The simulation of fluid flow in safety elements of longwall shield support hydraulic legs

S Rajwa\textsuperscript{1*}, T Janoszek\textsuperscript{1} and J Świątek\textsuperscript{1}

\textsuperscript{1}Central Mining Institute, Plac Gwarków 1, Katowice, Poland

*E-mail: jswiatek@gig.eu

\textbf{Abstract.} The paper presents the research on pressure variation in elements securing the shield support against overloading. The chosen constructions of connecting pipe were analysed and its flow characteristic were determined. The operational parameters of different connections were also considered in light of their interaction with whole hydraulic control system. The results allowed to determine of the optimal connecting pipe curve with regard to the pressure in whole pressure safety system.

\section{1. Introduction}

Longwall powered support intended for operation under the conditions of rock mass tremor hazards should be yielding, i.e. it should be able to take over dynamic loads \cite{6, 8}.

This characteristic is required according to the regulations included in the Ordinance of the Minister of Energy of 23 November 2016, Dz. U. No 2017, item 1118, para. 523, sec. 1, pt. 1. The evaluation of yielding supports as well as the guidelines for their construction are prepared at the Central Mining Institute according to own methodology. It is currently the only method of support yield evaluation that is applied broadly in practice. It is an analytical method utilising many years of operational experience and the technical parameters of the support units. This approach also constitutes a part of the support selection method for geological and mining conditions \cite{5, 7, 12}.

Its use is based on determining a safe operational range of a support unit for a given situation. Result verification of the application of the method in question is systematically conducted by GIG employees, including the use of increasingly more broadly utilised support monitoring systems \cite{3, 4}.

One of the key support unit safety elements defined in the GIG method is the fluid flow in the system securing the legs from overloading. Such a system is comprised of a pressure relief valve and a pipe supplying it with fluid from the leg working space. The pipe flow characteristics, understood as the relation between the pressure and the fluid stream volumetric flow, and the valve parameters determine the efficiency of securing the leg from overloading. Deteriorating mining conditions (increasing depths of the extracted deposits, rock mass tremor hazards) result in the necessity to increase the support unit load capacity by utilising legs with diameters $\varnothing \geq 0.32$ m. Additionally, the current trend of placing the leg control systems as high as possible above the floor considerably extends the distance needed for the fluid stream to flow from the leg working space to the pressure...
relief valve. The flows required to secure the high-diameter legs necessitate the optimisation of the fluid stream, directed at decreasing the losses in continuous and local flows.

The subject of this publication is an example use of numerical analysis using the SolidWorks Flow Simulation software to optimise the system securing the hydraulic leg from overloading, and particularly to optimise the shape of the fluid stream supplied by a connecting pipe from the leg working space to the pressure relief valve.

2. Leg securing system flow characteristics

The leg securing system is characterised by parameters that express continuous flow as a set of relationships with pressure at the system entry. These are non-linear relationships dependent on multiple factors, such as: fluid type (mono- or diphase), local and continuous losses (shape and geometry of the fluid stream, roughness, Reynolds number), characteristics of the fluid flow through the valve. Fluid leaks can be continuous or undergo periodic variations as a result of the two-state valve operation, i.e. open/closed. This publication presents issues related only to the steady continuous flow. The hydraulic system securing the leg from overloading is presented as a diagram in Figure 1a. The fluid stream is drained from the secured leg space via a port, usually a steel pipe with an outlet. The flow of the entire system can be described as a sum of the characteristics of the connector and the valve, presented as an exponential polynomial. This form makes it possible to sum the relationships for a series connection of system elements where the flow for individual elements is identical – Figure 1b [10].

![Figure 1](image_url)

**Figure 1.** Flow characteristics of the system securing the leg from overloading and its determination method [10]; a – general diagram, b – flow computational relationships. Where: \( P_{PT} \) – pressure under the piston, \( P_p \) – pressure drop in the connector for a specified mass flow, \( Q_{ukl} \) – system volumetric flow, \( P_Z \) – pressure drop in the valve for a specified mass flow, \( P_K \) – pressure at the valve exit, \( Q \) – volumetric flow.

