FLUID-STRUCTURE SIMULATION OF A VALVE SYSTEM USED IN HYDRAULIC DAMPERS

Piotr Czop[3a], Damian Gąsiorek[1b], Jacek Gniłka[1c], Damian Sławik[2], Grzegorz Wszołek[2d]

[1]Silesian University of Technology, Department of Applied Mechanics,
[2]Silesian University of Technology, Institute of Engineering Processes Automation and Integrated Manufacturing Systems
[3]AGH University of Science and Technology, Department of Robotics and Mechanics
e-mail: [a]pczop@agh.edu.pl, [b]Damian.Gasiorek@polsl.pl, [c]Jacek.Gnilka@polsl.pl, [d]Grzegorz.Wszolek@polsl.pl

Abstract

The aim of this paper is to develop a method for optimizing the design of a disc spring valve system by reducing the aeration and cavitation effect which negatively influences the performance of a shock absorber. A fluid-structure interaction (FSI) model is used in order to modify the geometry of the valve interior and, in turn, to achieve better performance of a shock absorber. The paper analyzes the pressure distribution along the flow paths inside the valve cavity to reduce the risk of aeration and cavitation, while other important engineering aspects are omitted, e.g. durability of disc-spring valve systems as discussed in [1]. A key measure of valve improvement was chosen as deterioration of the damping force level generated by a shock absorber vs. the number of cycles during continuous cycling of the damper .

The objectives of this work are as follows: (i) to present a process for reducing the complexity of the geometry of a disc spring valve system in order to perform a combined fluid-structure simulation, (ii) to show key steps of the simulation process focusing on interactions between fluid and structure domain and to review relevant simulation results, (iii) to describe practical aspects of the simulation process, including basic parameters and boundary conditions related to the applied commercial software, (iv) to make an optimization case study to show the application scope for the simulation methodology proposed in the paper, and to confront the simulation results with experimental investigations.

1. INTRODUCTION

The roles of a shock absorber working in a vehicle suspension are, in a sense, contradictory. Firstly, an ideal shock absorber should guarantee good road handling, secondly, it has to be designed for durability, thirdly, the radiated noise and emitted vibrations should have as little power as possible, and lastly, it should ensure passenger comfort. This work concentrates on improving road handling using advanced simulation methods. The goal is to optimize a shock absorber by providing a stable damping force characteristic versus disturbance factors which is essential to maintain good road handling. A damping force characteristic of a shock absorber is determined by tunable valve systems which are the key components of a shock absorber. A state-of-the-art theoretical and practical overview of shock absorber technologies is provided by Dixon [2], including the latest technologies available in the market, i.e. shock absorbers for active and semi-active suspension systems. This work proposes an advanced fluid-structure interactions (FSI) model to understand and optimize performance of a valve system assembled in a piston-rod of a double-tube shock absorber regarding aeration/cavitation phenomena. Advanced simulation methods are required since the simplified analytical approach widely presented in the literature does not provide enough detailed first-principle insight to perform optimization of the geometrical properties of a valve system. First-principle lumped-parameter dynamic models of conventional shock absorbers are discussed by Lang [3] or Talbot and Starkey [4] including detailed analysis of valve system models. Another study conducted by Yamauchi [5] developed the physical model of a shock absorber to understand and reduce the effect of self-excited vibration in the piston-rod assembly. The model was correlated to the available measurement data, including an experiment with a modified construction of the piston-rod assembly. A model intended to optimize the high-frequency dynamical behavior of a shock absorber was developed by Kruse [6]. There are other studies relevant to the scope of this paper concentrated on components of shock absorbers, namely valve systems. Valve component models were developed by Beyer et al. [7] and Choon-Tae and Byung-Young [8]. They used advanced measurement and validation methodology, i.e. a measurement set-up equipped with laser and pressure sensors. In the field of shock absorber research, Martins et al. [9] presents a development and validation of a CFD model to investigate the oil flow in a shock absorber. Guzzomi et. al [10] present an FSI approach towards prediction of a valve system performance by coupling the valve disc stack deflection with the CFD pressure results. That study reports a work using a similar methodology as proposed in this paper, while focusing on a prediction of valve performance instead optimization of a valve system. Other works presenting application of the CFD technique in understanding and optimization of other types of valve systems used in industry are, for example [11], where the analysis is also relevant to the topic of this paper. Other works also present important experimental approaches, e.g. [12-14]. The fundamental reference to fluid mechanics and coupled structure-fluid computations deployed in this paper to optimize a valve system can be found in [15-16].

