Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 2, 2011, vol. LVII
article No. 1887

Lukáš ZAVADIL[*], Sylva DRÁBKOVÁ[**]

DETERMINATION OF PUMP PERFORMANCE USING NUMERICAL MODELLING

PREDIKCE CHARAKTERISTIK ČERPADLA S VYUŽITÍM NUMERICKÉHO MODELOVÁNÍ

Abstract

This paper deals with the numerical modelling of the flow in the single-stage centrifugal pump. The main objective is to determine pump performance characteristics using two different geometries – the first one consisting only of the inlet part, impeller and volute, the second one including also the impeller casing, i.e. the gaps on both sides of the impeller. Both stationary and time dependent problems were solved to obtain data for comparison with experimental measurement. The influence of geometry reduction was evaluated and advantages and limitations of both approaches were depicted.

Abstrakt

Článek se zabývá numerickým modelováním proudění v jednostupňovém hydrodynamickém čerpadle. Hlavním cílem je určení charakteristik čerpadla pro dvě rozdílné geometrie: první obsahující pouze hydraulické části čerpadla, druhou zahrnující také statorovou část, tedy prostory kolem oběžného kola. Proudění bylo modelováno jako stacionární i jako nestacionární, s cílem porovnat získané výsledky s fyzikálním experimentem. Byl posouzen vliv redukce geometrie a výhody i omezení obou přístupů.

1 INTRODUCTION

Pump characteristic curves describe the variation of head, power consumption, efficiency and NPSH on the flow rate. They are obtained by measurement and provided by pump producers. During the process of pump design, numerical modelling can be applied as a tool for the pump performance prediction. This may be important especially in cases when the pump specific speed is out of the common range and can help to investigate pump performance under various operational conditions.

Numerical modelling has been used to investigate the flow in a single-stage centrifugal pump with horizontally mounted shaft and twisted blade. The design parameters are specified in table1.

Tab. 1 Design parameters of modeled centrifugal pump.

Head	H =	80.9	[m]	Outlet diameter	D_2 =	244	[mm]
Flow rate (capacity)	Q =	7	[$dm^3 \cdot s^{-1}$]	Number of blades	z =	5	
Rotational speed	n =	2900	[min^{-1}]				

As can be seen from design parameters, the pump provides high head and low flow rate, which yields a low value of non-dimensional specific speed

[*] Ing. Lukáš ZAVADIL, VŠB-Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Hydromechanics and Hydraulic Equipment, 17. listopadu 15/2172, 708 33 Ostrava, Czech Republic, tel.: (+420) 59 732 5753, e-mail: lukas.zavadil@vsb.cz
[**] doc. Ing. Sylva DRÁBKOVÁ, Ph.D., VŠB-Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Hydromechanics and Hydraulic Equipment, 17. listopadu 15/2172, 708 33 Ostrava, Czech Republic, tel. (+420) 59 732 4386, e-mail sylva.drabkova@vsb.cz

$$n_b = n\frac{Q^{0.5}}{Y^{0.75}} = \frac{2900}{60}\cdot\frac{0.007^{0.5}}{(80.9\cdot 9.81)^{0.75}} = 0.0263\ , \quad (1)$$

where:

n_b – specific speed $[1]$, n – rotational speed [s^{-1}], Q – flow rate [$m^3 \cdot s^{-1}$], Y – specific energy [$J \cdot kg^{-1}$]