3. Scope and description of the conducted leg port analyses

The first stage of the numerical modelling with the use of CFD (Computational Fluid Dynamics) methods is to define its purpose, i.e. to consider the physical phenomena, the conditions of the uniqueness of the numerical model solution, and the scale of the computational area. The second stage is to develop the geometry of the structure that will constitute a reflection of the actual subject of study. Once the geometric model is built, the next stage is discretisation, i.e. generating a computational grid that will constitute the area of the numerical solution of the analysed problem. The final stage of the modelling process is to define the correct settings for the analysed problem (the solver) and to perform numerical calculations with systematic monitoring of the convergences of the obtained numerical solution.

The analysed powered support unit leg port is presented in Figure 2. It is comprised of a specifically shaped pipe and a special tip for connecting the relief pressure valve. Both the elements are made from low-alloyed constructional steel, which is characterised by high strength and ductility, good treatment and fine-grained alloy structure.
Figure 2. Leg securing system. a - connector cross section, b - limiting pressure valve block, c - analysed longwall powered support leg securing system assembly model, where: 1 – limiting securing system block, 2 – connector.

The main goal of the analysis was the volumetric fluid flow through the analysed connector as a function of pressure. The following calculation variants were adopted:

- influence of the connector geometry on the pressure characteristics – variant I,
- influence of the connector cross-section on the pressure characteristics – variant II,
- influence of the air content in the fluid on the pressure characteristics – variant III.

A discretisation area was generated based on the developed connector assembly model in SolidWorks software, in the form of a numerical grid composed of 430535 fluid cells, where 87651 fluid cells contacting solids, which represents geometry occupied by the fluid – Figure 3 (volume of fluid is 0.000198 m³).
Figure 3. Leg securing system: 1 – pressure limiting valve block, 2 – connector, 3 – fluid volume.

The following boundary conditions were included for the purposes of the model:
1. For variants I and II, the calculations involved fluid in the form of pure water with the following parameters:
   - density $\rho = 999.76$ [kg $\cdot$ m$^{-3}$],
   - dynamic viscosity $\mu = 0.0017912$ [Pa $\cdot$ s],
   - specific heat $c_p = 4219.9$ [J $\cdot$ kg$^{-1}$ $\cdot$ K$^{-1}$],
   - volumetric flow variation range $V = 0 \div 2000$ dm$^3$ $\cdot$ min$^{-1}$,
   - thermal conductivity coefficient $\lambda = 0.56104$ [W $\cdot$ m$^{-1}$ $\cdot$ K$^{-1}$].
2. For variant III, the calculations involved fluid in the form of a mixture of water and air, with the following air parameters:
   - density $\rho = 1.2$ [kg $\cdot$ m$^{-3}$],
   - dynamic viscosity $\mu = 6.04 \times 10^{-6}$ [Pa $\cdot$ s],
   - specific heat $c_p = 1029.1$ [J $\cdot$ kg$^{-1}$ $\cdot$ K$^{-1}$],
   - volumetric flow variation range $V = 0 \div 2000$ dm$^3$ $\cdot$ min$^{-1}$,
   - thermal conductivity coefficient $\lambda = 0.00779$ [W $\cdot$ m$^{-1}$ $\cdot$ K$^{-1}$].

The results of the conducted modelling are presented on charts as a relationship of $P=f(Q)$, and additionally in the form of an exponential polynomial for a selected case.

4. Numerical analysis results
The analyses of individual leg securing system variants were conducted using the SolidWorks 2018 software with the Flow Simulation tool. SolidWorks is a program for computer-aided design, used for constructing 2D and 3D models. This simple very efficient computer program allows designers to create very detailed elements and assemblies at the design stage as well as working drawings. SolidWorks is an engineering tool that can support multiple stages of the development of a given product. It makes it possible to inspect a design directly before production by means of a broad range of available tools, from computational fluid dynamics (CFD) to the analysis of static and dynamic strength of a studied prototype construction, based on the finite element method (FEM) [2, 9].

