DETERMINATION OF THE FLOW CHARACTERISTIC OF A HIGH-ROTATIONAL CENTRIFUGAL PUMP BY MEANS OF CFD METHODS

SŁAWOMIR DYKAS AND ANDRZEJ WILK

Institute of Power Engineering and Turbomachinery,
Silesian University of Technology,
Konarskiego 18, 44-100 Gliwice, Poland
{*slawomir.dykas, awilk*}*@polsl.pl*

(Received 30 June 2008)

Abstract: The paper refers to a numerical analysis of the flow through a centrifugal pump working at high rotational speed. This kind of a pump is characterized by a high delivery head and a low discharge. The considered pump is a one-stage centrifugal pump with a rotational speed of 12 000 rpm. This pump has been manufactured and prepared for experimental research. The main purpose of the presented research has been to determine the flow characteristic of the pump by means of CFD methods. To this end the commercial CFD code, ANSYS CFX 11, has been used. The obtained numerical results have shown high usefulness of the CFD methods for the flow machinery design process. In the next step the obtained flow characteristic will be compared with the experimental investigations.

Keywords: centrifugal pump, CFD modeling, flow characteristic

Notation

b – blade width [m]
d – diameter [m]
H – delivery head [m]
Q – discharge (volumetric flow rate) [m^3/s]
s – gap width [m]
t – pitch [m]
z – number of blades [–]

Indices
1 – inner
2 – outer
s – introductory impeller

1. Introduction

Enhancement of the rotational speed of a centrifugal pump decreases its overall dimensions and increases the operation parameters, especially the delivery head by low discharges. The current trend in the world is to increase the rotational speed of the pump even up to 30 000 rpm. It permits increasing the area of applications of these pumps. Simultaneously, such pumps have much lower sensitivity to mechanical impurities of the working liquid and the maintenance of these pumps usually poses fewer problems.

Nowadays, a fast progress in the field of the metallurgy, manufacturing of rolling bearings and packing systems can be seen. Increasing the rotational speed reduces the overall dimensions, the mass and consumption of material. For these reasons, further progress in the use and application of high rotational speeds for centrifugal pumps can be expected.

Production of pumps working at a high rotational speed is controlled by several manufacturers which treat their achievements as intellectual property and do not publish the technical data, very often considering it as secret (military technology).

The pump presented in this paper has been designed for a rotational speed of 12 000 rpm. The main purpose of the design has been, first and foremost, a simple geometry which is crucial for the pump manufacturing cost. Complicated blade shapes increase the cost significantly, especially if only several pieces are manufactured.

2. Model pump specification

A high delivery head of a pump with a closed impeller (rotor) or a semi-open impeller causes serious problems connected with the axial forces acting on the impeller [1]. For this reason, it is better to use open-flow impellers.

Using blades of a curvilinear shape in open-flow impellers causes serious strength problems at high rotational speed. The blades must be of variable thickness so that the resultant stress in the blades should have a constant value along the blade.

Significant deformations can take place for curvilinear blades, hence the use of wider gaps is forced. Manufacturing of curvilinear blades poses additional technological problems. Therefore, it is much easier to apply radial blades with simple shapes.

The literature connected with the issues of open impellers of centrifugal pumps working at high rotational speed is very scarce. It can be concluded that the operation parameters of pumps with an open-flow impeller with radial blades depends on such parameters as the inner and outer diameter of blades, the impeller width at the inlet and outlet, the dimensions of the gaps between the edges of the blades and the pump casing, and the number of blades.

Having performed a design analysis, one centrifugal pump variant has been chosen. The pump has one stage with an open-flow impeller, a horizontal axis, a spiral casing [2] and an outlet diffuser. The impeller has radial blades with a decreasing width.

The pump construction makes it possible to carry out laboratory tests with various outer and inner diameters of blades. It is also possible to change the width of blades (decreasing the impeller passage). It is possible to use different introductory impellers before the main (centrifugal) impeller.

The pump shaft has its own bearings. It is connected to the motor through a transmission chain increasing the rotation speed to 12 000 rpm.

