

Keywords: shock absorber; Fluid-Structure Interaction; CFD; damping forces; piston valve

Daniel BUCZKOWSKI*

Tenneco Inc. (Eastern European Engineering Centre), CAE Department
Bojkowska 59B, 44-100 Gliwice, Poland

Grzegorz NOWAK

Silesian University of Technology, Department of Power Engineering and Turbomachinery
Konarskiego 18, 44-100 Gliwice, Poland

**Corresponding author.* E-mail: daniel.buczowski@polsl.pl

MODELLING THE SHOCK ABSORBER PISTON VALVE USING 2-WAY FLUID-STRUCTURE INTERACTION

Summary. The aim of this study is to examine the strongly coupled Fluid-Structure Interaction approach as a comprehensive method of predicting the performance of the shock absorber piston valve. For this purpose, numerical simulation and experimental testing are carried out. The coupled CFD-FEA numerical model described in this article, contrary to the attempts made so far, takes into account the influence of contact between valve discs and the initial conditions of the disc stack preload. The model is based on the actual valve geometry used in the shock absorber design. As a result, the described approach is intended for use in industrial applications in development works, in particular, at the conceptual stage. To prove the reliability of the model, two valve compositions are chosen to be measured on a test bench and modelled in FSI simulations. For both of them, a satisfactory level of correlation is achieved, with the correlation error below 10% and well-predicted valve opening points. As a result, it is proved that the 2-way FSI approach has great potential to be successfully used to investigate the damper valve operation.

1. INTRODUCTION

1.1. Industrial background

The shock absorber is a crucial part in the car suspension system. It is responsible for comfort, safety and car handling, which are ensured by controlled generation of the damping force-dissipating energy cumulated in a suspension spring.

A wide range of technologies are currently used to meet customers' and the drivers' expectations. Economy and mid-class cars are usually equipped with passive dampers, while premium-class cars may have semi-active or dynamically adjusting complex systems, which respond to the road profile adequately. Nevertheless, even in the most advanced systems, passive pressure-flow characteristics are pre-set by passive valves. All commonly used passive shock absorber valves are based on a similar working principle. They consist of a piston with passages (channels) that moves forth and back in a pipe filled with viscous oil. During the motion, the oil is pushed from the piston from one side (chamber) to another through the passages. Additionally, the piston passages are covered with a set of metal discs, restricting the flow between the oil chambers. Flow restrictions generate friction within the oil, which is then dissipated in the form of heat into the environment. A piston valve has two sets of disc stacks on both sides. Each of them is responsible for generating pressure during the opposite stroke, but they can also influence each other. Depending on the disc's composition, thickness and

shape, a variety of pressure characteristics can be obtained. This study focuses on a valve placed at the end of a rod with a set of clamped steel discs. The device, commonly referred to as a piston valve, is presented in Fig. 1.

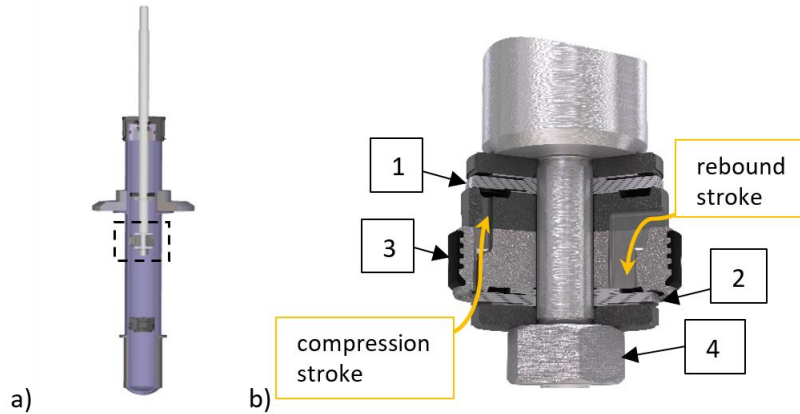


Fig. 1. a) – Piston valve location and the oil flow paths; b) – valve assembly: 1 – compression discs stack, 2 – rebound discs stack, 3 – sealing band and 4 – nut

Shock absorber valves operate in a non-linear manner. Their characteristics can be influenced by a number of parameters, such as the disc stack composition, the nut preload force, disc deflection limits, the design of piston channels and of the bleed disc (a disc with notches on the circumference for controlled leakage, as shown in Fig. 3), etc. To provide a satisfactory driving experience, these parameters must be well specified, together with the durability of components under operating pressure conditions. Collection of such data requires many physical tests using available and prototype components. Without a precise analytical tool, prototype components need many modifications before the final design is formulated. This makes the development process unduly expensive and time-consuming. To tackle these challenges, the automotive industry makes use of numerical modelling to pre-validate designs, propose enhancements and determine the design working conditions in the development stage, which shortens the development time and saves money¹.

The shock absorber valve operation is based on constant interaction between pressure conditions and deflection of the valve elastic components. At low rod (valve) velocities, the oil flow is restricted by bleed discs, which allows leakage for the damper soft response. The main disc stack opening is initiated by pressure overcoming its stiffness at higher rod velocities. Due to the asymmetrical load distribution caused by the piston channel's position and irregular bleed flow paths, the disc stack may open unevenly, which has to be taken into account. The opening point will also depend on the pretension clamping force, which counteracts the pressure. Because the flow conditions are strongly related to the structure deflection due to load, the fluid–structure interaction (FSI) analysis is required to perform reliable simulations of the valve operation.

1.2. Overview of FSI methods

Fluid–structure interaction is a multi-physics task involving the mutual influence between solid and fluid domains. It is understood as structural deformation under pressure forces present in the fluid domain. As a result, the fluid field changes to align with the new shape of the structure.

The piston valve is a common subject of FSI analysis. It consists of three domains: the oil–fluid domain, the valve disc–structure domain and the FSI interface region (surfaces of the structure domain in direct contact with oil). Interaction between the domains can be modelled in a few ways developed so far.