The purpose of the fluid flow process simulation supported by CFD methods in the space that constitutes the volume occupied by the analysed fluid is to obtain a solution to a set of differential equations which interpret the law of conservation of mass and momentum of the moving fluid (Navier-Stokes equation). These fundamental equations, which express the behaviour of a fluid flow through a given geometry in SolidWorks Flow Simulation, are relationships defined in the following forms [2, 9]:
- mass conservation equation:
  \[
  \frac{\partial p}{\partial t} + \nabla (p \nu) = 0
  \]
- Navier-Stokes equation:

\[ \rho \frac{\partial \mathbf{v}}{\partial t} = -\nabla p + \rho g + \mu \nabla^2 \mathbf{v} \]  

(2)

where:

- \( \rho \) – fluid density [kg·m\(^{-3}\)],
- \( \mu \) – fluid dynamic viscosity [Pa·s],
- \( \rho \) – fluid density [kg·m\(^{-3}\)],
- \( \mu \) – fluid viscosity [Pa·s],
- \( g \) – gravity acceleration [m/s\(^2\)],
- \( \mu \) – fluid dynamic viscosity [Pa·s],
- \( \rho \) – fluid density [kg·m\(^{-3}\)],
- \( \mu \) – fluid viscosity [Pa·s].

The influence of disturbances in the fluid transfer process within the given geometry were interpreted with a \( k-\varepsilon \) turbulence model. Solving this model comes down to determining the value of turbulent viscosity \( \mu_t \) using turbulence kinetic energy \( k \) and dissipation rate \( \varepsilon \) related to the energy dissipation resulting from the occurrence of internal resistances to motion of a fluid flow through a channel. The turbulent viscosity \( \mu_t \) model of the fluid is expressed using an equation defined in the following form in SolidWorks Flow Simulation:

\[ \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \]  

(3)

Fluid transport equations for turbulence kinetic energy \( k \) and dissipation \( \varepsilon \) are expressed in SolidWorks Flow Simulation as relationships in the following forms [2, 9]:

- for turbulence kinetic energy:

\[ \frac{\partial \rho k}{\partial t} + \frac{\partial \rho k \mathbf{v}_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \tau_{ij}^\varepsilon \frac{\partial \mathbf{v}_j}{\partial x_i} - \rho \varepsilon + \mu_t \mathbf{P}_b \]  

(4)

- for dissipation energy:

\[ \frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho \varepsilon \mathbf{v}_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial \varepsilon}{\partial x_i} \right) + C_\varepsilon \frac{\varepsilon}{k} \left( f_1 \frac{\partial k}{\partial x_j} + C_{\varepsilon P} \mathbf{P}_b \right) - f_2 C_\varepsilon \rho \frac{k^2}{\varepsilon} \]  

(5)

where:

- \( C_{\varepsilon 1} \) – empirical constant, \( C_{\varepsilon 1} = 1.44 \),
- \( C_{\varepsilon 2} \) – empirical constant, \( C_{\varepsilon 2} = 1.92 \),
- \( C_\mu \) – empirical constant, \( C_\mu = 0.09 \),
- \( k \) – velocity fluctuation (turbulence) kinetic energy [m\(^2\)·s\(^{-2}\)],
- \( P \) – local vorticity fluctuation production,
- \( \varepsilon \) – turbulence kinetic energy dissipation rate [m\(^2\)·s\(^{-3}\)],
- \( \mu_t \) – turbulent viscosity [Pa·s],
- \( \sigma_k \) – turbulent Prandtl number \( \sigma_k = 1.0 \),
- \( \sigma_\varepsilon \) – turbulent Prandtl number \( \sigma_\varepsilon = 1.3 \),
- \( \rho \) – fluid density [kg·m\(^{-3}\)].

