Optimization Study of a Centrifugal Pump in Cavitation

University Politehnica of Bucharest

Constanta Maritime University, Romania

Ioan CĂLIMĂNESCU[2], Liviu – Constantin STAN[1]

[1)] Engineering Sciences Department in Mechanical and Environment, Constanta Maritime University, Constanta, Romania

E-mail: liviustan14@yahoo.com

[2)] Engineering Nuclear Power Plant, Cernavoda, Romania

Keywords

CFD, Pump, Cavitation, Optimization

1. Introduction

Cavitation if often encountered in the functioning of centrifugal pumps and usually is manifesting itself via rattling or a knocking sound along with vibrations leading in the end to the pump rotor failure, internal damage, and leakage from the seal and casing, bearing failure, etc.

The centrifugal pumps cavitation is implying a dynamic process of formation of the bubbles inside the liquid domain, their growth and collapse as the liquid flows through the considered pump.

The purpose of this paperwork is to investigate the cavitation phenomenon inside a centrifugal pump with ANSYS 19 and to determine the influence of main process factors upon the pump cavitation using the Response Surface technology and optimization.

2. Material and methods

The 3D geometry of the centrifugal pump on the subject is created with SolidWorks 2018 and is like in the Figure 1:

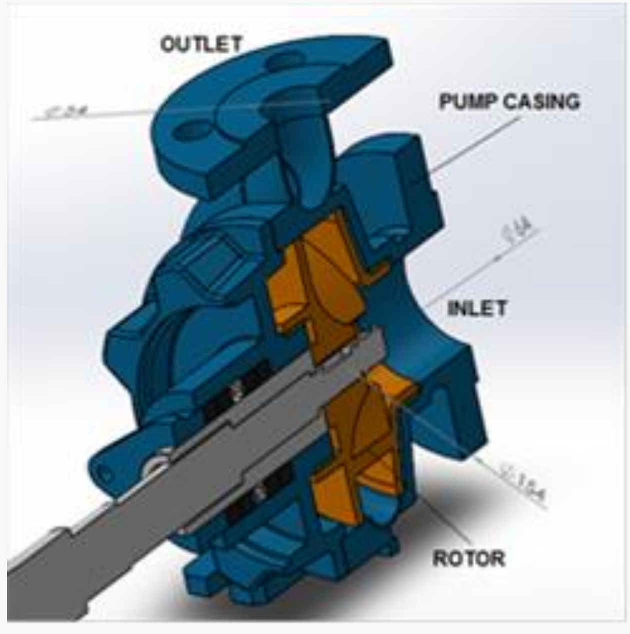


Figure 1. The CAD geometry of the pump on subject

The inlet diameter is 64 mm, the outlet is 34 mm and the rotor diameter is 154 mm so that we are talking about a small pump.

This geometry is then exported inside ANSYS 19 under the Computer Fluid Dynamics CFX module. Here with the Design Modeller module the geometry is further processed so that the final result will be the fluid domain. Inside the Computer Fluid Dynamics CFX module the finite volume mesh is generated as given in the Figure 2:

