NUMERIČKA ANALIZA PROTOKA I PRITISKA U KLIPNOM VENTILU

NUMERICAL ANALYSIS OF FLOW AND PRESSURE IN PISTON VALVE

***Ahmed Mujkanović*[1]**
***Armin Bošnjak*[1]**
***Martin Gehrmann*[2]**

[1]University of Zenica
Faculty of Mechanical Engineering,
Fakultetska 1, Zenica, B&H

[2]Germannius GmbH
Adalbert-Stifter-Weg 1
D-44534 Lünen, Germany

Ključne riječi:
numerička analiza, SolidWorks, klipni ventil

Keywords:
numerical analysis, SolidWorks, piston valve

Paper received:
16.05.2021.
Paper accepted:
26.06.2021.

Stručni rad

REZIME
Numerička analiza uz pomoć softvera CAD/CAE predstavlja veoma koristan alat za razne oblike proračuna i provjere konstrukcionih rješenja kao i optimizacije dizajna svakog planiranog idejnog rješenja. U okviru ovog rada, provedena je numerička analiza uz upotrebu softverskog modula SolidWorks Flow Simulation*. Pokazana je važnost upotrebe numeričkih simulacija pri razvoju novih idejnih rješenja. Kao dio ovog rada izvršena je numerička analiza protoka fluida kroz klipni ventil. Analiza je izvedena za nekoliko različitih protoka kako bi se dobio dijagram protoka i pritiska.*

Professional paper

SUMMARY
Numerical analysis with the help of CAD / CAE software is a very useful tool for various forms of calculation and verification of construction solutions as well as design optimization of each designing idea. As a part of this paper, a numerical analysis was performed using the SolidWorks Flow Simulation software module. The importance of using numerical simulations in the methodology of new idea solution development has been shown. As a part of this work, a numerical analysis of fluid flow through the piston valve was performed. The analysis was performed for several different flows in order to obtain a flow-pressure diagram.

1. INTRODUCTION

In modern designing and product development, software analysis and numerical calculations using various CAE (Computer Aided Engineering) software considerably replace the classic experimental analysis because, with a much lower cost and higher speed of analysis, give acceptable accuracy results.

Through various packages offered by the SolidWorks software, it is possible to implement the entire process of development and design of products, constructions and technology tools.

The Flow Simulation module is fully integrated into SolidWorks and is used to calculate fluid flow (gases or liquids), heat transfer through, from or within the model using computer fluid dynamics (CFD) technologies. The flow can be observed through or around the created 3D models, [1].

Flow Simulation has found application in many branches of industry, especially where product design optimization and system performance analysis are extremely important, such as flow simulations through valves, regulators, hydraulic and pneumatic components, [2-5].

As a part of this work, a numerical analysis of fluid flow through the piston valve was performed. The analysis was performed for several different flows in order to obtain a flow-pressure diagram.

2. TECHNICAL DESCRIPTION

2.1. Working principle and role of safety valve

In order to understand the functioning of the hydraulic supports, it is necessary to examine in detail the operation of the hydraulic cylinder in the roof racks themselves, Fig.1.

When the hydraulic support is installed, the pressure in the cylinders is about 350 bar. The weight that presses on the roof rack can easily raise the pressure to over 400 bar.

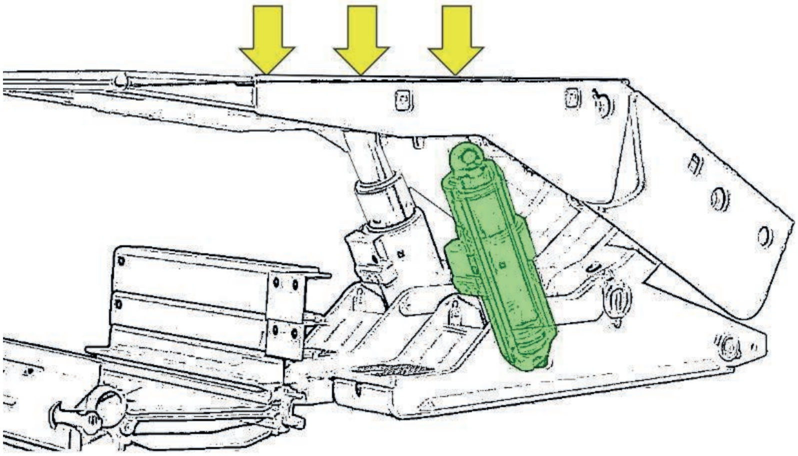

Figure 1 *Hydraulic cylinders as parts of roof rack*

In cases of a sudden increase in the load on the roof racks, the pressure rises sharply and fluid is released from the hydraulic cylinders, i.e. the safety valve is activated.
This drain will save the seals and cylinders from damage.