In theory, three methods of FSI modelling can be distinguished. They are divided into two groups, under the monolithic and the partitioned approach. The grouping of the methods is illustrated in Fig. 2.

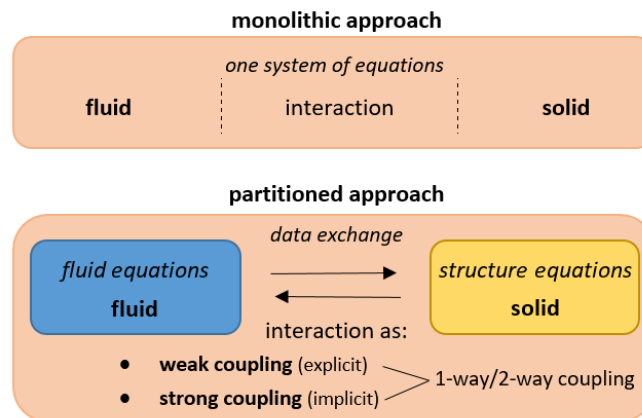


Fig. 2. FSI coupling methods

The monolithic approach to FSI is based on a single extended system of differential equations covering the fluid, the structure physics and their interaction ². This method is computationally expensive and difficult to linearize. It may also lead to convergence problems. The advantage is the lack of added-mass instabilities. Examples of the successful application of the approach are well described in ³, with their advantages and limitations, and then developed further in ⁴.

The partitioned approach divides the problem into individually solved sub-systems, which makes it possible to independently set up solving methods for each solver whose interaction is described by an external coupling algorithm ⁵. This method is more efficient than the monolithic approach and relatively easy to apply. However, using this method results in the added-mass effect of the fluid on the structure ⁶, which may cause numerical difficulties. The effect can be omitted for high structure/fluid density ratios ⁷. Within the partitioned approach, two methods of coupling can be specified: a weak and a strong coupling scheme. The weak coupling method is realized by solving the fluid and the structure once per time step based on a single data exchange between the two. The solution is not convergent between the sub-systems and the time steps in the systems may not be the same. The strong coupling approach means an iterative data exchange between solvers until convergence criteria are reached in each time step. This difference indicates that weak coupling can be considered an explicit method and strong coupling can be considered an implicit one.

The selected approach can further be described as one- or two-way coupling. One-way coupling means that solvers export data, but do not receive results in return. Two-way coupling is needed when the structure immersed in the fluid domain is highly deformed due to flow conditions. The data transfer is realized in two directions. Based on the structure deformation, the fluid pressure and the velocity field are updated and exert an impact on the structure domain in return.

The physical phenomena behind FSI are described by continuum mechanics equations. Therefore, the structural problem relations are material law, kinematics and the equilibrium conditions. The fluid dynamics is calculated based on four differential equations obtained from the momentum conservation (Navier-Stokes) equations (in three dimensions) and from the mass conservation equation. This set of equations completely describes an incompressible and isothermal fluid, which can be assumed for the purpose of this study and many other applications.

The differences in the FSI application results, depending on the needs and the selected approach, can be seen clearly by comparing the following studies on the shock absorber check valve. The coupled scheme is used in ⁸, which means that the flow and the structure calculations were performed independently. The results were then combined manually to identify the real points of operating conditions assuming the quasi-static behaviour of the valve. The authors claim to have achieved a good correlation with the physical test. However, the method is applicable only if the valve disc's position can be forecasted well and no dynamic effects are expected (low accelerations).

In contrast, the study of a check valve presented in ⁹ was conducted using strongly coupled fluid-structure interaction. As a result, the authors obtained a dynamic pressure response of the valve to sinusoidal flow excitation. The presented model reflects the valve disc deflection, and the disc opening

at a real flow rate is included in the results. As expected, a delay can be observed in the disc deflection due to the helical spring preload. The pressures achieved in the simulation were successfully used in further calculations of the piston rod oscillations and related to noise-generation issues. The coupled method described by the authors can be used successfully in a wide range of excitation functions and may yield reliable results in terms of both static and dynamic performance of the valve and identification of the structure loads. This type of study spurred other authors to perform FSI simulations of the clamped valve design.

In this work, the performance of a piston valve with the clamped design will be predicted using the strongly coupled partitioned FSI approach. The aim of this study is to develop a comprehensive modelling method, making it possible to calculate a variety of stack compositions and capture the impact of 3D geometric features without having to introduce excessive changes in the FSI model configuration. Currently performed investigations of clamped disc valves, such as 10 and 11, are focused on a mathematical description of the structure response and have very limited adaptation capabilities, without the possibility of resolving the effects associated with 3D geometry.

2. NUMERICAL MODELLING

The numerical models were prepared using ANSYS 2019 R3 software. The modelled object was a piston valve with concentric disc support surfaces, referred to hereinafter as “lands”. Two sets of discs were considered to check if their performance curves were in line with the expectations and the measurements. The general dimensions of the components are listed in Table 1.

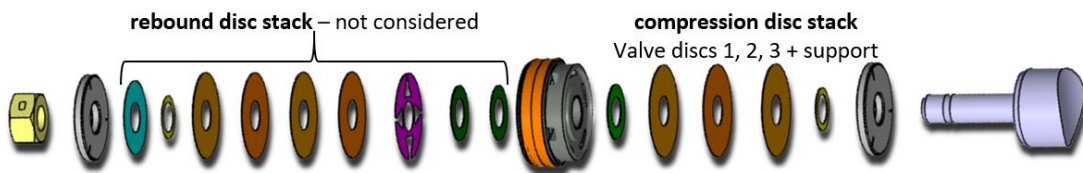


Fig. 3. Exploded view of the valve stack composition

Table 1

Modelled disc stack composition

Component:	Low stack (Case1)	High stack (Case2)
	disc outer diameter, mm	
Support disc	18.0	18.0
Valve disc 3	28.0	28.0
Valve disc 2	30.0	30.0
Valve disc 1	-	30.0

All the discs were solid and axisymmetric without any notches or holes (cf. Fig. 3 – discs 1, 2 and 3 in the compression disc stack). According to the piston cyclic symmetry, the prepared model was reduced to a 1/12 cyclic part consisting of half of the piston channel and half of the inter-channel area.