The work discussing aeration/cavitation in shock absorbers, which is the effect intended to be minimized using the FSI simulation approach proposed in this paper, is reported by Dyum [17], Dixon [2], Czop and Slawik [18], and Sławik et al. [19]. Iyer and Yang [20] also performed a relevant analytical study on the dynamics and hydrodynamic stability of liquid-vapor mixtures in the bubble-flow range in reciprocating motion through a horizontal channel applicable to optimization of shock absorber development. This section provides an introduction to the theory and triggering factors of aeration and cavitation in shock absorbers based on Dixon [2] and the analytical work conducted by the authors [18] and supported with experimental studies [19].

The aeration phenomenon in a shock absorber is defined as a process by which gas, typically nitrogen, is circulated through, mixed with, or dissolved in oil being used as a working fluid in shock absorbers. Gas is included in shock absorbers under certain pressure, separately from the oil, to provide compressibility to allow for the rod displacement volume compensation. Theory states [2,21] that a liquid exposed to a soluble gas (i.e. the liquid comes into contact with the atmosphere of a gas that can dissolve in it) is in one of three forms: liquid-gas solution, liquid-gas bubble emulsion or foam. The liquid-gas solution is prone to bubble formation when the pressure of the liquid-gas solution falls below the so-called saturation pressure. In this state, the liquid is no longer capable of retaining all the gas in its dissolved form and therefore bubbles occur. The solubility of gas in a liquid is directly proportional to the absolute pressure above the liquid surface (Henry's law), and normally decreases with rising temperature [2]. All of the mentioned liquid-gas mixtures can be considered as liquid with pockets of gas or vapor. The dissolved gas has a significant influence on the oil mixture and thus on the shock absorber's behavior. The presence of gas bubbles is the cause of the damping force loss in the shock absorber. It is an undesirable and negative effect visible as asymmetry of the force displacement characteristic and should be minimized. Fig. 1 shows the influence of aeration on the damper performance based on the force-displacement characteristic obtained for a sequence of 1500 cycles. The energy of hydraulic friction absorbed by a shock absorber caused an increase in its temperature. The damper was cycled with high velocity of 1.5 m/s, three sequences (the first, the middle, and the last) of 500 cycles each were plotted to show deterioration of the force-displacement characteristic.

Modeling the dynamics of gas bubble formation and transport is a task very difficult for several reasons. Most important ones are difference between time scales in which aeration processes occur (order of minutes) and the time scales of oil flow through a damper (order of seconds), existence of uncontrollable parameters on which bubble size depends and the bubble size itself (e.g. oil impurities and sharp edges), re-absorption of gas from bubbles surface, etc.

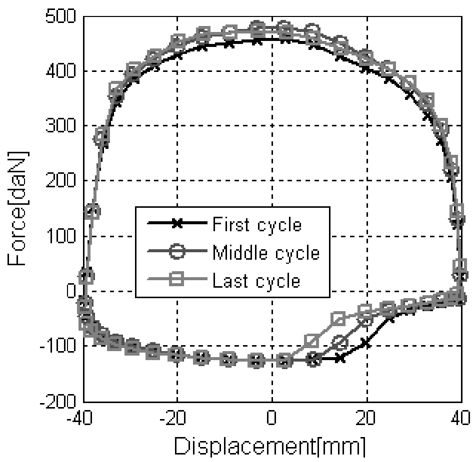

Fig. 1. The force-displacement characteristic of a shock absorber [19]

The cavitation phenomenon occurs, when oil does rupture under the influence of tensile stress, the rupture of the fluid manifests itself as a number of very small cavities in the oil [1]. The process of cavitation depends among other considerations, on the purity of the liquid and the rate at which the liquid is stressed. Cavitation is the formation of pockets of vapor in a liquid. When the local ambient pressure at a point in the liquid falls below the liquid's vapor pressure, the liquid undergoes a phase change to a gas, creating "bubbles," or, more accurately, cavities, in the liquid. Changing temperatures alter the vapor pressure of a liquid dramatically, making it easier or harder for the local ambient pressure to dip below the vapor pressure to cause cavitation.

The violent collapse of cavitation or aeration bubbles results in the production of noise as well as the possibility of material damage to nearby solid surfaces [1]. Noise is a consequence of the momentary large pressures that are generated when the contents of the bubble are highly compressed. This also results in a micro flow in the liquid caused by the volume displacement of a growing or collapsing cavity. A larger number of collapsing gas bubbles decreases the bulk modulus of the gas-oil mixture, and produces. Cavitation and aeration occur at restrictions where potential pressure energy is converted into kinetic energy increasing flow velocity and dramatically decreasing the pressure in the oil locally. Valve systems used in hydraulic dampers should be designed to minimize the possibility of occurrence of local low-pressure regions which contribute to the formation of gas or cavity bubbles.

