EPJ Web of Conferences **180**, 02043 (2018) https://doi.org/10.1051/epjconf/201818002043
EFM 2017

Dynamics of the nozzle valve with regard to the properties of the piping system

Roman Klas[1,*], *Vladimír* Habán[1], and *Pavel* Rudolf[1]

[1]Brno University of Technology, Faculty of Mechanical Engineering, Victor Kaplan Department of Fluids Engineering, Technická 2, 61669 Brno, Czech Republic

Abstract. It is obvious that the main function of the nozzle valve is to shut off the stream of fluid in the piping system. The response rate of the valve to the decreasing or reversing flow in the system will then depend on the valve properties and equally on the properties of the piping system. The interaction of these two elements is also important for the origin of pressure pulsations in the system. While the pressure pulsations were the cause for design of this particular valve it should be noted that the general design of the valve for any pipeline system is not possible. The valve cannot properly work under all circumstances and operating conditions. With respect to this, the dynamic properties of the valve will be assessed on the basis of the valve equation of motion and the pipeline model. An adequate response of the whole system can be obtained by combining both approaches. The valve equations of motion are also complemented by CFD simulations, which enable to capture the movement of the valve disc with respect to flow rate.

1 Introduction

The presented study follows the main simulation of the in-line check valve [1]. An attempt to predict the in-line valve poppet lift in dependence on the flow rate occurred in these simulations based on a one-dimensional model. Verification of the one-dimensional model was realized only partially through the 1DOF CFD simulations. This paper will complements the CFD simulations of the in-line check valve, but it will preferably focus on the nozzle check valve. The 1DOF CFD in-line check valve simulations will also serve to compare with 1DOF CFD by simulating the nozzle check valve. The fact that the designing of the in-line check valve and the nozzle check valve was underway within the same development of the check valves is the reason. Each of these valves was one of the concepts that had been part of a prototype search for a real device. Therefore, the comparison is actually expected. Another important factor is that experimental data are already available in the case of an in-line valve.

The other circumstance that will be considered is the fact that the dynamic testing of the valve can be realized in two directions. The first is the dynamic behavior of the check valve itself, i.e. the valve without the connected pipe system. Of course, it is necessary to note at this point that the obtained data obtained can hardly match the operational properties of the valve in the real pipeline. Such an approach is somewhat more favorable for CFD simulation, because CFD simulations in generally characterize considerable computational requirements. It would be inconvenient in common conditions to simulate the whole piping system through CFD and still consider a dynamic network, even in a small computational domain. Of course, it is also possible to combine CFD simulations and one-dimensional models representing the piping system with all its elements. However, if there was a one-dimensional model in the form of the valve equation of motion, it would lose that justification and the comparison could be made directly. The use of CFD simulations is particularly useful in the determination of those valve parameters that are elusive by the experimental testing. These data can then serve to accuracy a one-dimensional model. It is also necessary to say that the valve disc movement prediction through CFD brings problems of greater or lesser loss of control over the quality of the computational mesh, which varies depending on the remeshing or smoothing mode in the individual time steps. This has an impact on the choice of wall functions and the turbulent model in CFD simulations [2, 3].

The second option is to create a one-dimensional model of the pipeline system supplemented with a check valve model. Verification of the obtained results is possible from experimental data.

2 CFD simulations and description of the check valves and the piping system

All CFD simulations were carried out by using the commercial code ANSYS ANSYS Fluent 17.2. The component that is part of this software was used to realize the disc movement in the valve axis (1DOF) by considering the dynamic computational mesh. The valve geometry and computational mesh were created in ANSYS Design Modeler and Meshing. The computational mesh was conceived as mapped except the

* Corresponding author: klas@fme.vutbr.cz

nearness of the disc and the valve seat, i.e. the area of the dynamic mesh.

The computational mesh was also improved with respect to determine the exact magnitudes of the forces acting on the disc. Therefore, in incremental steps, the number of mesh cells was increased until the force magnitudes in the corresponding flow modes remained unchanged. However, it is necessary to mention at this point a certain complication in the selection of wall functions and in their correct setting with respect to the wall y + parameter. The right size of the wall y + parameter was the most conflicting with reaching the relevant force magnitudes. 2D CFD simulations were used as the basic mode of calculation, which at least in case of the in-line check valve proved to be interchangeable with 3D simulations [1]. In addition, it is to be noted that due to the number of computational cells of the three-dimensional computational mesh, the simulation of the valve's disc movement while maintaining the correct conditions is time-consuming. This is also why the aim of the paper is to minimize computational requirements on prediction of valve disc motion. Besides the above mentioned, the flow field parameters necessary to create a one-dimensional model were obtained by CFD simulations. The parameters of CFD simulations are presented in Table 1.

Table 1. Description of CFD numerical model.

	2D
Pipeline diameter	DN100
Number of computational cells	~ 400000
Turbulence model and near wall modeling	realizable k – ε non equilibrium wall function
Boundary conditions	Inlet: velocity inlet Outlet: pressure outlet
Calculation mode	unsteady, incompressible flow, 1DOF

The nozzle check valve is shown in Fig. 1 and the in-line valve on Fig. 2.

