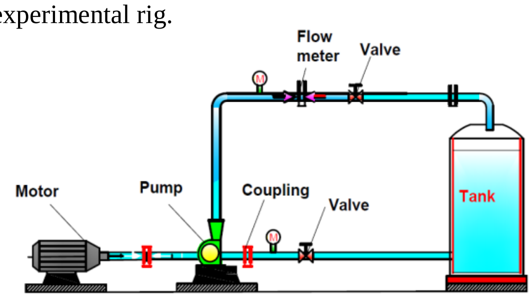

Figure 9. Assembly of experimental rig.

IV. Results and Discussion

In this research, the efficiencies, distribution of total pressures and head changing are investigated for a (3D) steady state flow regarding to the numbers of blades at a constant velocity (1000) rpm. Many parameters like the volute dimension as well as the clearances between volute and rotor have been taken in to confederation.

Contours of pressure distribution show the maximum pressure values were registered in the inlet suction in case of 6) blades impeller, but the distributions in other zones are uniform, while the maximum pressure values in 4 blades was at the discharge zone. This variation can be due to high dynamics circulation and fluid momentum in this case. “Fig. 10,” “Fig. 11,” and show the pressure contours in both (4) and (6) blades impeller.

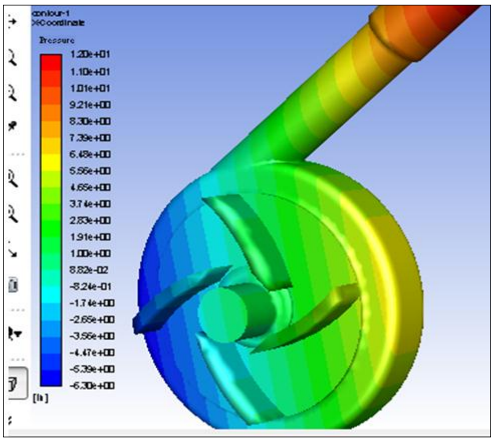

Figure 10. Pressure contours (4) blades impeller.

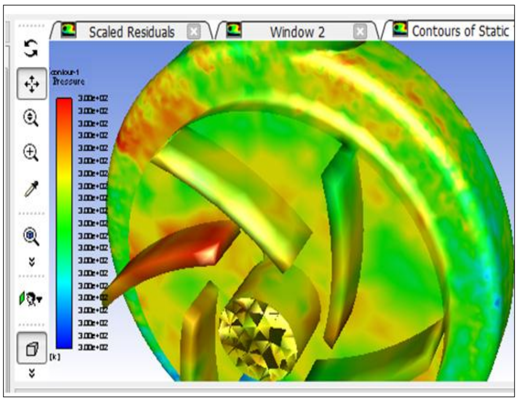

Figure 11. Pressure contours (6) blades impeller.

In high pressure and velocity values, many types of stresses will raised up. Even though each types of flow, whatever is the impeller type can cause stresses, but these values will be at maximum in section side and especially when the pressures and dynamic circulation is very high. Even more, these stresses will be the main causes of cavitation phenomena. Contours of mises stresses distributions are illustrated in “Fig. 12,” and “Fig. 13,”.

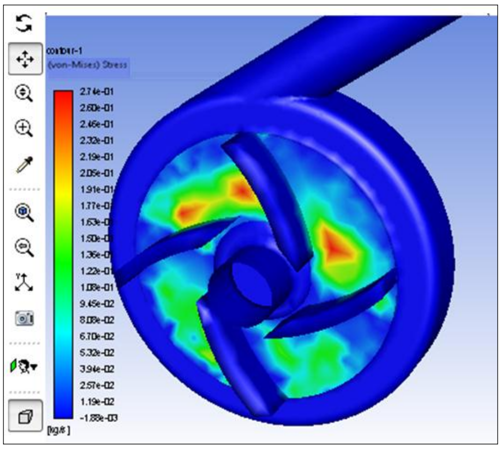

Figure 12. Contours of Von-Mises stresses distributions in 4 blades.

