The influence of a pipe impeller external shape on the pump parameters

J Skrzypacz¹, P Szulc¹

¹ Wroclaw University of Science and Technology, Faculty of Mechanical and Power Engineering, Wybrzeze Wyspianskiego 27, 50–370 Wroclaw, Poland

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

Abstract. Multi-piped impellers (name proposed by the authors) represent an innovative approach to the design of rotodynamic pumps operating at the specific speed \( n_q < 10 \). In such a construction the energy increase is caused by the flow of liquid through the internal impeller passages as well as by an external flow around the passages, and sensible designing of such a construction is extremely difficult. The study investigates the flow phenomena and impact of the external shape of a multi–pipe impeller on the efficiency of the process of energy transfer into liquid. The main research method involves computational fluid dynamics (CFD) simulations.

1. Introduction

Multi–piped impellers (the name is suggested by the authors) [1] develop the concept of pumping discs with drilled holes [2, 3] used in rotodynamic pumps with the specific speed \( n_q < 10 \). Such pumps are widely applied in the chemicals industry, machine lubrication systems, liquid gas technology and firefighting, among others.

From the point of view of the liquid flow through the impeller flow channels, both constructions – a multi–piped impeller and a drilled impeller – are identical (Fig. 1). The essential difference in operation results from the different phenomena at the external flow around both of the impellers. In the drilled impeller, the external flow around the body is accompanied with the friction of the rotating disks of the impeller and the liquid, which results in power losses proportional to the fifth power of the outer diameter of the impeller and the third power of the angular speed \( (d^5 \text{ and } \omega^3) \) [2].

Figure 1. Drilled impeller and multi-piped impeller pump.
In the case of multi–piped impellers, the external flow around the pipes generates a drag force, the value of which can be theoretically expressed with the following relationship [4]:

\[ P_x = C_x \frac{D \mu^2}{2} A \]  

(1)

The value of the real drag force will be different due to the mutual interaction of individual pipes, influencing the speed profile of the liquid flow around the impeller flow channels. However, it seems that some of the power necessary to overcome the drag force at a specific rotational speed will cause an increase in the liquid circulation inside the pump and, hence, an increase in the total head. Initial experimental tests revealed that a multi–piped impeller generates a 30% higher head as compared to a drilled impeller with identically sized flow channels at a comparable efficiency level [5].

The main aim of this project is to identify flow phenomena that occur during external flow around an impeller channels. The influence of the impeller external shape on the operating parameters were investigated.

2. Object of research

A five–passage multi–piped impeller, designed for the operating parameters presented in Tab. 1, was used as the base construction (I_Base) for the study. Typical dimensions of the impeller are shown in figure 2 and operating parameters in table 1. The impeller works with a constant cross–section stator, whose outlet ends with an outlet diffuser.

The impeller was manufactured by means of SLS method and investigated on the special test rig. Details about test rig one can find in [5, 6]. The results of measuring the performance curves of the base construction are presented in figure 3. Results of measurements were a base to numerical model validation.

![Figure 2](image)

**Figure 2.** Dimensions of a basic multi–piped impeller and real model used to test.

**Table 1.** Operating parameters of the base impeller.

| No. | Name             | Symbol | Value |
|-----|------------------|--------|-------|
| 1   | Flow rate        | Q [m³/h] | 4     |
| 2   | Total head       | H [m]  | 28    |
| 3   | Rotational speed | n [rpm]| 2870  |
Figure 3. The characteristics of the pump with a multi-piped impeller shown in Figure 2.

3. Numerical model
The CFD Ansys FLUENT software was used for analyzing the liquid flow through a multi-piped impeller. The geometrical model reflected the geometry of the impeller and the stator used in the pump in the test rig. The model consisted of the following volumes (Fig. 4): inlet, impeller and stator. The boundary conditions were defined according to Figure 4 as:
- INLET model – velocity at the inlet, corresponding to the assumed stream of volume and turbulence intensity $I = 2\%$.
- OUTLET model – static pressure $p = 400\,000$ Pa, corresponding to the expected head at the optimum operating point.

Figure 4. The geometrical model of the pump with a multi-piped impeller for numerical calculations.
The detail information about, grid, turbulence model, calculation parameters, model validation one can find in [5, 6].

The global parameters of the numerical pump operation were determined according to the formulas (1–5) [7]:

\[ H = \frac{P_{oa} - P_{oi}}{\rho g}, \]  
\[ P_w = M_i \omega, \]  
\[ M_i = M_i + M_o, \]  
\[ \eta_h = \frac{P_h}{P_w} = \frac{\rho g QH}{M_i \omega}, \]  
\[ \eta = \eta_h \eta_v \eta_m. \]

The total efficiency was determined assuming a fixed value of volumetric efficiency \( \eta_v = 0.92 \) and mechanical efficiency \( \eta_m = 0.95 \) within the entire flow rate range.

The compression of the numerical and experimental pump curves is presented in the Figure 5. The difference between the results of the experimental and numerical tests next to the BEP (Best Efficiency Point) does not exceed 0.5% (within the entire comparison range, the difference between the results of the experimental and numerical tests does not exceed 3%). The achieved accuracy of the numerical calculations’ results seems highly satisfactory [8, 9].