The remaining content of the paper is divided into five sections. Section 2 presents the working principles of shock absorbers and valve systems, while Section 3 provides an introduction to the methodology used to simulate structure-fluid interactions in a valve system and illustrates the process of fluid-structure model development. Section 4 discusses the proposed valve design optimization method, while Section 5 reports experimental results of this optimization process. Lastly, Section 6 presents the summary of the paper.

2. SHOCK ABSORBER AND VALVES WORKING PRINICPLES

This section presents the fundamental working principles of a hydraulic shock absorber. The hydraulic double-tube damper presented in Fig. 2 consists of a piston moving within a liquid-filled cylinder. As the piston is forced to move within the cylinder (pressure tube), a pressure differential is built across the piston and the liquid is forced to flow through valves located in the piston and the base-valve assembly. The presence of the piston divides the cylinder space into two chambers: (i) the rebound chamber, that portion of the cylinder above the piston and (ii) the compression chamber, that portion below the piston. The action of the piston transfers liquid to and from the reserve chamber, which surrounds the cylinder, through the base-valve assembly located at the bottom of the compression chamber. Two types of valves are used in the shock absorber: (1) intake valves and (2) control valves. The intake valves are basically check valves which provide only slight resistance to flow in one direction and prevent flow in the opposite direction when the pressure differential is reversed. Control valves are preloaded through a valve spring to prevent opening until a specified pressure differential has built up across the valve.

The two working phases of a hydraulic shock absorber are distinguished as the compression phase and rebound phase. During the compression phase the rod is tucked into the damper, compression chamber volume decreases and oil flows through the piston compression intake valve (piston intake) and the base compression control valve (base valve) accordingly, to the rebound and reserve chambers. During the rebound phase the rod is rejected from the damper, the compression chamber volume increases and oil flows through the piston rebound control valve (piston intake) and base rebound intake valve (base intake) accordingly, to the rebound and reserve chamber.

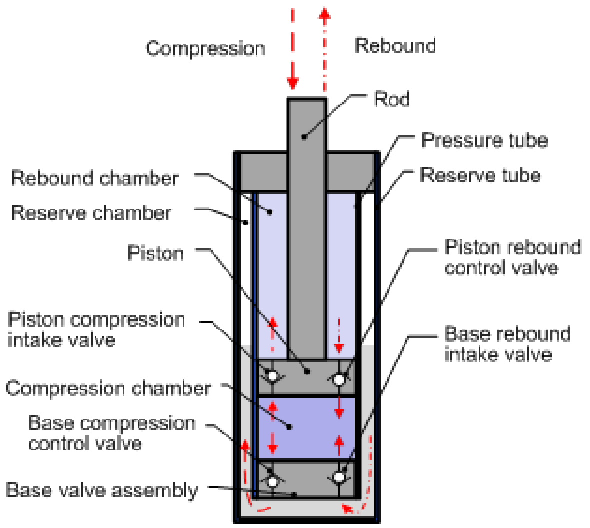

Fig. 2. Shock absorber working principle

The piston and base valves has to be balanced during operation which requires to maintain the differential pressure over the piston has to be greater than the sum of differential pressure over the base valve and the gas pressure in the reserve chamber. This requires to adjust the pressure-flow characteristics of piston and base valves to meet valve balance conditions during a compression stroke. Valve unbalance results in an effect that the pressure in the rebound chamber becomes lower than the atmospheric pressure during a compression stroke. This low pressure causes cavitation or gas release from oil-gas mixture in whole rebound chamber volume.

The paper considers a specific type of shock absorber valve, i.e. the clamped piston compression valve presented in Fig. 3. Such a valve system consists of a combination of disc springs, referred to further in the paper as a stack of discs or a disc stack. The number of discs, their diameters and thickness, directly affects the operational pressure-flow characteristics of the valve system.

A valve system operation can be split into three regimes. In the first regime, there is only a small flow through bleeds of a very small area below 1mm2 in the so-called orifice disc while the stack of discs is completely closed (Fig. 3a). The damping forces produced by the valve are therefore very small, similar to a drive along a smooth road such as a highway. The stack of discs starts opening in the second regime providing a typical range of damping forces (Fig. 3b).