There are sliding walls inside the spiral casing on the inlet side and behind the impeller. The walls permit axial gap control between the front casing wall and the impeller blades and between the impeller and the back casing wall. Mechanical sealing is provided at the shaft outlet from the pump casing (in the stuffing-box).

Numerical calculations of the discussed pump were carried out for 12 000 rpm for the impeller with $z = 12$ blades. The impeller dimensions were equal to the nominal values and were the following:

- outer diameter of the impeller $d_2 = 180$ mm,
- inner diameter of the impeller $d_1 = 60$ mm,
- blade width at inlet $b_1 = 18$ mm,
- blade width at outlet $b_2 = 9$ mm.

The following gaps were between the impeller blades and the pump casing:

- the width of the gap between the casing front wall and the impeller blades $s_1 = 3$ mm,
- the width of the gap between the casing back wall and the impeller blades $s_2 = 3$ mm.

An introductory one-coil impeller with a cylindrical hub and a cylindrical outer shape of the following diameters was used before the centrifugal impeller:

- outer diameter of the introductory impeller $d_{s2} = 40$ mm,
- screw line pitch $t_s = 21$ mm,
- number of screw line pitches $z_s = 2$,
- the width of the gap between the introductory impeller blades and the casing $s_s = 1$ mm.

Figure 1 shows pictures of the presented pump and its elements.

3. Numerical modeling

Due to the significant costs of experimental research a numerical analysis (CFD analysis) of fluid flow machinery is very attractive and promising. Additionally, thanks to a CFD analysis it is possible to find out more about the flow behavior which can be crucial for a future optimization of the flow channels in the considered fluid flow machinery, *e.g.* the pump. For that reason, efforts have been made to determine the flow numerical characteristics, *i.e.* the delivery head relation in the function of discharge $H = f(Q)$.

In the paper, tentative numerical calculations have been presented for a centrifugal pump with a double open main impeller (Figure 2, left) and an introductory impeller (Figure 2, right).

The rotational speed of the analyzed pump is 12 000 rpm. The outflow from the impeller takes place through a spiral collector and a diffuser (Figure 3).

The CFD modeling has been performed using the ANSYS CFX 11 commercial code on an unstructured numerical mesh consisting of 4 million nodes. The numerical mesh has been prepared in such a way so as to ensure a best solution for the

Figure 1. Pictures of the manufactured centrifugal pump

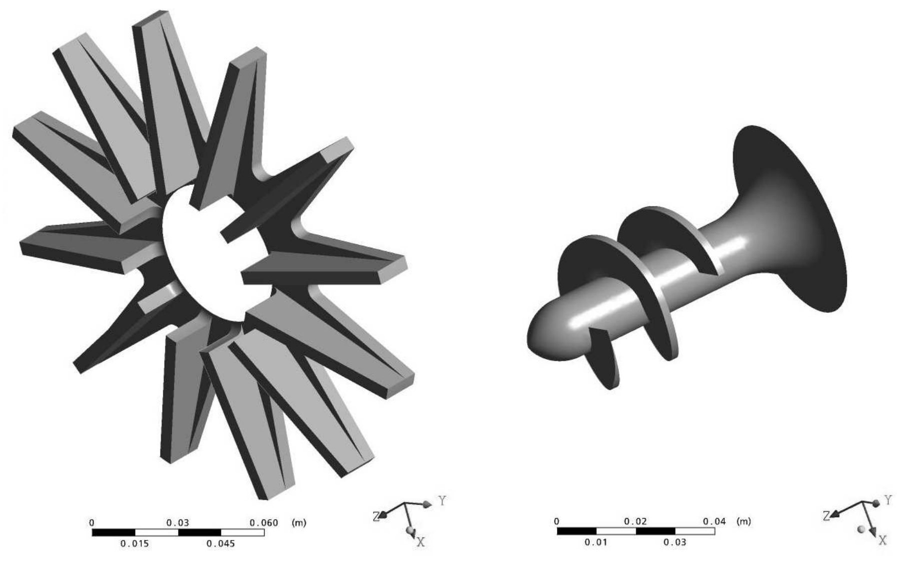

Figure 2. View of pump impellers: left – main impeller, right – introductory impeller

presented quantitatively calculations. For a more qualitative analysis planned for shape optimization of the pump flow paths (impeller blades, size of the gaps between the impeller and the casing, *etc.*) the numerical grid has to be finer, but then calculations will be made not for the whole pump.

