Modelling Fluid-Structure Interaction in a Pressure-Controlled Current-Limiting Valve

L. Fromme, N. Mügge, M. Petry, A. Waschke
Department of Engineering Sciences and Mathematics, Bielefeld University of Applied Sciences, Bielefeld, Germany

Abstract: This assignment deals with the modelling of a pressure-controlled current-limiting valve. For this purpose, the CAE-Software COMSOL Multiphysics® is used. Current-limiting valves are used as a measure of safety in many hydraulic applications. They are released in cases of a high decrease in pressure due to a burst pipe or leaks. The chief aim of this project is to find a beneficial modelling approach for the closing operation in order to get a detailed view of the mass-flow rate.

Keywords: Current-limiting valve, fluid-structure interaction (FSI), plate valve, burst pipe protection, gap modelling.

1. Introduction

As a part of safety engineering, current-limiting valves are used to avoid pipe bursts. Due to the safety aspect, there are high requirements set to the technical reliabilities.

If the local pressure gradient at the valve is increased, the gap will be closed to disturb the flow. This paper deals with the numerical analyzing of the closing operation regarding the fluid-structure interaction. Therefor the main aspects of the model structure and the modelling of the closing operation are explained in detail. Figure 1 shows a detailed view of the current-limiting valve model.

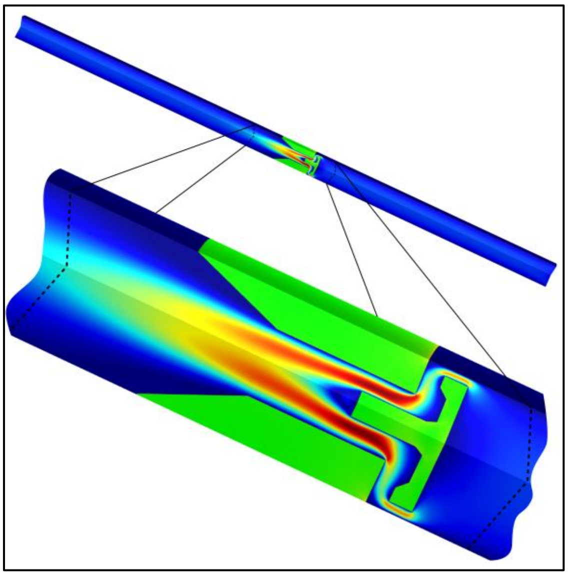

Figure 1: Current-limiting valve model

For the seat and the body of the closing valve steel has been chosen for the material specification. For the fluid, a high viscous oil is used to achieve an operating point with laminar and isothermal flow at 20°C. The interaction between the fluid and solid domains is modelled using the FSI (*Fluid-Structure Interaction*) interface provided by the CFD Module of COMSOL Multiphysics®.

Regarding the convergence, a structured mesh is beneficial. To avoid inverted cells during the closing process, the nodes are coupled to the valves displacement by using point probes. However, a complete closure cannot be achieved due to the mesh deformation, which would lead to a singularity. To minimize the mass flow through the closing gap, the viscosity is increased as a function of the clearance. The results of the transient solution are compared with steady state solutions of different back pressures using the parametric sweep study function.

2. Theory

The fluid-structure interaction is a coupling of fluid mechanics and structural mechanics. The interaction between the solid and the fluid takes place on the interface and works in both directions (two-way coupling). For the fluid, the Navier-Stokes equation for incompressible fluids

$$\rho_F\left(\partial_t\vec{v}_F + (\vec{v}_F\cdot\vec{\nabla})\vec{v}_F\right) = \vec{\nabla}\cdot\left\{-p\hat{I} + \mu\left[\vec{\nabla}\vec{v}_F + \left(\vec{\nabla}\vec{v}_F\right)^T\right]\right\} + \vec{F} \qquad (1)$$

is solved, with ρ_F as the fluids density, $\vec{v}_F$ the velocity of the fluid, p for the pressure, μ as the fluids dynamic viscosity and $\vec{F}$ as the sum of the external forces. Furthermore, for fluids with constant density the continuity equation can be expressed in the following form:

$$\vec{\nabla}\cdot\vec{v}_F = 0. \qquad (2)$$

The tension

$$\vec{f} = \vec{n}\cdot\left\{-p\hat{I} + \mu\left[\vec{\nabla}\vec{v}_F + \left(\vec{\nabla}\vec{v}_F\right)^T\right]\right\} \qquad (3)$$

which acts on a surface element of the solid with the normal vector $\vec{n}$, arises from the fluids stress tensor. Under consideration of the acting volume forces $\vec{F}_V$ the displacement $\vec{u}_s$ of the valve body can be

