Journal of Marine Technology and Environment
DOI: 10.53464/JMTE.01.2023.08
ISSN (Print): 1844-6116
ISSN (Online): 2501-8795
https://www. jmte.eu

NUMERICAL SIMULATION OF CENTRIFUGAL PUMP

SCUPI Andrei-Alexandru[1], PANAITESCU Mariana[1] & PANAITESCU Fanel-Viorel[1]

[1]Constanta Maritime University, Faculty of Naval Electro-Mechanics, 104 Mircea cel Batran Street, 900663, Constanta, Romania, e-mail address: andrei.scupi@gmail.com

Abstract : Centrifugal pumps are most commonly used pumping devices in industry their being fit for pumping water. There are several types of centrifugal pumps, their differences in construction is due to their size and how cloggy the pumped water is.

In the paper it was simulated a 2D centrifugal pump of small size used in daily activities. The simulation, performed with CFD (ANSYS-Fluent), was made for different inlet velocities of water. Different parameters, such as inlet pressure, outlet pressure, maximum velocity and flow rate, were calculated and graphically represented against inlet velocity. Also, graphical representation of pressure distribution and velocity distribution inside the impeller were presented for a better comprehension of the physical phenomena.

Key words : centrifugal pump, numerical simulation, finite element analysis.

1. INTRODUCTION

Hydraulic turbo-generators (turbo-pumps), creates an energy transfer at the impact between the rotor blades and the fluid flow, increasing the kinetic moment of the fluid. [2]

The fluid passes through the suction zone, enters the rotor, where a kinetic energy is imprinted on it, transforms it into potential energy in the spiral chamber and exist through the discharge zone. [1]

We will consider a 2D section of a centrifugal pump. It functionality, working principle and energy conversion its better visualized in two dimensions.

In Figure 1 is shown a representation of a centrifugal pump, Bernoulli's equation.

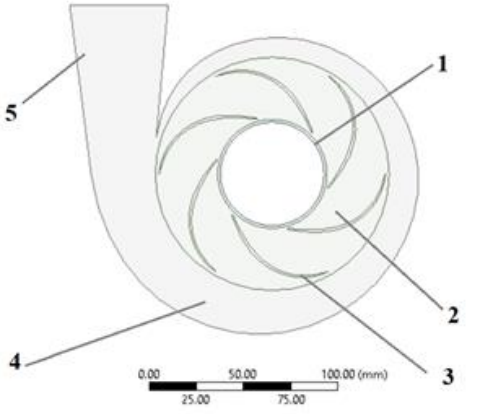

Figure 1 Representation of a centrifugal pump

where:

1 – inlet zone (ring);
2 – impeller;
3 – blade;
4 – spiral chamber;
5 – diffuser.

The spiral chamber has the property to convert kinetic energy into potential energy. The continuity equation (1) tells us that the flow rate is constant no matter the section. Of course, this equation has certain limitations and applicability. [1]

$$Q = v_1 \cdot S = const. \qquad (1)$$

In the same time, Bernoulli's equation (2), under pressure form, shows that and decrease of dynamic pressure will result an increase of static pressure, or vice versa. [1]

$$\frac{\rho v^2}{2} + p_s = const \qquad (2)$$

Since velocity is decreasing due to the increase of section, it means the dynamic pressure will decrease hence the static pressure will increase. The latter is necessary the overcome the loss of pressure due to friction.

The larger the section of the spiral chamber on the outlet part, the smaller the velocity, the bigger the static pressure will be. Hence, designing the centrifugal pump depends on its industrial purposes.

2. CENTRIFUGAL PUMP SIMULATION

2.1 Centrifugal pump characteristics:

Centrifugal pumps have a wide range of applications and they can vary in size from a few of centimetres in

Journal of Marine Technology and Environment
DOI: 10.53464/JMTE.01.2023.08
ISSN (Print): 1844-6116
ISSN (Online): 2501-8795
https://www. jmte.eu

diameter for domestic applications up to half a meter in diameter or more for industrial applications.

Main characteristics of the centrifugal pump:

- inlet diameter (ring) - 55.0 mm;
- interior diameter of the impeller - 58.0 mm;
- exterior diameter of the impeller - 124 mm;
- number of blades - 6;
- angular velocity - 2900 rpm.