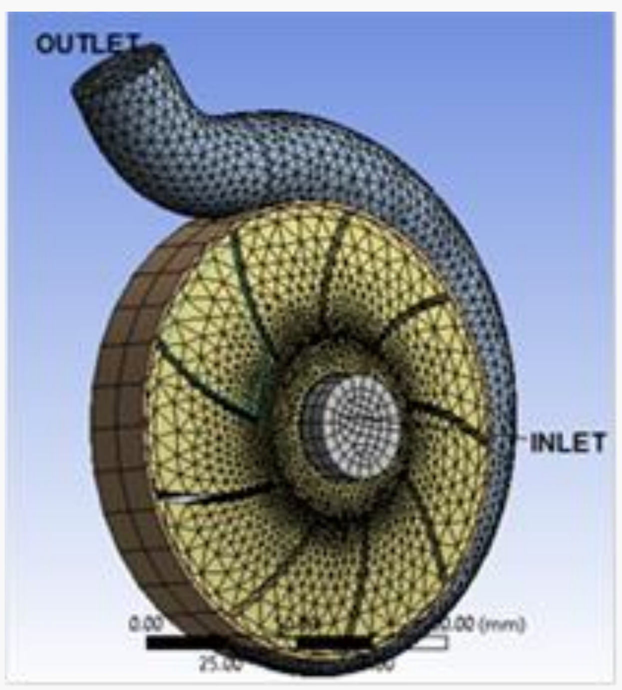


Figure 2. The finite volumes mesh

There are defined 5 fluid domains in which four are fixed: inlet, outlet and two intermediary domains, and one is rotating around the axis Ox (the Rotor) with the angular velocity of 2160 rpm=226 rad/s. In between all these domains Interface areas were defined with the mesh connection of GGI type.

The inlet fluid velocity was set to 7 m/s and is mimicking the NPSH (they are in direct proportionality relation) via an Inlet condition.

The outlet surface was set as Opening type with the entrainment relative pressure of 3e5 Pa.

The cavitation model used is that of Reyleigh-Plesset one with the saturation pressure of 2650 Pa(a). The relative pressure of the model is set to zero.

3. Results and Discussions

After running the model, some important results were retrieved in the CFD model.

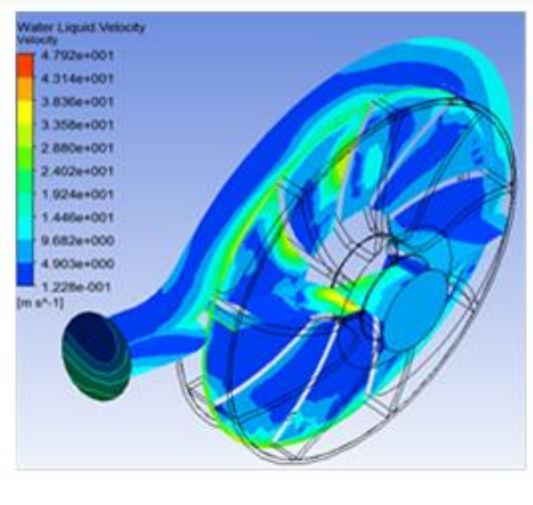
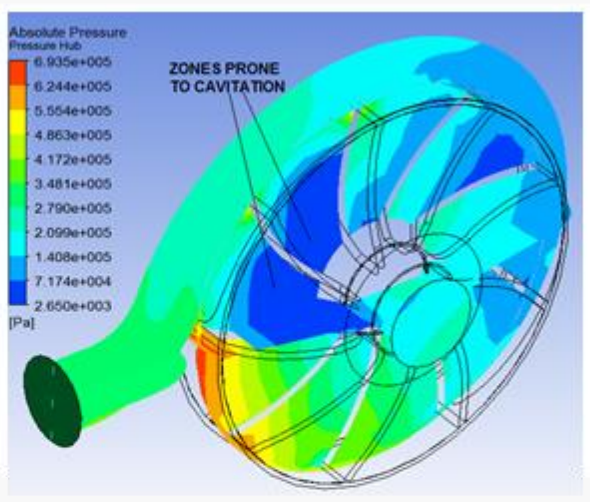


Figure 3. Water velocity calculated fields



Figures 4. Absolute pressure fields

The water velocity fields are in the Figure 3. The water is getting inside the pump with the normal velocity of 7 m/s and within the pump the rotating rotor is accelerating the water up to 48 m/s, the maximum occurring on the blades near the outlet region of the pump.

The absolute pressure fields are to be seen in the Figure 4. The lowest pressures of 2650 Pa(a), which is the vaporization pressure, are to be found in the same regions of the blades where the biggest velocities were recorded. These zones are prone to develop the cavitation phenomenon.

After running the model, some important results were retrieved in the CFD optimal model.

