ANALYSIS OF THE USE OF CENTRIFUGAL PUMP DESIGN TECHNIQUES IN AUTOMATED SYSTEMS

Vira Shendryk*, Olga Aleksenko*, Natalia Zinchenko*, Ivan Gordienko*

The article deals with the possibility of using different design techniques of centrifugal pumps designing in the context of their use in the computer-aided design. It is carried out a comparison of calculated and experimental pump characteristics.

Keywords: *centrifugal pumps, computer-aided design, the characteristics of pumps, pump design methods*

1. Introduction

Centrifugal pumps are the most common types of power machines. The design of these pumps is the selection of the internal parameters (combination of geometry of elements) of flowing parts according to the values of the output parameters (flow, pressure), due to the terms of reference for the designing of the pump.

In engineering practice the solution of this problem lies in the design calculation, often with a view to optimizing the geometrical parameters according to some optimality criterion (most often – the coefficient of performance).

Nowadays the following methods are used in the design of the flowing parts of centrifugal pumps [1]:

- conversion from a model to natural conditions (in the presence of the model flowing parts in full compliance with the requirements of the terms of reference);
- conversion from the model to natural conditions and subsequent considering not model changes (in case of deviation of the model flowing part from a complete geometric similarity);
- a new calculation of the flowing part involving generalized empirical data (if there are no model flowing parts);
- a numerical experiment using both modules developed within the organization, and the universal CAE-programs.

Years of experience in creating centrifugal pumps showed that the most reliable method of analysis and design of flowing parts of centrifugal pumps is the method based on a similarity theory, which is used in the presence of the flowing parts of the model with the required speed coefficient. This methodology is the most popular and doesn't require experimental verification.

Also frequently used are methods of translation from model to natural conditions with a subsequent consideration of not model changes, as they are a result of generalization of

* Assoc. prof. V. Shendryk, Ph.D., Assoc. prof. O. Aleksenko, Ph.D., N. Zinchenko, Ph.D. student, I. Gordienko, masters student, Department of Computing Science, Sumy State University, Sumy, Ukraine

a significant empirical experience and are easy to use (do not require a lot of processing procedures).

For all their popularity, simplicity and reliability the application of these two design methods is limited because they can be used only with the model flow parts, which in some cases can not be maintained.

There is a considerable interest in the possibility of creating new methods of flowing parts designing and verification of their reliability. When creating new flowing parts (in the absence of model flowing parts) there are widely used a theory of lattice, or generalized empirical dependence, or a calculation technique of an impeller using one-dimensional Euler's fluid dynamics equation. Most often, in this case, one has to consider several possible combinations of geometrics of the flowing part elements (alternatives), which provide the required operating parameters. The final choice of alternatives take place only at the stage of experimental development of pumps, which allows to accumulate most of the necessary information for a decision making.

The developers of modern CAE-modules claim, that they are multifunctional and can be used for engineering analysis [2] including pumping equipment. Results of designing typically contain data that can be used as input data for engineering analysis. At present there is a clear trend to complement CAD-tools with CAE-tools [3], which makes them very convenient for integration into the overall CAD-system. But still the issue of their veracity remains open.

Everything foresaid allows to draw a conclusion that a problem of the creation of computer-aided design of centrifugal pumps through the integration of new methods of designing hydraulics and CAD-, CAE-tools into a single design system is of current importance. In this context methods of designing of flowing parts can be considered as a methodology for zero level (designing calculations), the purpose of which is the accumulation of the necessary content of initial information, and CAE-Tools – as the methodology of the first level, the purpose of which is to substitute piloting of the pumps and to provide most of the necessary information for final selection of alternatives.

2. Problem definition

The objectives of this research are:

- Validation of CAE-tool COSMOSFloWorks, embedded in SolidWorks CAD system by comparing the results of calculations made by using this tool with the experimental characteristics of a real centrifugal pump;
- Verification of the range of design methods for flow parts of centrifugal pumps;
- Verification of the integration capability of a solid model of the design object and designing techniques of new hydraulics and COSMOSFloWorks CAE-tool into a single CAD system;
- making recommendations on sharing of designing techniques for new hydraulic flowing parts and COSMOSFloWorks CAE-tool.