If the roof support is loaded and pressed to the lowest position, the safety valve will let some of the fluid out of the cylinder and keep the pressure inside the cylinder within normal limits. This will save the lives of the miners in the workspace, Fig. 2.

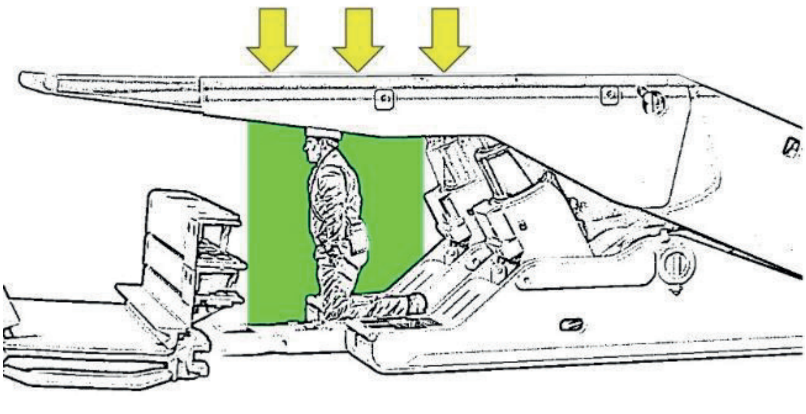

Figure 2 *Reducing the height of the working space and saving the life of the miners*

2.2. 3D model creation

After defining the parameters of the piston valve, the following is modeling procedure. Modeling of parts and assembly of valves was done in the SolidWorks, taking care not to overlap or excessive gaps at the places where the parts are joined.

The basic parts of a piston valve are, Fig. 3:

1. Connection,
2. Piston,
3. Spring disc,
4. Spring,
5. Spring sleeve,
6. Spring guide and
7. Adjusting screw.

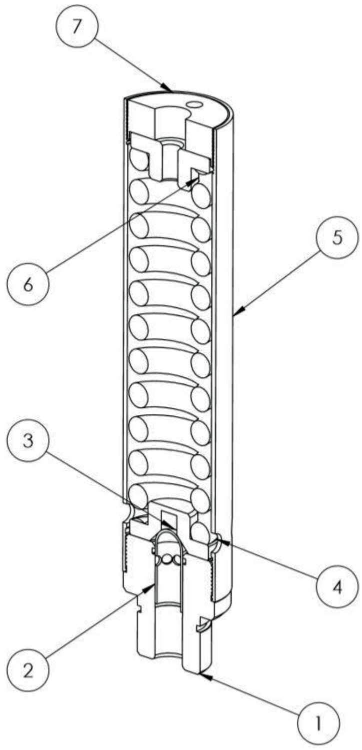

Figure 3 *Piston valve assembly cross section*

2.3. Piston valve assembly

Once all the necessary parts have been modeled, assembly into a single model follows, Fig. 4 and 5.

Figure 4 *Valve assembly without spring sleeve*

Figure 5 *Piston valve assembly*

3. DISTRIBUTION VALVES

With distribution valves, the flow of the fluids is stopped by setting the work element on the cased surface (sealing surface). The work element can be in the form of a sphere, conical or fungus-like. The working element is of the conical shape in most of the cases. This type of work element during working times deliver high grades of sensibility, sealing and reliability.

Piston is opened when the thin part of the work element is pressured by force (F) of higher value, than the force from the spring. When the condition F = 0 is met, the spring pressurizes the work element in the upper position, which is why the created contact between the edge of piston body and the conical surface of the sealing seals the surface and disallowes fluid flow. In those cases, force that is acting upon the piston work element is equal to:

$$F_p = p_1 \cdot A - p_2 \cdot (A - a) + F_{op} + F_T, \tag{1}$$

where are:

p_1, p_2 – pressure in the supply and drainage chamber,

$A = \frac{D^2\pi}{4}$; $a = \frac{d^2\pi}{4}$ – area surfaces,

F_{op} – spring force,

F_T – frictional force at rest.