Figure 3. View of spiral outflow collector and outlet diffuser

3.1. Boundary conditions and flow model

The computational domain for the considered configuration of a high-rotational centrifugal pump is shown in Figure 4. Both impellers rotate with a rotational speed of 12 000 rpm, the outflow diffuser and both impeller casings are treated as stationary. It has allowed us to take into consideration the impeller tip gaps in the numerical algorithm.

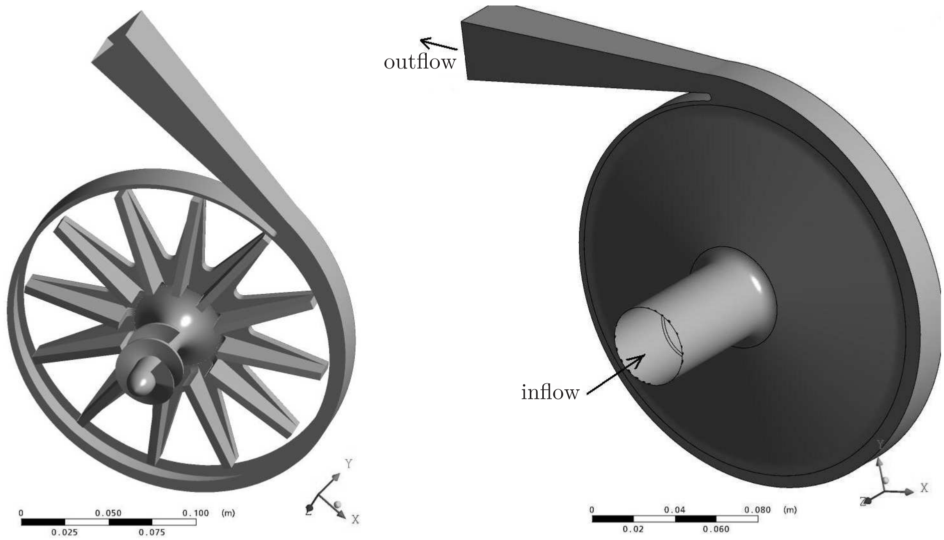

Figure 4. Numerical domain: left – view of impellers and outflow collector, right – view of casings with marked inflow and outflow planes

The RANS method available in ANSYS CFX 11 has been used for the CFD modeling. It means that stationary flow fields have been predicted. The calculations have been performed by means of an *Isothermal at 25°C* heat transfer model with k-ε turbulence model. At the impeller/diffuser connection the *Frozen rotor* interface model has been set up [3].

The total pressure at the inlet has been set up as a boundary condition, whereas the discharge value has been assumed at the inlet. For that assumption the delivery head has been a result of computations.

3.2. Numerical results

Two flow types have been considered, namely a single phase flow and a flow with cavitation. In the former case the main purpose has been to determine the pump flow characteristic for one rotational speed of 12 000 rpm. To this end five calculations have been done for the same constant total pressure at the inlet, $p_0 = 0.12$ MPa, and five different discharges at the outlet, $Q_{\rm out} = 1\cdot10^{-3}$, $2\cdot10^{-3}$, $3\cdot10^{-3}$, $4\cdot10^{-3}$, $5\cdot10^{-3}$ m^3/s. The Q change range has been assumed on the basis of the designed value of the discharge for the introductory impeller for the considered rotational speed. In the case of the introductory impeller, which has the shape of a screw, the designed discharge for a rotation of 12 000 rpm has been about $3\cdot10^{-3}$ m^3/s.

Figure 5 shows the pump flow characteristic determined from CFD flow simulations. The characteristic line has been drawn using spline approximation on the basis of five points. It is visible from the characteristic that the maximum delivery head for a discharge is a little less than 3 l/s. However, the obtained characteristic is relatively flat in the range between 1 and 5 l/s. The maximum difference in the delivery head is about 20 m.