**Figure 5.** Comparison of the characteristics determined numerically and experimentally.

The graphical results of the calculations are presented in the figures (6–9).
Figure 6. Distribution of static pressure on the model central plane, with a magnified impeller inlet section.

Figure 7. Distribution of the liquid velocity on the model central plane, with a magnified section between the blades.

Figure 8. a) Streamlines - the colours describe the speed value, b) total pressure distribution at the model mean plane.

Figure 9. Velocity distribution between the impeller flow channels.
It can be seen in the figures above that the distribution of (Fig. 7) and pressure (Fig. 6) is certainly correct. There are no visible areas of strong turbulence nor are any other non–stationary phenomena present. This conclusion confirms also the Figure 8. One can observe areas of the pressure dropping at the inlet to the impeller flow channels, which can result in a high value of dynamic depression and, hence, in a high value of the impeller NPSHR. Observing the velocity distribution on the plane between the impeller flow channels (Fig. 9), one can also observe liquid circulation generating an exchange of momentum, which seems to be the primary mechanism of an increase in the head, resulting from the external flow around the impeller flow channels.

4. Analysis of the impact of the outer profile of pipes on the operating parameters of a multi–piped impeller

In order to determine the influence of the outer shape of the pipes on the process of energy transfer into liquid, a numerical experiment was conducted. The experiment was aimed at analysing the impact of four additional outer profiles of pipes, presented in Table 2. The examination covered five–passage impellers at a constant value of \( Q = 4 \text{ m}^3/\text{h} \).

| Table 2. Outer profiles of passages (pipes). |
|---------------------------------------------|
| I_Base | Variant 1 | Variant 2 | Variant 3 | Variant 4 |
| ![Image](image1.png) | ![Image](image2.png) | ![Image](image3.png) | ![Image](image4.png) | ![Image](image5.png) |

**Figure 10.** Distribution of velocity for various external profiles of impellers: (a) I_Base, (b) Variant 1, (c) Variant 2, (d) Variant 3.

The results of the calculations are presented in graphic form in Figure 10. The greatest total head was obtained for the base impeller (I_Base), characterized by the highest velocity (Fig. 10a), whereas
the greatest hydraulic efficiency was obtained for the impeller corresponding to Variant 1, where the external profile of the impeller passages resembled an ellipse. The latter case reveals the most uniform velocity area in the pump chamber, measured in the radial direction (Fig. 10b). Variant 4 reached such low efficiency, that it was not presented in graphical way.

5. Conclusion
Multi-piped impellers make an interesting alternative to a construction of rotodynamic pump impellers used for the specific speed of \( n_q < 10 \) and flow rate \( Q < 10 \text{ m}^3/\text{h} \). The external shape of the passages has a significant impact on the achieved working parameters and efficiency. From a functional and technological point of view, it seems that using elliptical and circular profiles makes most sense. The elliptical profile ensures a slightly higher total pump head, as compared to the circular profile, at a comparable efficiency level.

References
[1] Skrzypacz J 2008 Wirnik pompy wirowej (Rotodynamic Impeller Pump), Patent PL 386135, Poland
[2] Gulich J 2008 Centrifugal Pumps (Berlin: Springer)
[3] Jędrach W 2001 Pompy Wirowe (Impeller Pumps) (Warsaw: PWN)
[4] Anderson J 2001 Fundamentals of Aerodynamics, third ed. (New York: McGraw–Hill)
[5] Skrzypacz J 2014 Numerical modelling of the flow phenomena in a pump with a multi–piped impeller Chemical Engineering and Processing: Process Intensification 75 58
[6] Skrzypacz J 2014 Investigating the impact of multi–piped impellers on the efficiency of rotodynamic pumps operating at ultra–low specific speed Chemical Engineering and Processing: Process Intensification 86 145
[7] The European Standard EN ISO 9906:2000, Rotodynamic pumps, Hydraulic performance, acceptance tests, Grades 1 and 2, BSI, 2003
[8] Kaewnai S, Chamaoot M, Wongwises S 2009 Predicting performance of radial flow type impeller of centrifugal pump using CFD J. of Mechanical Science and Technology 23 1620
[9] Cui B, Lin Y, Jin Y 2011 Numerical Simulation of Flow in Centrifugal Pump with Complex Impeller J. of Thermal Science 20(1) 47

Acknowledgements
Calculations have been made using resources provided by Wroclaw Centre for Networking and Supercomputing (http://wcns.pl), grant No. 444/2017.

Nomenclature
- \( A \) – area of cross–section [m²],
- \( H \) – pump total head [m],
- \( M \) – total momentum on the surface of the rotating impeller flow channels [Nm],
- \( n \) – rotational speed [rpm],
- \( n_q \) – kinematic specific speed \( (n_q = nQ^{0.5}/H^{0.75}) \) [rpm],
- \( P_h \) – hydraulic power [W],
- \( P_w \) – power on a pump shaft [W],
- \( P_c \) – drag force [N],
- \( Q \) – flow rate [m³/s],
- \( \eta_h \) – hydraulic efficiency,
- \( \eta_v \) – volumetric efficiency,
- \( \eta_m \) – mechanical efficiency,
- \( \eta \) – total efficiency,
- \( \rho \) – density [kg/m³].