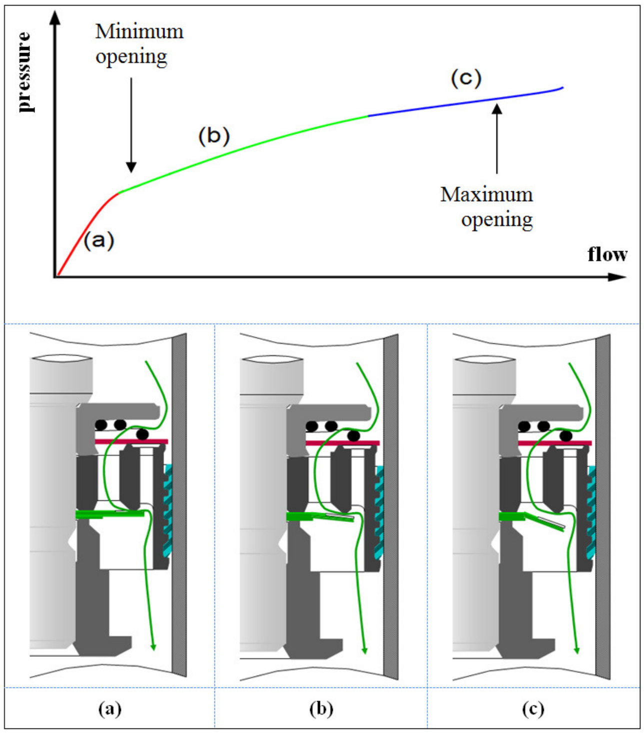

Fig. 3. Pressure-flow or force-flow characteristics and its regimes: a) bleed operation; b) normal operation; c) high-damping operation

The last regime corresponds to the case when the stack is fully opened and the restriction is provided by the profiled channels in the piston component (Fig. 3c). This regime covers off-road conditions or violent maneuvers on the road. This work focuses on the second and third regime, corresponding to the minimum (initial) opening and the maximum opening of the valve system.

Valve systems are characterized by the pressure-flow characteristics which are obtained during component-level tests with the use of a hydraulic test-rig equipped with a pump, a flow tool where the valves are assembled and measured, hydraulic lines, and a data acquisition system. The controller regulates the flow through the valve allowing a pressure-flow characteristic of a valve to be captured. The pressure is a differential pressure measured before and after the flow tool, evaluating the pressure drop across the valve assembly for a given flow rate.

3. FLUID-STRUCTURE MODEL DEVELOPMENT

A hydro-mechanical valve system model usable from an engineering application perspective should be able to reproduce essential properties of a valve system during operation in a shock absorber. This requires a combination of two sub-models: (i) a finite element mechanical (stress/strain) model, and (ii) a flow model. The mechanical model obtains (i) stress in discs, (ii) displacement between the orifice and a valve seat; both as a function of the pressure load. The opening of a disc stack can also be expressed as a function of an outflow area vs. pressure load. If the shock absorber geometry is known, then the flow model allows the outlet flow rate through the valve system as a function of the pressure load to be obtained. The input of the mechanical model is a pressure drop across the valve system, while the outputs are the critical stress and displacement of a disc stack over the valve seat. The input to the flow model is the pressure drop and the opening displacement, while the output is flow rate or velocity.

Fluid-structure simulations apply two major approaches, namely monolithic and partitioned. The equations governing the flow and the displacement of the structure are solved simultaneously, with a single solver in the monolithic approach, while the partitioned approach facilitates solution of the flow equations and the structural equations in two non-coupled simulation environments. The approach deployed in this work belongs to the second approach using iteratively two independent solvers where the obtained geometry is transferred between the two solvers.

On the other hand, the FSI methods are also classified according to the purpose of the simulation as one-way, two-way, or mixed methods. One-way FSI is based on the pure mapping of physical properties resulting from the analysis of a CFD-/FE-model to another FE-model. The two models typically do not rely on matching meshes The mapping of the physical properties does not include the modification of the meshes. Two-way FSI is based on the mapping performed in an iterative loop i.e. the results of the first model are mapped to the second model and these results are mapped back to the first model and so on until convergence is found or the process is stopped manually. 2-way FSI can include modification/morphing of the mesh of one or both of the models during the mapping phase. The method proposed in this paper is a modification of the two-way FSI method, where deformations of a stack of discs due to a pressure load are transferred to the CFD-model, and then re-evaluating the CFD-model in the deformed configuration. This method dramatically simplifies calculations and makes the process suitable for application in commercial engineering work, where hardware and software resources are limited, while lead-time for obtaining results is critical. The method is intended to perform quick-and-dirty calculations for a limited number of points/ranges in the pressure-flow characteristic to provide an approximate solution which is a starting point for further experimental studies using design of experiment and other data-driven techniques within six-sigma methodology.