Excerpt from the Proceedings of the 2017 COMSOL Conference in Rotterdam

calculated from the stress tensor $\hat{\sigma}_s$ of the solid domain in equation (4), with ρ_s as the solids density:

$$\rho_s \partial_t^2 \vec{u}_s - \vec{\nabla} \cdot \hat{\sigma}_s = \vec{F}_V. \qquad (4)$$

For the fluid, the valves movement acts as a moving wall boundary condition with the following velocity:

$$\dot{\vec{u}}_s = \partial_t \vec{u}_s. \qquad (5)$$

3. Usage of COMSOL Multiphysics®

The given task has been solved by using the FSI interface from the *CFD Module* of COMSOL Multiphysics®. The tensions are transferred at the boundaries between the fluid and the solid. To describe the fluids and solids behavior, two *Domains* with different material allocations are required. The solid is defined as a high-strength steel from COMSOL's® *Material Library*. The used *high-strength alloy steel* has a density of 7850 $\frac{kg}{m^3}$ and a Young's modulus of $200 \cdot 10^9\ Pa$. For the fluid, a high viscous hydraulic oil (ISO VG 68) with a 880 $\frac{kg}{m^3}$ density and the viscosity equals $0.2\ Pa\ s$ is defined. Due to the high viscous hydraulic oil and the small flow cross-sections, a laminar approach is used to solve the fluids equations of state. Figure 2 shows the setup of the current-limiting valve model. The different domains are colored. The solid is colored blue and the fluid is labelled grey. To avoid high amounts of computing time, the rotational symmetry is exploited to reduce the number of cells.

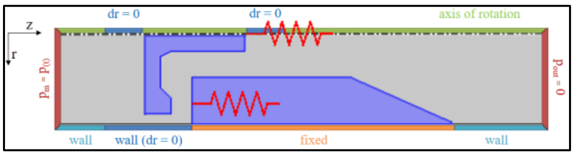

Figure 2: Boundary conditions

The axis of rotation is marked by the chain line. The spring, which controls the closing operation, affects the whole valve body as its modelled as a body force. It is shown Figure 2 in red color on the axis of rotation. The spring stiffness equals 5 $\frac{N}{mm}$. Caused by numerical issues, the complete closure cannot be modelled. Accordingly, the solid-state contact is modelled by an additional spring, which represents the materials Young's modulus. The simulation region has a length of 800 mm and the height equals 17 mm. To achieve a shake down flow at the beginning of the valves geometry, a distance of 340 mm is given. The valve is given with a 15 mm radius at the head and a 1.5 mm radius at the shaft with a length of 21 mm.

Due to the valves movement during the closing process, a moving mesh operation is used. Figure 2 shows the boundary conditions for the moving mesh as well as for the physics model. In red color, the pressure inlet and outlet are marked. Right beside the inlet on the left-hand side, a wall boundary condition (blue labelled) is chosen while on the opposite side, a rotational axis condition (green colored) is used. For mesh refinement and moving mesh operation, some additional conditions have to be determined. Those are labelled with an $dr = 0$ condition. This means, that the mesh cannot deform in the r direction. While the valve body can move in the z direction, the valve seat has a fixed constraint (orange colored).

The solutions convergence is in strong dependence to the structure of the mesh. For small gaps, a structured *mapped* mesh is used. Structured meshes can be built considering the deformation during the closing operation. Building the mesh requires some auxiliary lines as shown in Figure 3. Furthermore, at the auxiliary lines a boundary condition for the mesh deformation is defined. The nodal displacement is coupled with a *Domain Probe* on the valve body. The mesh deformation between the valve body shaft and the valve seat is determined by a *Coefficient Form Boundary PDE*.