Without depending on the surface body piston type, the true contact is created over the surface width, which is why the value of the force that acts upon the piston distributor depends on the contact surface. If the piston is located in secluded area, then, in order to open the piston, force that acts upon the work element should be calculated by:

$$F = p_1 \cdot A + F_{op} + F_T - p_2 \cdot (A - a). \tag{2}$$

After opening the piston, force (F_{op}) and pressure (p_1) remain unchanged, while the pressure in the opening through which the fluid is flowing, decreases proportionally. Force neccessary for further movement of the work element will be:

$$F_1 = p_1 \cdot A_1 + F_{op} + F_{T_2} - p_2 \cdot (A_2 - a) - p_{sr} \cdot (A_1 - A_2), \tag{3}$$

where are:

$p_{sr} = p_1 + \frac{p_2}{2}$ mid-value pressure;

F_T – frictional movement force;

a – surface that matches the diameter d;

A, A_1, A_2 – surfaces that match the diameters D, D_1, D_2.

Based on previously mentioned it can be concluded that, the force (F) will be higher than force (F_1) for $p_{sr} \cdot (A_1 - A_2)$, and also the value of the size of friction: in movement or at rest. After the separation of the work element from the casing, opening force will be reduced, while the pressure in the drainage chamber will increase up to the value of $p'_2 = p_2 + \Delta p$, where (Δp) is the increase in presure in drainage chamber. Now, the value of medium pressure will also increase, and it is:

$$p_{sr} = \frac{p_1 + p_2 + \Delta p}{2}, \tag{4}$$

and the neccessary opening force is equal to:

$$F_1 = p_1 \cdot A_1 + F_{op} + F_{T_2} + (p_2 + \Delta p) \cdot (A_2 - a) - \left(\frac{p_1 + p_2 + \Delta p}{2}\right) \cdot (A_1 - A_2). \tag{5}$$

3.1 Non-return valves

Non-return valves represent one of the simplier valve cased constructions. Regardless of the constructive solution, every one of these types of elements have the task to allow fluid flow in one direction, while in the other way, disallow the flow of fluids. Simplified section of non-return ball-type valve with conical work element has been shown at the Fig. 6. a). The diagram (p-Q) is presented at the Fig. 6. b), from which is noted that is out of importance to create enough values of pre-pressure (p_o), in order to subdue the spring force and open the valve. From the pointed section, it can be noticed that this type of valve does not allow fluid flow in the direction B -A.

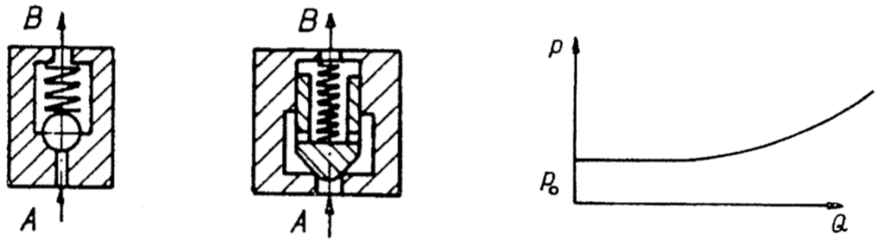

Figure 6 *Non-return section: a) types of valve, b) p-Q diagram*

4. NUMERICAL ANALYSIS

4.1 Analysis type and problem description

In any of the statistical analysis, in most cases it is neccessary to change the geometry of SolidWorks CAD model in order to perform the simulation. Same goes for flow simulations. SolidWorks Flow Simulation creates flow analysis in two different categories, internal and external flow analysis. Before even starting the model preparation, it's neccessary to decide in which category is the type of analysis planned to perform. In this case, internal or insider analysis is going to be performed. At Fig. 7. green arrow represents the location on which the fluid enters the valve, while red arrows represent the places, i.e. the openings on which the fluid, after the valve opening during the movement of the piston, exits from the valve. Valve is connected to the cylinder where, while pressuring the cylinder, the increase in pressure happens. After the pressure crosses the value on which the valve spring starts to compress, movement in the piston occurs and valve opening is achieved. During the valve opening, fluid exits through 9 holes in the piston, and, after that in most parts exits through 9 holes on the pipe, and on the upper oppening through the top of the valve.