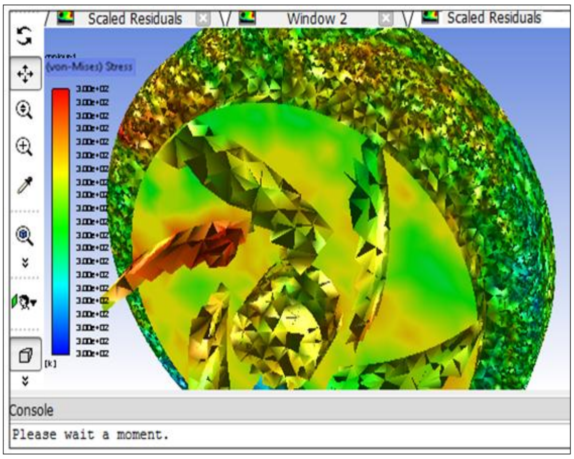

Figure 13. Contours of Von-Mises stresses distributions in six blades.

Displacements of volute diameter with respect to its width due to the pressure variations show that the maximum displacement in (4) blades impeller shaft is more than the similar one in (6) blades model.

These variations are observed in section side and the differences between the two model is due to the high impeller interference and small clearance in (4) blades model. Contours of displacement are shown in “Fig. 14”, and “Fig. 15”.

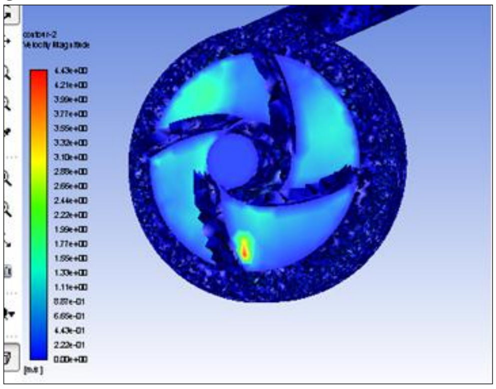

Figure 14. displacement Contours of 4 blade model.

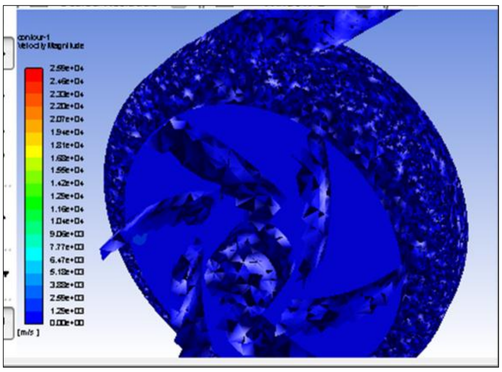

Figure 15. displacement Contours of 6 blade model.

Both of dynamic and hydraulic performance of the pump was evaluated for each blade configuration in addition to conventional configuration. Effect of each blade configuration was evaluated and compared with the conventional one. The evaluation and comparison of discharge with head and for the (4) blades model show that the conventional impeller head is more than simulation. Also the values of efficiency in a conventional impeller are more than in simulation. These differences can due to variations in boundary conditions and interactions. “Fig. 16,” and “Fig. 17,” Show these variations.

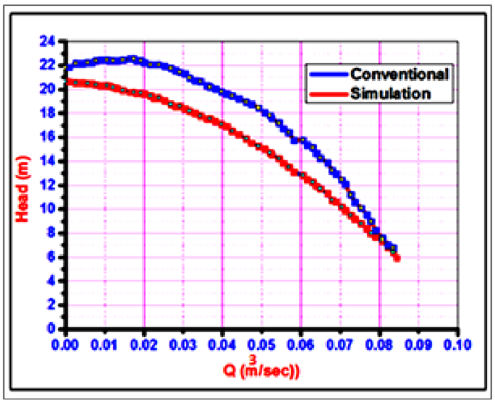

Figure 16. Head comparison between conventional and simulation in (4) blade model.

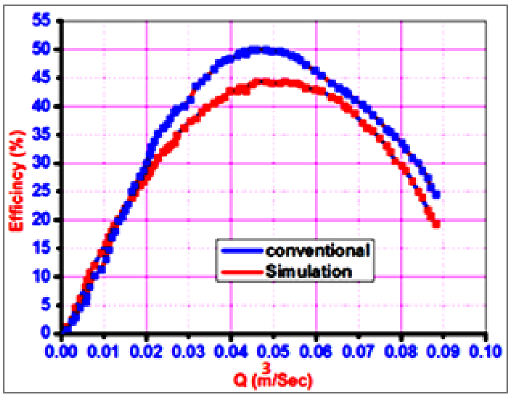

Figure 17. Efficiency comparison between conventional and simulation in (4) blade model

For the (6) impeller blades; its fund that the head and efficiency for conventional values is higher than simulation, but is better and higher than the values in (4) blades model. “Fig. 18,” and “Fig. 19,” shows these variations.