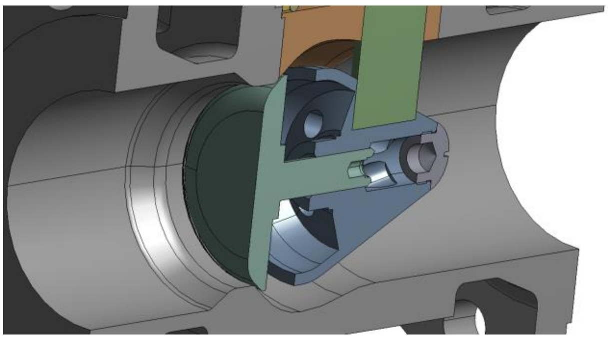

Fig. 1. Nozzle check valve.

The piping system (Fig. 3) used in the one-dimensional model should be as close as possible to the real valve testing circuit. However, due to the character of the circuit, its model is in principle universal.

Circuit consists of the connecting pipes, pressurized vessel, which is connected to vacuum pump (for cavitation testing), the valve itself and hydrodynamic pump for the case of turbine and valve testing. The hydrodynamic pump is driven by dynamometer, which also measures pump torque and speed. Flow rate is measured using induction flowmeter, pressure difference between valve inlet and outlet is assessed using pressure sensors. Measurement procedure and data evaluation is according to valid standard ČSN EN ISO 9906.

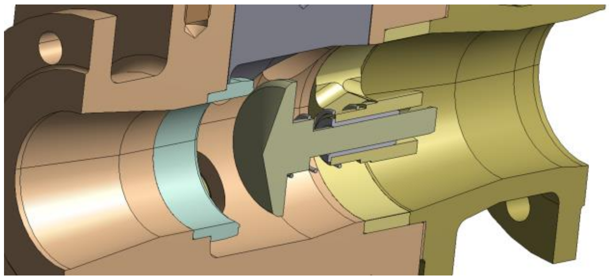

Fig. 2. In-line check valve.

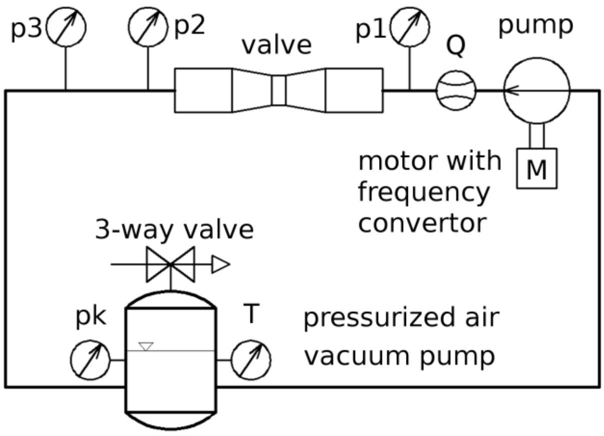

Fig. 3. The pipeline and the testing circuit diagram

The Table 2 lists the quantities and their markings used in the paper and in the following equations.

Table 2. List of symbols.

A_Y, B_Y, C_Y	coefficients of specific energy
b_t	damping
J	moment of inertia
k, k_S	gradient of the straight line, stiffness of the spring
K	fin and guide bushing
l_a	Sa surface center of gravity coordinates
l_b	Sb surface center of gravity coordinates
m_j, n_i, n_j	normal vectors
M_s	friction moment
n	pump speed
p, pa, pb	static pressure and static pressure on the inlet and outlet surfaces
P	solid wall of the valve housing

P_D, P_o	dissipated power, dissipated power at the shut-off point
Q, Q_m	volumetric and mass flow rate
R_{ab}	hydraulic resistance of the valve between the surfaces Sa and Sb
Sa, Sb, S	areas of the inlet and outlet surfaces of the valve
v	absolute fluid velocity
V_Γ	volume of the valve disc
x_0, x_3	minimum spring preload, disc position
Y	specific energy of the pump
Γ	moving wall of the disc
Π	strain rate tensor
η	pump efficiency
ρ_F, ρ_Γ	density of the liquid and the valve disc
ω	angular speed
1DOF, 1AM	one degree of freedom, one dimensional numerical model

From the Navier-Stokes equation (1), from the continuity equation (2) and from boundary and initial conditions we can obtain the fluid force acting on the valve disc.

$$\rho_F\left(\frac{\partial v_i}{\partial t}+\frac{\partial v_i}{\partial x_j}v_j\right)=\rho_F g_i-\frac{\partial p}{\partial x_i}+\frac{\partial \Pi_{ji}}{\partial x_j} \qquad (1)$$

$$\frac{\partial v_j}{\partial x_j}=0 \qquad (2)$$

The equation (3) defining the force magnitude is, of course, dependent on the choice of boundaries and boundary conditions. It seems logical to identify the position of the boundary conditions with the valve boundaries in particular as regards the location of its inlet and outlet parts. However, the problem of determining some of the members in the equation (3) increases. It is possible to accept the assumption of rotational symmetry and, above all, to simplify the integration of the static pressure on the surface **P**, but problems with the expression of strain rate tensor $\boldsymbol{\Pi_{ij}}$ persist. Any possible neglect of these members in the equation, of course, will reduce the accuracy of the obtained results.