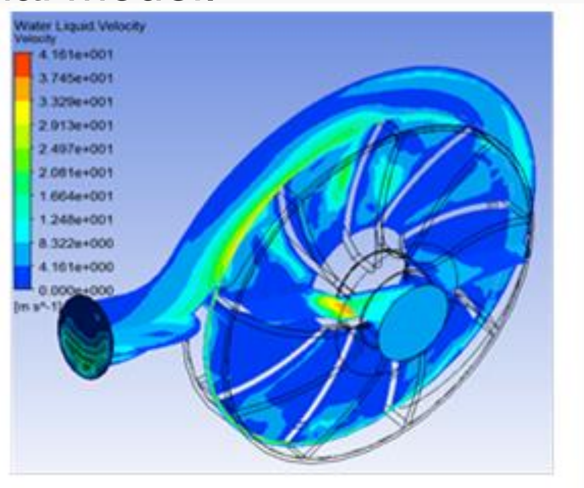
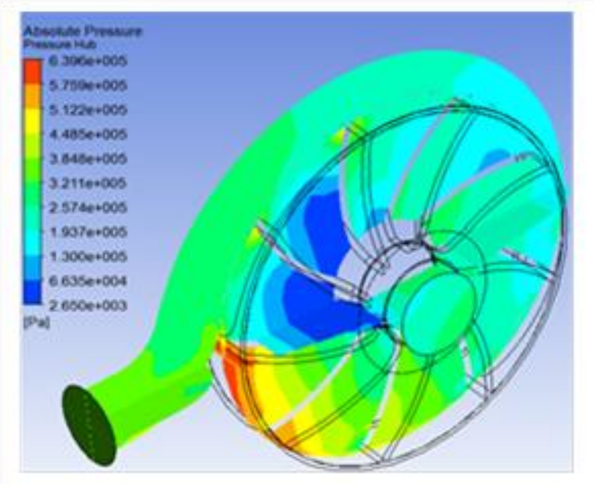


Figure 5. Water velocity calculated fields for the optimal model



Figures 6. Absolute pressure fields for the optimal model

The water velocity fields are in the Figure 5. The water is getting inside the pump with the normal velocity of 7.6 m/s and inside the pump the rotating rotor is accelerating the water up to 41.8 m/s, the maximum occurring on the blades near the outlet region of the pump. This is smaller than initial model.

For the absolute pressure fields for the optimal model, the affected zones are smaller than the ones calculated for the initial model.

4. Conclusions

This study is meant to underline a practical procedure to investigate the cavitation inside the centrifugal pumps and the way to have precise values of the process variables in order to get out the pump from the cavitation functioning zone.

The results retrieved in this study are in line with all known theoretical and experimental ones therefore it is a credible procedure.

The increasing of the outlet and NPSH pressures and the decrease of the rotor velocity is the solution to have any centrifugal pump out of the cavitation.

References

[1] Hofmann M., Lohrberg H., Ludwig G., Stoffel B., Reboud JL., Fortes-Patella R., "Numerical and experimental investigations on the self-oscillating behavior of cloud cavitation: part 1 visualisation / part 2 dynamic pressures", 3rd ASME/JSME Joint Fluids Engineering Conference, San Francisco, (1999)

[2] Lohrberg H., Stoffel B., Fortes-Patella R., Reboud JL., "Numerical and Experimental Investigations on the Cavitating Flow in a Cascade of Hydrofoils", Proceedings of the 4th Int. Symp. on Cavitation, Pasadena, California, (2001)

[3] Reboud J.L., Stutz B. and Coutier O., "Two-phase flow structure of cavitation: experiment and modelling of unsteady effects", Proceedings of the 3rd Int. Symp. on Cavitation, Grenoble, France, (1998)

[4] Takasugi N., Kato H., Yamagushi H., "Study on cavitating flow around a finite span hydrofoil", Cavitation and Multiphase Flow Forum, ASME-FED vol. 153, pp. 177-182, (1993).