The simulation was conducted for the minimum opening of the stack of discs in the range between 1.6 – 2.6 MPa with the step of 0.1 MPa, while for the maximum opening, the range was between 2.8 - 4.8 MPa with the step of 0.2 MPa.

A valve production design needs to be simplified to reduce numerical effort during simulations by decreasing the total number of finite elements used in the simulation. Therefore, the simulation model neglects the geometrical and physical details which less significantly influence the modeling accuracy regarding the major objective of the paper, i.e. optimization of the flow through the valve interior to avoid low-pressure regions. The maximum stress is considered as the measure of model quality after the reduction process of model numerical complexity [1]. The results of model reduction are summarized in Table 2. The intention was also not to include micro quality surface parameters into the model geometry, e.g. parallelism or flatness. The FS model proposed in this paper covers: (i) loading resulting from the valve assembly process and (ii) loading resulting from the operational conditions. It is important to correctly determine residual forces and stresses which are applied to a valve during its assembly process in a manufacturing process.

The step of initial loading includes a simulation of the valve assembly process where the washer (surface no. 1) is moved down, while the valve seat (surfaces no. 2-3) held fixed as shown in Fig. 4. The washer moves until the clamping force is equal to the specified preload distance. Fig. 4 shows also important surfaces in the loading process for the considered valve system. Surface no. 1 is a head surface limiting the discs D1, D2, D3. Surface no. 2 determines the edge (valve seat) supporting the initially loaded disc, while Surface no. 3 is the hub surface towards which the discs are moved after applying the preload force. The elements to be deformed during simulations were not simplified.

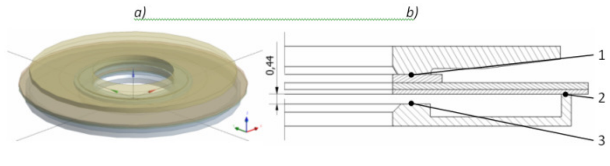

Fig. 4. CAD model of a valve: (a) 3D overview b) a cross-section with the important surfaces

The model applies a three dimensional (3D) non-axisymmetrical geometrical discretization with finite element mesh defined by QuadraticHexahedron elements (NASTRAN name: CHEXA [22]). Disc thickness is decomposed at a minimum of 3 to 5 finite elements respectively for discs D1-D3. The boundary conditions were defined as a rigid fixation of the valve assembly, and cylindrical fixation of the limiting disc and discs D1-D3 including free rotation and axial movement. The discs were moved in the simulation at a distance of 0.44 mm towards the valve assembly to generate the initial pre-load force. The symmetrical properties of the valve were considered in the next step of the model reduction process. The symmetry of holes in the piston solid allows the reduction of the model geometry considering only 1/8 of the piston component. The reduction allows the number of finite elements to be reduced proportionally by 8 times. The reduced 3D model was simulated and compared to the full 3D model. The summarized results are presented in Table 1. The reduced model showed higher stress values of about 5% during the preloading step which is, according to the authors, acceptable for obtaining correct simulation results

Tab. 1. Simulation results for the original and reduced model

	Disc D1	Disc D2	Disc D3
The total applied displacement [mm]	0.44	0.44	0.44
Maximum von Mises stress value (complete 3D model) [MPa]	719	1203	861
Maximum von Mises stress value (reduced 3D model) [MPa]	715	1158	897
Change between complete and reduced model [%]	99.44	96.25	95.98

Operational loading is applied to the preloaded spring discs to achieve the minimum and maximum opening regimes of the valve as presented in Fig. 3bc. An additional coordination system was added to enable the measurement of the deflection of the disc surface (mesh nodes) and to transfer the results to the flow domain solver.