2.2 Pump discretization :

After the geometric representation of the centrifugal pump, the discretization is shown in Figure 2.

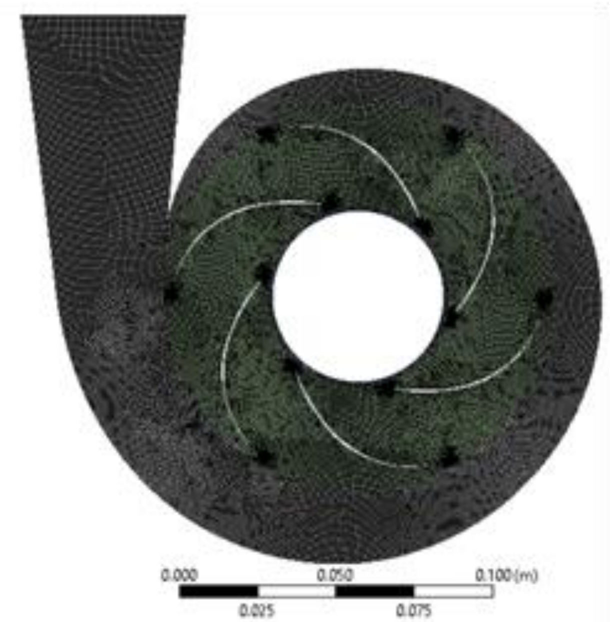

Figure 2 Centrifugal pump discretization

The centrifugal pump is discretised in approximately 27.500 polyhedral cells with 29.000 knots.

2.3 Boundary conditions :

Flow rate can be deducted using equation (3). Length of the circle was used instead of surface, due to 2D simulation.

$$Q = v_1 \cdot \pi D_i \left[\frac{l}{s} \right] \quad . \qquad (3)$$

Following conditions were used in ANSYS – Fluent v.22:

- water was selected as fluid, with a density of $1000\ \frac{kg}{m^3}$;
- inlet velocity of water was set to 0.5 m/s, 1 m/s. 1.5 m/s, 2 m/s;
- The fluid model used was k-omega is turbulent with a Prandtl number equal to 0.667.
- the water dynamic and cinematic viscosity are constant and are equal to $10^{-3}\frac{kg}{ms}$, $10^{-6}\frac{m^2}{s}$, respectively;
- The turbulence viscosity ratio is set to 10.

There were performed 4 different simulation regarding the inlet velocity imposed at 0.5 m/s, 1 m/s, 1.5 m/s and 2 m/s and observed how the suctions pressure, outlet flow and maxim velocity and pressure changes at constant angular velocity of 2900 rpm.

2.4 Results and discussions:

All the simulations were performed using the boundary conditions presented above.

The solutions converged after all the residuals reached the value of 10^{-4}.

Four scenarios were performed where only the inlet velocity was changed. For instance, pressure distribution, velocity distribution and vectors distribution are presented below for an inlet velocity of 1.5 m/s.

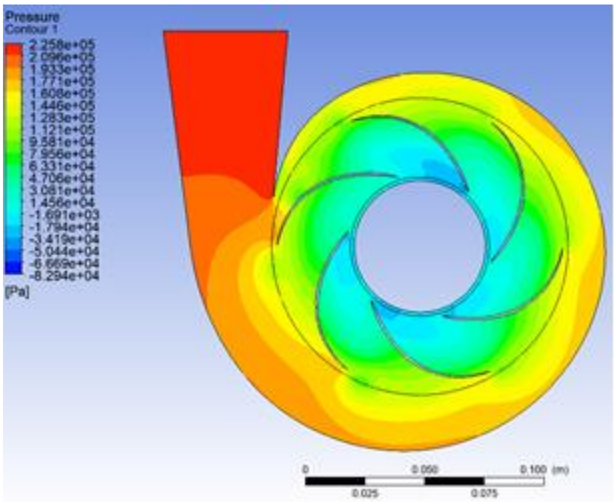

Figure 4 Pressure distribution

In parallel to Figure 4, velocity distribution from Figure 5 and vectors distribution in Figure 6 shows an agreement in Bernoulli's equations (where pressure is higher, velocity is lower). Also, it can be observed that maximum velocity reaches a value close to 26 m/s.

