54. *CSRID Journal, Vol. 11 No. 1 Februari 2019, Hal. 54-61* *ISSN: 2085-1367*
https://www.doi.org/10. 22303/csrid.11.1.2019.54-61 *eISSN :2460-870X*

Computational Fluid Dynamics: Flow Field of an Impeller

Yaser A. Jasim[1], Senan Thabet[2], Thabit H. Thabit[3]
[1]Cihan University-Erbil, Iraq,
[2]University of South Wales Cardiff, UK,
[3]Ninevah University, Mosul, Iraq
e-mail: [1]yaser.jasim@cihanuniversity.edu.iq, [2]stc@englandmail.com, [*3]thabit.acc@gmail.com

Abstract
This paper aims to develop a centrifugal pump with variable speed controlled by inverter to adjust the flow and the head by the operator and to observe the velocity profile and pressure distribution by using computational fluid dynamic simulation program using Pheonics software. According to the simulation results, the researchers concluded that the pressure increases gradually from impeller inlet to impeller outlet and centrifugal pumps have reasonable price and maintenance as well as high efficiency comparing to the price and been more advanced by coating the inner surfaces with anti-corrosion and smooth material to reduce the friction and raise the efficiency.

Keywords— Centrifugal pump, CFD, Flow Field, Ansys

1. Introduction

Centrifugal pump is a machinery or device for raising or transferring fluid. Pumps are one of the most often used mechanical equipment and can be found in almost every industry. The family of pumps is separated in to positive displacement pumps and kinetic pumps as shown if figure 1.

Kinetic pumps are centrifugal pumps. Pumps are designed to overcome the resistance and height. There are too many examples of wide range industries using pumps especially centrifugal pumps as they are cheap to run, simple and easy to maintain.

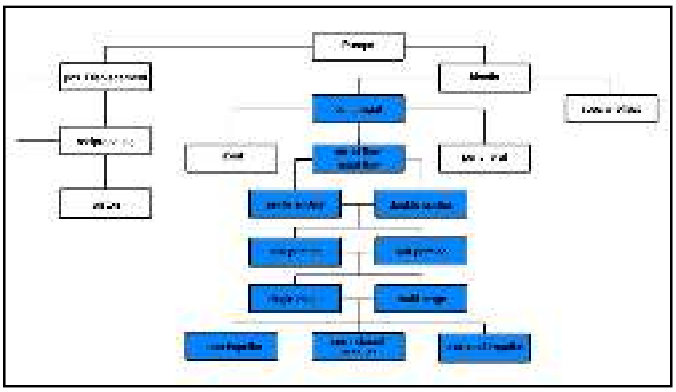

Fig. 1. The Family of Pumps

2. Literature Review

A centrifugal pump is a pump that uses a rotating impeller to increase the pressure of a fluid. The fluid enters the pump near the rotating axis which holding the impeller, streaming into the rotating impeller. The impeller consists of a rotating disc with several vanes attached in a certain angles. The vanes normally slope backwards, away from the direction of rotation. When the fluid enters the impeller at a certain velocity due to the suction system, the rotating impeller vanes capture it. The fluid is accelerated by pulse transmission while following the curvature of the impeller vanes from the impeller centre (eye) outwards. It reaches its maximum velocity at the impeller's outer diameter and leaves the impeller into a diffuser or volute chamber as shown in figure 2.

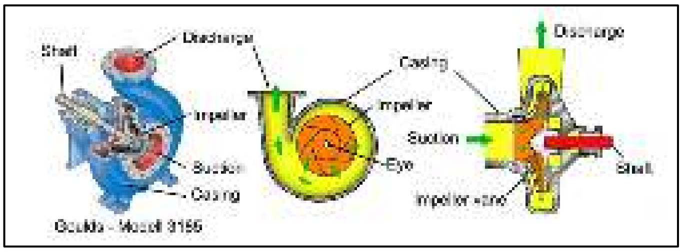

Fig. 2. Goulds – Modell 3185

Therefore, the centrifugal force helps accelerating the fluid particles because the radius at which the particles enter is smaller than the radius at which the individual particles leave the impeller. Now the fluid's energy is converted into static pressure, assisted by the shape of the diffuser or volute chamber. The process of energy conversation in fluids mechanics follows the Bernoulli principle (eqn.1) which states that the sum of all forms of energy along a streamline is the same on two points of the path. The total head energy in a pump system is the sum of potential head energy, static pressure head energy and velocity head energy.

As a centrifugal pump increases the velocity of the fluid, it is essentially a velocity machine. After the fluid has left the impeller, it flows at a higher velocity from a small area into a region of increasing area. Therefore, the velocity is decreasing and so the pressure increases as described by Bernoulli's principle.

