BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 62 (66), Numărul 2, 2016
Secţia
CONSTRUCŢII DE MAŞINI

CFD ANALYSIS REGARDING THE INFLUENCE OF IMPELLER PARAMETERS ON THE PERFORMANCE OF A SIGLE-STAGE CENTRIFUGAL PUMP

BY

EUGEN-VLAD NĂSTASE*

“Gheorghe Asachi” Technical University of Iaşi,
Department of Fluid Mechanics, Fluid Machinery and Fluid Power Systems

Received: December 5, 2016
Accepted for publication: December 20, 2016

Abstract. The objective of the application is to determine, through Solid Works Flow Simulation, the influence of impeller parameters on the performance of a centrifugal pump. Rotor parameters whose influence on efficiency will be studied are: the shape blade geometry and number of blades respectively. For this purpose will design a pump impeller for working with a flow Q = 0.083 m^3/s, head H = 50 m, at a rotation speed n = 1450 rev/min. The rotor blades are constructed in three ways: simple arc method, double arc method, and point by point method. Also the number of blades will change to the value obtained by calculation, to see the influence of the number of blades on pump efficiency. For each type of rotor design, simulations will be made and tracked finding the optimal solution.

Keywords: centrifugal pump; flow simulation; turbo machinery; impeller design; computational fluid dynamics.

1. Introduction

Centrifugal pumps are widely used in many applications. This type of pump is used in various field such as in industries, agriculture and domestic

*Corresponding author; *e-mail*: nastase_eugenvlad@yahoo.com

applications. The input power of centrifugal pump is the mechanical energy, applied to the shaft, and the output energy is hydraulic energy of the fluid. Two main components of a centrifugal pump are the impeller and the casing (Fig. 1). The impeller is a rotating component and the casing is a stationary component. Water enters axially through the impeller eye and exits radially from impeller to casing in virtute of the centrifugal force produced by the impeller's speed. (Gundale and Joshi, 2013). The role of casing is to leads the liquid, from the impeller discharge to the outlet of pump and transform into pressure, a part of the kinetic energy of this fluid.

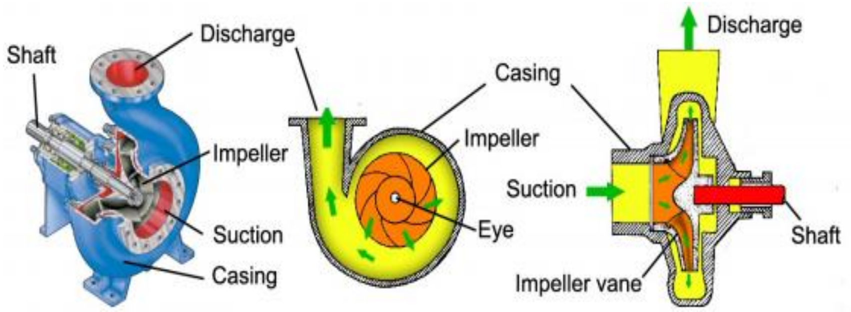

Fig. 1 – Centrifugal pump.

The impeller is a main part of a pump and the performance of this machine depends on the impeller diameters and design (shape of impeller vane). The impeller is a complex structure and the conventional method of verification is time consuming and expensive, the designer maybe used a CAD software in constructing the geometrical profile and CFD analysis for final model (Rajendran and Purushothaman, 2012).

2. Design of Impeller

General methods available to design an impeller vanes are simple arc method, double arc method, and point by point method (Gundale and Joshi, 2013; Wu *et al*., 2008). The main geometric parameters of impeller determined by classical methodology (Wu *et al*.,2008), are shown in Table 1.

Table 1
The Geometric Characteristics of Impeller

Pump head: H [m]	50	Outlet diameter : D_2 [mm]	400
Flow rate: Q [m^3/h]	300	Inlet blade angle: β_1[°]	17
Angular velocity: n [rad/s]	151.8	Outlet blade angle: β_2[°]	22
Specific speed: n_s	22.3	Blade inlet height: b_1 [mm]	40
Inlet diameter: D_1[mm]	175	Blade outlet height: b_2 [mm]	20

Number of rotor blades (z) were determined using Pfleiderer relationship (Budea and Carbune, 2012):

$$z = 6{,}5\,\frac{D_1 + D_2}{D_2 - D_1}\sin\frac{\beta_1 + \beta_2}{2} \tag{1}$$

With the parameters of impeller (Table 1) result for the number of blades z = 5.9. In conclusion, we consider for geometrical model and simulation, z = 5 and z = 6. Depending upon the calculated parameters the modelling of the impeller is done in Solid Works (three rotors with six blades and three rotors with seven blades) and then the CFD analysis is performed.