The second option is to move the boundaries of the area to the limit positions of the valve disc. The above mentioned difficulties would persist, though the inaccuracy of the one-dimensional model would be less, for example, by neglecting shear stresses. Of course, we plan to use CFD methods to determine them in both cases. However, the simplest description of the physical phenomenon should be the objective of the analytical model. Therefore, it would be more acceptable to keep the boundaries on the boundary surfaces of the valve and to evaluate the valve losses as if they were only generated by the valve disc. Thus, it is necessary to determine only the forces acting on the disc, not on the remaining part of the valve, although it is expedient to evaluate these forces from equation (3). This gives us an idea of the synergy of the valve disc and the housing with regard to the designing of the valve interior.

$$\begin{aligned}F_F &= \rho_F \ddot{x}_i V_\Gamma - \rho_F\left(l_a+l_b\right)\frac{\partial Q}{\partial t}+p_a S_a - p_b S_b + \\ &\iint\limits_{S_a\cup S_b\cup P\cup K}\Pi_{ij}n_j dS+\iint\limits_{S_a}\rho_F v_i v_j m_j dS-\iint\limits_{S_b}\rho_F v_i v_j n_j dS \\ &-\iint\limits_{P\cup K} p\, n_i dS\end{aligned} \qquad (3)$$

The fluid force acting on the disc can be simplified in equation (4) by defining the boundary conditions according to Fig. 4.

$$F_F=\rho_F \ddot{x}_i V_\Gamma-\rho_F\left(l_a+l_b\right)\frac{\partial Q}{\partial t}+R_{ab}Q^2 S \qquad (4)$$

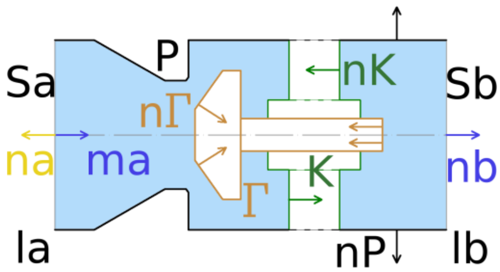

Fig. 4. The markings of definition areas.

In most common cases, the hydrodynamic pump is a feeding unit in a real pipeline system, except the gravity water system. The design of this pump may be axial, semi-axial or radial depending on the impeller type. We will focus on further considerations regarding the different properties of these pumps to systems with hydrodynamic pumps radial and peripherally also semi-axial [4].

Pump and fluid in piping are a source of inertial effects in the pipeline, which has a significant impact on the check valve dynamics and its closing times. It can be expected that the hydrodynamic pump will have favorably effects in terms of the danger of water hammer, and vice versa, it may have a negative effect on the long closing times of the valve. It is also important differentiate low and high pressure systems with regard to simulation of compressible media [5].

It is necessary to consider the operation of the system in order to assess the effect of the pump. We can expect a piping system failure and a subsequent decelerating of impeller rotation, which is the most frequent consequence of a power outage. In addition, we can consider a pump failure consisting of an immediate stop of the impeller, or pump speed control and its slow shutdown [6]. The last possibility will not be further discussed as it offers full control over pipeline dynamics.

Of course, it is necessary to complement the previous considerations with the relevant equations that include the effect of the hydrodynamic pump, the fluid in the pipeline, the elements of the piping system and the check valve itself. The described system will be simulated until a possible water hammer can occur and the incompressible liquid will be further considered. Fig. 3 in combination with Tab. 2. includes the parameters of the fluid and the piping system.

$$\omega J \frac{d\omega}{dt} + P_D = 0 \tag{5}$$

The equation (5) is the basic relationship describing the balance between the power consumption of the machine and its inertial effects. Determining the power consumption $\boldsymbol{P_D}$, which in terms of dissipated power in the pump interior except the spiral case, is the main difficulty of this equation. There are basically two main possibilities in this respect. The first is a detailed analysis of a centrifugal pump and a subsequent determination of the losses in the main parts of the pump interior. The literature offers several basic resources in this respect, among which the most well-known are [7-13]. In [13] is possible find a description of the hydraulic losses in the spiral case.

Operation of centrifugal pumps is usually assumed at constant speed. However, even at constant impeller speeds, parameters such as specific energy and flow rate may change. The pump power is, of course, changing as a result of these parameters. Changing pump speeds, such as machine downtime until it stops completely, complicates this situation. The power of the pumping unit is derived from the inertia effects of the engine and pump assembly. This power consumption then decides the time at which the impeller stops. It is necessary to consider that the impeller and fluid flow stopping time will most likely not match. It is obvious that the remaining parts of the piping system, including the check valve, will affect the fluid flow. In the event of an undesired backflow through the valve, which is a negative factor for the possible high peak pressure peak in the pipeline, consideration should also be given to possible backflow of the fluid to the impeller. The impeller can rotate in the pump mode or even operate at a low speed like a turbine for a short time. However, the turbine modes of operation will no longer be the focus of our research. The pump is therefore analysed primarily in terms of its impact on the check valve dynamics.

