

Dynamic Analysis of a Centrifugal Pump using CFD and FEM Methods

PhD Student Eng. **Nicolae IACOB**¹, Assoc. Prof. PhD Eng. **Nicușor DRĂGAN**²

¹ Institute of Solid Mechanics, Romanian Academy Bucharest, nicolaeiacob84@yahoo.com

² "Dunarea de Jos" University Galati, Engineering and Agronomy Faculty of Braila, The Research Center of Machines, Mechanic and Technological Equipment - MECMET, ndragan@ugal.ro

Abstract: The article analyses the performances of a centrifugal pump using the Computational Fluid Dynamic (CFD) method, a computer numerical simulation method that was performed with specialized software in dynamic fluid flow analysis, ANSYS Workbench, using the FluidFlow (CFX) module. With this, the performances parameters of a centrifugal pump were determined: the inlet pressure and the outlet pressure of the pump, the flow of the pump, the speed of the fluid through the pump, the cavity of the pump. Using the finite element method (FEM) for solid bodies, the stresses and deformations occurring in the rotor blades were determined. For this part of the study, the Static Structural module from the ANSYS Workbench work platform was used. The 3D geometry of the centrifugal pump was modelled with the Siemens PLM NX 7.5 CAD/CAM software, then imported into the ANSYS Workbench to perform the analysis. The results of numerical simulations were interpreted and the conclusions of the study were drawn. This study emphasizes the advantages of using numerical modelling and simulation by CFD method and by the finite element method.

Keywords: Dynamic analysis, current lines, centrifugal pump, CFD, FEM

1. Introduction

Centrifugal pumps are rotary hydraulic machines that create hydraulic energy based on the centrifugation of the liquid passing through the pump rotor. Centrifugal pumps are widely used in technical applications.

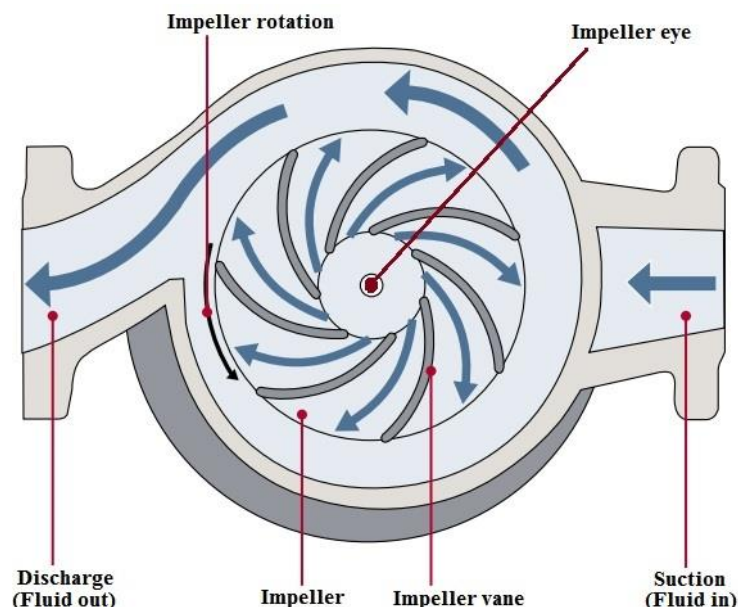


Fig. 1. Centrifugal pump - operation principle [1]

Due to the development of the centrifugal pump construction industry, the field of use of piston pumps has been narrowing lately, being used only for low flows and high pressures, an area where the use of centrifugal pumps is uneconomical, but even this area it narrows. The principle of operation of centrifugal pumps is relatively simple. This can also be seen in Fig. 1. The centrifugal pump creates a increased pressure inside it by transferring the mechanical energy from an engine

through an axis to the pump rotor. The fluid flows from the inlet to the center of the rotor, then flows along the blades of the rotor outward due to centrifugal force. The kinetic energy created by the rotation of the rotor is converted to pressure at the output of the pump (discharge area).

2. Performance parameters of centrifugal pumps

The performance parameters of the centrifugal pumps can be determined by a set of performance curves diagrams. Figure 2 shows the performance curves of the centrifugal pumps. The first diagram shows the curves for the height of the fluid rise in relation to the flow produced by it and the yield curve. The optimum power consumption of a centrifugal pump is found at the intersection of the height curve and the pump yield curve.