This results in an increased pressure at the discharge side of the pump. As fluid is displaced at the discharge side of the pump, more fluid is sucked in to replace it at the suction side, causing flow.

Back in 1475, the Italian renaissance engineer Francesco di Giorgio Martini describes a water or mud lifting machine in one of his treatises that can be characterised as the first prototype of a centrifugal pump.

The French physician Denis Papin invented the first true centrifugal pump in 1689, when he was experimenting with straight vane impellers. British inventor John Appold introduced the first curved vane impeller in 1851. Nowadays only curved impellers are used in 3 different types. There are pumps with open, semi-open and enclosed impellers.

These days there are three different types of impellers as seen in figure 3. :

1. Open impeller.
2. Semi Open Impeller.
3. Closed impeller.

Fig. 3. The Different Types of Impellers

So, centrifugal pump consists mainly of the following parts:

- Casing

https://www.doi.org/10. 22303/csrid.11.1.2019.54-61 *eISSN :2460-870X*

- Impeller
- Shaft
- Sealing

In this paper the researchers are going to concentrate on casing and impeller as they are the main parts of the impeller.

A. Casing:

The casing designed to force the fluid to discharge from the pump and convert velocity in to pressure and reduce friction losses. The pump casing designed in spiral shape to avoid turbulences and increasing the pressure. The material of the pump's casing should be selected neatly, cast iron mainly used to manufacturing standard pump casing, many casing pumps are coated or made from more wear resistance material due to the cast iron is not that resistance against cavitations as shown in figure 4.

Fig. 4. The Spiral Shape of Pump

B. Impeller:

The researchers chose the semi open impeller for the design which is a compromise between open and enclosed impeller, it is normally made from few vanes located on a round disk in a certain angles.

A big advantage of semi-open impellers compared to open ones is that the impellers axial position can be adjusted to compensate for wear. Semi-open impellers are also easily to manufacture as all sides of the impeller are easy accessible for manufacturing processes as well as for applying surface hardening treatments. In combination with wear compensation applications as shown in figure 5.

Fig. 5. Semi-Open Impeller

C. Inner Fluid Friction:

The fluid in contact with the rotating impeller causes flow friction including the interior surfaces in the pump casing. The flow friction causes a pressure loss that reduces the head of the pump. The size of the friction loss rely on the roughness of the surface, the more rough the more friction loss and head loss resulting to nowadays manufacturers coating the interior surfaces to overcome the friction loss.

3. CASE STUDY

In this paper the researchers used Pheonics software to examine the designed centrifugal pump, they tried three speeds

- 30 rpm
- 300 rpm
- 3000 rpm

D. Pressure at 30 rpm on 100000 mesh:

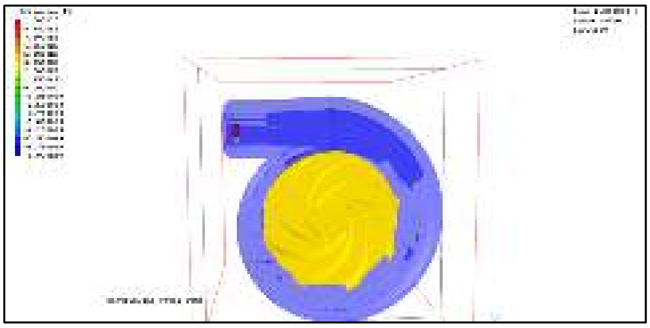

Fig. 6. Case One (30 rpm)

E. Velocity at 30 rpm:

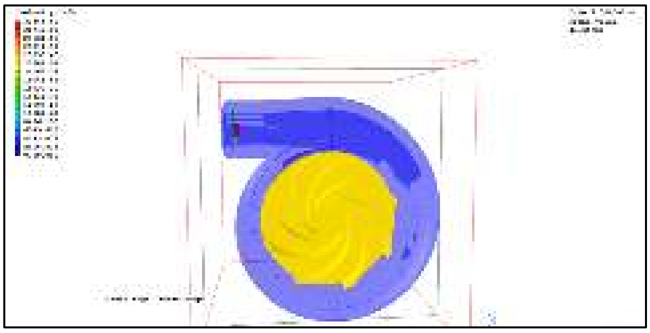

Fig. 7. Case One (30 rpm)

