Experimental Measurements and Mathematical Modeling of Static and Dynamic Characteristics of Water Flow in a Long Pipe

J Jablonska¹, M Kozubkova ¹

¹Departament of Hydromechanics and Hydraulic Equipment, VSB Technical University of Ostrava, 17. listopadu 15/2172, CZ- 70830 Ostrava

E-mail: jana.jablonska@vsb.cz

Abstract. Static and dynamic characteristics of flow in technical practice are very important and serious problem and can be solved by experimental measurement or mathematical modeling. Unsteady flow presents time changes of the flow and water hammer can be an example of this phenomenon. Water hammer is caused by rapid changes in the water flow by means the closure or opening of the control valve. The authors deal with by hydraulic hammer at the multiphase flow (water and air), its one-dimensional modeling (Matlab SimHydraulics) and modeling with the use of the finite volume method (Ansys Fluent) in article. The circuit elements are defined by static and dynamic characteristics. The results are verified with measurements. The article evaluates different approaches, their advantages, disadvantages and specifics in solving of water hammer.

1. Introduction

The dynamics of hydraulic systems is often accompanied by a hydraulic hammer, and it has been investigated experimentally and mathematically by a number of authors who have used in the past many partially simplified approaches. It has to be said that the quality of results always depended on obtaining high quality data regarding the physical properties of liquids, such as the modulus of elasticity and, of course, the viscosity and density of the flowing liquid. The article focuses on the possibilities of solution of the hydraulic hammer, the creation of more precise mathematical models and the presentation of the solution approach.

A typical example of a non-stationary isothermal flow of compressible fluid is a hydraulic hammer which is typical by the pulsations of pressure and noise. In experimental hydraulic hammer measurements, it is necessary to record the time series of pressures at various points of the circuit, respectively flow rate to the computer and to devote their evaluation.

Numerical simulation is used to examine the dynamic properties of the system. Computer processing and simulation of mathematical models to describe variables of fluid flow has a number of advantages. With the massive development of computational capacity, it is useful to use mathematical modeling in designing and constructing of hydraulic devices in many engineering applications. However, some caution should be exercised when using the simulation results, as the issue of turbulent flow is not yet fully resolved. Therefore, the simulation can be used as an exploration tool,
which must be further verified and tested. A great emphasis must be put on the flow prediction as a result of the simulation.

For the experimental investigation of the hydraulic hammer, the hydraulic circuit was created where the water was as the flowing medium. By quick closing or opening the valve at the end of the long pipeline, a hydraulic hammer was simulated. Because this is a dynamic task, pressure sensors were used to record to a computer [1]. The results were used as boundary conditions for mathematical simulations and subsequently for their verification.

Due to the electrohydraulic analogy, hydraulic elements and systems can be examined by the theory of electrical circuits that have similar transient properties and the circuit is defined as a one-dimensional mathematical model. For hydraulic circuits, the following hydraulic resistances have been specified [4, 7]:

- The local and frictional resistances:  
  \[ R = \frac{d(\Delta p)}{dQ} \]

- The resistance to acceleration (inductance):  
  \[ L = \frac{\Delta p}{\frac{dQ}{dt}} \]

- The resistance to deformation (capacity):  
  \[ C = \frac{Q}{\frac{d(\Delta p)}{dt}} \]

Resistance sorting (serial, parallel) is the same in fluid systems as in electrical systems (the most famous is so called RLC T-cell). For combined sorting, this is a resistance network, the solution of which is described by the node law and the circuit law.

The software Matlab-Simulink-SimHydraulics was used to solve the hydraulic circuit by electrohydraulic analogy. The program allows to model systems consisting from hydraulic and mechanical elements, called multi-domain systems. It uses a direct analogy with real system elements (SimHydraulics blocks correspond to the real elements in the circuit). Each single connection between Matlab blocks corresponds to the power transmission as in the actual system. This makes it possible to simulate systems by describing their physical structure and it is not necessary to derive mathematical relationships between controlled quantities [1, 3, 5].