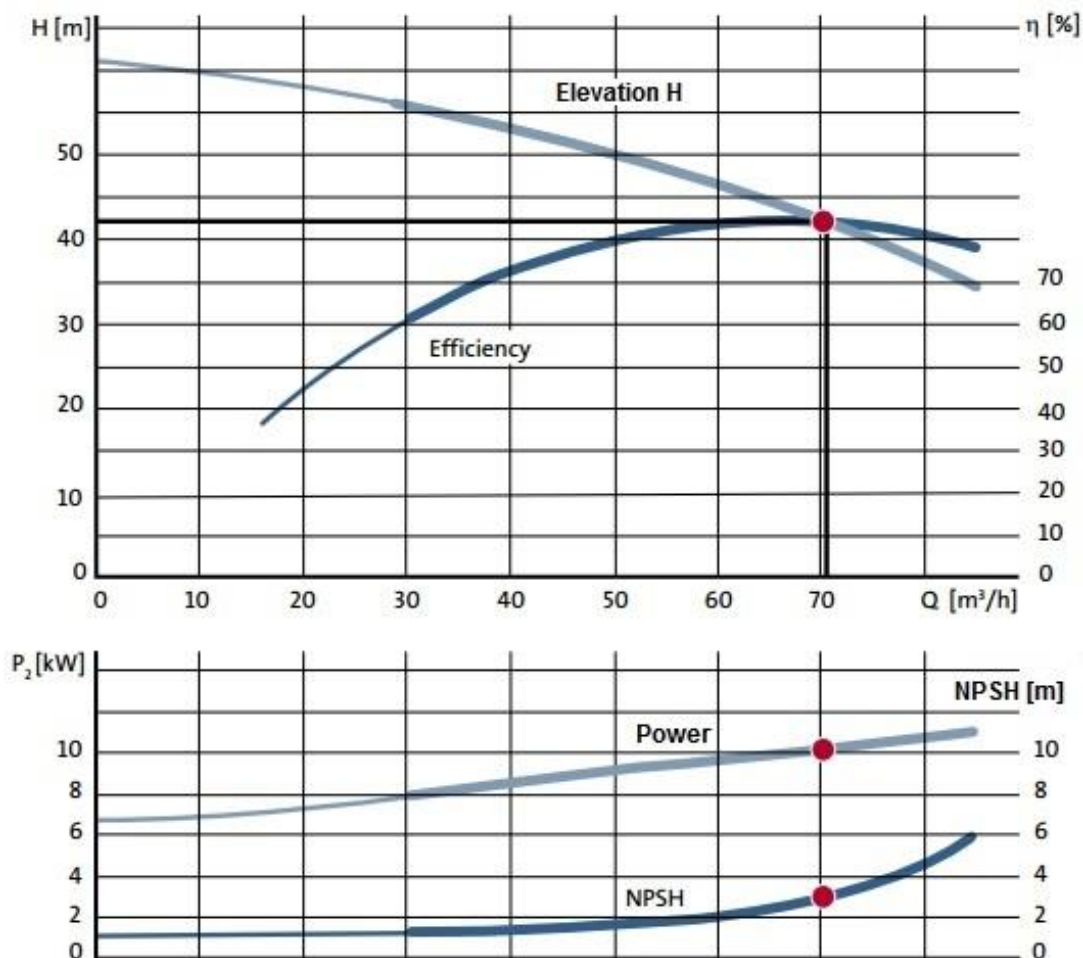


Fig. 2. Centrifugal pumps - performance curves diagrams [1]

The second diagram shows the power curves and the suction height of the pump. The NPSH (Net Positive Suction Head) abbreviation in the second diagram represents the suction height. For example, for a suction height of 2.5 m, a power consumption of 10 [kW] is required, as can be seen in the second diagram by marking the points in red.

The performance of centrifugal pumps is characterized by static pressure, dynamic pressure, fluid flow rate, suction height and fluid rise height. All of these performances are based on more complicated mathematical computing formulae, determining the performance of centrifugal pump by an analytical becoming almost impossible.

Another method of determining the performance of a pump is the experimental one, but this means consumption of material, time and money. The method proposed in this paper is the numerical simulation through the dynamic CFD fluid analysis performed with the ANSYS software.

3. Dynamic Analysis of a Centrifugal Pump using CFD Method

In order to determine the performances of the centrifugal pumps in this work, the CFD method was proposed. For this, the Fluid Flow (CFX) module from the Workbench Ansys 14.5 work platform was used. There are numerous work programs that can perform numerical simulation but Fluid Flow (CFX) is easy to use and has an algorithm with a very high computing power compared to other dynamic flow analysis software. The pump geometry was performed in the 3D modelling software NX 7.5 from the company Siemens PLM (Fig. 3, Fig. 4).

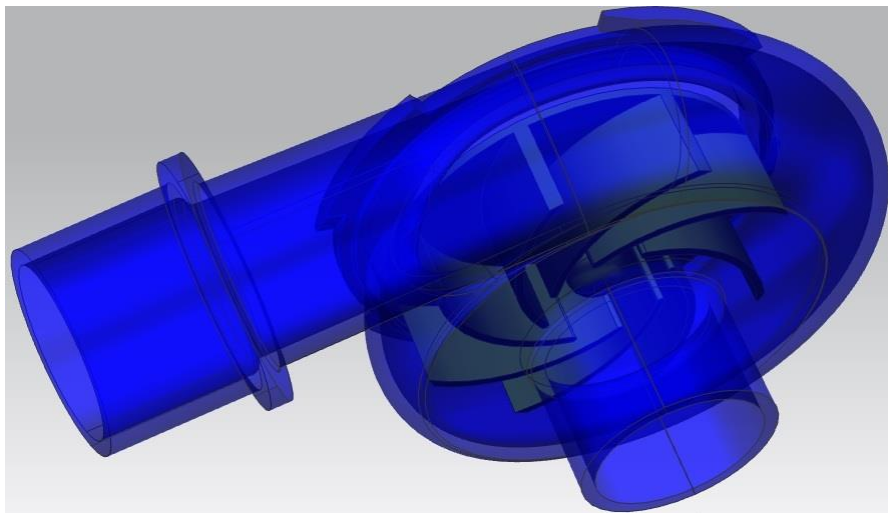


Fig. 3. Centrifugal pump geometry (model assembly)