4 . VALVE DESIGN OPTIMIZATION

The analysis of the flow through the valve cavity was performed for the minimum and maximum opening of the stack of discs (cf. Fig. 3). Figs. 5-6 illustrate turbulent behavior of the oil inside the valve cavity. The flow is more turbulent in the case of the larger (maximum) opening and also its velocity is higher compared to the minimum opening of the valve

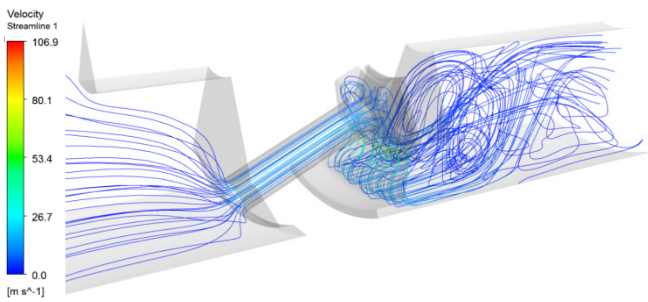

Fig. 5. Visual presentation of the flow through the valve cavity with the use of velocity streams (minimum opening)

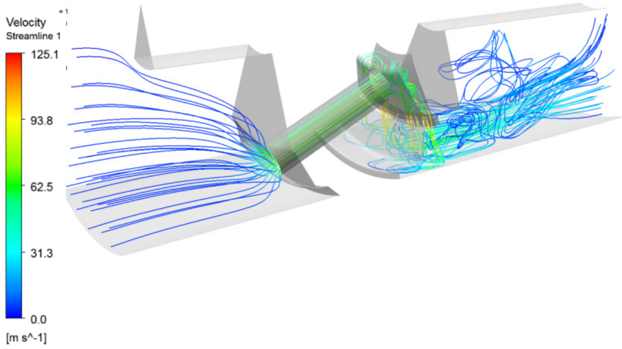

Fig. 6. Visual presentation of the flow through the valve cavity with the use of velocity streams (maximum opening)

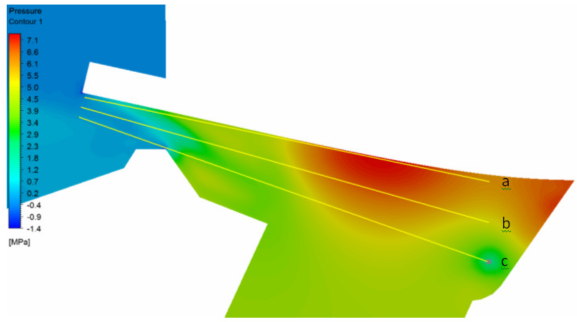

Fig. 8. Region of pressure decrease inside the valve cavity and further over the valve seat edge (a,b,c) – maximum opening

The next analysis is presented using the velocity maps. The visual inspection shows larger regions of the higher velocities for the maximum opening compared to the minimum opening of the valve. The velocity is basically determined by the pressure drop across the valve assembly, which is 2.3 and 7 MPa corresponding to the minimum and maximum opening of the valve, respectively (Fig. 7).

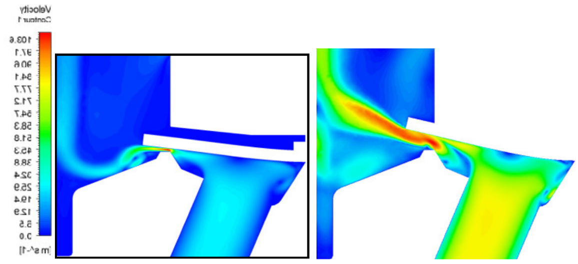

Fig. 7. Visual presentation of the flow through the valve cavity with use of velocity maps (the left plot – minimum opening; the right plot – maximum opening)

Visual inspection of velocity and pressure maps confirms that pressure energy turns into kinetic energy as a high velocity jet of oil. There are visible high-pressure gradients of the magnitude of 4MPa at the most narrowed geometry of the flow channel, i.e. at the valve seat. The pressure in the channels first drops significantly to 0 MPa and then returns again to a level of 6MPa. Visual inspection of Fig. 8 shows two regions of significantly lower absolute pressure drop. Those regions are located in the channel formed under the deflected disc surface and at the opposite side to where the disc is clamped. The presence of such regions with dramatically lower (or negative) pressure can indicate the possibility of cavitation or aeration.

The first region of the pressure drop (-0.9MPa) is formed between the external edge of the circular valve seat and the deflected disc surface (cf. Fig. 8), while the second region is located at the outlet of the flow channel in the piston (~1MPa). The pressure variation profiles along the flow path were obtained in Fig. 9 using the three supporting lines showed in Fig. 8.