Velocity vectors distribution shows the way the particles leaves the ring zone, enter the impeller where their velocity magnitudes increases up to 26 m/s and the magnitude drops linear due to the increase of section of the spiral chamber.

Journal of Marine Technology and Environment
DOI: 10.53464/JMTE.01.2023.08
ISSN (Print): 1844-6116
ISSN (Online): 2501-8795
https://www. jmte.eu

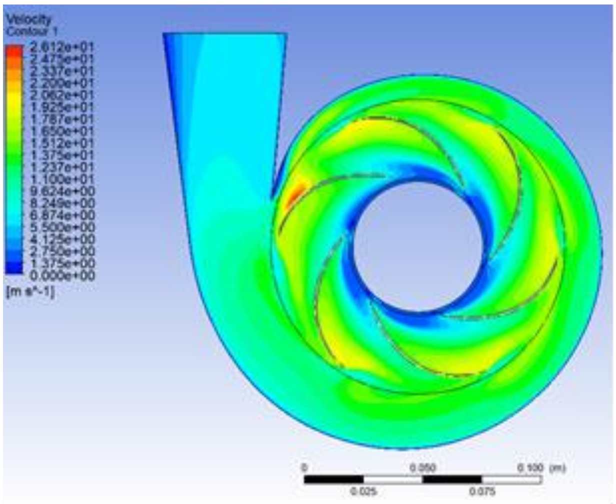

Figure 5 Velocity distribution

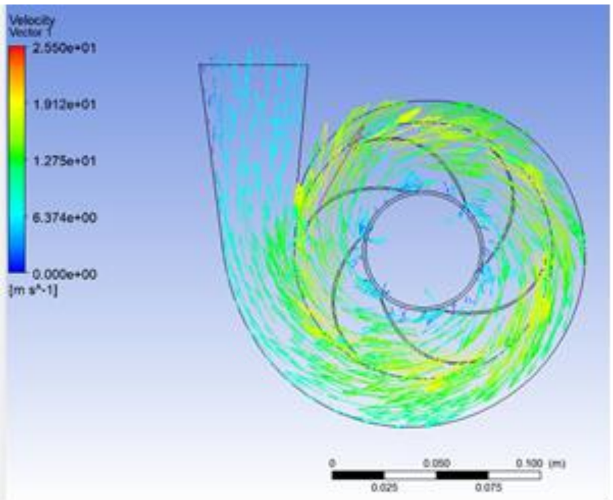

Figure 6 Velocity vectors distribution

For others scenarios was chosen an inlet velocity of 0.5 m/s, 1 m/s and 2 m/s respectively. Than were compared different parameters for all four simulation: inlet pressure, outlet pressure, maximum velocity, flow rate.

2.5 Discussions about results

In Table 1 it is presented the values of the main parameters with respect to inlet velocity.

Table 1. The results of simulation

Inlet velocity [m/s]	0.5	1	1.5	2
Inlet pressure [Pa]	6400	3200	-6000	-11500
Flow rate [l/s]	0.08	0.17	0.26	0.345
Maximum velocity [m/s]	27	26.8	26	20
Outlet pressure [bar]	2.25	2.14	2.10	1.8

Than graphs were created accordingly for a better comprehension of the phenomena implied.

3. CONCLUSIONS

Finite Element Analysis is a method to solve the mathematical model and to simulate the complex phenomena of Fluid Mechanics.

Therefore the person who uses a simulation program of a real physical phenomenon by finite element method using computer must well understand the physical processes that take place and to be able to correctly interpret the data that the computer has to offer.

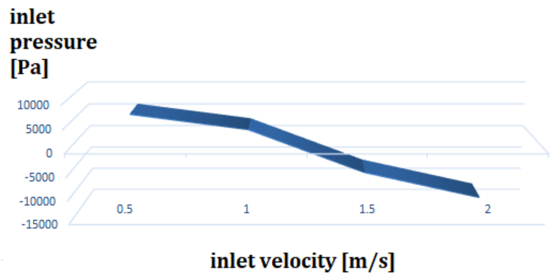

Figure 7 Inlet pressure vs. inlet velocity

According to equation 3 flow rate is directly proportional with velocity, thus higher inlet velocity implies a higher flow rate (Figure 8).