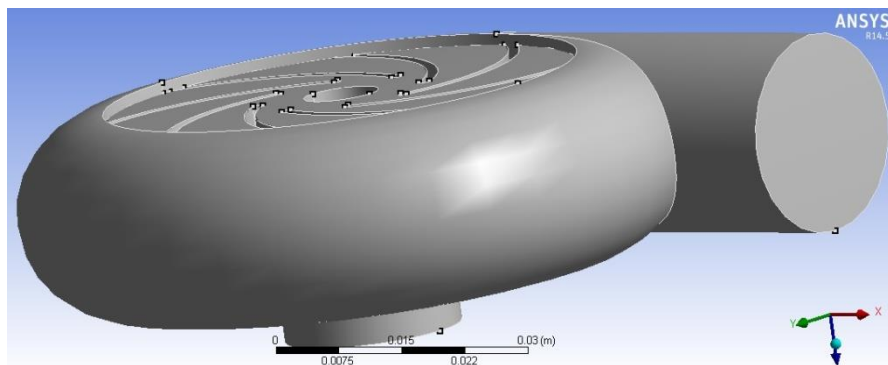


Fig. 4. Centrifugal pump - fluid flow geometry

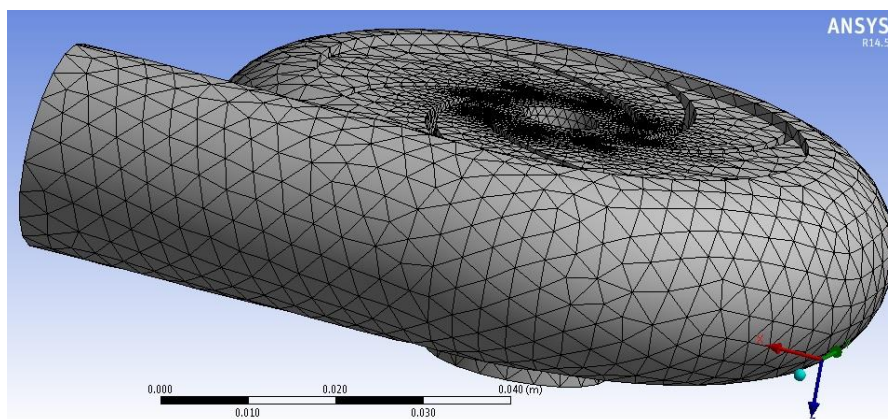


Fig. 5. Centrifugal pump - 3D mesh model of the flow area

After the discretization of the working model (Fig. 5), we proceeded to the next stage in which the boundary conditions for the dynamic flow analysis were imposed. The limit condition that was imposed on the analysis was the rotational speed of the rotor of 1500 rpm under the conditions of atmospheric pressure. This condition was imposed due to the operation of the centrifugal pumps. The centrifugal pumps as mentioned in the first part of the work are hydraulic machines that operate on the basis of fluid centrifugation. And centrifugation is a phenomenon that is obtained under the conditions of high angular velocities. In the case of the centrifugal pump, it is the rotor that drives the fluid in rotational motion and generates the phenomenon of centrifugation. After imposing a boundary condition, the order for analysis was executed, and after the analysis was completed, the results were exported to a special post-processing module within the WORKBENCH ANSYS 14.5 platform. to be interpreted. This module gives the user the opportunity to observe the results of the analysis both numerically and through graphical illustrations.