Affinity laws (6) will be used to overcome the problem of determining change of power with a change in speed.

$$\frac{P}{\overline{P}} = \frac{n(t)^3}{\overline{n}^3},\ \frac{Q(t)}{\overline{Q(t)}} = \frac{n(t)}{\overline{n}},\ \frac{Y(Q)}{\overline{Y(Q)}} = \frac{n(t)^2}{\overline{n}^2} \tag{6}$$

Further, it is assumed that the hydraulic and volumetric efficiency is constant, depending on pump speed (7). The mechanical efficiency is then considered for the corresponding pump speed at the mode of its operation.

$$\begin{aligned} \eta_{h,o} = \overline{\eta_{h,o}} = \frac{\overline{\rho Q(t) Y(Q)}}{\overline{P_o + k\, Q(t) - M_s \omega}} = \\ \frac{\rho Q(t) Y(Q)}{P_o + k\, Q(t) - M_s \omega} \end{aligned} \tag{7}$$

It is also possible to consider approximately linear load dependence on pump flow rate (8) for radial flow pumps.

$$P_D = P_o + k\, Q(t) \tag{8}$$

The equation (8) becomes the equation (9), which can be used to determine the speed of the impeller. Speed will represent basic information about the mode of pump operation.

$$\begin{aligned} &(2\pi)^2 J\, n(t) \frac{d\, n(t)}{dt} + \\ &\frac{n(t)^3}{\overline{n}^3} \left(\overline{P_o} + \overline{k}\, \overline{n} \frac{Q(t)}{n(t)} - \overline{M_s}\, 2\pi\, \overline{n} \right) = -2\pi\, M_s\, n(t) \end{aligned} \tag{9}$$

Equation (8) can also be used when power consumption is not a linear dependence of pump flow rate. Just consider the variable coefficient $\boldsymbol{k}$. The ability to determine pump consumption and efficiency $\boldsymbol{\eta_{h,o}}$ is also offered. In this way, it would be difficult to determine the pump consumption at the shut-off point, see (8).

In order to be relatively simple to descrive the pump's specific energy, it is possible to approximate it by the corresponding curve (10). It would be possible to use a more accurate substitution, but this would not mean a greater contribution to the problem analysis. The dependence of the specific energy on the speed of the impeller corresponds again to the affine laws (6). A typical dependence of specific energy can be described at a constant pump speed according to (10). More can be found in [6].

$$\overline{Y(Q)} = A_Y - B_Y \overline{Q(t)}^2 - C_Y \overline{Q(t)} \tag{10}$$

Similarly, the specific energy Y(Q) corresponds to the equation (11) at variable speeds.

$$Y(Q) = A_Y \left(\frac{n(t)}{\overline{n}} \right)^2 - B_Y Q(t)^2 - C_Y \frac{n(t)}{\overline{n}} Q(t) \tag{11}$$

The equation (12) describes the movement of the valve disc.

$$\begin{aligned} &\rho_\Gamma V_\Gamma \ddot{x}_i + b_t \dot{x}_i + k_S (x_0 + x_i) = \\ &\rho_F \ddot{x}_i V_\Gamma - \rho_F (l_a + l_b) \frac{\partial Q}{\partial t} + R_{ab} Q^2 S \end{aligned} \tag{12}$$

The following equation (13) symbolically describes the pipeline in which the valve was connected.

$$\frac{1}{S} \sum_j l_j \frac{dQ}{dt} + \frac{Q|Q|}{2S^2} \left(\sum_i \xi_i + \sum_j \lambda_j \frac{l_j}{d_j} \right) = \pm Y(Q) \tag{13}$$

It is advisable to consider improvement the one-dimensional circuit model by simulating unsteady friction as the flow is unsteady. This issue is more clearly explained e.g. in [14]. However, unsteady friction in the pipeline model had almost no effect on the obtained results, and therefore unsteady friction was further neglected.

3 Numerical simulations

As already mentioned, valve testing involves two approaches and combines CFD methods and one-dimensional models of the pipeline system and the valve itself. However, it would not be possible to create a one-dimensional model without CFD simulations. The meaningfulness of a one-dimensional model, of course, lies in smaller computational demands and in the analysis of parameters influencing the movement of the valve plug. Of course, the meaningfulness of a one-dimensional model is based on lesser computational requirements, as well as in the analysis of parameters influencing the movement of the valve disc.

Several results will be presented. 1DOF CFD simulation of the axial valve itself without the pipe system Fig. 5, which is not finished in [1], will be completed first. The change of flow rate Q is considered linear. CFD simulation is also complemented by experimental data.

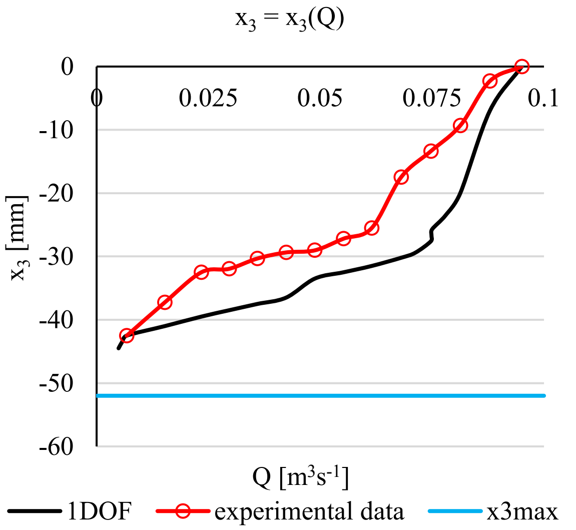

Fig. 5. 1DOF CFD In-line check valve simulation and comparison with experimental data, Ts = 1.5s.