Comparison and verification of the adequacy of methods and CAE-tool were made with the experimental characteristics of the pump AX 65-40-200K, which was manufactured and tested at PJSC 'Sumy Frunze NPO' (Sumy).

The pump AX 65-40-200K is chemical, single-stage, horizontal, cantilever, with an open impeller, with axial supply of liquid, which is designed for pumping chemically active and neutral liquids without inclusions or those that contain solid inclusions with bulk concentration up to 1.5 % and particle size up to 1 mm. The pump is designed in accordance with ISO 2858. The tests were performed on a dedicated test bench for centrifugal pumps of the type X (AX) 1.6010-78, which runs an annual certification to the National Centre for Standardization, Metrology and Certification of Sumy.

3. Results

The study had several phases.

In the first phase was implemented calculation of the basic elements of the flow, based on the procedure described in [4].

The required flow rate, pressure and rotational speed of the pump were the input data for calculation. At first the main geometric parameters of the flowing part elements were selected. At the stage of the main parameters selection the values of efficiency and pressure were estimated in the calculated point (point of maximum efficiency). Preliminary evaluation of hydraulic efficiency of the pump was performed according to the formulas, using calculation and experimental method of losses separation [5]. After the definition of the pump main parameters (outlet diameter, the width of the impeller at the inlet and outlet, angles of the blade at the inlet and outlet, the number of blades, etc.) the element wise calculation of losses in the flowing part was made in the context of one-dimensional Euler's equation. Loss of pressure in the pump is determined by the sum of losses in its individual parts (impeller, volute and volute diffuser). Then the pump parameters were refined based on these calculations. The results were compared with the pump performance, obtained from tests (Table 1).

Flow rate, m^3/hour	Head pressure, m		Deviation, %
	Experiment	Prediction	
25	49	48.6	0.8

Tab.1: Results of prediction of pressure characteristics according to the procedure [1, 5]

The analysis of the calculation results showed that the error in the calculation was 0.8 %.

In the second step the prediction of the pump performance was made. The procedure, which is given in [1, 5], is based on the analytical calculation of the pump performance $H = f(Q)$.

At first head pressure rate was defined from the dependence:

$$\Psi = (0.025\,\beta_2 - 0.6)\,q^2\,\delta_{\mathrm{m}}^2 + (10.6 - 0.23\,\beta_2)\,q^2\,\delta_{\mathrm{m}} + \\ + (2.03\,\beta_2 - 189)\,q^2 + (0.001\,\beta_2 - 0.105)\,\delta_{\mathrm{m}} + (0.002\,\beta_2 + 1.18)\ , \tag{1}$$

where $q = Q/(\pi\,D_2\,b_2\,u_2)$ – delivery rate; $\psi = 2\,g\,H/u^2$ – head pressure rate; β_2 – the angle of the blade at the outlet of the impeller; δ_{m} – the relative end clearance between the blade edge of the impeller and pump casing; D_2 – external diameter of the impeller; b_2 – the width of the impeller at the outlet; u_2 – portable velocity at the outlet of the impeller.

In this calculation the geometrical parameters δ_{m}, β_2, D_2, b_2, u_2 possessed fixed values obtained by calculation using the methodology [4]. The values Q were set in accordance with the experimental characteristics of the pump. The pressure was defined according to the obtained values of head pressure rate, and deviation of the design from the experimental data was calculated.

Flow rate, m^3/hour	Head pressure, m		Deviation, %
	Experiment	Prediction	
5	56.40	52.888	6.23
10	55.8	52.448	6.01
15	54.00	51.715	4.23
20	52.10	50.688	2.71
25	49	49.369	0.75

Tab.2: Results of the pressure characteristics prediction by the method [1]

The analysis of the results showed that the average deviation of calculation is 3.986 %, which is within engineering accuracy.

The next stage of the research was a simulation of the pump working process by means of Computational Fluid Dynamics (CFD). CFD is a powerful tool for fluid dynamics investigation in hydraulic machines. The hydraulic calculations were performed by COSMOS FloWorks CAE-software since the pump had been designed in SolidWorks2008. This software allows to set initial and boundary conditions, create mesh, solve and evaluate the results.