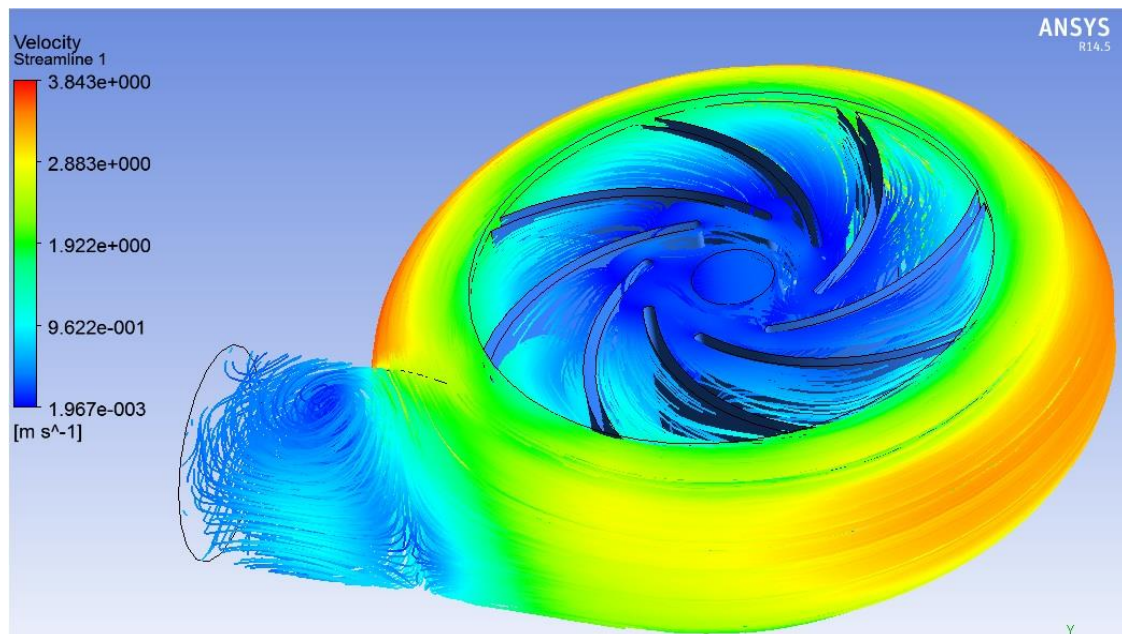


Fig. 6. Streamlines and flow velocity distribution inside the centrifugal pump [m/s]

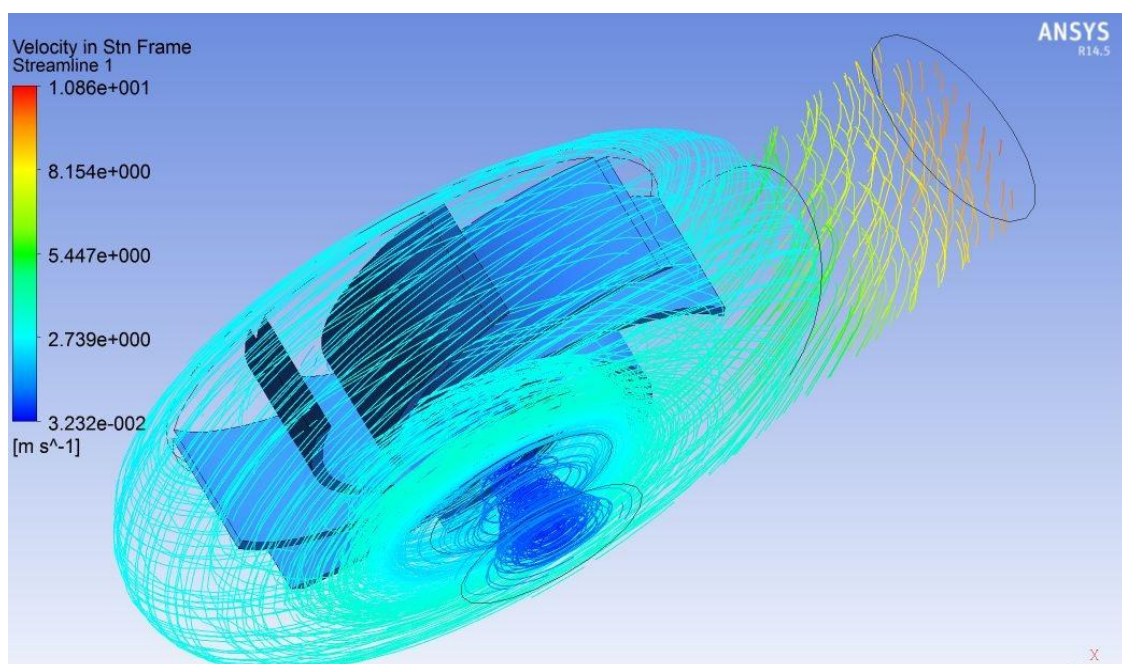


Fig. 7. Streamlines inside the centrifugal pump [m/s]

Fig. 6 and fig. 7 show the current lines of the fluid. As noted, the fluid ascends into the centrifugal pump having a cycloid motion. The fluid is sucked in due to the Coandă effect [2]. This can be seen very well in fig. 6 where in the middle of the rotor there is a decrease in the flow velocity of the fluid, which leads to the decrease of the dynamic pressure and the increase of the static pressure.

In the middle area of the rotor a depression is formed that sucks the fluid and raises it in the pump. Once reached the centrifugal pump the fluid is pushed outward from the pump walls and compressed due to the centrifugal force. With the compression of the fluid, there is an increase in pressure inside the pump. This pressure is much higher than the atmospheric pressure, and when it reaches the pump outlet, the fluid is discharged out.

