The CFD calculations as a main tool for the mixed–flow pump modernization

J Skrzypacz¹, P Szulc¹

¹ Faculty of Mechanical and Power Engineering, Department of Design Fundamentals and Fluid–Flow Machinery, Wroclaw University of Science and Technology, ul. Wybrzeze Wyspianskiego 27, 50–370 Wroclaw, Poland

E-mail: janusz.skrzypacz@pwr.edu.pl

Abstract. The primary aim of the project was to increase cavitation performance of a big mixed–flow pump. The CFD software was used as a main research tool. The two kinds of numerical models were used to solve this problem – both were presented. The results were verified on the basis of comparison with experimental results obtained for real and model pumps. The interesting way of estimating cavitation performance was presented. The pump characteristics before and after the modernisation were shown.

1.   Introduction
Pumps are the biggest (after electrical motors) consumers of power produced globally. It is said that the range of this consumption could reach even 30%. In accordance with this estimation the efficiency of the pump should be as high as possible [3], of course, in compliance with BAT (Best Available Technology). The literature [2, 6–9] describes the design method that meets this significant requirement. However, there is another important parameter which should be taken into consideration – suction performance of the pump. This crucial aspect should be analysed, especially for a high specific speed pump.

When a designer designs or modernizes flow elements of turbo machinery there is always a dilemma if a new structure fulfills all requirements. The answer could be obtained by performing a test in the laboratory. A problem arises when it is not possible to realize the experimental investigation due to, e.g. power, flow rate or the pipeline connector dimensions. In such a situation the law of hydrodynamic affinity can be applied. It means that the model machine has to be designed (according to the affinity law), next an experimental test should be carried out and after wards the results of the test should be recalculated for real machine dimensions [2, 9]. This way is very expensive and time-consuming.

Owing to CFD a designer has new possibilities in the area of the turbo machinery design. He can analyze many variants in a „virtual laboratory” and choose the best solution. His actions are not limited by the machine dimensions or operating parameters.

The results shown in this article are part of a bigger project aiming at modernization of the large mixed–flow pump. It was decided that CFD would be the primary research method. The main goal of the project was to increase the cavitation performance with no changes in the level of efficiency.
2. Research object
The research object was a mixed–flow pump (Figure 1) applied, e.g. in a cooling tower. The pump dimensions and operating parameters are shown in Table 1. Presented parameters exceed the measurement possibilities of the testing station.

From the hydraulic point of view this pump consists of: the inlet element, the impeller and the diffuser.

| Parameter                                      | Symbol | Value | Unit  |
|-----------------------------------------------|--------|-------|-------|
| 1 Flow rate                                   | $Q$    | 23000 | m$^3$/h|
| 2 Head                                        | $H$    | 23    | m     |
| 3 Rotational speed                             | $n$    | 370   | rpm   |
| 4 Power                                       | $P$    | 1.7   | MW    |
| 5 Impeller external diameter                   | $d_2$  | 1.412 | m     |
| 6 Diameter of pump discharge connector         | $d_d$  | 1.6   | m     |

Figure 1. The analysed mixed–flow pump.

3. Numerical model
Considering complexity of the problem numerical calculations were performed in two stages:

- **STAGE I** – The main aim of this stage was to determine: the characteristics of the performance, efficiency of different kinds of the inlet elements and identify flow phenomena inside the hydraulic channels of the pump. At this stage the numerical model consisted of the inlet element, impeller, diffuser and the auxiliary outlet volume (see Figure. 2). All impeller and stator channels were
modelled due to the asymmetric variants of the inlet element. The Fluent [4] software was applied for the calculation. The pump volumetric delivery was taken into account in calculations.

![Figure 2. Numerical model. Model for stage I with different inlet elements.](image)

- STAGE II – The main aim of this stage was to determine the influence of individual geometrical features of the impeller inlet on the pump cavitation performance. At this stage numerical model consisted of a single channel of the impeller and auxiliary the inlet and the outlet volumes (Figure 3). The pump volumetric delivery was taken into account in calculations. The Fluent [4] software was used for the calculation. The research object was the mixed–flow pump (Figure 1) applied, e.g. in a cooling tower.