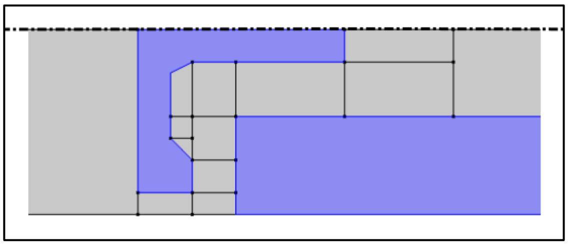

Figure 3: Auxiliary lines for mesh generation

Those regions which are not affected by the valve movement and the solids themselves are discretized with a triangular mesh (*Free Triangular*). For small gaps and small flow cross-sections, a mesh refinement is used, as shown in Figure 4. The mesh deformation caused by the valves movement, is countervailed by an initial elongation of the cells. In Figure 5 the mesh inside the closing gap is shown. The cells have a rectangular form in the default state, which nearly provides a rectangle with reversed aspect ratio in the closed state, so that the deformation of the cells is not too intense.

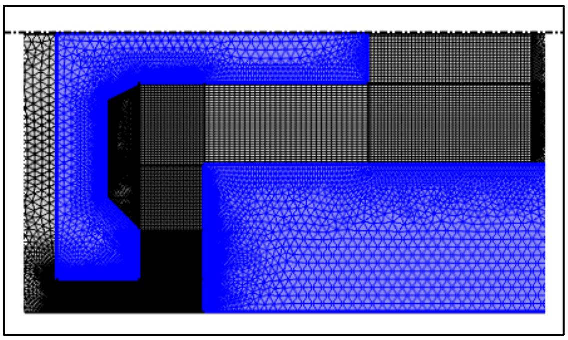

Figure 4: Mesh refinement

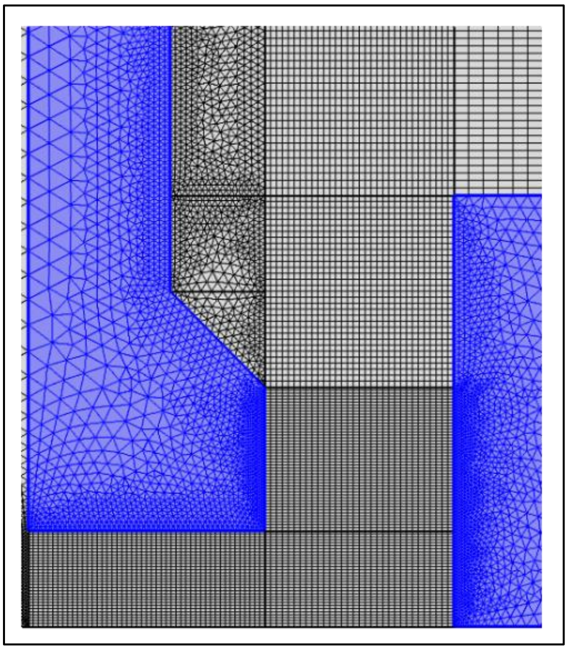

Figure 5: Mesh inside the closing gap

A full closure of the closing gap cannot be achieved, like mentioned before. If the valve would close completely, the mesh deformation would result in a single line. Such a singularity demonstrates the numerical limits. So, the closing process is approximated. The gap between the valve seat and the valve body constitutes 5 mm in the full opened state. To avoid direct contact of the valve body and the valve sear, an additional spring is modelled, which acts after a displacement of 4.95 mm. This equal 99 % of the total range. With a stiffness of $10000\ \frac{N}{mm}$, the spring essentially represents the solid-states contact and their Young's modulus. The mass flow through the left gap is limited by increasing the viscosity inside the gap as a function of the displacement. Therefore, an additional *Domain* is modelled in between the valve body and the valve seat as shown below in Figure 6. The increase in viscosity starts, if the clearance is below 1 %.

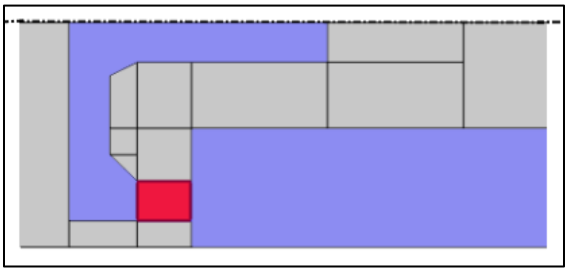

Figure 6: Fluid domain for a local increase in velocity

The fluid stream is pressure controlled. Both, the inlet and the outlet have Dirichlet boundary conditions regarding the pressure. While the pressure at the outlet is constant at 0 Pa, the inlet pressure is given by a time dependent function. The function is designed to represent an opening process as well as a closing operation. Figure 7 shows the time dependent function for the pressure boundary condition at the inlet.