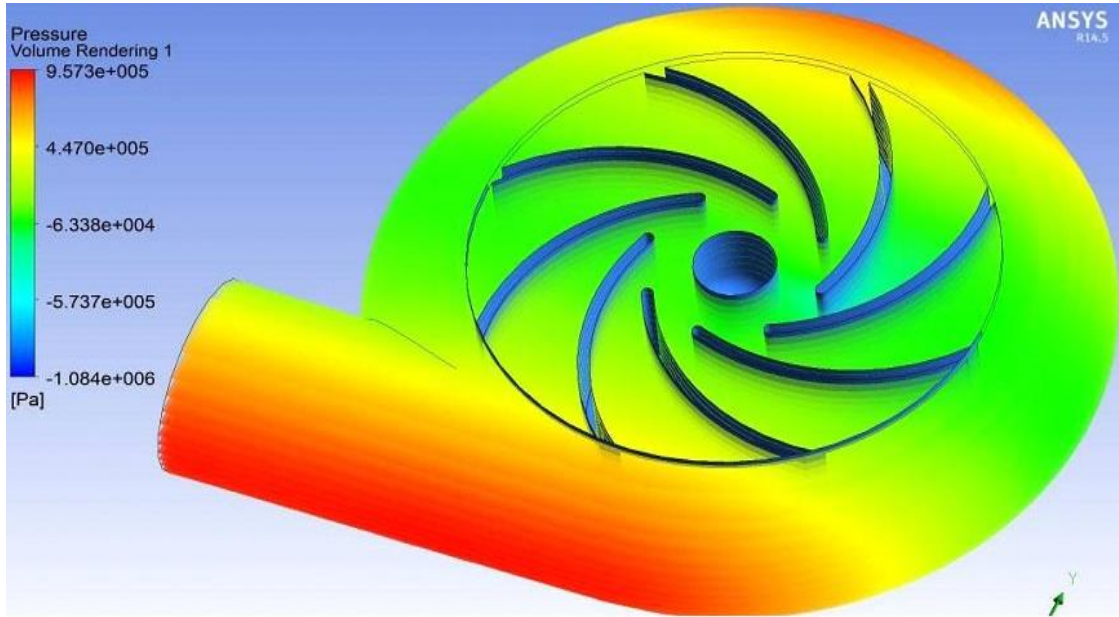


Fig. 8. Static pressure distribution [Pa] - fluid volume rendering

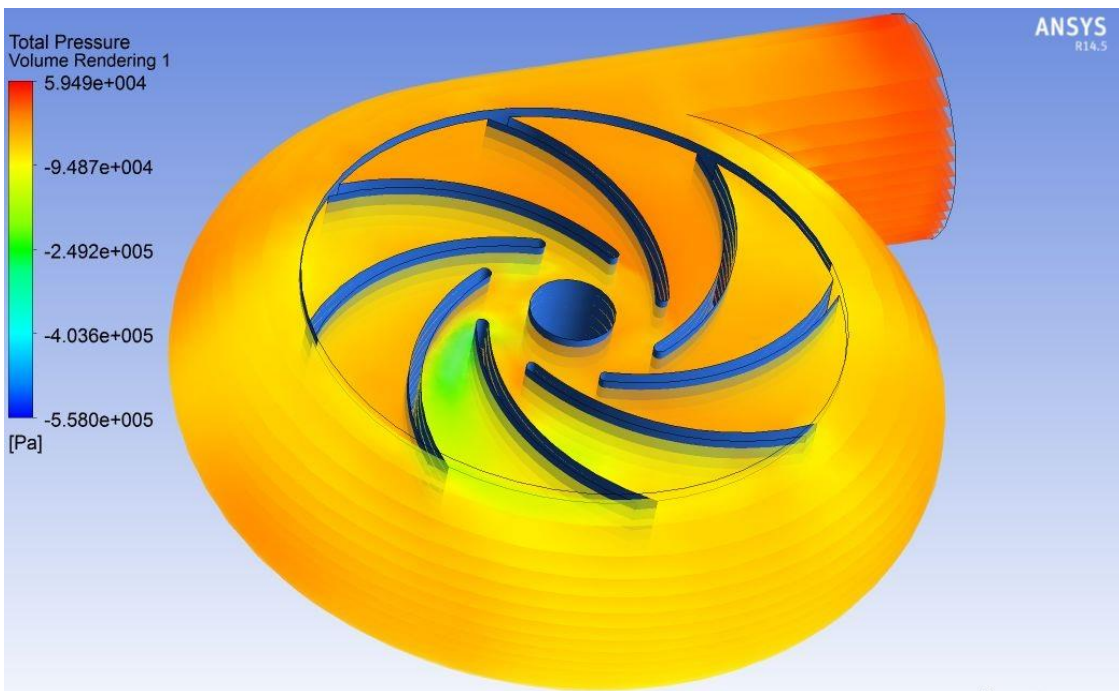


Fig. 9. Total pressure distribution [Pa] - fluid volume rendering

Figures 8 and 9 show the distribution of static pressures and total pump pressure. Static pressure is formed at the inside walls of the pump. As shown in figure 8, in the middle of the rotor a depression of -6.3×10^4 Pa is formed. This is the suction pressure of the pump.