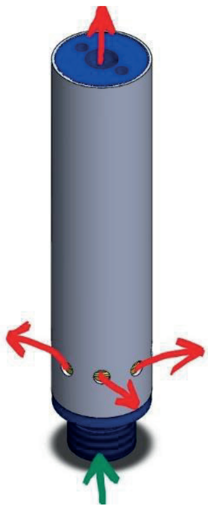

Figure 7 *Entry and exit points of the fluid*

4.2. Lid creation

Lids are used in internal flow analysis. During this types of analysis, all holes on the model need to be closed by using lids as a type of special elements in the constructional tree of SolidWorks CAD model. Lid surfaces (that are in direct fluid contact) are used for setting borderline conditions, that are defining the values of mass flow, volume flow, static/total pressure and etc. On the border of fluid injected areas. One of the ways of creating lids on curved surfaces is by using the surface tools, Fig. 8.

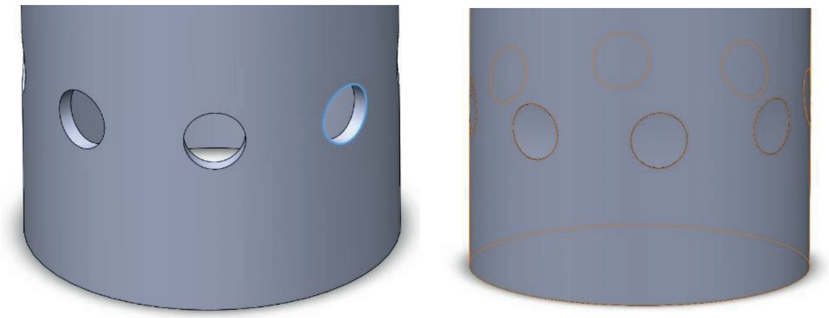

Figure 8 *Curved surface lids*

4.3. Simulation settings

Computer domain is defined as volume in fluid flow field, stationary in opposing to the coordinate system. Even though fluid flows in and out of the computer domain, it by itself remains fixed in the area. SolidWorks flow simulation analises model geometry and automatically creates the computer domain in the form of rectangular prism that connects the entire model. For the external analysis of treshold levels, computer domains are automatically moved from the model, and that way they limit the specific fluid area surrounding the model. During the internal threshold level analysis, computer domains are automatically set in a way that domain covers only the analysed model. On the surface of the entry opening we insert the volume flow, while on the surfaces of every exit valve opening we insert the condition known as Enviroment Pressure, Fig. 9.

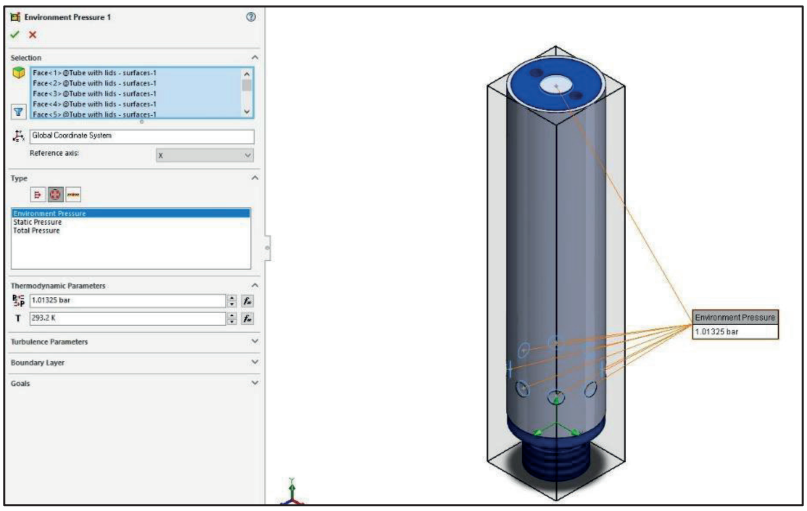

Figure 9 *Boundary conditions*

4.4. Results analysis

The valve is set to open at 400 bar, i.e. at a pressure of 400 bar the spring compresses and the piston moves through which fluid begins to flow, Fig. 10. For an inlet volume flow of 1750 l/min, the maximum pressure that occurs in the valve is 454,97 bar, Fig. 11. Fig. 12 shows an enlarged view of fluid flow trajectories.