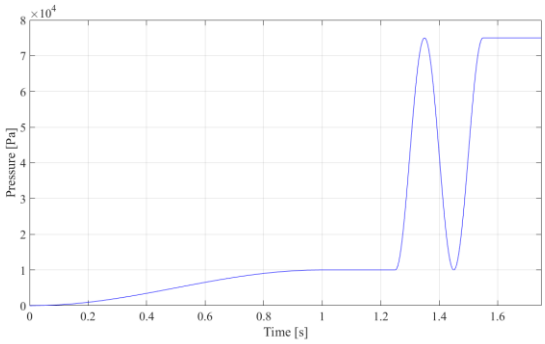

Figure 7: Time dependent function for the inlet pressure

In the first second, the pressure rises slowly by a cosine function to a target value of 10000 Pa. Afterwards for 0.25 s the pressure is constant. Then a cosine formed surge over 0.2 s with an amplitude of 75000 Pa acts on the inlet. At the end of the function, the pressure rises to 75000 Pa and remains at that value. With this probe function, the systems behavior is tested in case of a short pressure pulse and whether the valve remains in closed position while the pressure reaches constant high values. To observe the given pressure condition, the *Suppress Backflow* function inside COMSOL Multiphysics® must be deactivated for the inlet and outlet. Otherwise the dynamic portions would be considered. For the contact between the fluid and the solid bodies a fixed boundary condition without slip is set, so that the fluidal velocity is set to the walls velocity of the valve body. As the valve body is not fixed, its movement is given by Newton's second law of motion as shown in eq. (6):

$$\sum_i \vec{F_i} = m\vec{a}. \qquad (6)$$

The forces acting on the valve are the spring force, friction forces and pressure forces which results from the circulation of the valve body. Under the acceptance that the valves mass is constant, the valves acceleration can be calculated from eq. (6).

The given problem is solved in a transient simulation due to dynamic influences and effects of inertia. The simulation time to solve is 1.75 s and the maximal time step is set to 0.001 s. To guarantee a resolution of the high accelerations during the impact on the valve seat, an adaptive time-step regulation is chosen. During the simulation, a minimum time-step in the dimension of 10^{-8} s is observed.

To achieve better convergence, the solver is set to *Nonlinear-Newton*. This solver provides good results for the nonlinear Navier-Stokes equation. The current-limiting valves spring has a damping coefficient of 2.5 $\frac{Ns}{m}$ to provide numerical stability. Additionally, the program function *Anderson Acceleration* is activated to shorten the calculation time. The *Anderson Acceleration* uses the solution of the previous time-step to solve the current time-step.

4. Results

In the following evaluation, the results of the simulation are considered. Besides, there will be a closer look at the valves displacement, velocity and acceleration. In addition, the time behavior of the mass flow is regarded to compare the results of transient solution to a series of steady state simulations at different valve displacements.

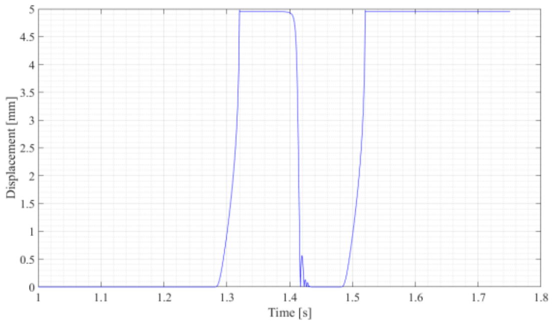

Figure 8: Time-Displacement diagram

In Figure 8 the valves displacement over time is shown. The movement of the valve body begins at $1.28\,s$ and is delayed compared to the excitation function. This is caused by the oils inertia. The closing operation takes place within four tenths seconds. Besides, an overshooting up to $4.98\,mm$ is observed. On the one hand, this is due to the inertia of the accelerated valve body. On the other hand, there is an increase in velocity during the closing operation by which the pressure must decrease regarding the Bernoulli equation. Thereby an undertow arises which accelerates the closing process. Afterwards the gap remains closed for $0.07\,s$ until it is opened again. With the impact on the stop, the system gets in oscillations. Besides, the gap closes up to $0.5\,mm$ after the first impact and opens again. The oscillation process ends after six cycles. After $0.05\,s$ in the opened state, the closing operation starts again. At the end of the simulation the valve remains in the closed state at the position of $4.9501\,mm$.