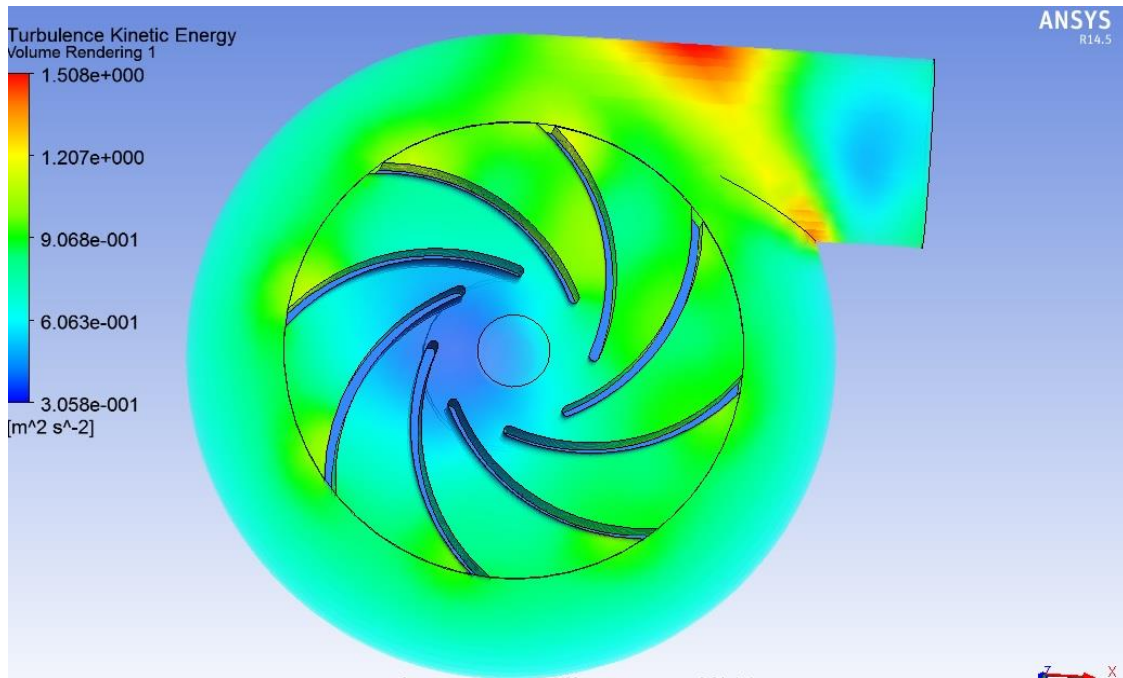


Fig. 10. Turbulence kinetic energy distribution [m^2/s^2] - fluid volume rendering

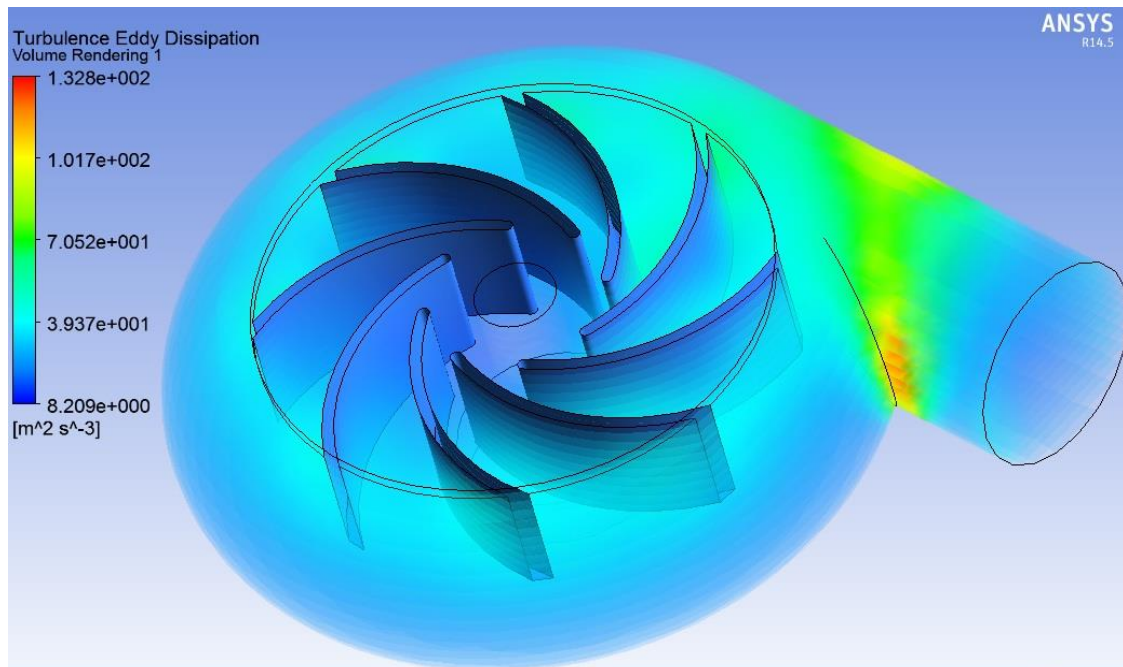


Fig. 11. Turbulence Eddy dissipation of kinetic energy [m^2/s^3] - fluid volume rendering

Figures 10 and 11 show the distribution of the turbulence occurring at the centrifugal pump level. These turbulences produce the phenomenon of cavitations, an undesirable phenomenon because it leads to damage to the pump. As shown in Figure 10, the cavitations appear in the area of the pump rotor.