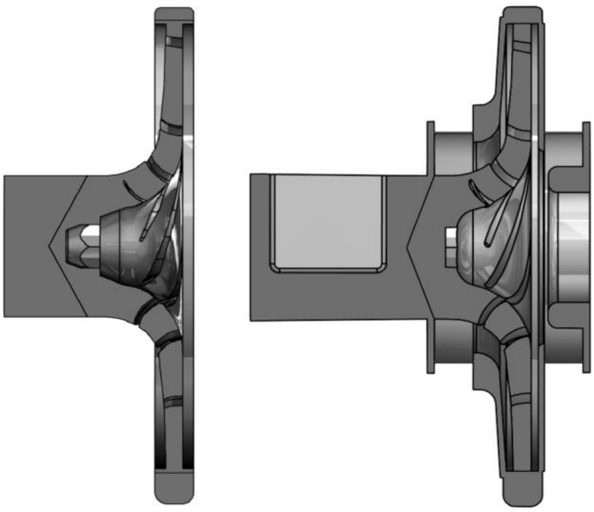
The obtained value of specific speed is very low and it is below the specific speed range considered for the radial type of impeller. It is caused by the high values of the head in connection with the low values of the flow rate. Such parameters can be reached by twisting the blades of the impeller, as was designed at the Victor Kaplan Department of Fluid Engineering, Energy Institute, Technical University of Brno. In practice, prediction of pump performance can be approximated using the empirical formulas based on hydrodynamic similarity, i.e. specific speed. In this case of very low value of specific speed the theoretical assumptions often fail. That is why numerical modelling of the complex flow in the pump can be used to predict the performance of the designed pump before the construction of the prototype is realized.

2 NUMERICAL MODELLING OF THE FLOW IN CENTRIFUGAL PUMP

Viscous fluid flow can be described by partial differential equations that must be solved by numerical methods. Using this approach, it is possible to solve in an appropriate manner the 3D Navier-Stokes equations in complex geometries.

2.1 Definition of the computational geometry

At the beginning of the numerical modelling a 3-dimensional geometric model of the pump components must be created which describes the calculation domain by coordinates. This calculation domain is then subdivided into a large number of cells, i.e. a grid is generated, the quality of which is essential for the reliable numerical solution. Large number of cells brings better accuracy but also higher demands on computational time. In this work two different geometries were used. The modelled assembly of hydraulic components (inlet part, impeller and volute) was the same, but the impeller casing was included in the geometry only in second case. Both geometries are presented in Fig.1.



a) geometry without impeller casing b) geometry with impeller casing

Fig. 1 Definition of the pump geometry.

As can be seen, the second geometry marked by b) contains also the impeller sidewall gaps, including the clearances between impeller and casing.

The simplified geometry in Fig.1a) was divided into three parts (volumes) separated by „*interface*“, two of them stationary and one rotating. The complex geometry Fig.1b) was divided into six volumes (inlet part, impeller, volute with impeller sidewall gaps, clearances between impeller and casing on the front and rear side and the space behind the rear shroud under the annular seal. In the first case the geometry consisted of 1700000 of cells, in the second case the number of cells was about 4300000.

2.2 Numerical modelling

FLUENT release 6.3.26 [1] was applied for numerical modelling of the flow through the pump. The incompressible, time dependent flow (and steady flow to compare) was modeled in complex 3D geometry. The problem involves multiple moving parts as well as stationary surfaces. In Fluent, two approaches can be applied for the modelling of such cases:

- Multiple Rotating Reference Frames
  - *Multiple Reference Frame model (*MRF*)*
  - *Mixing Plane Model (*MPM*)*
- Sliding Mesh Model (SMM)

Both the MRF and MPM approaches are steady-state approximations, and differ primarily in the manner in which conditions at the interfaces are treated. In this case results are given for one position of blades of the impeller against volute. Simulations take less of a time but dynamic effects of flow in this approach are neglected.

The “*Sliding mesh*” model (SMM approach) is unsteady due to the motion of the mesh with time. This approach was applied in most cases. Data sampling for time statistics was applied which enable to compute the time average (mean) of the instantaneous values and root-mean-squares of the fluctuating values sampled during the calculation. The time step was constant during simulation with value Δt = 0.001s. Turbulent model k-ω SST was applied that is recommended for the flow in rotating geometries [3]. This model employs the k-ε model in the core flow, the k-ω model near wall and it modifies the relation for the eddy viscosity.

Using numerical methods (mostly finite volume method), the partial differential equations are converted into algebraic equations and solved by different algorithms which differ in convergence, accuracy and time of computation. The convergence of the steady solution can be observed by evaluation of residuals and also by the monitoring the value of the outlet pressure. Unsteady calculation enables to account for the rotor-stator interaction but it requires sufficient number of the time steps to eliminate initial oscillations of the flow and to establish the flow which exhibits periodical behaviour [1].