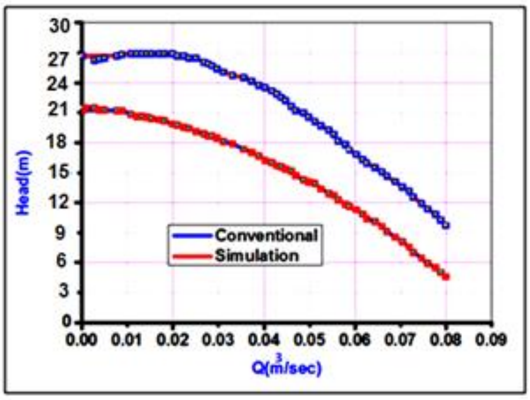

Figure 18. Head comparison between conventional and simulation in (6) blade model.

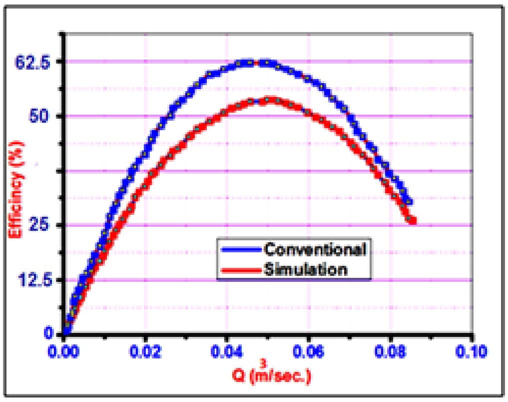

Figure 19. Efficiency comparison between conventional and simulation in (6) blade model.

Results comparison was made between the heads values of conventional and simulation for the (4) and (6) model. “Fig. 20,” below illustrate this comparison.

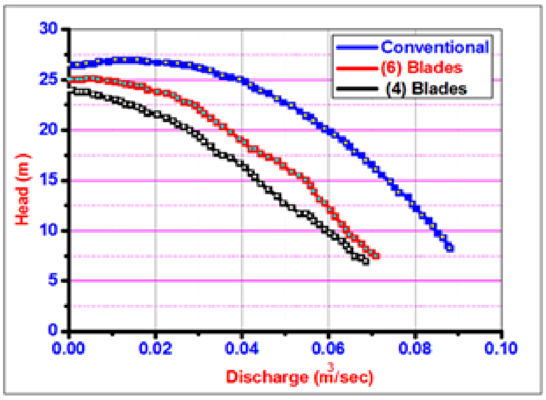

Figure 20. Comparison between heads values of conventional and simulation.

As a results summery; for this design method (single arc curved blades) it have been estimated that pump head and efficiency will be increase whenever blades number is increase with little constrains. Evermore, the pressure and velocity distribution is uniform in suction side due to high space for fluid circulation between blades, and this will lead to eliminate the cavitation chance. The data base of developed model and numerical analysis are created to be used for many similar impeller geometries, and this will help to repeat the analysis with simple modification. This property is very important and can used for results optimization.

The evaluation of hydraulic and dynamic performances was per-formed by measuring the pump head, discharge, power, and vibration. Then these measurements were analyzed to indicate the effect on each impeller configuration. So, the changes in flow rate and heads which associate with each blade configuration are recorded accordingly.

V. Conclusions

A 3D CFD model has been developed to model two different impellers in the centrifugal pump. Base on the experiment found in the literature, the boundary conditions and operating parameters are precisely implemented. A grid independency test is carried out in this CFD study in order to provide a high accuracy simulation in estimating the distribution of surface pressure and speed values. Two different impellers are modelled and investigated, one has four blades and the second has six blades in the same size of the centrifugal pump. In short, this research paper is revealed for some conclusions; they are:-

- Due to many shortage in using these types of pumps; like cavitation, less head, and parts fitting problems; it is necessary to study these problems to find out a suitable solutions.
- Any increase in blades number will help to avoid the stagnation phenomenon and terminate or reduce the head losses.
- Cavitation phenomena are proportional with the pump head; when head decrease the cavitation is decrease and vice versa.
- Pump head is also proportional with blades numbers for specific limits that should not exceed, otherwise many undesirable phenomena will arise like cavitation and pressure stagnation.
- When the blade number increase, the zone with low pressure at suction side in the inlet will raise up gradually, and the distribution of pressure become not uniform, while it was uniform at diffusion side.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Malik N. Hawas conducted the research; Akeel Abbas Mohammed analyzed the data and Audai Hussein Al-Abbas wrote the paper; all authors had approved the final version.