![Figure 3. Numerical model. Model for stage II.](image)

In both stages the calculations were carried out as stationary with the second order discretization scheme. The boundary conditions in both stages were defined as velocity on the inlet surface and the static pressure on the outlet surface (Figure 2). Grid in both stages was based on the hexa elements and concentrated next to the blades. The grid on the blades is regular (Figure 4). The number of cells equals, for:

- Stage I – 2340000,
- Stage II – 195000.
To obtain the cavitation resistance of the pump a multiphase cavitation model was applied. The cavitation performance was calculated in accordance with the equation (1), which was verified in [1]. This is a simple and fast way to determine the cavitation performance based on a CFD calculation.

\[
NPSH_i = p_{\text{sin}} - p_{\text{s min}} + \frac{c_0^2 - c_{\text{s min}}^2}{2g}
\]

where:
- \(NPSH_i\) – NPSH for inception,
- \(p_{\text{sin}}\) – static pressure on the impeller inlet surface,
- \(p_{\text{s min}}\) – minimal static pressure into the impeller channel,
- \(c_0\) – velocity on the impeller inlet surface,
- \(c_{\text{min}}\) – velocity at the point where the static pressure is minimal.

4. Numerical model verification

Whenever the results of numerical calculations are analysed their quality raises doubts. Conducted measurements (according to [5]) of the real and the model pump allows to compare physical results with the ones obtained from the CFD analysis. The comparison is shown in Figure 5. In the vicinity of an optimum operating point the average error of calculations is less than 4%. This divergence of the results was mainly caused by the fact that the numerical model did not include the element carrying the fluid from the stator outlet to the pipeline connector of the pump.

It is worth to mention that the difference between the real pump and the model pump characteristics is similar to the difference between the real pump characteristic and the virtual pump characteristic. It means the CFD analyses are a good alternative to experimental tests for both constructions: the real and the model pumps.
5. Results
The examples of calculation results of the stage I are shown in Figures 6 and 7.

The analysis show, that in the best efficiency point the pump operates without any swirls, nevertheless, the backward flows and other disturbing flow phenomena occur in the other operation points. However, the main goal of the research was to optimise the impeller inlet structure to obtain a better cavitation performance.

Figure 6. Examples of I–st stage calculation. Static pressure distribution.

Figure 7. Examples of I–st stage calculation. Velocity distribution.

In course of the project the following geometrical features of the impeller inlet were tested:
- The location of the leading edge in a meridional plane.
- The inlet attack angle (inlet blade angle).
- The numbers of blade.
- The splitters.
- Mix of the above.

Figure 8 shows the comparison of the performance curves calculated for the pump with the impeller before the modernisation and for the best version of the impeller after the modernisation. The model of the stage I was used in both characteristic calculations. It must be emphasised that the NPSH values shown in figure 8 should not be considered as the crucial ones. These values correspond to the initial stage of the cavitation (formula (1)), therefore, the crucial values are considerably lower. Figure
8 shows that the applied modifications have not resulted in the change of optimum operating point parameters (BEP) (the pump head increased slightly), however, the dramatic increase of the cavitation performance of the pump was observed.

![Image](chart.png)

**Figure 8.** The flow and the cavitation characteristics before and after the impeller modernization (for the best impeller solution).

6. **Conclusions**

The presented research proved that the CFD software is a very useful tool for industrial applications. Due to numerical calculations realized for many variants the new design of the mixed–flow pump impeller of a much better cavitation performance and a little better performance curves was developed. The new impeller made and it has been operating well in one of the European power plants. Thanks to the calculations some contradictory ideas presented in professional literature were verified, for instance, the ideas of influence of the inlet attack angle on the mixed flow pump cavitation performance.

**References**

[1] Misiewicz A and Skrzypacz J 2006 Obraz zachowań kawitacyjnych wirników pomp wolnobieżnych projektowanych wg zasady tzw "ciasnego wlotu" International conference Hydroforum 2005 pp 291–299
[2] Lazarkiewicz S and Troškolański A 1965 *Impeller pumps* (Warsaw: Wydawnictwo Naukowo–Techniczne)
[3] Shankar V, Umashankar S, Paramasivam S and Hanigovszki N 2016 A comprehensive review on energy efficiency enhancement initiatives in centrifugal pumping system *Applied Energy* 181 pp 495–513
[4] Ansys Fluent documentation
[5] The European Standard EN ISO 9906:2012 2012 *Rotodynamic pumps – Hydraulic performance acceptance tests – Grades 1, 2 and 3*
[6] Varchola M and Hlbocan P 2014 Interaction between impeller and axial vane diffuser of mixed flow pump *Applied Mechanics and Materials* 360
[7] Jin–Hyuk K and Kwang–Yong K 2011 Optimization of Vane Diffuser in a Mixed–Flow Pump for High Efficiency Design, *International Journal of Fluid Machinery and Systems*
[8] Urban G and Varchola M 2006 Hydraulic relation between the impeller and the diffuser of the mixed–flow pump *Mechanical Engineering (Bratislava)*
[9] Gulich J F 2008 *Centrifugal pumps* (Berlin Heidelberg: Springer–Verlag)