Definition of the pump head in case of steady solution is based on the formula

$$H = \frac{p_{\text{out}} - p_{\text{in}}}{\rho \cdot g} + \frac{v_{\text{out}}^2 - v_{\text{in}}^2}{2 \cdot g} + \left(y_2 - y_1\right) \tag{2}$$

and for time dependent solution we can define

$$H = \frac{\sum_{i=1}^{N}\left(p_{\text{out}_i} - p_{\text{in}_i}\right)}{N \cdot \rho \cdot g} + \frac{v_{\text{out}}^2 - v_{\text{in}}^2}{2 \cdot g} + \left(y_2 - y_1\right) \tag{3}$$

where:

p_{out}, p_{in} – are the outlet and inlet pressure values [Pa],
v_{out}, v_{in} – are the outlet and inlet velocity [m·s^{-1}],
y_2, y_1 – potential heads $[\text{m}]$,
ρ – fluid density [kg·m^{-3}],
g – gravity acceleration [m·s^{-2}].

During the pump performance hydraulic forces and moments are generated which act on the rotor. In case of steady incompressible flow calculation, the momentum on the pump rotor can be evaluated after reaching convergence according to the well known formula:

$$P = M \cdot \omega, \tag{4}$$

where:

P – power input [W],
ω – angular velocity [s^{-1}],
M – momentum on the rotor [N·m].

In case of unsteady solution, the *moment coefficient* C_M is saved at each time step. Momentum on the rotor can be calculated according to (5):

$$M_i = \frac{1}{2} \cdot C_{Mi} \cdot \rho \cdot S \cdot v^2 \cdot L, \tag{5}$$

where:

C_M – moment coefficient [1],
ρ – reference density [kg·m^{-3}],
S – reference area [m^2],
v – reference velocity [m·s^{-1}],
L – reference length [m].

Using the mean value of the moment, we can derive the power consumption P:

$$P = \frac{\sum_{i=1}^{N} M_i}{N} \cdot \omega \tag{6}$$

Hydraulic output P_h can be determined from the calculated parameters:

$$P_h = H \cdot \rho \cdot g \cdot Q \tag{7}$$

The ratio of both values is the pump efficiency:

$$\eta_c = \frac{P_h}{P} \tag{8}$$

where:

η_c – overall pump efficiency [1]
P – power input [W],
P_h – hydraulic output (useful power) [W].

The power supplied at the pump shaft is always bigger than the useful power. It is a result of losses in a pump which can be divided into mechanical, volumetric, hydraulic and disk friction losses. The accuracy of the losses determination depends on the applied computational geometry of the pump. In case of the simplified geometry (see Fig.1a) disk friction losses and leakage losses are not included into the model, the evaluated efficiency can be defined as hydraulic efficiency. The value of hydraulic efficiency can be approximated using the empirical formula according to Wislicen [2]:

$$\eta_h = \sqrt{\eta_c} - 0.03 \tag{9}$$

For comparison with the test data, hydraulic efficiency based on (9) was calculated from the measured efficiency curve and compared with the results of numerical simulation (see Fig. 5). In case of the complex geometry (see Fig.1b) also the annular seal leakage and disc friction losses are considered. The value of efficiency obtained by numerical modelling is close to that obtained by measurement (however we do not account for the mechanical losses in bearings and shaft seals).

2.3 Comparison with measured data

A pump performance curve was available from actual tests performed at Sigma Lutín. Relationship between capacity (volumetric flow rate) and head, power consumption and the efficiency was measured. Test data were used for comparison with the results obtained with CFD application. Comparison was done for the flow rate ranging from 0 to 10 l·s^{-1}. Results obtained with both pump geometries (Fig. 1) as well as with steady and unsteady solution are presented in Fig. 2.