The fluid motion is modeled by the Navier-Stokes equations describing the laws of conservation of mass, momentum and energy in a time-dependent formation of the operating environment. The Reynolds averaged Navier-Stokes equations for steady-state flow were used for the calculations. Water with a reference temperature of 10 °C was given as the working fluid to comply with the calculation parameters of the physical experiment. The turbulence was modeled using the K-ε turbulence model. The turbulence parameters were set so that the turbulence intensity was 2 % and the turbulence length was set default in compliance with model dimensions.

To determine the boundaries of the analysis, COSMOSFIoWorks automatically creates the fluid volume using SolidWorks software geometry.

The pump casing walls were set as the stator. The environmental pressure was applied as inlet boundary condition. The pump volume flow rate was applied as outlet boundary condition.

As long as centrifugal pump analysis studies fluid behavior not only around the impeller but also inside the casing, a method of multiple rotating reference frame was used in the research. The local rotating region enclosed the pump impeller. The shape of the rotating region (Fig. 1) was chosen using next recommendations:

– the rotating region boundaries must be axisymmetric;
– the rotating region boundary are placed within a solid body to reduce negative influence on the calculation of the flow disturbances within the narrow gap.

The axis of the rotating region coincides with the impeller axis. The rotational speed for the rotating region is specified as equal to the speed in the physical experiment ($\omega = 1450$ RPM) that allows to validate computational results on experimental data.

The walls of the pump casing are specified as stator. The ambient pressure was set as inlet boundary condition. The pump volume flow was set as outlet boundary condition.

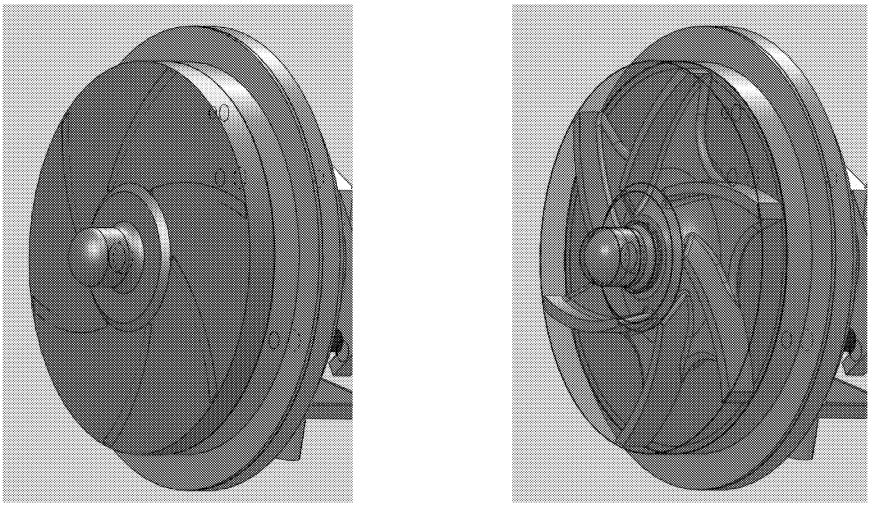

Fig.1: Local rotating region

The mesh for calculation was built with regard to the need of condensation in the local rotating region and in the areas of abrupt curvature change.

The calculating of flow parameters in COSMOSFloWorks is iterative, therefore, in the research there were identified the criteria for convergence, which ensure adequate results. The main performance of the pump is a head pressure, therefore the parameters, by which it is determined, should be classified as basic criteria for results convergence. Thus total pressure of fluid at the inlet and outlet of the pump was chosen as the first criteria of convergence. It should be also mentioned that pump workflow leads to the vortex at the inlet of the pump. Adequate calculation results of pump performance can be expected only when the vortex is fully formed. That's why the volumetric flow rate at the pump inlet was chosen as an additional criterion for the calculation of the convergence. These criteria were set as calculation goals.