REFERENCES

[1] D. Campbell, N. Brown, and J. Cox, "Centrifugal Pumps". [Online]. Available: (2016). https://us.grundfos.com/content/dam/Global.

[2] L. Hedi, K. Hatem, Z. Ridha, "Numerical flow simulation in a centrifugal pump," *International Renewable Energy Congress, IEEE Trans, Electron Devices*, vol. ED-11, pp. 34-39. 2010.

[3] M. Asuaje, F. Bakir, S. Kouidri, R. Noguera, and R. Rey. "Validation d'une d´emarche de dimensionnement optimize des roues centrifuges 2D par comparison avec les outils de la simulation nu´erique (CFD)," in *Proc. 10`eme Conference Annuelle de la Soci ́et ́e Canadienne de la CFD*, pp. 560–565, Windsor,Ontario, Canada. (2002).

[4] S. Chakraborty and K. M. Pandey. "Numerical studies on effects of blade number variations on performance of centrifugal pumps at 4000 RPM," *IACSIT International Journal of Engineering and Technology*, vol. 3, no. 4. (April, 2011).

[5] J. L. Parrondo-Gayo, J. Gonz ´alez-P ´erez, and J. Fern ´an-dez-Francos. "The effect of the operating point on the pressure fluctuations at the blade passage frequency in the volute of a centrifugal pump," *Transactions of the ASME, Journal of Fluids Engineering*, vol. 124, no. 3, pp. 784–790, 2002.

[6] K. V. Karanth, N. Y. Sharma. "Numerical analysis on the effect of varying number of diffuser vanes on impeller-diffuser flow interaction in a centrifugal pump," *World Journal of Modelling and Simulation*, vol. 5, pp. 63-71, 2009.

[7] Y. Tian, J. Wang, L. Wang, "Optimal design schemes of centrifugal pump compound impeller at super-low specific speed," vol. 46, 2015.

[8] A. Goto, M. Nohmi, T. Sakurai, Y. Sogawa, "Hydrodynamic design system for pumps based on 3-D CAD, CFD, and inverse design method," published by EBARA Corporation, Tokyo, Japan. (2002).

[9] E. Blanco-Marigorta, J. Fern ´ andez-Francos, J. L. Parrondo-Gayo, and C. Santolaria-Morros, "Numerical simulation of centrifugal pumps," in *Proc. ASME Fluids Engineering Summer Conference (FEDSM '00)*, Boston, Mass, USA, paper FEDSM00 11162. (June 2000).

[10] A. F. A. Hassan, H. M. Abdalla, and A. Abou El-Azm Aly. "Centrifugal pump performance enhancement by blade shape modification," GT2017-63023, pp. V02BT41A001, 2007.

[11] V. A. Gundale and G. R. Joshi, "A simplified 3d model approach in constructing the plain vane profile of A radial type submersible pump impeller," *Research Journal of Engineering Sciences*, vol. 2, no. 7, pp. 33-37, 2013.

Malik N. Hawas was born in Baghdad, IRAQ in 1968. He received the B.S. degree in Mechanical engineering from Al– Furat Al-Awsat Technical University/ IRAQ and M.Sc degree also in Mechanical engineering from KUCHURAT University/ INDIA, in 2009. At 2015 he got Ph.D. degree in mechanical engineering from Bucharest University/ Romania.
From 2007 to 2008, he works as a Senior Engineer in mechanical Laboratory. Now, he is a lecturer in Mechanical Engineering Department, Al– Furat Al-Awsat Technical University/ IRAQ. He has many publications in high impacts factor journals.

Akeel Abbas Mohammed now, is a lecturer in Mechanical Engineering Department, Al– Furat Al-Awsat Technical University/ Iraq..
He has many publications in high impacts factor journals. He got Ph.D. degree in mechanical engineering from The University of Technology / Iraq.
Reviewer for the journal of thermal engineering.

Audai Hussein Al-Abbas Now is a lecturer in Mechanical Engineering Department, Al– Furat Al-Awsat Technical University/ IRAQ.
He got Ph.D. degree in mechanical engineering from Milburn University / Australia
He has many publications in high impacts factor journals.
Reviewer for the following journals

- Journal of Petroleum Science and Engineering.
- Renewable Energy.
- Experimental Thermal and Fluid Science.