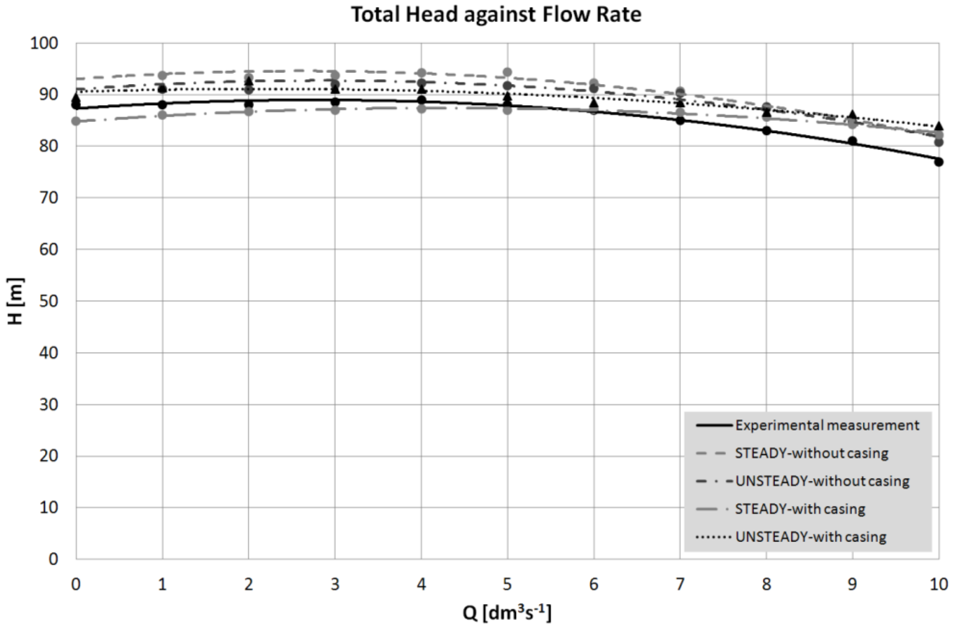


Fig. 2 *H-Q* curve.

It can be seen, that the Head-Capacity (*H-Q*) curve is slightly unstable, which can correspond to the very low value of specific speed. Most precise results were obtained with unsteady solution of the flow in complex pump geometry including the impeller casting. The deviation between the measured and modeled results is in the range 1.6-9%, the mean value is about 3.9%. Results obtained with simplified and full geometry are comparable, as the main contribution of losses belongs to hydraulic losses. The effect of the disc friction losses on the pump head is not very significant.

Power-Capacity (*P-Q*) curve (Fig. 3) exhibits the significance of pump geometry definition on the power input prediction. The simplified geometry ignores the disc friction, which leads to lower values of the moment on the rotor and so lower value of the power input. Results obtained for complex pump geometry are in good agreement with physical experiment.

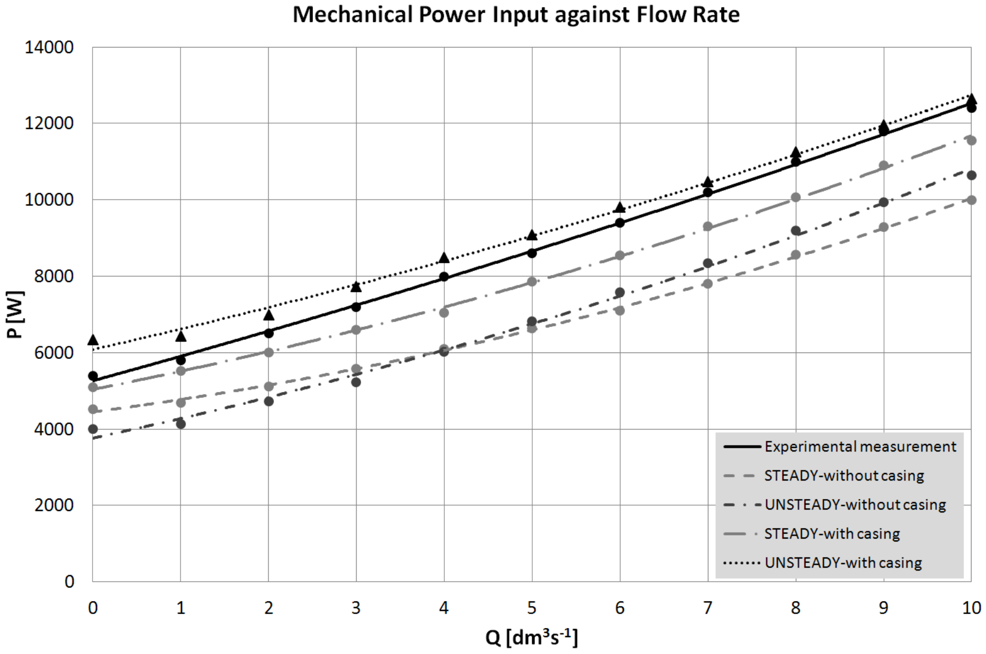


Fig. 3 *P*-*Q* curve.