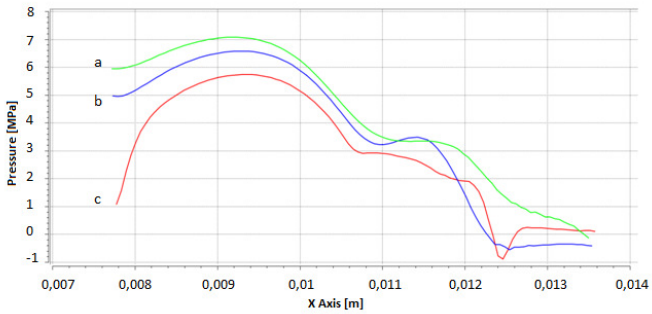

Fig. 9. Extracted pressure profiles (a,b,c) from the pressure map shown in Fig. 8

Theoretical analyses and experimental investigations [1,17,20,23] indicate the possibility of aeration/cavitation in the local regions where the pressure significantly falls approaching the reference zero value, which is 0.3-0.6 MPa above the absolute zero pressure due to initial pressurization of the shock absorber with the nitrogen gas. In consequence, the presence of aeration/cavitation causes a deterioration of damping force and increases the risk of high-frequency vibrations transferred through the suspension system to the vehicle interior [17,18]. In turn, passengers can perceive a lower ride comfort, noise and harshness. Moreover, a high-damping lag as discussed in Section 1 can influence the vehicle ride/handling properties including safety risk.

It is important to increase the margin of stable operation of shock absorbers improving performance of valve systems without exciting any side-effects. The valve opening should be smooth and the flow through the valve cavity should be characterized by a uniform pressure decrease along the flow path without tendencies to self-excited whirls. This is achievable by removing internal sharp edges, obstacles, and small cavities where the oil flow can initiate turbulences at high velocities. The optimization of the geometry should result in an increase in oil flow efficiency and ensure more uniform pressure distribution without the tendency to form local low-pressure regions.

The authors proposed the modification of the piston component differently shaping the valve seat and interior of the valve in the location indicated by markers, cf. Fig. 8. A few proposals were verified by means of numerical simulation to achieve improvement minimizing the pressure drop. The optimal results were obtained for the shape shown in Fig. 10.

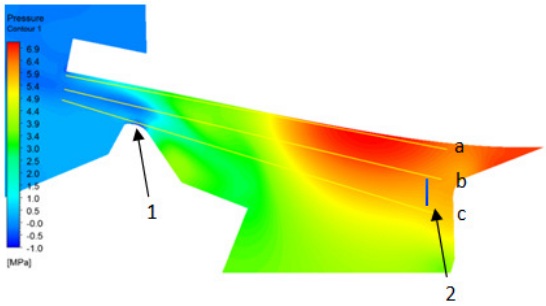

Fig. 10. A pressure distribution after the applied modifications in the design

The modification provided a visible improvement for the second region, where the pressure increased to a level of 5.6MPa (Fig. 11). The improved characteristic showed fewer differences between the curves, resulting in a more uniform pressure decrease along the flow channel. Visual inspection of the differential pressure distribution map in Fig. 8 versus Fig. 10 shows an improvement by means of numerical simulation.

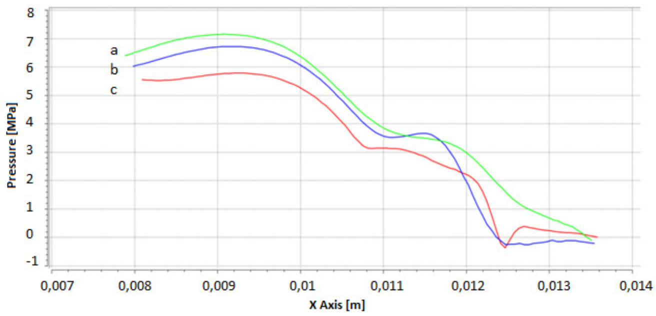

Fig. 11. Region of pressure drop inside the valve cavity and over the valve seat edge (a,b,c) after design modification (Fig. 10)

5. SUMMARY

This paper discusses a method of fluid-structure modeling in order to optimize a disc spring valve system. The proposed methodology deploys a fluid-structure (FS) model to indicate the local pressure drop regions which are the major root cause of aeration/cavitation.

This work describes a process of reducing the complexity of geometry of a disc spring valve system which is discussed in Sections 3. This process allows the reduction of the full 3D model to 1/8-th equivalent 3D model reflecting the must geometry, i.e. the three base surfaces. The accuracy of the simplified model differs less than 5% from the original full 3D model (Table 1). The authors evaluated that the model simplification allowed 80% of the computational time to be saved, shortening calculations to 2 days per single case, thus the optimization method can be recommended for industrial applications. The fluid-structure simulation process is presented including the process of model preparation (Section 3), and the analysis of the obtained pressure/velocity characteristics for the flow through the valve cavity (Section 4).