It was possible to simulate about 90% of the maximum opening of the valve, considering the real closing time of the valve Ts = 1.5s. However, the valve is usually only open from 20 to 30% in real operation. The 1DOF CFD simulation was therefore realized for this opening, Fig. 6. The opening corresponds to 30% of its maximum opening. This simulation is also complemented by a one-dimensional model (1AM) following the equation of motion (12).

The agreement of both simulations is basically good, but their comparison does not exist near the valve seat. The computational stability of 1DOF CFD was week, as in the previous case. Similarly, data 1DOF CFD and 1AM simulation of the nozzle check valve itself are shown in Fig. 7. The closing time Ts = 1.5s was set for the second check valve so that both valves could be compared.

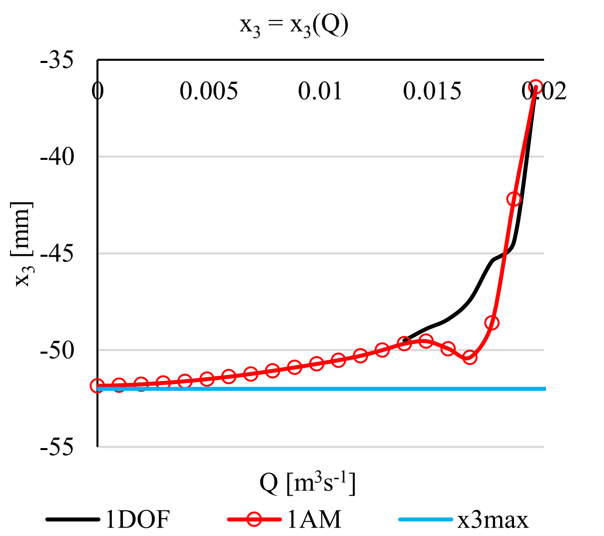

Fig. 6. 1DOF CFD and 1AM in-line valve simulations, Ts = 0.5s and initial opening 30%.

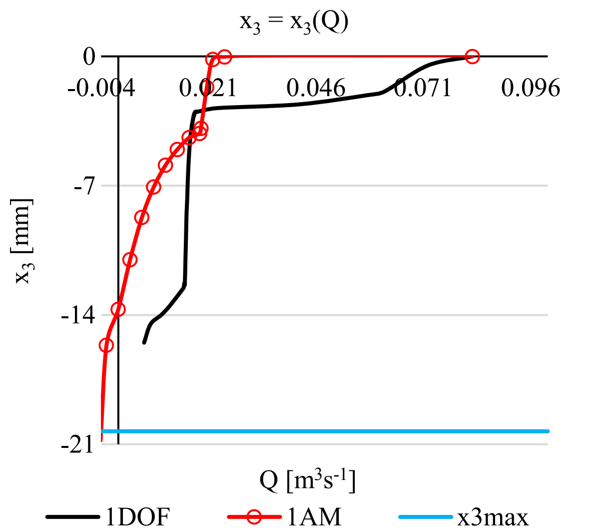

Fig. 7. 1DOF CFD and 1AM simulations of the disc position of the nozzle check valve, Ts = 1.5s.

The biggest difference in Fig. 7 is visible at maximum valve opening. The length of time the disc does not move differs slightly. Rapid closing is characteristic in both cases. The fact that full valve closure occurs with a small backflow through the valve could be unfavorable. The backflow may be undesirable in relation to the pressure peak size if a water hammer occurs. It is shown with respect to the time delay at the beginning of the valve closing that the exact determination of the hydraulic resistance of $\boldsymbol{R_{ab}}$ is decisive in the times approaching t = 0s. Even a very small change in $\boldsymbol{R_{ab}}$ can be very significant in the length of time the valve disc remains in its initial position.

The other simulations already concern the movement of the disc in the check valve that is inserted into the piping systém. 1AM in-line and nozzle check valve simulations in the piping system are shown in the following Figs. 8 and 9.

Note: The literature [1] contains only 1AM simulation of the in-line check valve itself.

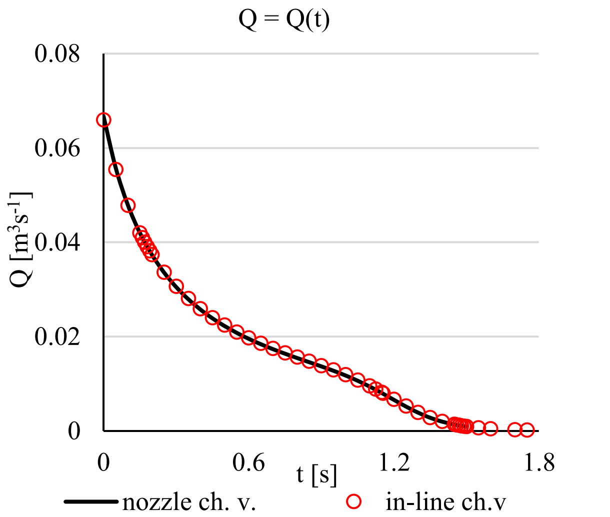

Fig. 8. 1AM simulations of the in-line and nozzle check valve in the piping system, the flow rate.