Hydraulic power is plotted against the capacity in Fig. 4. The deviation between calculated and measured data grows with the increasing flow rate. This corresponds to the predicted *H*-*Q* curve. As the hydraulic power is generated inside the hydraulic parts, results obtained with simplified and complex geometries are similar.

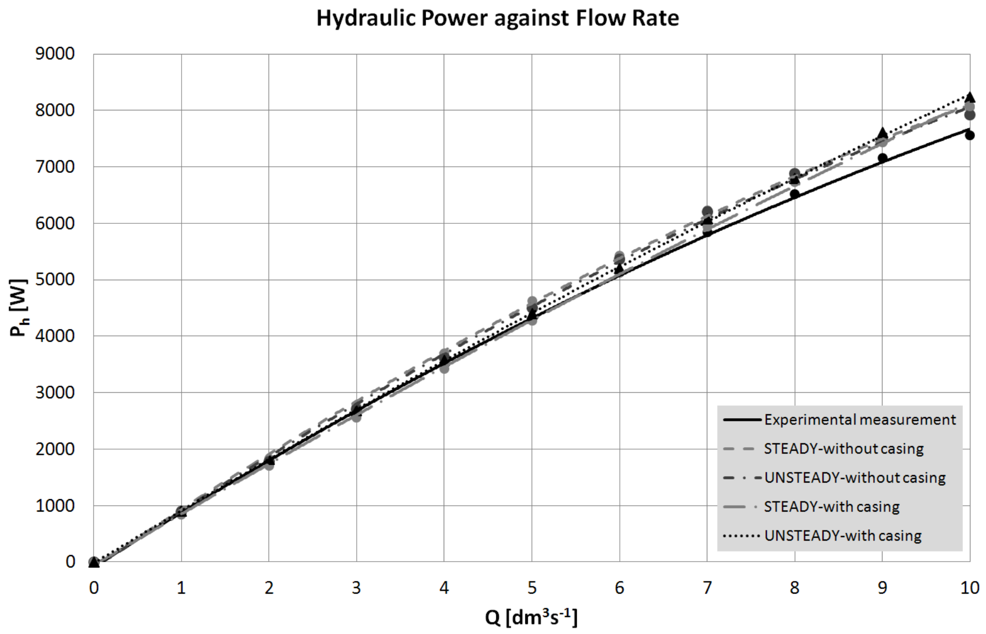


Fig. 4 P_h-*Q* curve.

The last of the evaluated parameters is the efficiency. As explained above, we must consider all phenomena contributing to the overall efficiency: hydraulic, mechanical, volumetric and disc fiction losses. Simplified geometry enables to model the 3D flow inside the hydraulic parts of the pump and so to determine the hydraulic losses. In Fig. 5 we can see comparison of calculated hydraulic efficiency with that approximated from measured data according to (9). A good agreement was reached in optimum flow rate. For lower values of the flow rate the hydraulic efficiency is underestimated and above the optimum flow rate it is overestimated, especially in case of the steady solution.

Full geometry containing also the impeller casing enables also to account for disc friction losses and leakage through annular sealing. Values of calculated efficiency are compared with measured values (Fig. 5). Steady solution leads to over prediction of efficiency. In this case the simulation is carried out only for one position of blades. Unsteady solution enables to evaluate mean values of variables using statistical approach.