2.1. Structural model

The structural model was prepared and calculated using the Ansys Mechanical preprocessor and solver. The valve discrete model consists of a hexahedral mesh for flexible discs, where 4 elements are used across the disc thickness. All the components were meshed using the sweep method along the circumference to ensure a high-quality index of the element, above 0.7. The piston deflection is insignificant and was not considered. The model was simplified to hub and land rigid bodies, fixed on their bottom surfaces.

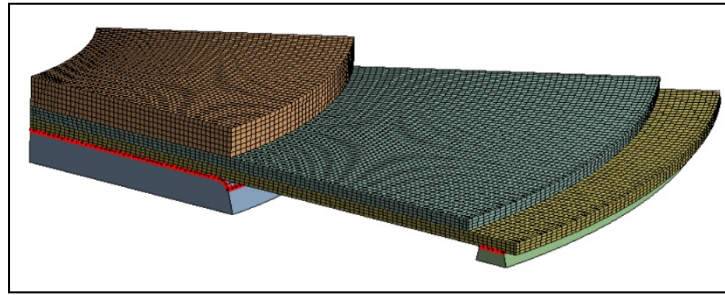


Fig. 4. Valve model structural mesh

The discs are made of spring steel and are expected to operate in the range of elastic material properties at the considered loads (the yield stress for such steel is about 1500MPa). Young's modulus and Poisson's ratio are 210GPa and 0.3, respectively.

The valve assembly is tightened with a specified torque applied to the nut. In the model, the 14.0kN axial force was applied on the top surface of the support disc. Such a force corresponds to the standard torque used for the considered valve type. Due to the difference (0.10mm) in height between the piston hub and land, the discs are initially deformed. As a result, the valve preload is achieved. This is reflected on the flow characteristic curve as delayed initiation of the opening and a shift of the curve towards higher pressures.

2.2. Fluid model

The modelled flow is isothermal and incompressible. It can be fully described using Navier-Stokes Eq.(1) and continuity equations Eq.(2), Eq.(3) 12. The assumption of incompressibility results in a constant value of density ρ over time. These equations with the Eulerian-view approach are commonly used by CFD solvers.

The Navier-Stokes Equation is

$$\rho \partial v_i / \partial t + \rho v_j \partial v_i / \partial x_j = -\partial p / \partial x_i + \mu \partial^2 v_i / \partial x_j \partial x_j + \rho b_i \quad (1)$$

where ρ is the fluid density, v is the flow velocity, μ is the dynamic viscosity, p is the pressure, t is the time and b_i represents the external forces.

The continuity equation and the incompressibility condition are as follows:

$$\partial \rho / \partial t + \rho \nabla \cdot v = 0 \quad \text{and} \quad \partial \rho / \partial t = 0 \quad (2)$$

Hence, mass conservation is described by zero divergence of the velocity field:

$$\nabla v = 0 \quad (3)$$

The CFD model was prepared using ANSYS Fluent software. The oil domain was prepared by extraction of the valve components' geometry from the oil volume inside the damper tube. The volume was divided into 3 regions. The inflow and the outflow regions were separated and modelled as static meshes to provide an accurate flow profile in far fields. These volumes were swappable bodies, meshed using a structural mesh of hexahedral elements to reduce the overall size of the discrete model. The oil volume in contact with the valve discs that has to handle their deflection was covered with an unstructured mesh composed of tetrahedral elements. The discrete model view of the fluid domain is presented in Fig. 5. The gap of 0.01 mm between the disc and the piston land was retained to reflect surface roughness and manufacturing tolerances. Such a pre-filled gap makes it possible to tackle the challenging aspect of closed contact between surfaces, filled with oil after opening. A similar approach was used successfully by authors of 9 for the same purpose. The gap size was assumed to be small enough to have no relevant influence on the flow.

To ensure a mesh-independent solution, the mesh size was adequately checked. Several mesh sizes were investigated with a steady-state CFD case. The adopted final size of the element provided the best relation between the computation time and the influence of the mesh on the results, considered as the pressure drop variation of less than 1% between the mesh models.

The next step was to define the mesh-deformation method, which may be critical for the solution convergence. For this purpose, 2 mesh-deformation methods were enabled: smoothing and remeshing.

Smoothing is the basic method to adjust the mesh while the boundary surface is deforming or moving. The surface nodes lying exactly on the boundary are moved to be in line with the new surface position or shape. Their displacement affects the position of neighbouring internal nodes according to the assumed smoothing model and its parameters. The ANSYS documentation recommends the diffusion-based model for Fluent-Mechanical FSI 13. However, the authors found that the *Spring* model performed better and more intuitively in defining parameters for the investigated case. Due to the high deformation of the mesh in comparison with the element size, a remeshing operation had to be used. Both Region and Local Remeshing methods were enabled to allow changes in the mesh density based on length scales and element skewness. The selected combination of spring smoothing and remeshing is a commonly used dynamic mesh technique, implemented in 22 in the partitioned FSI case of a pump check valve.