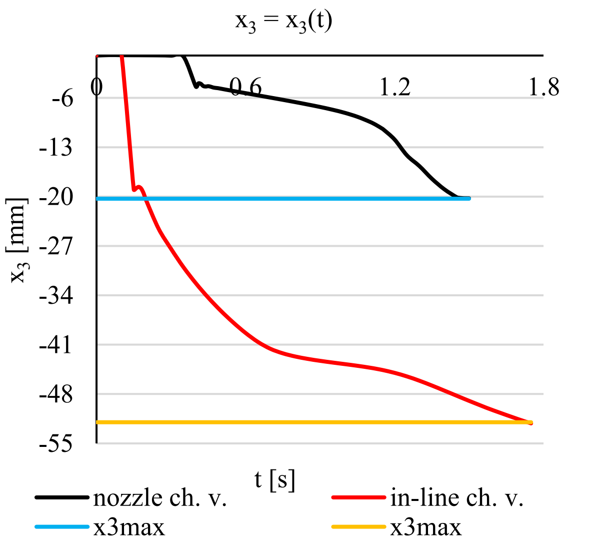

Fig. 9. 1AM simulations of the in-line and nozzle check valve in the piping system, the disc and the poppet positons.

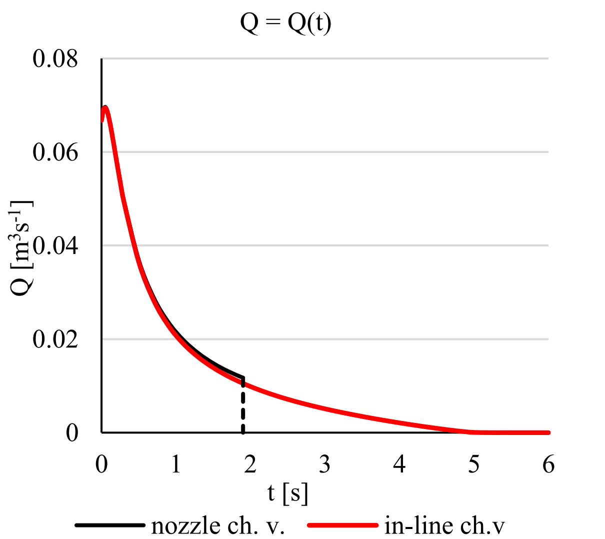

Fig. 10. 1AM in-line valve and nozzle check valve simulations in the piping system in case of a sudden failure of centrifugal pump during a power outage, the flow rate.

The effect of inertia in the system and, primarily, other hydraulic resistances in the circuit appears in the case of both valves (Figs. 8 and 9). Also, the closure curves of both valves will be plotted in addition to the above mentioned characteristics in case of a sudden failure of centrifugal pump during a power outage, Figs. 10 and 11.

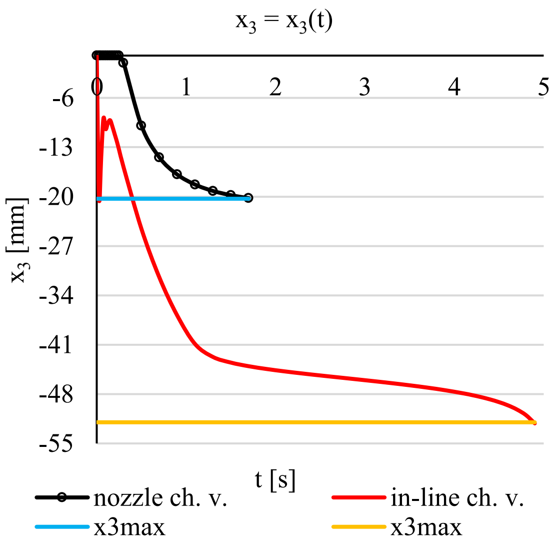

Fig. 11. 1AM in-line valve and nozzle check valve simulations in the piping system in case of a sudden failure of centrifugal pump during a power outage, the disc and the poppet positons.

It is evident from Figs. 10 and 11 that due to the inertia moment of the pump impeller and the inertia effects of the fluid in the pipeline, the closing times of both valves are relatively long. The influence of the hydrodynamic pump or its characteristics also appears at times close to t = 0s. There may be a slight increase in the flow rate for a very short time period and thus the movement of the poppet in the direction of flow. This movement is eliminated in the case of the nozzle check valve due to its maximum operational opening. But it is more important that the in-line valve closes in all cases without any backflow. Also, there is no backflow in the case of a nozzle check valve, unlike the simulation of the nozzle check valve itself. On the contrary, the nozzle check valve closes suddenly in low flows. The amount of this flow is determined by the moment of inertia of a pump impeller, the friction torque on the pump shaft and the inertia effects of the fluid. Also, the magnitude of the spring preload, which is of course given by the opening pressure, has a similar effect. The moment of inertia of the impeller is related to the appropriate choice of pump parameters and optimal operation. This can affect the size of the moment of inertia, which may be small or unnecessarily large depending on the pump. Not considering a flywheel. Also, the efficiency of the pump will affect the minimum flow rate at which the valve closes. The dependence of efficiency on flow rate is, of course, related to the charakteristic of power consumption but also to the stability of the head – flow curve. The disk friction losses in the pump have a stabilizing effect with respect to the head – flow curve, and just the magnitude of the disk friction and mechanical and spiral case losses at the shut-off point is determined by the gradient of efficiency curve at the shut-off point and thus the stability of the head-flow curve.