Fig. 8. Results of Case One

https://www.doi.org/10. 22303/csrid.11.1.2019.54-61 *eISSN :2460-870X*

F. Pressure at 300 rpm:

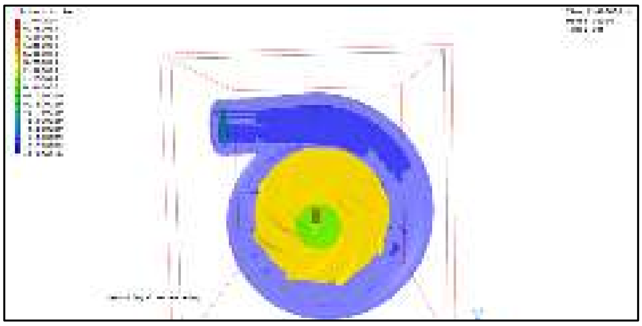

Fig. 9. Case Two (300 rpm)

G. Velocity at 300 rpm:

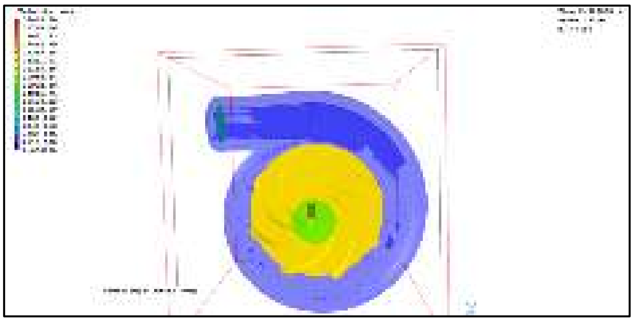

Fig. 10. Case Two (300 rpm)

Fig. 11. Results of Case Two

H. Pressure at 3000 rpm:

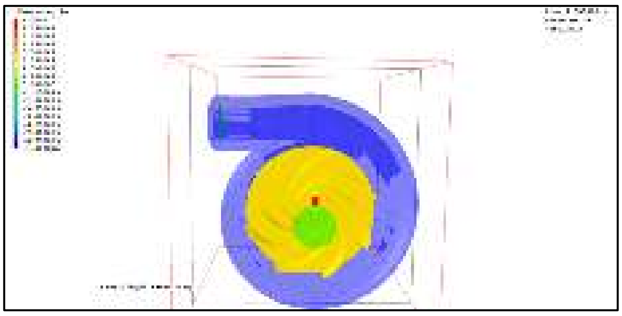

Fig. 12. Case Three (3000 rpm)

I. Velocity at 3000 rpm:

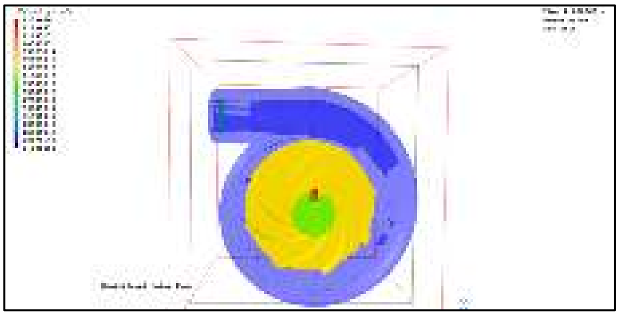

Fig. 13. Case Three (3000 rpm)

Fig. 14. Results of Case Three

4. DISCUSSION

Three different speeds have applied on the centrifugal pump and 30 m/s inlet velocity has applied as well.

In Fig.6 we applied a 30 rpm shows the pressure at the outlet and it is higher than the inlet pressure which acceptable as well as the speed as shown in Fig.7, but as shown in Fig.8 the conversion diagram unfortunately doesn’t meet the criteria.

In Fig.9 a 300 rpm has been applied, we can notice the pressure at the inlet is to low due to normal velocity as shown in Fig.10 which is also acceptable, Fig.11 shows the conversion diagram which the best of our result even though doesn’t meet our criteria.

A 3000 rpm has been applied in Fig.12 which shows the pressure at inlet too low as well as the speed in Fig 13 is too high, Fig.14 shows the conversion diagram again doesn’t meet.

Shown below couple of pictures to compare our result with other design been run on Ansys as we can see in Fig.15 the pressure at the outlet is much higher than the pressure at the inlet of the pump. Fig.16 shows the velocity is much higher at the inlet than the outlet of the pump.