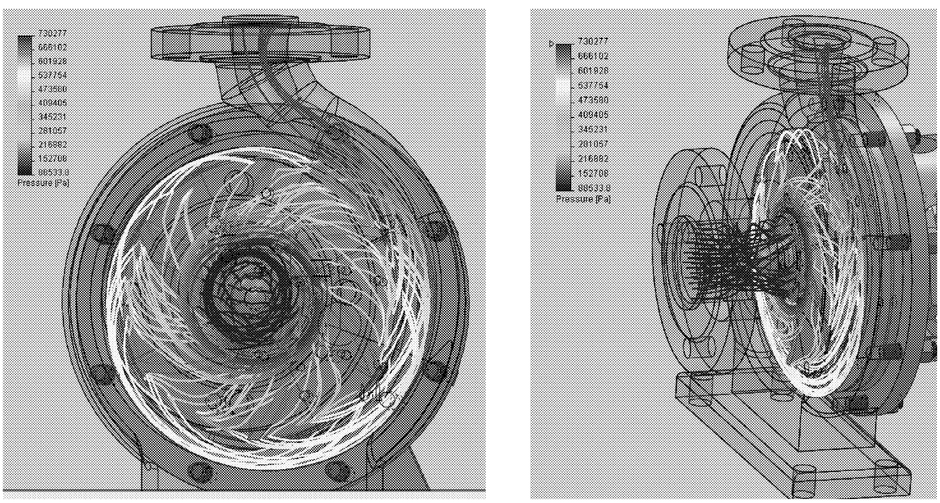

Fig.2: Visualization of lines of stream, painted by pressure values

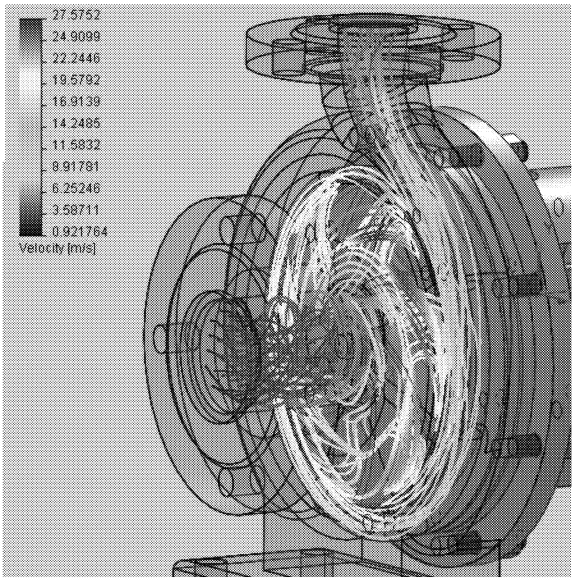

Fig.3: Visualization of streamflow lines, painted by velocity values

A 3-D pump model, built in SolidWorks 2008 on the basis of calculation results of the previous stage was used for fluid flow analysis (Fig. 2).

Fig. 3 shows increase in the rate on the blade edges of the impeller, which transfer energy to the workflow.

Model-based analysis showed that in case of convergence of all three selected criteria $H = f(Q)$ was the pump performance, calculated from the results of the design experiment and agreed with $H = f(Q)$ – the actual performance with a sufficient accuracy. But the convergence of all three selected criteria is achieved more than once. Therefore, the question of the results of which exactly cycle of convergence criteria should be selected for the final determination of the pump characteristics, remains open.

In the result of the design experiments we got the values of fluid pressure at the outlet of the pump, based on which we calculated the values of head pressure, generated by the pump. The results of comparison of the design values and the values obtained during full-scale experiments, are presented in Table 3.

Comparison of the results of modeling using COSMOSFloWorks tools with the experimental results showed that the established model has a sufficient accuracy.

Volume flow, m^3/hour	Head pressure, m		Deviation, %
	Experiment	COSMOSFloWorks model	
5	56.40	43.1	23.58
10	55.8	51.18	8.28
15	54.00	51.06	5.44
20	52.10	49.82	4.38
25	49	47.57	2.92

Tab.3: Results of calculation of head pressure parameter, carried out by COSMOSFloWorks