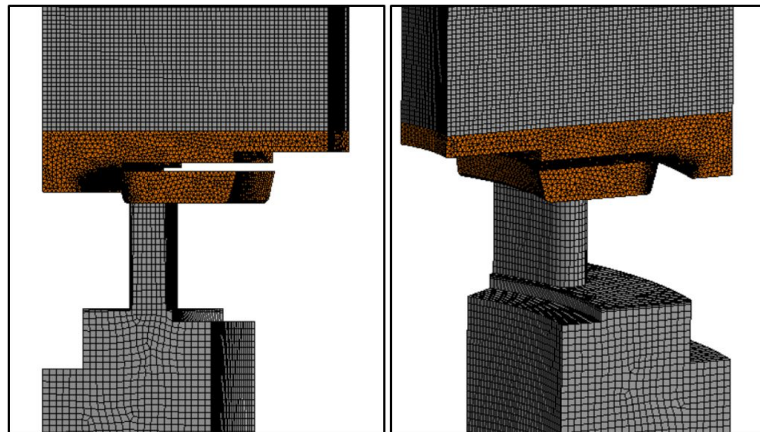


Fig. 5. Fluid domain mesh

The parameters of the oil used in the simulations were defined based on the material specification, according to the values listed in Table 2. This incompressible, viscous Newtonian fluid is assumed to have temperature-independent density and viscosity. Such an assumption seems to be suitable for the type of study performed and is widely used in research on damper valves (17 and 18). A velocity boundary condition was set at the inlet and defined using a UDF external file (user-defined function). Its value was linearly ramped in time up to 2.3 m/s, which corresponds to the maximum possible flow rate for a test bench where some of the measurements were carried out. The flow through the valve was considered in one direction, corresponding to the damper compression stroke. Therefore, the valve behaviour during unloading was not considered.

Based on previous studies 15, the realizable $k-\varepsilon$ turbulence model was selected. It is a variant of the standard two-equation $k-\varepsilon$ model – one of the most common options used in industrial CFD applications 13, 19. The realizable $k-\varepsilon$ turbulence model is an improved standard model for strong vortices and stream curvatures 16, recommended from the $k-\varepsilon$ family. In addition, Enhanced Wall Treatment was enabled to make the viscous model less sensitive to the near-wall mesh resolution. Adequate near-wall element size is evaluated using a dimensionless y^+ parameter, whose value depends on the first element height, local flow velocity and fluid viscosity. Where the mesh is fine enough, $y^+ \approx 1$, a traditional two-layer zonal model is used to predict the boundary layer behaviour. For a coarser near-wall mesh, an enhanced wall function formulation is implemented 14. This method makes it possible to overcome the computational requirements imposed by the standard near-wall resolving turbulence model, which requires a sufficiently fine mesh everywhere 21.

A non-slip condition was applied to the fluid domain external walls. This condition ensures that the fluid velocity is assumed to be zero in relation to the structural boundary surface. Furthermore, this boundary condition makes it possible to generate the near-wall boundary layer.

2.3. Fluid-Structure Interaction

To achieve the aim of this study, the 2-way FSI approach was used. The structure and the fluid subsystem were strongly coupled with the implicit method. As a result, the solution is convergent in each time step (the time step in the two systems must be the same).

Table 2

Fluid physical properties

Property	Value
dynamic viscosity	0.015 Pa·s
density	840 kg/m ³
reference temperature	25 °C

FSI simulations require interface identification. The structure surfaces taken into account in the FSI data exchange are selected in the ANSYS Mechanical solver as the Fluid Solid Interface. Then, using the ANSYS System Coupling tool, they are linked to the set of the fluid surfaces in the Data Transfer tab. These surfaces may not match up completely; coupling will proceed, and feedback is given on what percentage of the domain has been used in the data transfer. In this way, interfaces are created at which dynamic and kinetic conditions must be satisfied.

The kinematic condition is that the structure deflection velocity and the fluid velocity are equal, as expressed by

$$v_f = v_s \quad (4)$$

The dynamic equilibrium condition ensures the balance of forces acting on the FSI interface, which is defined as stress equilibrium and expressed as

$$\tau_f n = \tau_s n \quad (5)$$

where $\tau_{f/s}$ is the stress tensor and n is the distance.

In the Workbench system coupling interface, convergence criteria can be specified, or the number of data exchange iterations can be limited. For the study in question, the number of data exchange iterations in each time step was limited to 10, which was accepted as the optimal relation between the computation time and the level of convergence.

3. TEST BENCH

The numerical models were validated on the test bench used in 15. The test bench enables the measurement of flow restrictions through a variety of designs of the shock absorber valve. The measurement method guaranteed obtaining pressure values generated by a valve isolated from the damper system. Such an approach is intended to eliminate factors that could affect the comparison, such as the damper pipe deformation, the rod-sealing friction and the uncertainty of the damper gas fill pressure. The above factors are not considered in numerical models and may have an undesirable effect on results.

The isolated valve performance was measured using a metal column corresponding to the inner diameter of the damper tube in which the valve was placed. It was initially fastened to a purpose-made section of the rod shortened to fit the chamber. This assembly was closed inside the column with a sealed nut as shown in Fig. 6b. Two channels were connected to the column, above and below the valve, through which the oil was pumped. The stand was equipped with a three-way valve to enable easy control of the flow direction. The test bench hydraulic system used the same oil as the actual shock absorbers. The flow was provided by two pumps working in series. The pressure drop on the tested valve was measured on the inflow and on the outflow channel.

Two compression valve settings were assembled according to the information in Table 1. The rebound valve was closed with a set of solid discs to prevent leakage through the rebound piston channels. Each of the two valve assemblies was measured three times. The tested specimens were

twisted with a specific identical torque applied to the nut. The measurement was carried out in the range of the flow volumetric rate of 0-90 l/min. Ramped flow excitation took approximately 2.0s to reach the maximum flow. As a result, a p-Q (pressure-flow rate) diagram of the steady-state pressure drop against the volumetric flow rate was obtained. After the valve measurements, an additional measurement was carried out of the flow resistance of the test bench itself (with no valve in the metal column). It must be taken into account and subtracted from the measurement results to obtain the real values of the pressure drop.