The time dependencies of the flow rate and the valve position for different sizes of the moment of inertia ***J*** and for different efficiency curves of the pump (Fig. 12) are shown in Figs. 13 - 16. The pump parameters at the operating point remain the same. The difference of the efficiency curves is based only on the different magnitude of the dissipated power ***Po*** at the shut-off point. This consideration is related to the upcoming detailed analysis of losses in the centrifugal pump and to the effect of these losses on the dynamic behavior of the valve. The moment of inertia and the dissipated power in the figures are related to their nominal values.

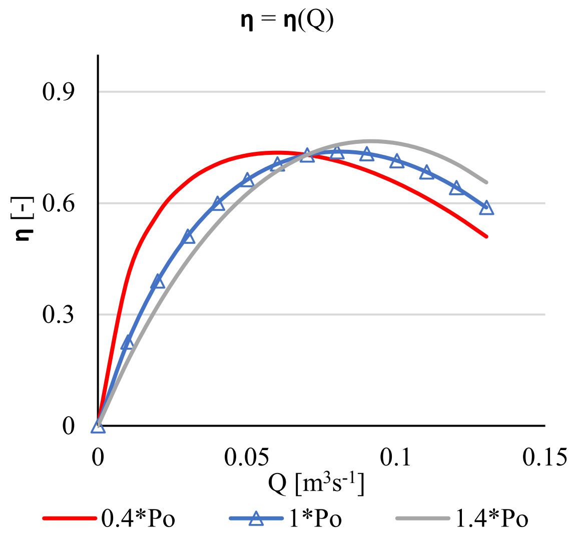

Fig. 12. The pump efficiency curves.

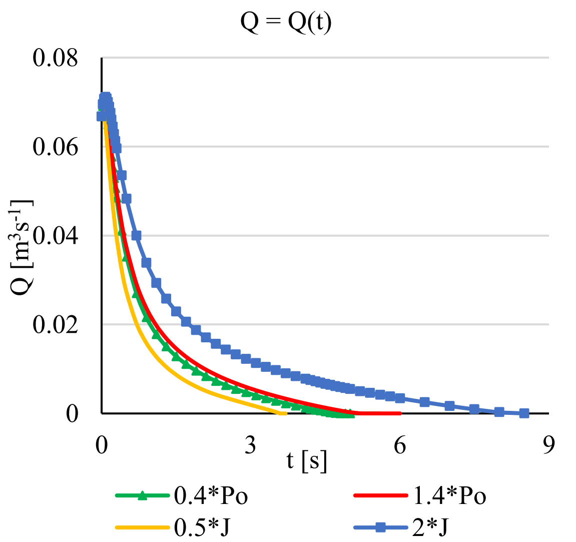

Fig. 13. 1DOF CFD and 1AM simulations of the in-line check valve, the flow rate.

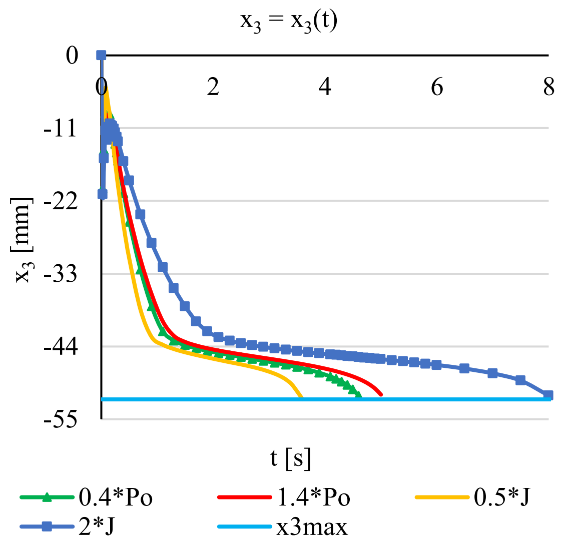

Fig. 14. 1DOF CFD and 1AM simulations of the in-line check valve, the positions of the poppet.

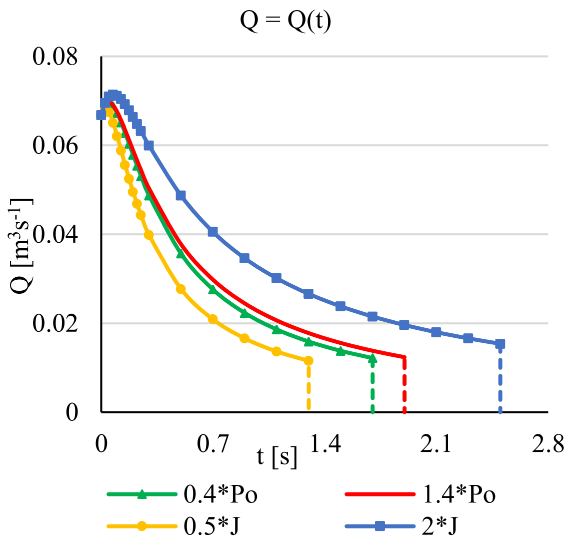

Fig. 15. 1DOF CFD and 1AM simulations of the nozzle check valve, the flow rate.