When the pumped fluid reaches the impeller, because the inlet section is small, a speed increase occurs, accompanied by a drop in pressure. The higher the flow rate of the pump, the more this

pressure drop is more pronounced. If this pressure drop exceeds a certain threshold or if the fluid temperature is high enough (or a combination of the two) the fluid vaporizes (the local pressure drops below the saturated pressure of the pumped fluid). The vapour bubbles get trapped on the rotor and after it exits, it enters a high pressure area, where it suddenly passes from the vapour state to the liquid state (it is destroyed), that means cavitations phenomenon.

4. FEM Analysis of Tension and Deformation Distributions in the Rotor Blades

The determination of the tensions and deformations in the rotor blades was performed using the FEM method (Static Structural module from the ANSYS Workbench platform).

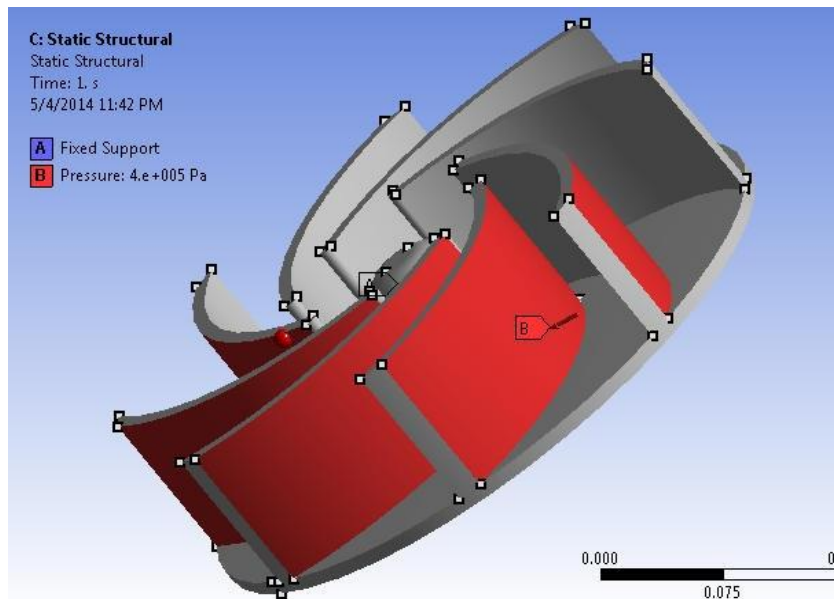


Fig. 12. The boundary conditions imposed to the rotor

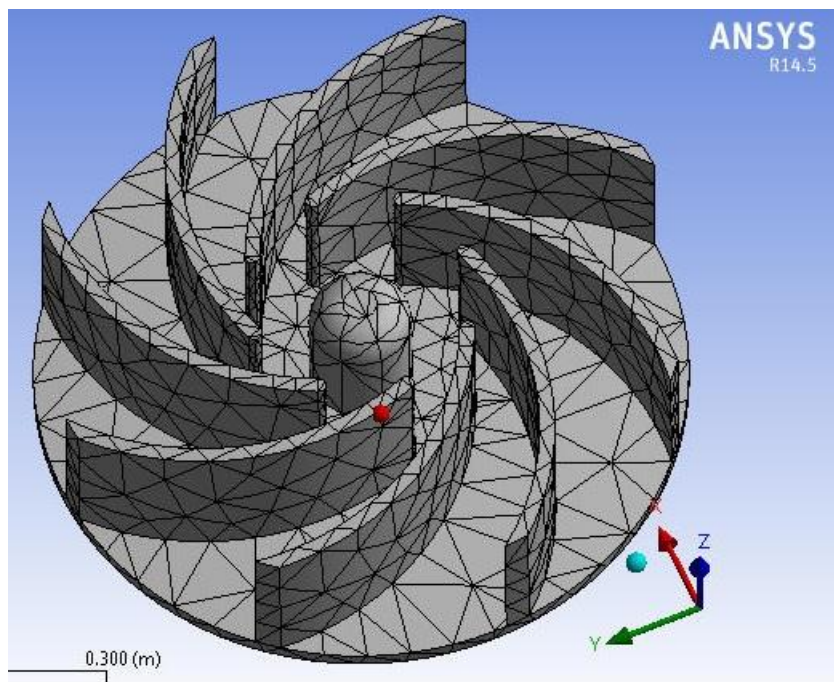


Fig. 13. Mesh model of the rotor

The geometry of the rotor was realized in the software Siemens PLM NX 7.5. The imported model

was discretized and the material from which the rotor is made was declared as alloy steel. The boundary conditions imposed for the structural analysis of the rotor can be seen in figure 12. The maximum static pressure resulting from the CFD analysis was applied to each rotor pallet surface. After applying the boundary conditions, the command to solve the calculation was launched. The 3D model discretion of the rotor can be seen in figure 13. The model was discretized into 51517 knots and 192526 tetrahedron type finite elements [3]. The rotor was analyzed under the conditions in which it is blocked and the maximum static pressure in the pump acts on the pallets. The maximum static pressure value on the rotor blades was taken from the results of the CFD analysis performed in §3.