Literatura

1. Czop P., Sławik D., Sliwa P.: Static validation of a model of a disc valve system used in shock absorbers. “International Journal of Vehicle Design” 2010, Vol. 53, No 4, p. 317 – 342.
2. Dixon J.C.: The shock absorber handbook. Chichester: John Wiley & Sons, Ltd, 2007
3. Lang H.: A study of the characteristics of automotive dampers at high stroking frequencies. PhD thesis, Michigan 1997.
4. Tallbot M. S., Starkey J.: An experimentally validated physical model of a high performance mono-tube shock absorber. In: Proceedings of the 2002 SAE Motorsports Engineering Conference and Exhibition. US,

SAE Technical Paper, 2002-01-3337, 2002.

5. Yamauchi H., Sughara T., Mishima M., Noguschi E.: Theoretical analysis and proposition to reduce self-excited vibrations of automotive shock absorber. In: Proceedings of Noise & Vibration Conference and Exhibition. Michigan, US, SAE Technical Paper, 2003-01-1471.
6. Kruse A.: Characterizing and reducing structural noises of vehicle shock absorber systems. "SAE Technical Paper" 2002-01-1234.
7. Beyer O., Becher B., Stüwing M., Zimmermann G.: Measurement and simulation of the hydraulic behavior of the piston valve in a monotube shock absorber. In: Proceeding of the 7th International Conference ATA, Italy, 2002.
8. Choon-Tae L., Byung-Young M.: Simulation and experimental validation of vehicle dynamic characteristics for displacement-sensitive shock absorber using fluid-flow modeling. "Mechanical Systems and Signal Processing" 2006, No. 20 373–388.
9. Martins F. P., Siqueira C., Spogis N.: Development and validation of a CFD model to investigate the oil flow in a shock absorber. "SAE Technical Paper" 2005-01-4030.
10. Guzzomi F. G., O'Neill P. L., Tavner A. C. R.: Investigation of damper valve dynamics using parametric numerical methods. In: 16th Australasian Fluid Mechanics Conference 2007.
11. Borghi M., Milan M., Paoluzzi R.: Transient flow force estimation on the pilot stage of a hydraulic valve. In: Proceedings of the ASME-IMECE FPST-Fluid Power Systems & Tech. 1998, Vol. 5, p. 157–162.
12. Johnston D. N., Edge K. A., Brunelli M.: Impedance and stability characteristic of a relief valve. In: Proc. Inst.. Mech. Engrs. 2002, Vol. 216, Part I. "Journal of Systems and Control Engineering" 2002.
13. Johnston D. N., Edge K. A., Vaughan N. D.: Experimental investigation of flow and force characteristic of hydraulic poppet and disc valves. In: Proc. Instn. Mech. Engrs. 1991, Vol. 205, p. 161-171.
14. Provoost G. A.: A critical analysis to determine dynamic characteristics of non-return valves. In: 4th International Conference on Pressure Surges 1983, p. 275-286.
15. Potter M. C., Wiggert D. C.: Mechanics of fluids. Third edition. Thomson Learning, 2002.
16. Wylie B. E., Streeter V. L.: Fluid transient in systems. Upper Saddle River: Prentice Hall, 1993.
17. Duym S. W., Stiens R., Baron G. V., Reybrouck K. G.: Physical modeling of the hysteresis behaviour of automotive shock absorber. In: International Congress and Exposition 1997, Detroit, Michigan, p. 125-137.
18. Czop P., Sławik D.: A high-frequency model of a shock absorber and servo-hydraulic tester. "Mechanical Systems and Signal Processing" 10.1016/j.ymssp.2011.01.011, http://dx.doi.org/10.1016/j.ymssp.2011.01.011
19. Sławik D., Czop P., Król A., Wszołek G.: Optimization of hydraulic dampers with the use of Design For Six Sigma methodology "Journal of Achievements in Materials and Manufacturing Engineering" 2010, Vol. 43, No. 2, p. 676-683.
20. Iyer C. O., Yang W.: Analysis on liquid-vapor bubbly-flow systems in reciprocating motion. "ASME Journal of Fluid Engineering" 1999, p. 185-190.
21. Brennen C. E.: Cavitation and bubble dynamics. Oxford: Oxford University Press, 1995.
22. MSC.Software, http: http://www.mscsoftware.com
23. Hodges P. K. B.: Hydraulic fluids, New York: John Wiley & Sons Inc., 1996.

The authors gratefully acknowledge the financial support of the research project N N504 494239 funded by the Polish Ministry of Science (MNiI).