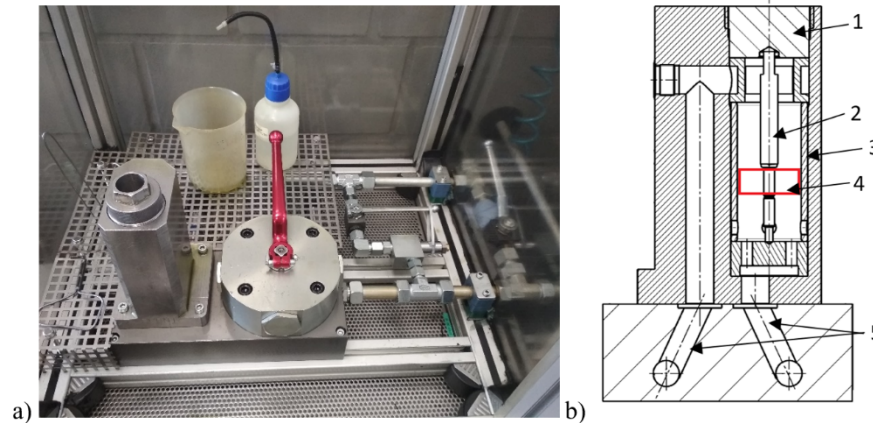


Fig. 6. a) Flow test bench, b) cross-section of the metal column: 1 – sealed nut, 2 – pin (shortened rod), 3 – metal sleeve (damper tube diameter), 4 – piston valve location and 5 – inflow/outflow channel 15

The measurement results of the valve settings (Case 1 and Case 2) are presented in Fig. 7 with the tooling restriction curve. Good repeatability of measurement results was observed between specimens of the same assembly (no differences can be seen in the graph between the specimens).

4. RESULTS AND DISCUSSION

The measured pressure drop characteristics of three specimens were averaged for each valve assembly. Before a comparison with the FSI model, the tooling restriction was extracted from the total pressure drop curves. The valve performance characteristics corrected by the tooling restriction are presented in Fig. 8.

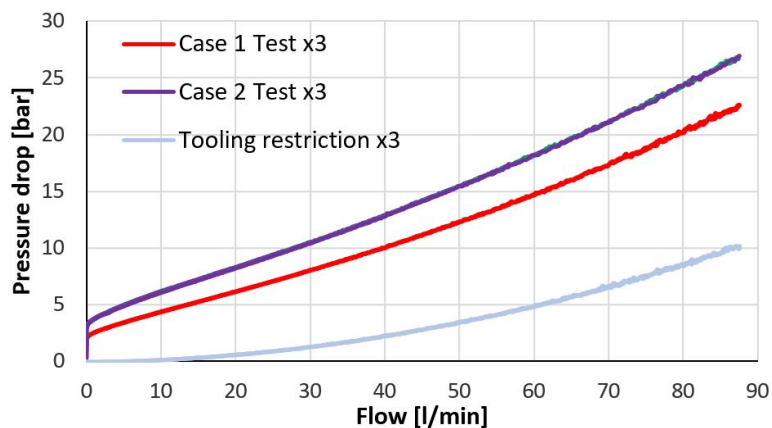


Fig. 7. Pressure drop measurement results – Case 1 and Case 2, and the tooling restriction curve

Because there are no bleed discs in the disc stack, the p-Q curves are linear. The pressure initial value at zero flow is due to the preload force, which must be overtaken by the pressure force to trigger the flow. As expected, a stiffer disc stack has a higher blow-off point and generates ~25% higher pressures.

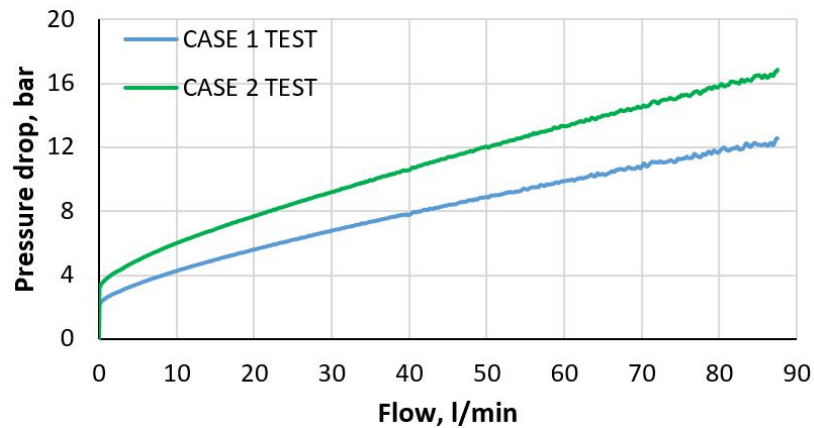


Fig. 8. Measured valve performance curves, p-Q characteristics

The correlation between the pressure drop curves obtained from the FSI simulations and the experimental testing results is shown in Fig. 9. The achieved consistency in the results is satisfactory, with the discrepancy level lower than 10%. The slope of the performance curves and the relation between the soft and the stiff disc stack were well predicted by the numerical model. The authors of 20 predicted the performance of a piston valve consisting of one disc using the coupled approach and achieved a similar level of correlation, although a very coarse mesh was used for the fluid discrete model. However, the numerical model results were overestimated compared to the obtained pressure drops. According to the authors, this might have been due to the very high stiffness of the clamped boundary condition in comparison with the real conditions. On the other hand, the coarse fluid mesh of the piston inlet region may have influenced the flow conditions, which affected the total pressure drop value.

In the presented results, it can be seen that Case 2 (green) generates slightly underestimated pressure drops, in the entire flow range, while Case 1 is in line with the experimental results. Such a tendency may indicate that too low a clamping force of the discs was taken into account or that the preload application method was inadequate.