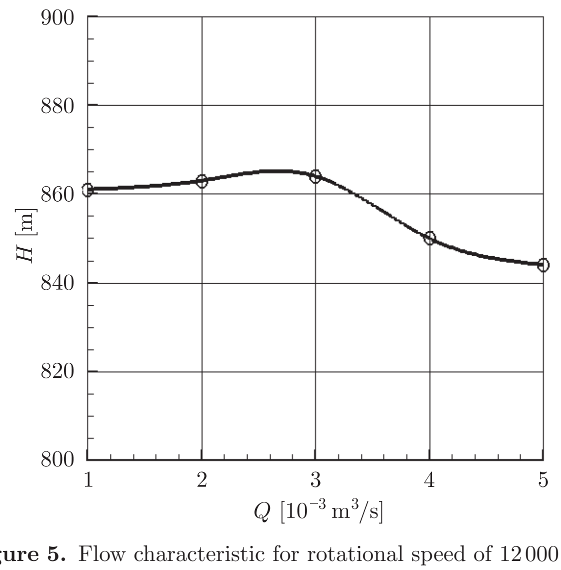

Figure 5. Flow characteristic for rotational speed of 12 000 rpm

A further analysis of the flow field has allowed us to asses the flow behavior in the pump. The pressure increase in the introductory impeller is relatively low and amounts to about 0.5 MPa (for $Q = 3$ l/s). This pressure increase is sufficient to prevent the cavitation process at the inlet to the main impeller. It has been confirmed in the calculations with a cavitation model. The flow characteristic for

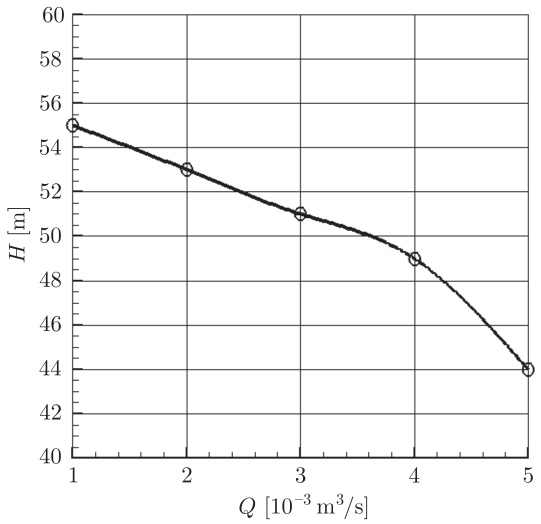

Figure 6. Flow characteristic introductory impeller for rotational speed of 12 000 rpm

the introductory impeller is presented in Figure 6. It is visible that the delivery head is strongly decreasing with the increasing discharge value.

A significant pressure increase is observed at the outlet from the main impeller, but the main pressure increase is realized in the spiral casing and the outlet diffuser (Figure 7).

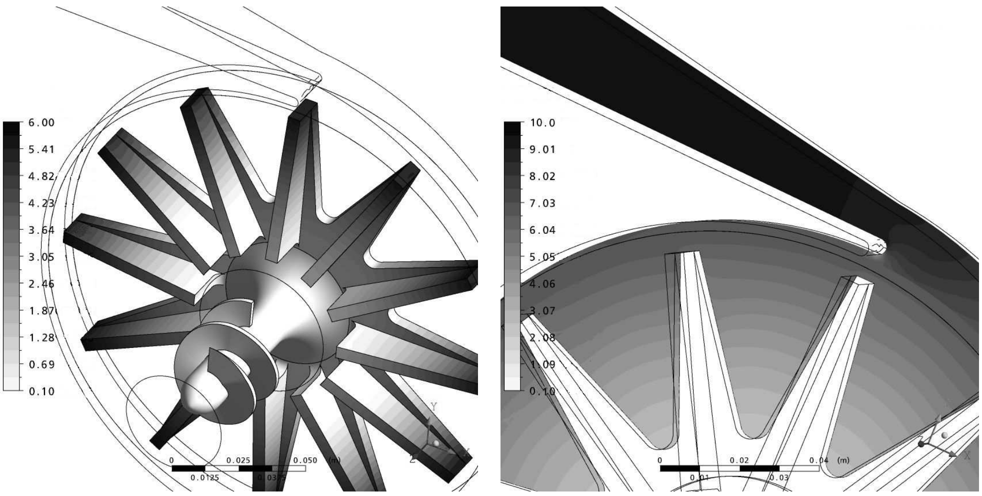

Figure 7. Pressure [MPa] contours on the impeller surface (left) and at the inlet to the diffuser (right)