4.1. Stream geometry influence

In order to demonstrate the influence of the variations in fluid geometry on the pressure distribution along the analysed leg securing system (Figure 2), numerical simulations were performed for the model, where the bend radius of connector was modified from the value of \( R=15 \) mm (Figure 2a) to a value of \( R=40 \) mm (Figure 4a). Figure 4 presents the basic dimensions of the modified connector (Figure 4a) and the view of the spatial model of the leg securing system in the assembly (Figure 4b).
Figure 4. View of the modified powered support leg securing system: a – connector cross section, b – spatial view.

The numerical calculations were conducted within the scope of water volumetric flow variations at a range of 0 to 2000 dm$^3\cdot$min$^{-1}$, simulating the fluid supply to the modified (Figure 4) and unmodified (Figure 2) leg securing system, as depicted in Figure 4. Total pressure variation measurements were performed at the outlet of the pressure limiting valve. The obtained results of the conducted numerical simulations are presented in Figure 5 in the form of a $P=f(Q)$ chart, where $P$ is the total pressure distribution variation along the powered support leg securing system.

Figure 5. Characteristics of the total pressure variation at the powered support unit leg securing system outlet depending on the fluid volumetric flow.
Additionally, maps of the total pressure variation distribution along the studied powered support unit leg port geometries for a fluid volumetric flow of 1000 dm³·min⁻¹ are presented in Figure 6 as well as for a fluid volumetric flow of 2000 dm³·min⁻¹ in Figure 7.

**Figure 6.** Maps of the total pressure variation distribution along the studied powered support leg securing system for a fluid volumetric flow of 1000 dm³·min⁻¹: a – pressure variation distribution for a powered support leg securing system geometry from Figure 2, b – pressure variation distribution for a powered support leg securing system geometry from Figure 4.

By analysing the curves in Figure 5 it can be observed that increasing the leg securing system bend radius by 60% results in an increase of the stream total pressure at the powered support at the outlet of pressure limiting valve by 3 MPa on average. Furthermore, by observing the maps of the total pressure variations along the analysed connector variants, it can be concluded that increasing the connector bend radius decreases the fluid resistances at the inlet by 10% on average for a volumetric flow of 1000 dm³·min⁻¹ and by 40% on average for a volumetric flow of 2000 dm³·min⁻¹.
4.2. Stream cross-section influence

In order to analyse the variant involving the influence of the fluid cross-section variation on the pressure distribution along the powered support leg securing system, numerical simulations were conducted for the port assembly model, where the pipe internal diameter ($d_{in}$) and external diameter ($d_{out}$) were modified. The basic diameter values, i.e. $d_{in} = 20$ mm and $d_{out} = 30$ mm, were increased to $d_{in} = 30$ mm and $d_{out} = 40$ mm. Figure 8 presents the basic dimensions of the modified connector (Figure 8a) and the view of the model in the assembly with the pressure limiting valve block (Figure 8b).

![Figure 8](image)

Figure 8. View of the modified powered support leg securing system geometry:
(a) – connector cross section, (b) – view of leg securing system

The numerical calculations were conducted within the scope of water volumetric flow variations at a range of 0 to 2000 dm$^3$·min$^{-1}$, simulating the fluid supply to the modified and unmodified leg securing system, as depicted in Figure 8. Total pressure variation measurements were performed at the outlet of the limiting pressure valve block during the conducted modelling.

The results of the numerical analysis of the modified port are presented in Figure 9 in the form of powered support leg securing system total pressure variation characteristics.
Figure 9. Characteristics of the total pressure variation along the studied geometries at the outlet of leg securing system depending on the fluid volumetric flow.

Additionally, maps of the total pressure variation distribution along the studied powered support leg securing system for a fluid volumetric flow of 1000 dm$^3$·min$^{-1}$ are presented in Figure 10 as well as for a fluid volumetric flow of 2000 dm$^3$·min$^{-1}$ in Figure 11.