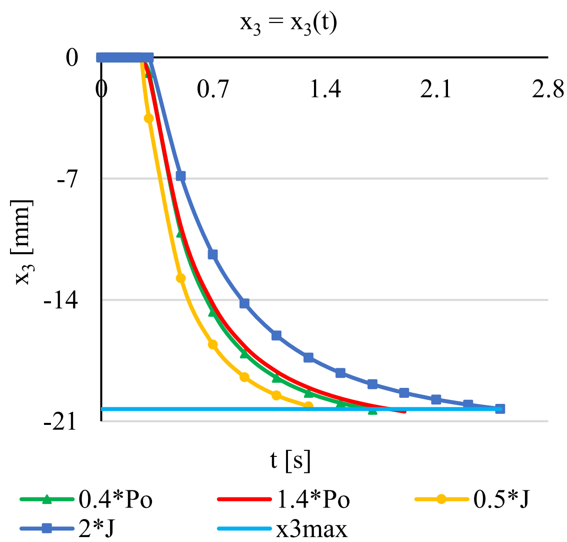

Fig. 16. 1DOF CFD and 1AM simulations of the nozzle check valve, the positions of the disc.

4 Conclusions

Finding a one-dimensional check valve model in the piping system should provide a basic analysis of the dynamic properties of the nozzle check valve. Since this is a follow-up study, the previous design (i.e. the in-line valve) was also reassessed in this respect.

The possibilities of positioning the boundary surfaces defining the valve were dealt with in the first phase. This step is important with respect to the definition of the magnitude of force acting on the disc by a one-dimensional model. However, the boundary surfaces of the valve can not be moved considerably due to the attempt to create a one-dimensional model of the check valve itself supplemented by a piping system. At the same time, it is necessary to use CFD methods to describe the variable velocity and pressure fields at the boundary surfaces of the valve.

The centrifugal pump as a typical part of the hydraulic system was simulated cumulatively by the power consumption, head - flow curve approximation, and the pump affinity laws. Also, the volume efficiency of the pump, even with the constant hydraulic resistance of the sealing rings, has been considered. In the future, the pump will be divided according to its basic parts and the losses determined in individual parts of the pump. However, this procedure does not have to increase the accuracy of the one-dimensional model because the pump power consumption used in the 1AM model can be determined quite accurately, leaving only defining spiral case losses The advantage will be to specify the spontaneous rotation of the impeller depending on the flow, which is essential with respect to the prediction of the water hammer in the pipeline. However, this study did not examine the water hammer.

A comparison of experimental data of in-line check valve and CFD simulation results in a relatively good agreement. The prediction of the poppet movement near the valve seat differs most. The practically non-existent backflow through the valve is important in assessing valve behavior. The nearness of the valve seat appeared problematic for 1DOF CFD simulations for both valves. An experimental testing in the form of the disc's position dependence on the flow was not realised for the nozzle check valve. The study of the valve was therefore realised with 1AM and 1DOF CFD simulations. The steeper gradient of closure curve of the valve depending on time, and the existence of backflow results from these simulations. However, the backflow was detected only in the simulation of the nozzle check valve itself without the connected pipeline. During the creation of the one-dimensional model, a considerable sensitivity of the hydraulic resistance was shown in extremely short times after the simulation started, when the disc was moved from the fully open position to the closed position. The magnitude of this hydraulic resistance has a significant effect on the time delay in which the valve remains in its fully open position. The dynamic behaviour of the pump with respect to full valve closure is shown in the pipeline system model with centrifugal pump. This may result in a rapid valve closure depending on inertia and mechanical friction in the pump.

It is also important to mention the necessity of experimental testing, which will allow the validation of the 1AM model and its further improvement.

This work has been supported by Technology Agency of the Czech Republic under the project Innovative research of check valves for extreme operating conditions in energetics TH01011352.

References

1. R. Klas, V. Habán, P. Rudolf, EPJ Web of Conferences, **143** 02051 (2017)
2. D. C. Wilcox, *Turbulence Modeling for CFD* (DCW Industries, Incorporated, 1994)
3. V. C. Patel, W. Rodi, G. Scheuerer, *Turbulence models for near-wall and low Reynolds numbers flows: A review*, AIAA J. **23(9)**, pp. 1308-1319 (1985)
4. J. F. Gülich, *Centrifugal Pumps* (Heidelberg Springer-Verlag, 2010)
5. P. Beierer, *Experimental and Numerical Analysis of the Hydraulic Cirquit of a High Pressure Common Rail Diesel Fuel Injection System* (Tampere, Tampere University of Technology, Publication 715, 2007)
6. J. Záruba, *Water Hammer in Pipe – Line Systems* (Academia Praha, 1993)
7. J. F. Gülich, World Pumps **1999(395)**, pp. 30 – 34 (1999)
8. J. F. Gülich, World Pumps **1999(396)**, pp. 39 – 42 (1999)
9. J. F. Gülich, Forsch Ingenieurwes **68**, pp. 87 – 95 (2003)
10. A. Nemdili, D. H. Hellmann, Forsch Ingenieurwes **71** pp. 59 -67 (2007)
11. A. Ladouani, A. Nemdili, Forsch Ingenieurwes **77 (3-4)** pp. 59 – 69 (2013)
12. M. H. Shojaeefard, B. Salimian Rizi, A.Khalkhali, M. Tahani, JAFM **8(4)**, pp. 673- 681 (2015)
13. A. K. Omar, A. Khaldi, A. Ladouani, Australian J. of Mech. Eng. pp. 1 – 12 (2016)
14. B. Brunone, U. M. Golia, M. Greco, J. of Hydr. Eng. **121(12)** pp. 906 – 912 (1995)