Figure 9 shows the development of the velocity over time. Positive values of the velocity correspond to a closing process and negative values correspond to an opening movement. At $1.28\,s$ the velocity of the valve body increases. After an intense rise at the beginning, the velocity increases moderately at $1.3\,s$. At the end of the closing process the velocity rises strongly which is caused by an undertow referring Bernoulli. Besides, a maximum of $1.137\,\frac{m}{s}$ is reached. The overshoot is to be recognized by the negative rash at $1.32\,s$. After $1.4\,s$ the openings process begins, whereas the valve is accelerated to a speed of $-0.819\,\frac{m}{s}$. The amplitude of the impact oscillation reaches a value of $0.605\,\frac{m}{s}$. The six oscillations after the impact on the opening stop are clear to recognize in Figure 9. At $1.48\,s$ the last closing operation begins after which the valve body remains at rest.

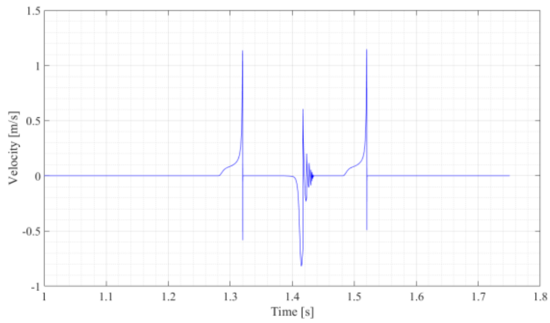

Figure 9: Time-Velocity diagram

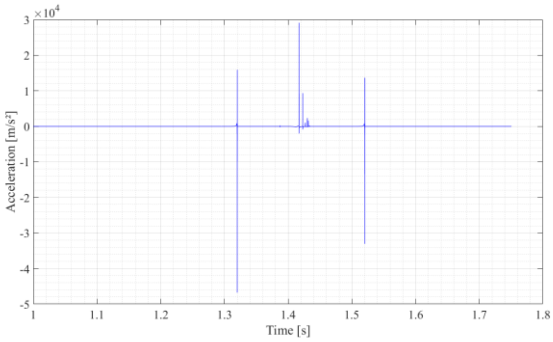

Figure 10: Time-Acceleration diagram

Figure 10 shows the acceleration acting on the valve over time. During the closing operation, the valve body is accelerated with the 85-fold of the gravity acceleration. At the impact onto the valve seat, the body is dragged with $4500\,g$. The acceleration during the opening is relatively small in comparison to the closing operation. Impacting on the opening limit stop results in an acceleration of $2950\,g$. As shown in Figure 10 the acceleration during the first closing process is even lower than in the second one. One possible reason for this is the adaptive time-step control, in the meaning of different resolution in time. This would lead in numerical inaccuracy, because the acceleration is a derivative, with respect to time.

Figure 11 shows the overall mass flow through the inlet over time. In the first stage, a mass flow of $0.205\,\frac{kg}{s}$ passes the inlet. Between $1.258\,s$ and $1.318\,s$ the mass flow is increased due to the increase in pressure. A short period of time before the valve closes, the mass flow drops down and reaches even negative values for just a moment. The negative mass flow means that some fluid inside the closing gap is pressed back to the inlet. In the closed state, the mass flow remains at $4.77 \cdot 10^{-5}\ \frac{kg}{s}$. As the valve start the opening process, some fluid has to be displaced, which results in a negative mass flow at $1.4\ s$. Afterwards the mass flow leads to the initial mass flow of $0.205\ \frac{kg}{s}$. At the end, the flow rate increases again caused by the increase in pressure and the valve closes.

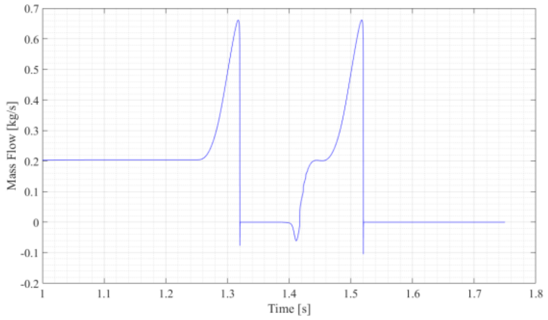

Figure 11: Time-Mass flow diagram

In Figure 12, a steady-state solution is compared to the transient solution regarding the mass flow development over the pressure difference. The red line represents the transient solution while the blue line shows different steady-state solutions at constant pressure difference condition using the *Auxiliary Sweep* function in COMSOL Multiphysics®. Both simulations are almost identical until they reach $10000\ Pa$. This occurs because the difference pressure increases slowly in the transient simulation. So that the effects of inertia are negligible until this point. At higher differences in pressure, the distinction of the curves are huge. Because effects of inertia are not considered in the steady solutions, the steady-state mass flow overlaps the transient simulation.