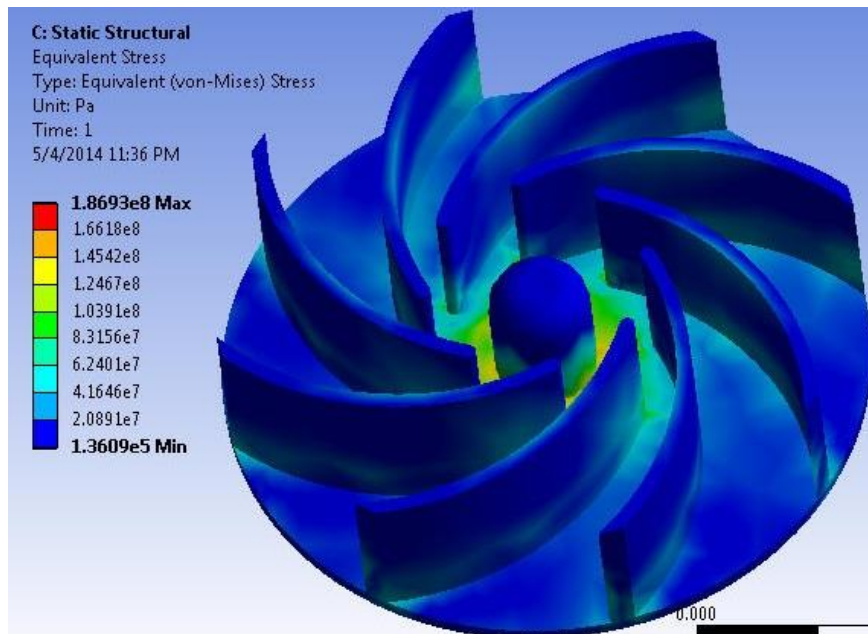


Fig. 14. Tensions distribution in the rotor [Pa]

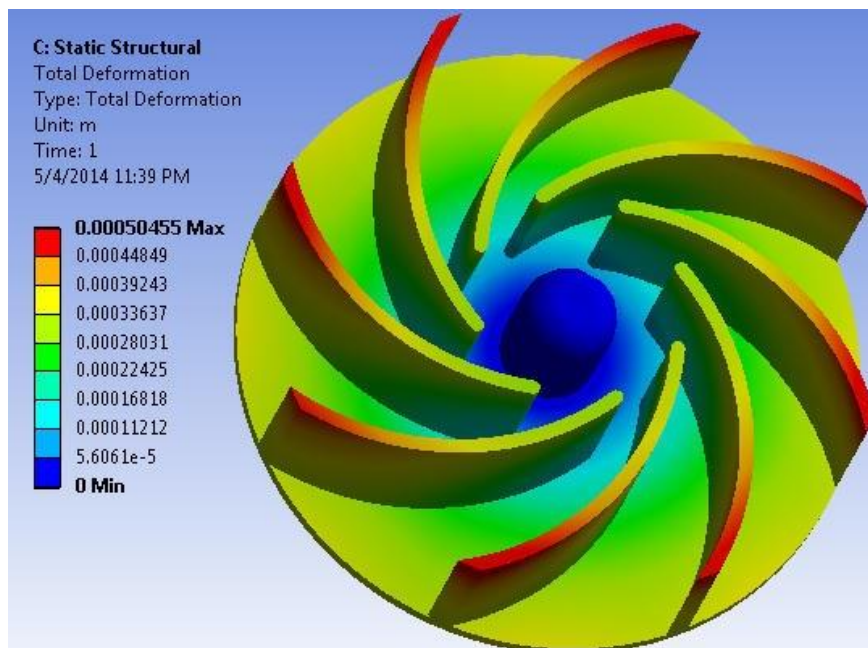


Fig. 15. Deformations distribution in the rotor [m]

Figure 14 shows the tensions distribution in the rotor blades, and figure 15 shows the deformations

that appear in the rotor. In the rotor blades there is a maximum tension of 41.6×10^6 Pa, that means 41 N/mm^2 . The maximum tensions in the rotor are below the material breaking limits. And the maximum deformations that occur in the rotor blades are 5.04×10^{-4} m, that means 0.504 mm. These deformations are very small (of the order of tenths of a millimeter). Maximum deformations occur in the end zones of the rotor.

5. Conclusions

In this paper, the functional parameters of a centrifugal pump were determined using CFD analysis and at the same time showed the efficiency of the finite element analysis method. Also with the CFD method, the Coanda effect created by the rotor of the centrifugal pump was highlighted.

References

- [1] Anton, Viorica, Mircea Popoviciu and Ioan Fitero. *Hydraulics and hydraulic machines/Hidraulică și mașini hidraulice*. Bucharest, Didactic and Pedagogical Publishing House, 1978.
- [2] Coanda, Henri. *Perfectionnement aux pompes centrifuges*. Brevet d'invention no. 1.030.088/23.12.1953, France.
- [3] Moeykens, Shane. "From CAD to CAE. FLUENT software now offers support for Autodesk Inventor" *ANSYS Advantage, Volume II, Issue 1*, 2008. Accessed September 30, 2019. <https://www.ansys.com/-/media/ansys/corporate/resourcelibrary/article/aa-v2-i1-from-cad-to-cae.pdf>.