3. Results of Simulation

CFD analysis were carried out to predict the efficiency of impeller for the given input model. The efficiency is specified as a equation goal shown in next relation:

$$\eta = \frac{(P_{outlet} - P_{inlet})\cdot Q}{\omega \cdot M} \tag{2}$$

where, P_{inlet} is the pressure at the pump's inlet [Pa], P_{outlet} – the pressure at the impeller's outlet [Pa], Q – the volume flow rate [m^3/s], ω – angular velocity [rad/s], and M – the impeller torque [Nm].

Simulation results are presented as graphs of pressure distribution. In Figs. 2 and 3 we can see the distribution of pressure to the rotor built by the method of arc.

In Figs. 4 and 5 are shown distributions pressure to the rotor whose blades are constructed by double-arc method. In Figs. 6 and 7 can be observed the distribution of pressure for impeller with blades constructed using point by point method.

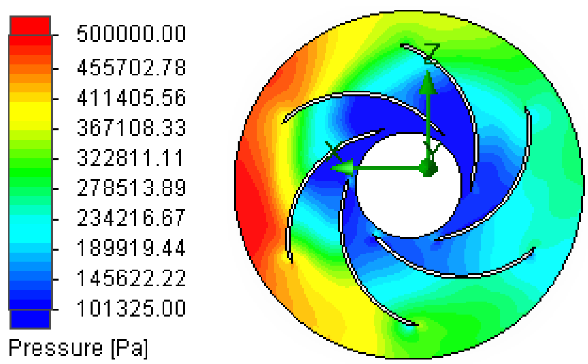

Fig. 2 – Contour of pressure (simple arc method, six blades).

Solid works flow simulation allows exporting the numerical results into Excel files. With data of Excel file was drawn chart efficiecy for all six impellers (Fig. 8).

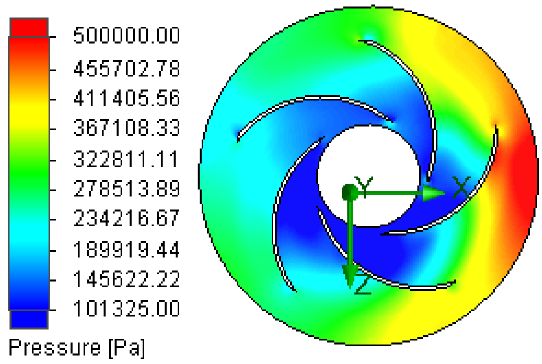

Fig. 3 – Contour of pressure (simple arc method, five blades).

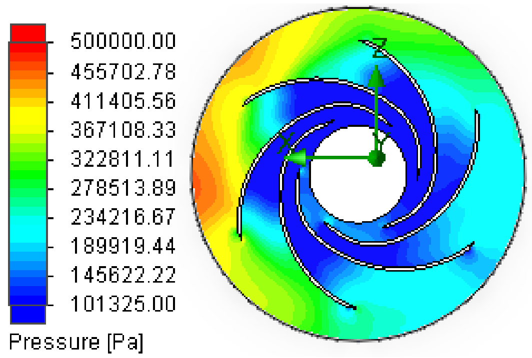

Fig. 4 – Contour of pressure (double arc method, six blades).

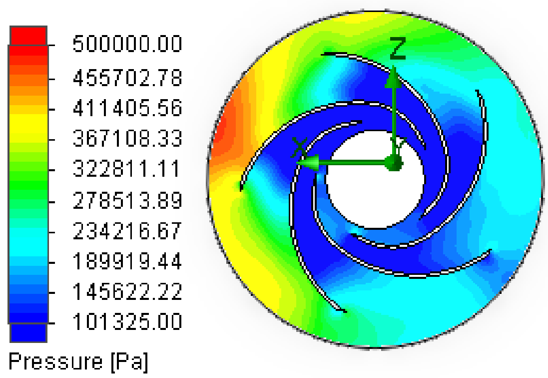

Fig. 5 – Contour of pressure (double arc method, five blade).

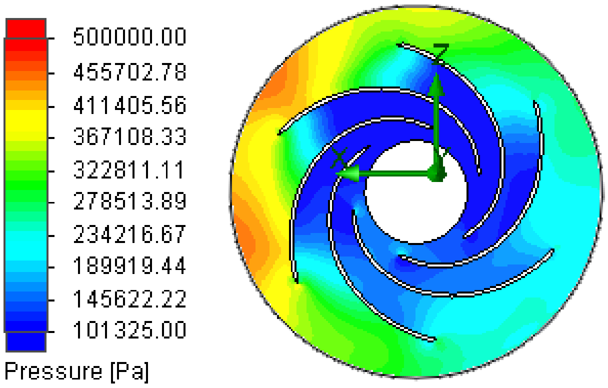

Fig. 6 – Contour of pressure (point by point method, six blade).