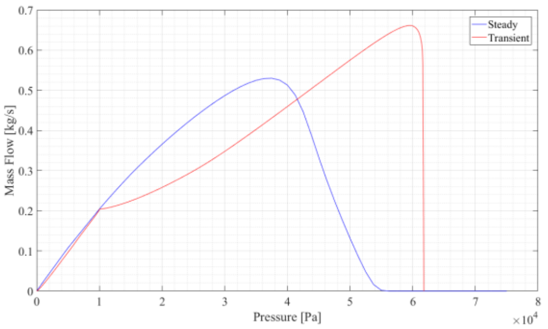

Figure 12: Pressure-Mass Flow diagram

The steady-state solution reaches its maximum of $0.53\ \frac{kg}{s}$ at about $38000\ Pa$ and then starts to decrease rapidly. A full closure of the valve is observed at $55000\ Pa$. If the inertia of the fluid and the valve body is considered, a change in the characteristic curve is observed as shown in Figure 12. The fluid and the valve body has to be accelerated by the potential difference which results in a mass flow advance of the steady-state simulation. A decrease in mass flow is observed at $60000\ Pa$. Then the mass flow drops to $0\ \frac{kg}{s}$ as the pressure increases by $2000\ Pa$. Though the maximal mass flow is $0.66\ \frac{kg}{s}$. Overall, the blue curve is useful to observe the mass flow in steady-state situations, where there is no variation in pressure over time. The red curve strongly depends on the input pressure function and can be used to give a forecast of the transient behavior.

In Figure 13 a detailed view of the mesh deformation at two discrete time is shown. The solid is colored blue and the fluid is shown white.

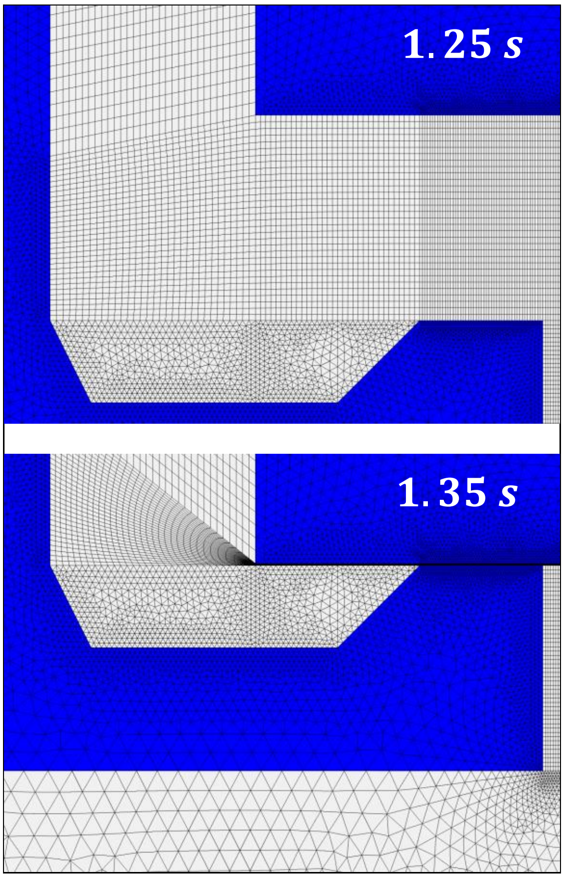

Figure 13: Mesh deformation at two different time states

At the time of $1.25\ s$ the valve body stays at the opening limit, while at $1.35\ s$ it reached the closing stop. Exclusively the triangular region on the left-hand side of the $1.35\ s$ state has a free deformation due to the *Hyperelastic smoothing PDE*. The connection between immediate neighbor cells remains.

The pressure distribution in the valve at the time of $1.3\ s$ is shown in Figure 14. A continuous pressure reduction is observed. A low-pressure area is seen at the tearing edge at the end of the valve shaft due to a change in streaming direction.