A more precise model is defined as a multidimensional mathematical model or a multidimensional multiphase model. The basic equations describing the flow of real liquids, expressing the basic physical laws of conservation of mass, momentum and energy, physically demonstrate the balance of a certain variable \( \zeta \) in the control volume as a continuous environment and in certain coordinate system. According to this balance, the time change (accumulation) of the variable \( \zeta \) in the given control volume \( V \) is equal to the flow of this variable by the control surface \( S \) limiting this volume and its production within the volume \( V \) [7].

\[
\frac{\partial}{\partial t} \int_V (\rho \zeta) dV + \int_S \rho \vec{F}(\zeta) \cdot \vec{n} dS = \int_V P(\zeta) dV
\]  

(1)

where

- \( \zeta \) – the balancing variable,
- \( \vec{F}(\zeta) \) - the flow density vector of variable \( \zeta \) by area \( dS \),
- \( \vec{n} \) - the vector normal to the area \( dS \),
- \( P(\zeta) \) - the density of production of variable \( \zeta \) (production per unit of time in volume \( V \)).

The ANSYS Fluent software enables the solution of the above equation system by the finite volumes method from its setting, through stationary and transient calculation, to subsequent display of
results in two-dimensional, axially symmetrical and generally three-dimensional space. It is important to choose the correct mathematical model of laminar or turbulent flow, or multiphase flow with or without admixture. Furthermore, the physical properties of used media, initial and boundary conditions are set. The following is a numerical calculation and evaluation of results [1, 2].

2. Experiment

Figure 1 illustrates a hydraulic circuit on which a number of static and dynamic measurements have been made. The result was the transition characteristics that served to define boundary conditions and verify mathematical modeling, see [1].

![Figure 1. Experimental equipment for measuring the hydraulic hammer and circuit diagram.](image)

The circuit is composed of following elements: water tank (T), pump (P), digital manometer (M), orifice (O) with a differential manometer of gauge for measuring the flow rate (DM), liquid manometer (LM), ball valve (V), long tube (T), three electrical pressure sensors \( p_1, p_2 \) and \( p_3 \) (connected to the sampling points 1 (behind the pump), 2 (behind the orifice) and 3 (before the closing ball valve)) and the connecting elements of the pipeline.

On the circuit for measuring the hydraulic hammer, the static characteristics of the hydrodynamic pump for the maximum speed of 2000, the orifice and the ball valve at full opening, and their loss / flow coefficients were measured. Subsequently, the hydraulic hammer was measured for comparison with the mathematical model.

![Figure 2. Static pressure behind pump, behind orifice, before valve.](image)

The circuit can be considered a low-pressure system. Figure 2 evaluates the measured transient characteristics where the curves present steady states in the left and right parts of the graph and the hydraulic hammer is evident in the middle [4, 7]. The results are summarized in Table 1.
Table 1. Measured and calculated results.

|                     |       |          |
|---------------------|-------|----------|
| flow rate $Q$       | m$^3$/hour | 1.120    |
| wave run time $T$   | s     | 0.86     |
| real velocity of sound $a_s$ | m/s | 111.63   |
| real bulk modulus of elasticity $K_s$ | Pa | 12 438 360 |
| closing time of the valve $t_u$ | s | 0.11     |
| mean pressure behind the pump $p_1$ | Pa | 29599    |
| mean pressure behind the orifice $p_2$ | Pa | 23783    |
| mean pressure before the valve $p_3$ | Pa | 4985     |

3. One-dimensional model (electro-hydraulic analogy)

For hydraulic hammer modeling the hydraulic circuit was created in the Matlab-SimHydraulics, see Figure 3. The tank is defined by the given level of the water, the energy source is the pump given by the static characteristic, the orifice is like the resistance to flowing, the long pipe is given as segmented pipe, which is presented approximately by number of T-cells, the ball valve is controlled by the closing signal and the short pipe feeds the water back to the tank. There are four pressure sensors and a flowmeter in the circuit. An important element is the custom hydraulic fluid defining the parameters such as density and kinematic viscosity of the fluid, the measured or the theoretical modulus of water elasticity and air content [6]. The numerical parameters and the solution method are entered in the configuration and simulation parameters.