It is the dimensions of the gaps between the main impeller and the impeller casing that are very important for the high efficiency and correct (safety) operation of the pump. It is also the tip impeller gap and the gaps ahead and behind the impeller blades that are important because the analyzed pump has an open-flow impeller. The tip impeller gap (between the impeller's outer diameter and the volute tongue) has been 10 mm and both the remaining gaps have been 3 mm each. In Figure 8 the velocity vectors in all gaps can be seen. Very high velocity values are observed for the tip flow and for the inlet to the outlet diffuser. A secondary flow is visible close to the walls of the outlet diffuser in the initial part. The flows in the gaps between the impeller and the front and back walls of the casing are relatively small. A conclusion can be drawn that a decrease in the size of gaps will not increase the pump's efficiency significantly and can increase the risk of the impeller's contact with its casing, which is of course undesirable.

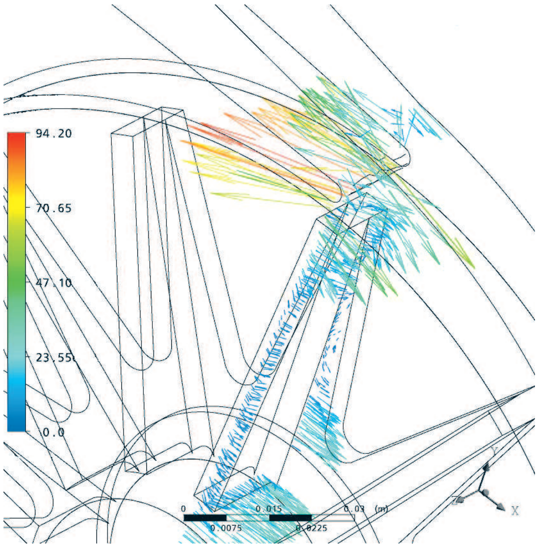

Figure 8. Velocity [m/s] vectors on the YZ plane along the main impeller blade and at the inlet to the outlet part of the diffuser

The performed CFD analysis has allowed us to determine the forces as well as the torques acting on the impellers and the outlet diffuser on the basis of static pressure distributions and areas. For the test case with the charge of 3 l/s the force acting on the main impeller in direction Z has been about 900 N. It shows that the axial thrust for an open impeller is not high. The torque for the main impeller in direction Z has been about 77 Nm, whereas the value for the introductory impeller has been less than 0.5 Nm. These quantities have shown that the the pump design ensures proper operation.

4. Summary and future work

The presented paper deals with the numerical determination of the flow characteristic of a centrifugal pump rotating at high rotational speed, 12 000 rpm. The obtained results have been consistent with the assumed and designed parameters. The CFD flow simulation has allowed us to make a detailed assessment of the flow field in the parts where measurement is very difficult or even impossible.

The developed procedure for numerical determination of the flow characteristic of the presented centrifugal pump will be further verified using the measurement data obtained in an experimental facility at the Institute of Power Engineering and Turbomachinery of the Silesian University of Technology. Next, following the presented analysis, a numerical shape optimization of the main impeller as well as introductory impeller geometry are planned.

Acknowledgements

The authors would like to thank the Polish Ministry of High Education and Science for the financial support of the research project N N502 4643 33.

References

[1] Wilk A 2004 *SYSTEMS J. Transdisciplinary Systems Science* **8/2003** 520 (in Polish)

[2] Wilk A 2007 *V Konferencja Naukowo-Techniczna Racjonalizacja Gospodarki Energetycznej REGOS'2007*, Łódź, Poland, pp. 297–304 (in Polish)

[3] *ANSYS CFX 11 documentation*, http://www.ansys.com/products/cfx.asp