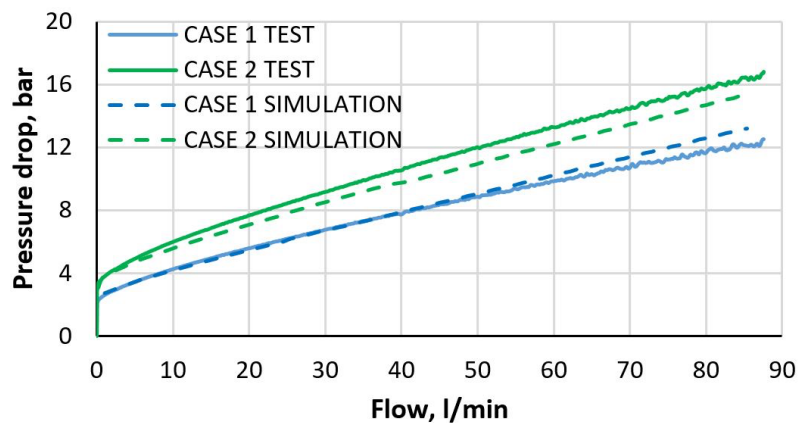


Fig. 9. Valve performance curves, pressure-flow characteristics

The observed phenomena are typical of preload loss issues of excessively tested parts, which manifest as a drop in damping forces in the entire range of flow rates. It should be expected that the discrepancy will increase with the number of discs and, consequently, with the disc stack stiffness. To overcome this problem, the valve clamping should be realized using the bolt pretension method instead of applying a force. Such an approach would reflect the preload from the nut torquing irrespective of the disc stack stiffness. Unfortunately, the Bolt Pretension feature available in ANSYS cannot be used for the FSI simulation due to the one-step-only limitation in defining mechanical loads.

Properly defined pretension requires at least 2 steps to apply a load and lock the bolt displacement. For this reason, a new approach must be developed for the valve preloading (discs clamping).

Besides the valve performance, its strength can be evaluated based on the structural model results. Due to the FSI coupling, stresses on the valve each disc can be identified as a function of the flow rate. However, most physical tests of the damping level are controlled by a specific velocity of the rod. In such a case, to facilitate the interpretation of results, the oil flow rate can be connected to the rod velocity using formula (6):

$$\dot{Q}/(A_{bore} - A_{rod}) = v_{rod} \quad (6)$$

where: A_{bore} – tube inner surface area; A_{rod} – rod cross-section surface area; \dot{Q} – volumetric flow rate, v_{rod} – damper rod velocity

The considered flow rate range can be related to rod velocities of 0-2.2m/s, which are commonly used damper velocities in road conditions. Within this range of velocities, some example results of maximum von Mises stresses are plotted in Fig. 10.

The discs work within elastic material properties, which is intended to maintain acceptable design durability. A bend can be noticed on each stress curve, at 1.1m/s for Case 1 and 1.2m/s for Case 2. This non-linear behaviour is recognized as the moment of change in the disc bending mode. Below the above-mentioned rod velocities, the discs open symmetrically around the circumference. Higher flows lead to buckling due to compression of the disc material on the suction side. As a result, the disc acquires a wavy shape in the circumferential direction. The outer disc edge measured deflection can be observed in the graph presented in Fig. 13. This phenomenon is additionally shown in Fig. 11 and Fig. 12 as an uneven stress distribution along the disc stack and the disc stack deflection at high rod velocities.

The shape of deflected discs is determined by the piston channel position and thus by the uneven pressure load distribution. As the flow rate increases, the effect of the piston channel position on pressure conditions becomes stronger. An example pressure distribution on the disc bottom surface at the rod velocity of 2.2 m/s is presented in Fig. 14. The pressure field is partially covered with the piston geometry to better visualize the relation between the channel and pressure concentration.

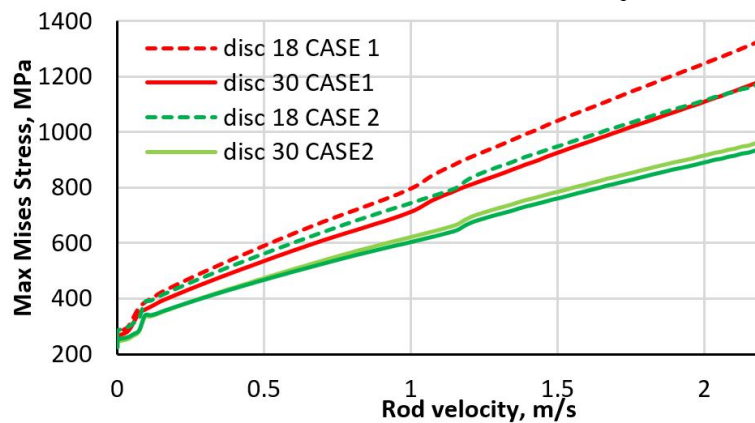


Fig. 10. Maximum von Mises stresses on the disc

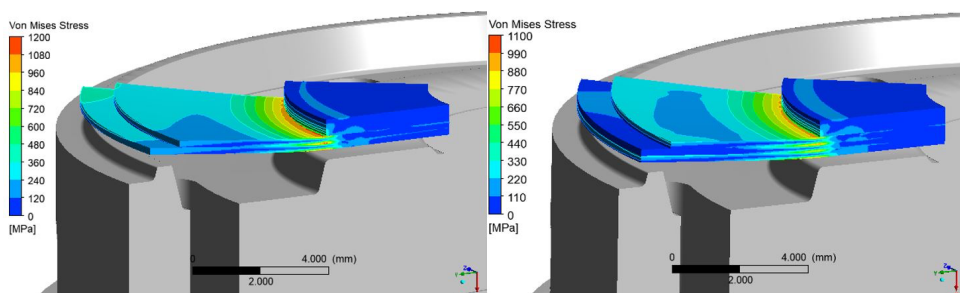


Fig. 11. Stress distribution over the disc stack at 2.2m/s rod velocity (left – Case 1, right – Case 2)

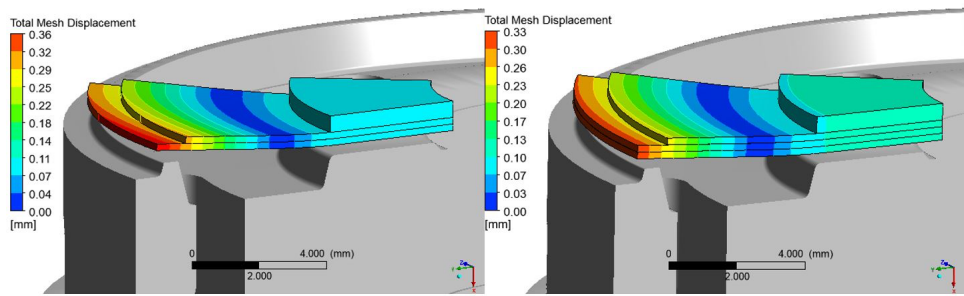


Fig. 12. Disc deflection at 2.2m/s rod velocity (left – Case 1, right – Case 2)