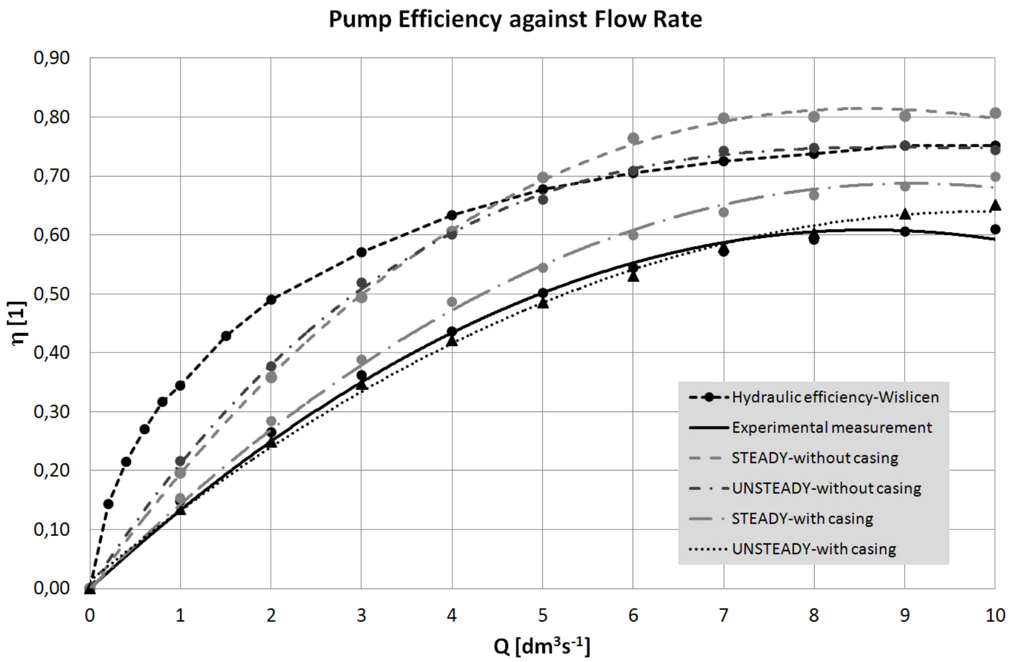


Fig. 5 η-Q curve.

It can be concluded that the most precise results were reached with unsteady solution of the flow in complex pump geometry.

3 CONCLUSIONS

The pump performance was investigated by means of numerical modelling using two different geometries – first one without impeller casing, the second one including also the impeller casing, i.e. the gaps on both sides of the impeller. Both stationary and time dependent solution was carried out and obtained values of head, power input and efficiency were compared with experimental measurement.

The reduction of geometry leads to lower number of grid cells and thus lower computational time. Steady solution enables to get fast prediction of the main pump parameters, which can be advantage in many practical studies. Efficiency prediction is limited, only hydraulic losses can be included.

To obtain more accurate prediction, complex geometry must be defined. This brings difficulties in grid generation, especially in the region of annular seals and gaps on both sides of the impeller. Number of cells grows rapidly. The solution requires long computational time, especially in case of unsteady solution. Overall efficiency can be predicted and analysis of dynamic behaviour of the flow through the pump can be achieved. Obtained results are in "good agreement" with measured data. The mean deviation between calculated and measured data is about 5%.

The correct loss prediction is a prerequisite of a pump component optimization. There are many factors that must be taken into account: grid quality, turbulence model, wall treatment, roughness. The applied modelling procedures bring some restrictions and that is why they need extensive validation with experimental data.

REFERENCES

[1] FLUENT 6.3.26 – *User´s guide*. Fluent Inc. 2003 [online]. Datum poslední revize 3.10.2006 [cit. 2008-12-08]. Dostupné z : < http://spc.vsb.cz/portal/cz/documentation/manual/doc.vsb.cz/Aplikacni%20software/FlueFl_6.3.26/pdf/ug/flug.pdf >.

[2] BLAHA, J.; BRADA, K. *Hydraulické stroje.* 1st ed. Praha: SNTL, 1992. 747 pp. ISBN 80-03-00665-1.

[3] GÜLICH, JOHANN, F. *Centrifugal Pumps*, Second Edition: Springer, 2010. 964 pp. ISBN 978-3-642-12823-3.

[4] TUZSON, J. *Centrifugal pump design* 2000. 320 pp. ISBN 0-471-36100-3.

The results presented in this paper were achieved within the specific research project SP2011/61 "*Zkoumání fyzikálních vlastností materiálů a jejich vlivů na dynamiku proudění*" solved at Faculty of Mechanical Engineering, VŠB – Technical University of Ostrava in 2011.