Figure 10. Maps of the total pressure variation distribution along the studied powered support leg securing system for a fluid stream volume of 1000 dm$^3$·min$^{-1}$: (a) – pressure variation distribution for leg securing system geometry from Figure 2, (b) – pressure variation distribution for leg securing system geometry from Figure 8.
Figure 11. Maps of the total pressure variation distribution along the studied powered support leg securing system for a fluid volumetric flow of 2000 dm$^3$.min$^{-1}$: a – pressure variation distribution for leg securing system geometry from Figure 2, b – pressure variation distribution for leg securing system geometry from Figure 8.

It can be observed from the courses of the curves in the charts (Figure 9) that increasing the cross-section of connector by 25% can result in an increase in the total pressure at the outlet of limiting pressure valve block by 80% on average.

4.3. Stream air content influence

In order to analyse the variant involving the possibility of the occurrence of leaks in the leg securing system assembly, a computational variant of the influence of the air content in the fluid volumetric flow on the total pressure distribution along the leg securing system was considered, measured at the outlet, at the pressure relief valve attachment point. The calculations were conducted for a mixture of water and air with air mass fraction range of 0 to 4%. A mixture was modelled in proportion of air to water: 1:49 and 1:24. The results are presented in Figure 12 in the form of total pressure variation characteristics at the outlet of the limiting pressure valve block, as a function of the fluid volumetric flow variation and the air content in the fluid (mixture of water and air).
Figure 12. Characteristics of the total pressure variation at the powered support at the outlet of leg securing system depending on the fluid volumetric flow and the air content in the fluid (mixture of water and air).

Figure 13. Maps of the total pressure variation distribution along the analysed leg securing system according to Figure 2 for a fluid volumetric flow of 1000 dm³·min⁻¹: a – pressure variation distribution for an airtight powered support leg securing system assembly, b – pressure variation distribution for a powered support leg securing system with an air mass fraction of 2%, c – pressure variation distribution for a powered support leg securing system with an air mass fraction of 4%.
Figure 14. Maps of the total pressure variation distribution along the analysed powered support leg securing system according to Figure 2 for a fluid volumetric flow of 2000 dm$^3$·min$^{-1}$: a – pressure variation distribution for an airtight powered support leg securing system assembly, b – pressure variation distribution for a powered support leg securing system with an air mass fraction of 2%, c – pressure variation distribution for a powered support leg securing system with an air mass fraction of 4%.

By analysing the chart of the total pressure variations at the outlet depicted in Figure 12, it can be concluded that an increase in the fluid of air mass fraction within a range of 0 to 4% results in a total fluid (mixture of fluid and air) pressure drop measured at the outlet by 30% on average.

5. System safety characteristics
Flows in the leg hydraulic securing system have a significant influence on the limitation of powered support unit overloads generated as a result of rock mass tremors. The flow values are determined for the entire system encompassing the pressure relief valve and the applied port connecting the space under the leg piston with the valve. The flow value has an influence on vibration damping in the computational model used for the evaluation of the longwall powered support unit yield. The pressure relief system flow in the leg should secure the leg up to a value 1.5 times as great as the base yield load [10, 11]. The flow in the system is presented below in the form of an exponential polynomial for a case of the modification of the internal and outer diameters of the applied pipe. Connector flow determined analytically by means of SolidWorks Flow Simulation was considered for two cases of connector diameters. Valve characteristics are presented according to the tests conducted at TLO Opava in Czech Republic [10]. After obtaining the characteristics of the leg securing system and the valve, it was possible to characterise the entire system, as depicted in Figure 15.
Figure 15. Flow analysis for the evaluation of the entire system securing an example leg from overloading ($P_r = 38$ MPa, $Q$ for $1.5\cdot P_r = 57$ MPa); $P_{wyp}$ – resultant pressure of the entire system for a defined mass flow, $P_z$ – valve characteristics, $P_p$ – connector characteristics, $P$ – pressure, $Q$ – flow.