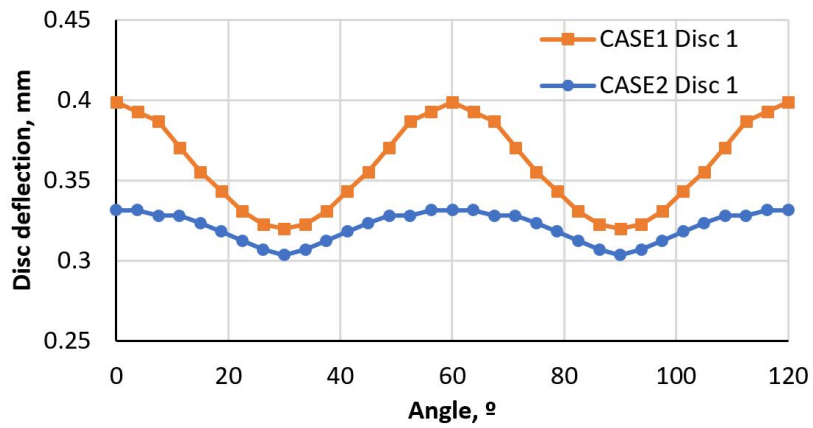


Fig. 13. Circumferential deflection of Disc 1 (Case 1/Case 2) – rod velocity: 2.2m/s

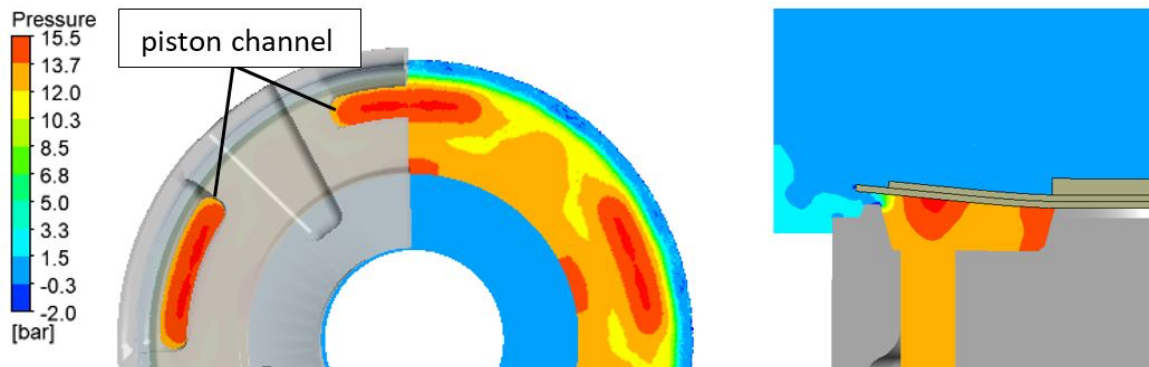


Fig. 14. Pressure distribution on the disc bottom surface (piston channels indicated) and cross-section through the channel

The distribution of pressure acting on the disc becomes complex as the flow rate increases. For rod velocities exceeding 1.5m/s, the influence of the outflow from the channels can be seen in Fig. 14 as fields of high pressure on the disc surface. The flow velocity distribution obtained from the models is presented in

Fig. 15. The identified pressure conditions may become the basis for the piston shape optimization for homogeneous or controlled distributions to achieve the desired blow-off point or smooth opening behaviour.

5. CONCLUSIONS

This paper presents a strongly coupled FSI model of the shock absorber piston valve. A satisfactory correlation was achieved between the valve performance curves generated numerically and those

obtained from experimental tests. However, a new method of the valve preload application was indicated to be developed to achieve a better pressure drop prediction. The structural results show that the stresses arising on the discs did not exceed the Yield point, which means that the discs operated within elastic material properties. Such results are in line with the post-testing visual assessment of the state of the components, which showed no plastic deformations of the discs.

The analysis performed did not cover bleed discs or the disc opening limiters, which are commonly used. For this reason, the next step of the research will naturally be to include these aspects to generate characteristics sensitive to all valve features.

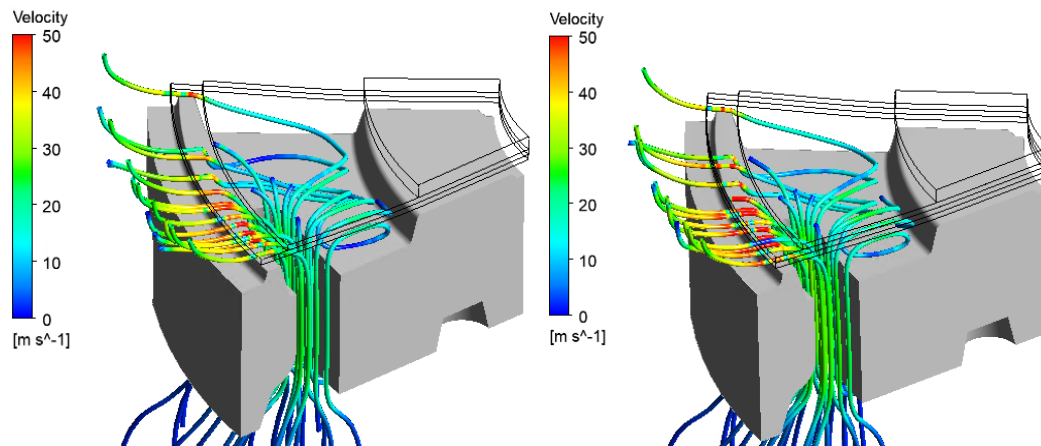


Fig. 15. Flow streamlines from the models at a rod velocity of 2.2 m/s (left – Case 1, right – Case 2)

It is proved that the proposed approach has potential to predict the valve performance and estimate the structural loads. The implementation of this approach may lead to added value in the development stage to analyse concept designs that are not physically available or to simulate conditions that cannot be tested at the moment. This will lead to significant financial benefits and may shorten the product development phase.