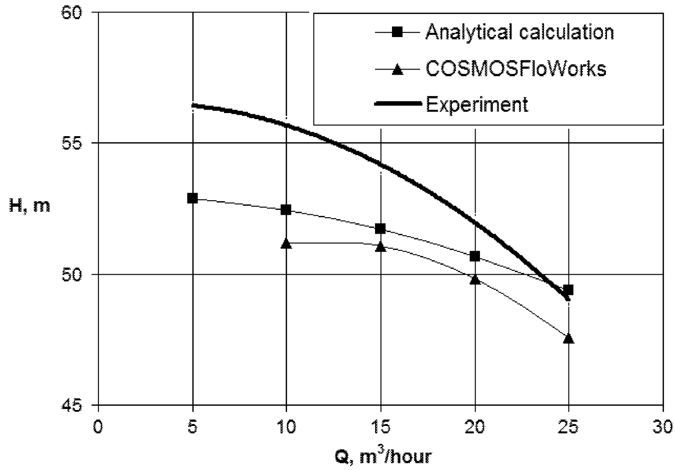

Fig.4: Comparison of experiment data calculation results

The comparison of experimental data with the results of analytical calculation and the calculation of the proposed model is shown at the Fig. 4.

The graph shows that the results, got by COSMOSFloWorks have a lower accuracy in the performance below optimum, than in the optimal one. But simulation data by COSMOSFloWorks are closer to experimental data in the greater than optimal performance.

One can also note that the slope of the curve $H = f(Q)$, obtained as a result of the numerical experiment in COSMOSFloWorks, best matches the slope of the curve $H = f(Q)$, obtained through a full-scale experiment.

The methodology, which is described in the work [4], gives the highest accuracy among the considered methods. Therefore, we suggest designing of a new flowing part of a centrifugal pump in the following way:

First, we determine the geometric parameters of flowing part using specified design values of head pressure and flow rate by the method [4]. Then we transfer the received values of geometrical parameters for the construction of the solid geometric model to the software SolidWorks. After building a geometric model we carry out a numerical experiment to get the curve $H = f(Q)$ using COSMOSFloWorks.

4. Conclusions

To ensure high energy efficiency of the pump with open impeller design it is necessary to repeat iteratively calculations in COSMOSFloWorks, which requires the use of the automation tools. The paper presents the methodology, which allows to get the optimal geometrics of the flowing part of the pump under given input conditions, to reduce time and material costs for design and experimental refinement.

The design procedure for the centrifugal pump with open impeller is as follows: by given parameters Q and H we determine the geometry of the pump flowing part with optimum efficiency by the method given in the paper [4]. The curve $H = f(Q)$ is predicted for the designed pump and a simulation is performed in COSMOSFloWorks.

To get optimal values of geometrics of the flowing part, the calculation in COSMOSFloWorks is carried out recursively. After each calculation the geometric parameters are changed according to the best received option and the curve $H = f(Q)$ is predicted. Such actions reiterate until the curve $H = f(Q)$ intersects at the point of the optimal efficiency, calculated by the method [4].

The research shows a potentiality of using simulation in COSMOSFloWorks to predict the performance of a centrifugal pump with open impeller on the basis of the geometric model, which has been received as a result of preliminary calculations and design [4, 5]. Carrying out the numerical experiment in COSMOSFloWorks allows to get results that agree with actual experimental data. Designing with COSMOSFloWorks is more flexible. It allows to predict energy performance of the similar pumps at the different stages of their designing.

References

[1] Rzhebaeva N.K, Rzhebaev E.E.: Calculation and design of centrifugal pumps: Manual, 220 p., SSU Publishing House, Sumy 2009 (in Russian)
[2] David C. Planchard, Marie P. Planchard: Engineering Design with SolidWorks 2008, Schroff Development Corporation, 2007 – 674 p.
[3] Kunwoo Lee: Principles of CAD/CAM/CAE, Prentice Hall; 1 edition, 1999 – 640 p.
[4] Rzhebaeva N.K, Shendryk V.V., Boroday M.V.: Methodology of calculating of the pumps with semi-open and open impellers, In: Vestnik NTU "KPI": Mechanical Engineering, Kyiv, 2002, No. 42, V. 2, pp. 166–170 (in Russian)
[5] Shendryk V., Vashchenko S.: The optimal design of pumps with open and semi-open impellers, Proceedings of the International conference 'HYDROTURBO 2008', Hrotovice, Czech Republic, 2008, pp. 59–68

Received in editor's office: June 15, 2013
Approved for publishing: September 7, 2013