The first step was to verify the stationary solution, i.e. the conformity of measured and calculated values of pressure and flow rate, and then to resolve and verify transient solution of hydraulic hammer.

Figure 3. Hydraulic circuit for solving the hydraulic hammer in Matlab-SimHydraulics.
Table 2. The parameters for definition the fluid.

|                         | Variant 1 | Variant 2 | Units   |
|-------------------------|-----------|-----------|---------|
| Fluid density           | 998       | 998       | kg.m⁻³  |
| Kinematic viscosity     | 0,0000001 | 0,0000001 | m².s⁻¹  |
| Bulk modulus            | 12 401 859| 2 300 000 000| Pa      |
| Relative amount of trapped air | 0     | 0,02      | 1       |

4. Two-dimensional modeling of final volum method

For two-dimensional modeling, the geometry, physical properties and boundary conditions were defined based on the experiment. Pipeline geometry has been simplified as axially symmetrical region for simulation in software Ansys Fluent. The computational mesh was refined near the wall and contained a smaller number of cells (300 970). Then the calculation was not time consuming. The area diagram and seven created boundary conditions are shown in Figure 4.

![Figure 4: The area diagram and boundary conditions.](image)

The task was solved as axially symmetrical and time-dependent with the selected time step of 0.001 s. Due to the lower Reynolds numbers, a two-equation k-ε RNG turbulent model was chosen as the computational model, the wall function was setting as "Standard Wall Function" [2]. The flowing medium (water) was defined as a single-phase compressible fluid with the bulk modulus of elasticity obtained from the measurement and as a multiphase mixture of water and air (density is given by the equation of state), see table 2.

The inlet boundary condition called "Pressure-Inlet" was defined as total pressure of 2 693.8 Pa (static pressure was calculated as a hydrostatic pressure, caused by a column of water in the tank and the dynamic pressure depending on the flow velocity in the tube). Turbulence was defined by: Turbulent Intensity of 1% and Hydraulic Diameter of 0.0232 m.

The pump boundary condition has been defined as "Fan" and allows input of static characteristic by polynomial, i.e.

\[
\Delta p = a_2v^2 + a_1v + a_0 
\]  

(2)

On the basis of measured pump characteristic for the revolution 2000 min⁻¹, the graph of the pressure drop versus flow velocity was created and the regression method determined the coefficients as \(a_0=41 053.24422, a_1=-20 942.39698, a_2=2 099.41419\).

The orifice boundary condition was specified by simplification as "Porous-Jump" as it represents a pressure drop due to flow around obstacle. The porous thin-walled medium of final thickness represents the pressure loss given by Darcy’s relationship and added inert loss according to the relationship

\[
p = \left(\frac{\mu}{\alpha}v + C_2 \frac{I}{2} \rho v^2 \right) t
\]  

(3)

From the values obtained when measuring the orifice static characteristic, the regression of the pressure loss versus flow velocity was created \(\Delta p = C_2 v^2 + C_1 v\). Subsequently, face permeability \(1/\alpha=2\) 694 205 m⁻² and pressure-jump coefficient \(C_2=46,87922\) m⁻¹ at porous medium thickness \(t=0.49\) m was determined [2, 8].
The output boundary condition was set as "Velocity-Inlet". The size of velocity was calculated from the measured volume flow rate $Q_v$ and tube internal diameter ($v = 0.633$ m$^3$s$^{-1}$). Turbulence was defined by turbulent intensity by value of 1% and a hydraulic diameter of 0.0232 m. The velocity value was set to 0 for closed valve. The closing time is assumed to be 0.11 s. The boundary condition was specified using the table as instructed [1, 2]. On the walls, the roughness has been estimated so that the pressure drop of the stationary result matches the measurement.

5. Comparison of results
The results of the mathematical modeling of the two variants with respect to the modulus of elasticity obtained from simulations using Matlab-SimHydraulics and Ansys-Fluent were compared with the experimental results with the same specification of boundary conditions, element characteristics and wall roughness. Figure 5 compares the pressure values before the valve, which is the most interesting for the hydraulic hammer and the simulation of the flow rate is evaluated in Figure 6. The dynamic flow rate was not measured with the use of existing measuring devices.