Figure 10 Fluid flow trajectories

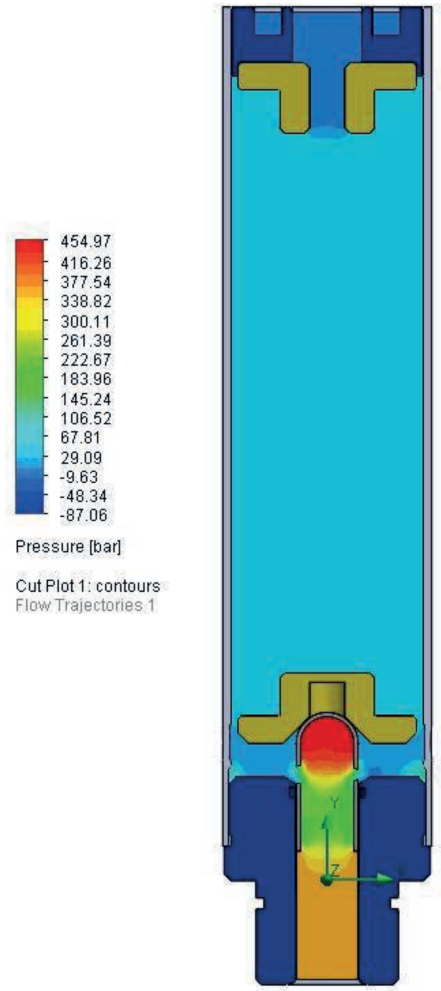

Figure 11 *Pressure distribution in valve*

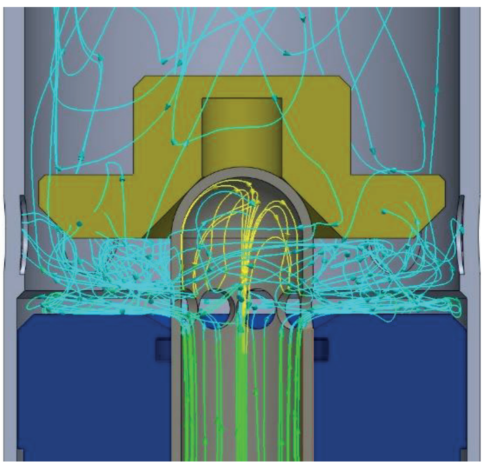

Figure 12 *Enlarged view of fluid flow trajectorie*

5. CONCLUSION

The aim of this paper was to present the process of making a piston valve in software SolidWorks, and to show the importance of numerical calculations in the development of a new idea solution. In general, it can be concluded that the results of numerical analysis are similar with the values of analytical calculation performed and some experimental testing. This confirms the validity of the numerical simulation and the correct definition of the boundary conditions. So, in the process, three ways of analysis and calculation are combined: analytical, numerical and experimental method. After a series of simulations, the results of the numerical calculation were compared with the results obtained by experimental testing at the Institute in Opava, Czech Republic, because of the intention of checking the procedure of CAD design and numerical analysis on exact results. Since the procedure of checking is based on a copyright-protected project, specific values and diagrams cannot be shown. The continuation of this study could be seen in the fact that parametric analysis and complete optimization of the piston valve could be performed.

REFERENCES

[1] Miloš, Marko i Grbović, Aleksandar: *Softverski alati u dizajnu*, Mašinski fakultet Beograd, 2017.

[2] Jonuskaite, Akvile: *Flow simulation with SolidWorks*, Bachelor thesis, 2017.

[3] SolidWorks, "3D CAD Design Engineering Software Tools," Dassault Systems, [Online]. Available: https://www.solidworks.com/sw/3d-cad-design-software.htm.

[4] Emad Q. Hussein et al. *Fluid Structure Interaction of Non–Return Valve Using CFD Technique*, Journal of Mechanical Engineering Research and Developments Vol. 43, No. 7, pp. 137-148, 2020.

[5] Filo, G., Lisowski, E., Rajda, J.: Design and Flow Analysis of an Adjustable Check Valve by Means of CFD Method. Energies 2021, 14, 2237. https://doi.org/10.3390/en14082237.

Corresponding author:
Ahmed Mujkanović, grad. mech. eng.
Faculty of Mechanical Engineering
University of Zenica
Fakultetska 1, Zenica
Email: mujkanovicahmed@gmail.com
Phone: +387 62 751 874