6. ACKNOWLEDGMENTS

This work was conducted in collaboration with Tenneco Inc. Company and funded by the Silesian University of Technology, grant no. BK-233/RIE5/2021. The research was performed as part of the Industrial Doctoral Program and co-financed under grant no. 11/DW/2017/01/1 supported by the Ministry of Science and Higher Education in Poland.

References

1. Moser, A. & Schweiger, R. Prospects & barriers for up-front CAE simulation in the automotive development. *Virtual Vehicle Research GmbH (ViF)*. No. 98830. P. 1-12.
2. Becker, P. & Idelsohn, S.R. & Oñate, E. A unified monolithic approach for multi-fluid flows and fluid-structure interaction using the Particle Finite Element Method with fixed mesh. *Computer Mechanics*. 2015. Vol. 55. P. 1091-1104.
3. Ryzhakov, P.B. & Rossi, R. & Idelsohn, S.R. & et al. A monolithic Lagrangian approach for fluid-structure interaction problems. *Computational Mechanics*. 2010. Vol. 46. P. 883-899.
4. Ryzhakov, P. A modified fractional step method for fluid-structure interaction problems. *Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería*. January-June 2017. Vol. 33. Nos. 1-2. P. 58-64. [In Spanish: *International Journal of Numerical Methods for Engineering Calculation and Design*].

5. Hou, G. & Wang, J. & Layton, A. Numerical methods for fluid-structure interaction – a review. *Communications in Computational Physics*. August 2012. Vol. 12. No. 2. P. 337-377.
6. Causina, P. & Gerbeau, J.F. & Nobile, F. Added-mass effect in the design of partitioned algorithms for fluid-structure problems. *Computer Methods in Applied Mechanics and Engineering*. 15 October 2005. Vol. 194. No. 42-44. P. 4506-4527.
7. De Nayer, G. & Apostolatos, A. & Wood, J.N. & Bletzinger, K.U. & Wüchner, R. & Breuer, M. Numerical studies on the instantaneous fluid-structure interaction of an air-inflated flexible membrane in turbulent flow. *Journal of Fluids and Structures*. October 2018. Vol. 82. P. 577-609.
8. Shams, M. & Ebrahimi, R. & Raoufi, A. & Jafari, B.J. CFD-FEA analysis of hydraulic shock absorber valve behavior. *International Journal of Automotive Technology*. October, 2007. Vol. 5. No. 8. P. 615-622.
9. Shu, H.-Y. & Guo, C. & Luo, S. & Wang, M.-M. Fluid-structure interaction of shock absorber for structure borne noise based on compensation valve opening. In: *ISBDAI '18: Proceedings of the International Symposium on Big Data and Artificial Intelligence*. December 2018. P. 129-134.
10. Xu, J. & Chu, J. & Ma, H. Hybrid modeling and verification of disk-stacked shock absorber valve. *Advances in Mechanical Engineering*. 2018. Vol. 10(2). P. 1-12.
11. Skačkauskas, P. & Žuraulis, V. & Vadluga, V. & Nagurnas, S. Development and verification of a shock absorber and its shim valve model based on the force method principles. *Eksplotacija i Niezawodność – Maintenance and Reliability*. 2017. Vol. 19(1). P. 126-133.
12. Gryboś, R. *Mechanika płynów* Silesian University of Technology. University. Scripts No. 1966. Gliwice. [In Polish: *Fluid mechanics*].
13. ANSYS Fluent User's Guide Release 2019 R3. ANSYS Inc. Jun. 2020.
14. ANSYS Fluent Theory Guide Release 2019 R3. ANSYS Inc. Jun. 2020.
15. Buczkowski, D. & Nowak, G. Increase in Tuning Ability of a Car Shock Absorber Valve using CFD. *Journal of Applied Fluid Mechanics*. 2019. Vol. 12. No. 6. P. 1847-1854.
16. Shih, T.-H. & et al. A new k- ϵ eddy viscosity model for high Reynolds number turbulent flows – Model development and validation. *Computers & Fluids*. 1995. Vol. 34(3). P. 227.
17. Guzzomi, F.G. & O'Neill, P.L. & Tavner, A.C.R. Investigation of Damper Valve Dynamics Using Parametric Numerical Methods. *School of Mechanical Engineering*. University of Western Australia. 6009 Australia. 2007.
18. Pelosi, M. & Subramanya, K. & Lantz, J. Investigation on the Dynamic Behavior of a Solenoid Hydraulic Valve for Automotive Semi-Active Suspensions Coupling 3D and 1D Modeling. In: *The 13th Scandinavian International Conference on Fluid Power*. June 3-5. 2013.
19. Launder, B. & Spalding, D. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*. 1974. Vol. 3(2). P. 269.
20. Le Talleca, P. & Mourob, J. Fluid structure interaction with large structural displacements. *Computer Methods in Applied Mechanics and Engineering*. March, 2001. Vol. 190. Nos. 24-25. P. 3039-3067.
21. De Santis, D. & Shams, A. Scaling of added mass and added damping of cylindrical rods by means of FSI simulations. *Journal of Fluids and Structures*. July, 2019. Vol. 88. P. 241-256.
22. Menéndez-Blanco, A. & Manuel, J. & Oro, F. & Meana-Fernández, A. Unsteady three-dimensional modeling of the Fluid-Structure Interaction in the check valves of diaphragm volumetric pumps. *Journal of Fluids and Structures*. October, 2019. Vol. 90. P. 432-449.