![Figure 5. Comparison of static pressure versus time between simulation results and experiment result.](image)

![Figure 6. Comparison of volume flow versus time from simulation results.](image)

6. Conclusion
This work was devoted to hydraulic hammer simulation in the low-pressure hydraulic system. Two time-dependent mathematical models were tested by simulation methods and verified by experimental measurement. Two variants of the solution were made. The first variant supposed the fluid with zero air content and the volume modulus of elasticity obtained from hydraulic hammer measurement. The second variant solved the fluid defined by the theoretical volume modulus of elasticity (2 300 MPa) and the volume fraction of air $\alpha = 0.02$.

The first one-dimensional flow model was compiled in the Matlab-SimHydraulics software. Circuit design in this program is simple and intuitive, the resulting model is very similar to a real hydraulic scheme. It is important to choose the correct numerical solver in setting. The number of segments in the segmented pipeline also has a significant influence on the accuracy of the calculation. The more
segments you choose, the more accurate the calculation and more the calculation time is longer. The second mathematical model, allowing simulation of multi-dimensional flow, was created in Ansys Fluent software under the same boundary and initial conditions. For reasons of shortening calculation times, geometry was defined as 2D and axially symmetrical. Simulated pressures were compared with the measured values and the results are evaluated. It can be said that the amplitudes and periods of static pressure are almost the same and in good accordance with experiment. Multi-dimensional modeling also enables to solve problems with complex geometries.

An important benefit of this work is that simulation of the hydraulic hammer by a one-dimensional model (SimHydraulics) and a multi-dimensional model (Fluent) are nearly of the same quality. Both programs model the real circuit, inlet parameters of the hydraulic elements are determined by static characteristics, theoretical modulus of elasticity and content of air, no auxiliary parameters are estimated or calculated. This allows to model the dynamics of the newly designed hydraulic circuit and to predict the behaviour of the pressure and flow rate at various points in the circuit, to optimize the circuit elements and subsequently prepare the design of the circuit for implementation. Further research will be focused on solving the dynamics of oil flow in hydraulic circuits.

References
[1] Fajka M 2017 Measurement of losses in the low-pressure hydraulic system with hydrostatic pressure Diplomová práce (Ostrava)
[2] ANSYS Fluent User's Guide [online]. Release 15.0. 2013, 2692 s. [cit. 2017-05-05] Dostupné z: http://148.204.81.206/Ansys/150/ANSYS%20Fluent%20Users%20Guide.pdf
[3] Wang C, Nilsson H, Yang J, Petit O 2017 1D–3D coupling for hydraulic system transient simulations Computer Physics Communications 210 pp 1-9
[4] Noskievič J 1995 Dynamics of fluid mechanisms (VŠB - TU Ostrava, Ostrava)
[5] Urbanowicz K, Firkowski M, Zarzycki Z 2016 Modelling water hammer in viscoelastic pipelines: short brief. In: Journal of Physics: Conference Series (IOP Publishing) p 012037
[6] Jablonská J 2014 Compressibility of the fluid In EPJ Web of Conferences (EDP Sciences) 67 p 02048
[7] Kozubková M 2009 Mathematical models of cavitation and hydraulic hammer (VŠB - TU Ostrava, Ostrava)
[8] Štigler J, Klas R, Kotek M, Kopeccký V 2012 The Fluid Flow in the T-Junction. The Comparison of the Numerical Modeling and Piv Measurement Procedia Engineering 39 pp 19-27
[9] Brennen E 1995 Cavitation and bubble dynamics Cambridge University Press

Acknowledgment
The work presented in this paper was supported by a grant SGS „Výzkum v oblasti dynamiky tekutínových systémů“ SP2017/103.

This work was supported by The Ministry of Education, Youth and Sports from the Large Infrastructures for Research, Experimental Development and Innovations project “Innovations National Supercomputing Centre – LM2015070”.