As can be seen in the above example, increasing the pipe internal diameter results in an increase in the flow, which is particularly visible at the higher volumetric flow range.

6. Conclusions
The numerical determination of longwall powered support unit leg safety system volumetric flows is useful when evaluating support yield for cases of the extraction of deposits at risk of rock mass tremors, as a way of fulfilling the requirements of the Ordinance of the Minister of Energy of 23 November 2016, Dz. U. No 2017, item 1118, para. 523, sec. 1, pt. 1. They also constitute a significant benefit for designers in terms of the optimisation of the construction of the safety elements that secure the leg against overloads.

The conducted numerical analyses have shown that the fluid stream geometry and its cross-section, resulting from the diameter of the applied pipe, have a significant influence on the limitation of losses in fluid flow from the secured leg space to the pressure relief valve. These parameters attain particular importance at high flow values.

The air content in the analysed stream, up to a value of several percent, had minor influence on the conducted analyses. However, additional calculations should be performed for higher air concentrations in the stream in order to verify the influence of the air content.

It should also be noted that the value of the losses in stream flows is dependent on multiple parameters, the majority of which exhibit non-linear characteristics, which is why it would be the most rational to consider every case individually.

Acknowledgement
The article was prepared based on research conducted within the Research Project: Productivity and Safety of Shield Support (PRASS III)-co-financed by European Commission-Research Fund for Coal and Steel (No. 752504) and Polish Ministry of Science and Higher Education.

References
[1] Janoszek T, Łączny MJ, Stańczyk K, Smoliński A, Wiatowski M 2013 Modelling of gas flow in the underground coal gasification process and its interactions with the rock environment (J. Sustain. Min.) vol. 12, no. 2, pp. 8–20
[2] Jonuskaite A 2017 Flow simulation with SolidWorks (Bachelor Thesis, Plastics Technology)
[3] Małkowski P 2017 Management of Mining Hazard Monitoring (J. Polish Miner. Eng. Soc.) vol. 18, no. 2, pp. 215–224

[4] Prusek S, Płonka M, Walentek A 2016 Applying the ground reaction curve concept to the assessment of shield support performance in longwall faces (Arab. J. Geosci.), vol. 9, no. 3, pp. 1–15

[5] Prusek S, Rajwa S, Wrana A, Alicja K, 2017 Assessment of roof fall risk in longwall coal mines, (Int. J. Mining, Reclam. Environ) vol. 31, no. 8, pp. 558–574

[6] Rajwa S, Janoszek T, Prusek S 2019 Influence of canopy ratio of powered roof support on longwall working stability – A case study (International Journal of Mining Science and Technology), Volume 29, Issue 4, pp 591-598

[7] Rajwa S, Masny W, Wrana A 2017 A comprehensive method for the selection of powered roof support in conditions of the rockburst hazard Wiadomości Górnicze No. 1 pp 2-7

[8] Rajwa S, Prusek S, Stoiński K 2016 Opis metody upodatkowania zmęczonowanej obudowy ścianowej Bezpieczeństwo Pracy i Ochrona Środowiska w Górnictwie No 12 pp 3-8

[9] SolidWorks Help SolidWorks Flow Simulation Overview (https://help.solidworks.com/2018/english/SolidWorks/flopress/c_flow_simulation_overview.htm)

[10] Stoiński K et al. 2018 Zmechanizowane obudowy ścianowe dla warunków zagrożenia wstrząsami górotworu Katowice: Główny Instytut Górnictwa, 2018

[11] Stoiński K, Prusek S, Rajwa S 2015 Dobór przepływów w układzie zabezpieczającym stojak zmechanizowanej obudowy ścianowej – opis przypadku (XXIII International Conference TEMA G, Gliwice)

[12] Świątek J Stoiński K 2019 Case Analysis of Damages to Control Hydraulics of the Leg in the Powered Roof Support Section (IVth International Innovative Mining Symposium, E3S Web Conf.) Volume 105