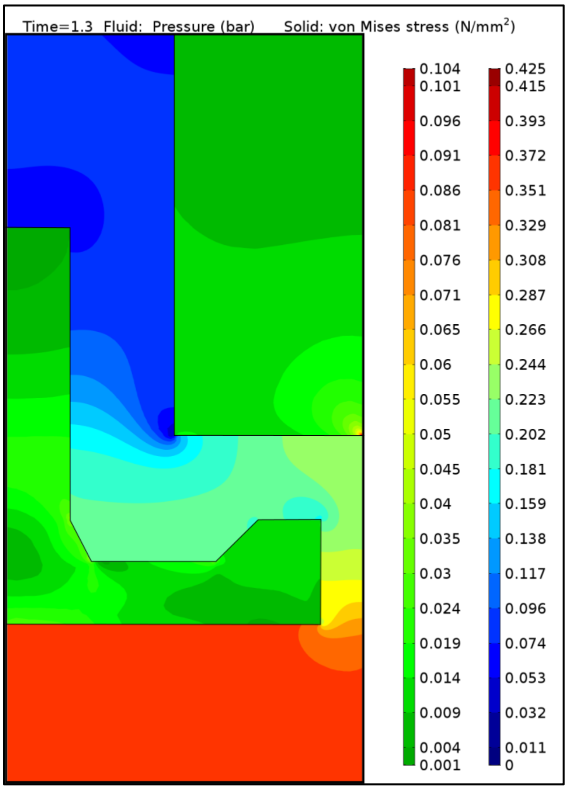

Figure 14: Pressure and van Mises stress at the time of 1.3 *s*

The valve body shows a homogenous stress distribution. At the valve seat an expected singularity close to the wall can be observed. The shown stress is low, compared to the stress during the closing operation. The maximal stress values reached at the valve seat is 200 $\frac{N}{mm^2}$.

5. Conclusion

Processing the previous defined task leads to some challenges regarding the modelling of the system. The exact process, which takes place in the reality, cannot be represented due to numerical circumstances. A modelling always illustrates an abstraction of the real object and contains therefore necessary changes. One of those modifications is the local increase in viscosity inside the closing gap. This entails a divergence of the acceleration or the pressure gradient. The effects should be low on account of the low spatial expansion of the affected area. Nevertheless, no measurements are given for comparison. Therefore, the mentioned effects cannot be neglected. At a rise of the viscosity above 200 %, negative effects on the convergence were observed, which lead to a demolition of the solving process.

Grieved acceptances with regard to the spring stiffness and the pressure function can be adapted to real system measurements. On account of the direct influence on the acceleration and therefore also on the time-step regulation, it can come to a strong increase of the calculation duration.

The subject of the project: *Modelling Fluid-Structure Interaction in a Pressure-Controlled Current-Limiting Valve* could be handled successfully with the use of COMSOL Multiphysics®. Closing operations with very small rest gaps of 0.05 mm were implemented. Besides, the remaining mass flow was reduced to a minimum of $4.77 \cdot 10^{-5}\ \frac{kg}{s}$. The demands for the numerics developed to the biggest challenge. The project could be finished successfully by the right choice of solution procedure as well as a skillful modelling.

Future works could extend the available model with a turbulence model. Therefor a structured mesh within boundary layers has to be built. Additionally, a measurement system is of high interest to validate the model. To make an adjustment, a high-speed camera could track the valves displacement and the difference pressure should be observed. The resulting pairs of difference pressure and displacement can be compared to simulations solution.

6. References

Surek, D.; Stempin, S.: *Angewandte Strömungs-mechanik.* Wiesbaden: Teubner Verlag, 2007

COMSOL AB: *COMSOL Multiphysics® Reference Manual Version 5.2*, 2016

COMSOL AB: *CFD Module User's Guide Version 5.2*, 2016

COMSOL AB: *Fluid-Structure Interaction Using COMSOL Multiphysics®*. Internet, 05.11.2016. (URL: https://www.comsol.com/video/simulating-fluid-structure-interaction-comsol-multiphysics)

COMSOL AB: *Fluid-Structure Interaction*. Internet, 01.11.2016. Application ID: 361 (URL: https://www.comsol.com/model/fluid-structure-interaction-361)

VDI-Gesellschaft Verfahrenstechnik und Che-mieingenieurwesen: *VDI-Wärmeatlas*. 11. Auflage, Berlin: Springer Verlag, 2013