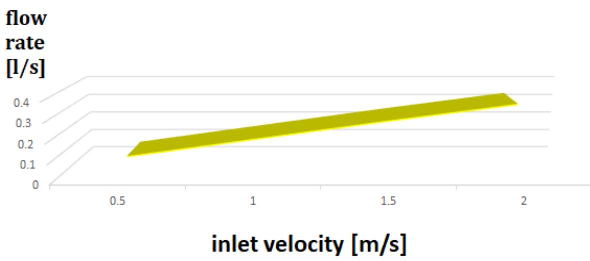

Figure 8 Flow rate vs. inlet velocity

Maximum velocity is reached at the end of the rotating blades. From there, in spiral chamber kinetic energy is converted into pressure energy, thus static pressure will increase (Figure 4). In Figure 9 It is observed a slowly decrease of maximum velocity with respect to inlet velocity up to 1.5 m/s, than a drastically decrease of the maximum velocity. This is due to the fact that inlet velocity enters radially while maximum velocity exists tangentially. Thus, inside the impeller we

Journal of Marine Technology and Environment
DOI: 10.53464/JMTE.01.2023.08
ISSN (Print): 1844-6116
ISSN (Online): 2501-8795
https://www. jmte.eu

have a loss of pressure due to friction and due to the fact more energy is consumed to change the direction of the flow.

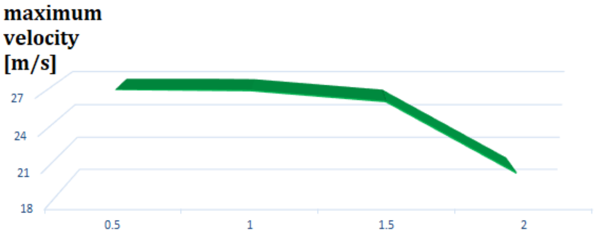

Figure 9 Maximum velocity vs. inlet velocity

Similar, in Figure 10 it is observed a slow variation of the outlet pressure up to velocity 1.5 m/s, than a more acute decrease of the pressure. Here, maximum velocity and outlet pressure are proportional due to the fact that loss of kinetic energy implies a loss in potential energy.

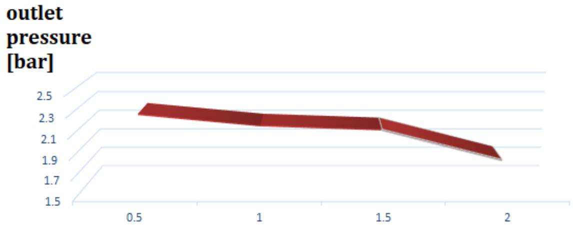

Figure 10 Outlet pressure vs. inlet velocity

More simulations were performed for higher inlet velocities, but it was noticed that the inlet pressure reaches values below (-1) bar, which implies it is not physically possible. So inlet velocity has a maximum value (depending on the dimensions of the pump, revolution per minute) that can be imposed.

4. REFERENCES

[1] DINU, D., 2010, *Fluid mechanics for sailors (Mecanica fluidelor pentru navalistic)*, Nautica Publisher House,Constanta, Romania.

[2] DINU, D., 1999, *Hydraulics and hydraulic machines*, SIGMA Publisher House,Bucharest, Romania.

[3] DINU, D., PETRE F., 1993, *Hydraulics and pneumatic machines (* Masini hidraulice si pneumatice), I.M.C. - Constanta, Romania.

[4] GEORGESCU, A.M., GEORGESCU, S.C., 2007, *Hydraulics of Pipe Networks and Hydraulic Machines* (*Hidraulica Reţelelor de Conducte şi Maşini Hidraulice)*, Printech Publisher House, Bucharest, Romania.

[5] DINU, D., SCUPI, A.A., 2019, *Theoretical study for air-lift pump system. Application,s proposal-offshore light fishing*, Journal of Marine Technology and Environment, Vol.2, pp.45-48, Nautica Publisher House, Constanta, Romania.