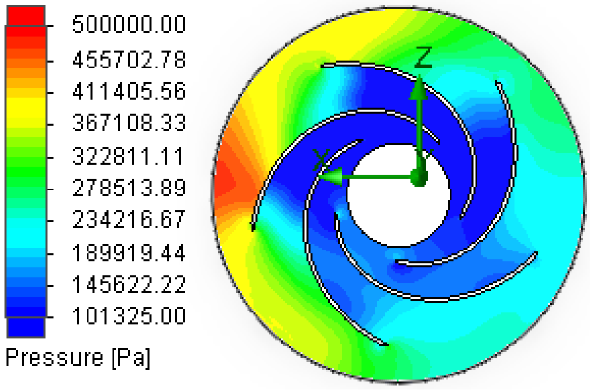

Fig. 7 – Contour of pressure (point by point method, five blades).

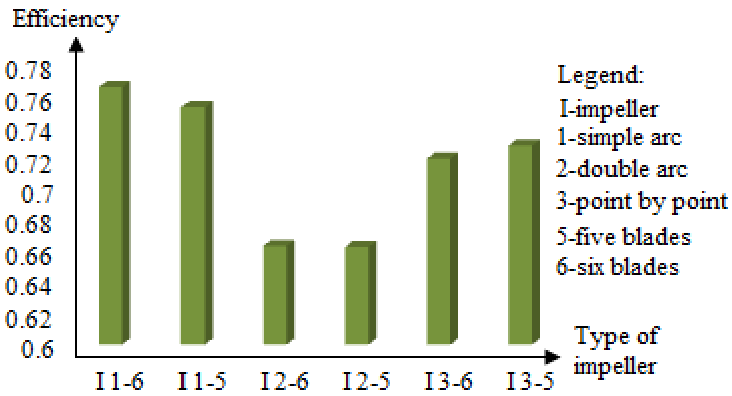

Fig. 8 – Efficiency chart for all impellers.

4. Conclusions

The pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller.

After analyzing the chart efficiency can be seen as the optimal variant is the impeller with six blades and blade profile built by the method of simple arc.

From the results obtained it is found that by decreasing the number of blades decreases and rotor's efficiency. For current conditions it is found that the optimum number is six blades.

CAD and CFD analysis are useful tools that reduce considerable time that is usually lost in physical testing.

REFERENCES

Budea S., Carbune V.D., *The Influence of the Inlet Angle Over the Radial Impeller Geometry Design Approach with Ansys*, Journal of Engineering Studies and Research, Vol. **18**, No. *4*, 32-39, 2012.

Gundale V.A., Joshi G.R., *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**, ISSN 2278 – 9472, 33-37, 2013.

Rajendran S., Purushothaman K., *Analysis of a Centrifugal Pump Impeller Using ANSYS-CFX*, International Journal of Engineering Research & Technology, Vol. **1**, ISSN: 2278-0181, 2012.

Wu K.,H., Lin B.J., Hung C.I., *Novel Design of Centrifugal Pump Impellers Using Generated Machining Method and CFD*, Engineering Applications of Computational Fluid Mechanics, Vol. **2**, No. *2*, 195-207, 2008.

ANALIZA CFD PRIVIND INFLUENȚA PARAMETRILOR ROTORULUI ASUPRA PERFORMANȚEI UNEI POMPE CENTRIFUGE MONOETAJATE

(Rezumat)

Scopul lucrării este de a determina, utilizând software-ul Solid Works Flow Simulation, influenţa parametrilor rotorului asupra randamentului unei pompe centrifuge. Parametrii rotorului care se vor studia, pentru a determina influența acestora asupra randamentului, sunt geometria paletei rotorului şi respectiv numărul de palete rotorice. În acest scop se va proiecta rotorul pentru o pompă centrifugă ce lucrează la un debit Q = 0,083 m^3/s, o sarcină H = 50 m şi o turaţie n = 1450 rot/min. Paletele rotorului vor fi construite prin trei metode: cu un arc de cerc, cu două arce de cerc şi respectiv prin puncte. De asemenea numărul de palete se vor modifica faţă de valoarea obţinută prin calcul, pentru a vedea influenţa numărului de palete asupra randamentului pompei. Pentru fiecare tip de rotor proiectat se fac simulări şi se compară randamentele obţinute.