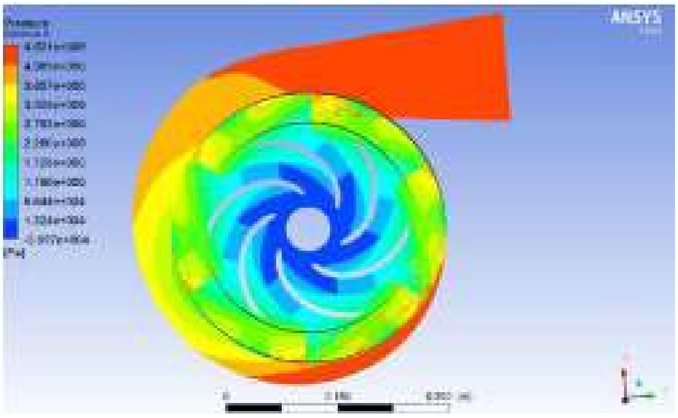

Fig. 15. Comparing Results by Ansys

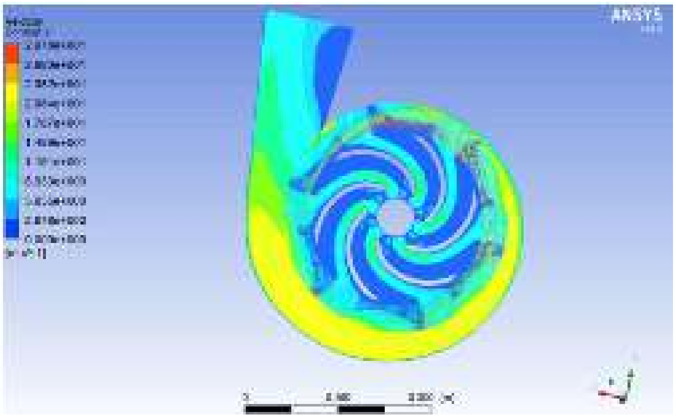

Fig. 16. Comparing Results by Ansys

5. CONCLUSION

The most effective to study pump performance is by computational fluid dynamics, now these days centrifugal pumps are mature devices due to wide range of sold pumps. Centrifugal pumps have reasonable price and maintenance as well as high efficiency comparing to the price and been more advanced by coating the inner surfaces with anti-corrosion and smooth material to reduce the friction and raise the efficiency.

REFERENCE

[1] Anderson, J. D. et al., 2009. *Computational Fluid Dynamics: An Introduction.* Berlin: Springer Berlin Heidelberg.

[2] ANSYS, 2010. *Introduction to ANSYS FLUENT - Lecture 5: Solver Settings.* [Online] Available at:

[3] Banga, S. et al., 2015. CFD Simulation of Flow around External Vehicle: Ahmed Body. *IOSR Journal of Mechanical and Civil Engineering,* 12(4), pp. 87-94.

[4] Gandhi, Priya, Brager, Gail, and Dutton, Spencer, 2014, Mixed Mode Simulation Tools, Internal Report, University of California.

http://imechanica.org/files/fluent_13.0_lecture05-solver-settings.pdf
[Accessed 20 April 2017].

[5] Raase, Sebastian, and Nordström, Tomas, 2015, On the Use of a Many-core Processor for Computational Fluid Dynamics Simulations, Procedia Computer Science, Vol. 51

[6] Thabet, Senan, and Thabit, Thabit H., 2018, CFD Analysis of a Backward Facing Step Flows, International Journal of Automotive Science and Technology, Vol.2 , No.3 , pp.10-16.

[7] Thabet, Senan, and Thabit, Thabit H., 2018, CFD Simulation of the Air Flow around a Car Model (Ahmed Body), International Journal of Scientific and Research Publications, Vol.8, No. 7, pp. 517-525.

[8] Thabet, Senan, and Thabit, Thabit H., 2018, Computational Fluid Dynamics : Science of the Future, International Journal of Research and Engineering, Vol.5, No. 6, pp. 430-433.

[9] Thabit, Thabit H., and Younus, Saif Q., 2018, Risk Assessment and Management in Construction Industries, International Journal of Research and Engineering, Vol. 5, No. 2, pp. 315-320.

[10] Thangam, S., & Hur, N. (1991). A highly-resolved numerical study of turbulent separated flow past a backward-facing step. International journal of engineering science, 29(5), 607-615.

[11] Tsuboi, N., Daimon, Y., Hayashi, A., 2008, Three-dimensional numerical simulation of detonations in coaxial tubes, Shock Waves, 18: 379-392.

[12] Yoder, D. A., & Georgiadis, N. J. (1999). Implementation and validation of the chien k-epsilon turbulence model in the wind navier-stokes code.

[13] Thabet, Senan, Jasim, Yaser A., and Thabit, Thabit H. (2018). Critically Evaluate the Capabilities of Ultrasonic Techniques Used for Tracing Defects in Laminated Composite Materials, International Journal of Engineering & Applied Sciences, 10(3), 237-251.

[14] Jasim, Yaser A., Thabet, Senan, and Thabit, Thabit H. (2018). Inspection of the Composite Materials, Computer Science Research and Its Development Journal, 10(3), 146-152.