Design hydrodynamic analysis of cavitation in narrow channels of the open-pit dump truck's hydraulic system

D A Panasenkov¹, A P Zaycev¹, A B Kartashov¹, N A Pikalov¹, D M Dubinkin², and A B Efremenkov³

¹ Wheeled Vehicles Department, Bauman Moscow State Technical University, 2nd Baumanskaya St., 5 105005, Moscow, Russian Federation
² Department of Metal Cutting Machines and Tools, Kuzbass State Technical University, Vesennaya St., 28 650000, Kemerovo, Russian Federation
³ Yaroslav-the-Wise Novgorod State University, B. St. Peterburgskaya str., 41 173003, Veliky Novgorod, Russia

Abstract. The paper deals with the processes that occur when the hydraulic fluid flows in the channels of the hydraulic block of the brake drive of the open-pit dump truck. The main attention is paid to the phenomenon of cavitation caverns emergence and hydro-mechanical losses depending on the hydraulic fluid parameters and the geometric parameters of hydraulic channels. The SST-HL and SSG turbulence models are presented in this paper. The Rayleigh Plesset cavitation model built into the ANSYS CFX software package is also used.

1. Introduction
When designing the hydraulic drive of the open-pit dump truck, it is often necessary to ensure the most compact and efficient placement of cartridge valve equipment that facilitates the operation of brake mechanisms, hydraulic cylinders for folding the frame and lifting the cargo platform. In order to minimize the range of hydraulic components, they are combined into hydraulic blocks. The main task in the development of such blocks is to ensure the speed of the braking system, steering and lifting of the cargo platform. This task is particularly acute when developing the hydraulic system of open-pit dump trucks with articulated frame, because the efficiency and safety of using such vehicles in mining works depends largely on the speed and reliability of the components of the overall hydraulic system.

Electromagnetic spool distributors are widely used in the hydraulic system of open-pit dump trucks, since they allow implementing various control schemes [1, 2]. In addition to the friction process, which leads to the gradual destruction of the moving parts of the spool distributor [3], there is an inevitable erosion wear that occurs due to the fact that metal particles enter the spool. One of the reason why metal particles get into the hydraulic fluid is cavitation erosion [4]. During cavitation,
there is a violation of the continuity of the fluid flow in the low-pressure zone, accompanied by local hydraulic shocks that occur when cavitation caverns disappear. Prolonged exposure to such erosion leads to the gradual destruction of the channel walls and the separation of small metal particles that fall into the spool distributors of the hydraulic drive, which has an erosive effect on the spool components. In [5] it is shown that the wear effect in hydraulic valves leads to deterioration of the spool distributor, reduces its service life, and results in leaks in the system.

Since the erosive effect on the spool is a consequence of cavitation in hydraulic channels, special attention should be paid to reducing the probability of cavitation phenomena when developing hydraulic blocks. Thus, the paper [6] demonstrates a computational model for evaluating cavitation-erosion damage in the solenoid valves of a diesel injector. In [7], the authors managed to reduce the calculation error when modeling the cavitation process in the throttle channel by approximating the design model to the actual operating conditions of the hydraulic drive.

It should be noted that the process of selecting rational parameters of hydroblock channels is complicated by a large number of them, since the phenomenon of cavitation depends on: the geometric parameters of the channels, the material of the hydraulic block housing, the pressure of the hydraulic fluid, the type of hydraulic fluid and the quality of the internal surfaces of the channels (roughness).

The tests carried out in [8, 9, 10] showed that the choice of product material has a significant impact on the process of cavitation, and in [11] it is shown that the stage of computer simulation of the cavitation process is necessary to exclude the method of trial and error during design and reduces the time of the development process in general.

One of the criteria for effective operation of the brake hydraulic drive is its speed that is closely related to such characteristics of the working fluid flow as the flow rate and fluid consumption. In [12, 13, 14, 15], studies showing the relationship between the parameters of the fluid flow: fluid consumption, pressure drop, speed and the parameters of the hydraulic fluid itself: temperature and viscosity are presented. Analysis and calculation of the geometry of hydraulic channels, taking into account the parameters of the hydraulic fluid, makes it possible to select the necessary value of the diameters of the working channels in order to achieve the maximum speed of the brake mechanism.

2. Evaluation of the hydraulic fluid and channel cavitation parameters of the hydraulic block
As previously noted, when designing the hydraulic block of the brake drive of the open-pit dump truck, it is necessary to take into account a significant number of factors that affect the operation of the executive mechanisms. Figure 1a shows a geometric model of the developed hydraulic block for the open-pit dump truck with a total weight of 75 tons. The channel parameters of this block were determined to predict the nature of the fluid flow at different viscosity values, which, in turn, depend on the temperature. Pressure values on the block channel walls were also obtained, which can be used in strength calculations for rational selection of the hydraulic block housing material and, as a result, reduce the value of deformation to reduce the volume of leaks. In addition, the influence of roughness at different viscosity values on output parameters (hydro-mechanical losses) is considered. The latter will help you choose the best way to process the block channels to find a balance between cost, efficiency, and the impact on the probability of cavitation.

Figure 1b shows a diagram of the section of the hydraulic block for which the calculated study was performed. The red color shows the channels that will be used in the model, provided that the valves 2, 3, 4, 5 are normally open, and the liquid flows through the valve 1 with a constant flow rate and pressure of 70 bar. In addition to calculating the resistances and studying the influence of viscosity on the flow of the working fluid in the channels, the dangerous section of the hydroblock channel is studied in order to predict the occurrence of cavitation.
2.1. Description of theoretical prerequisites

In order to determine the nature of the fluid flow in the channels of the hydraulic unit at the design stage and to prevent undesirable phenomena such as cavitation, as well as to account for hydro-mechanical losses, the design hydrodynamic analysis (CFD analysis) in the Ansys CFX 19.2. software package was performed.

In the first approximation, the SST-HL (Shear Stress Transport) model was chosen as the design turbulence model, which simultaneously uses two turbulence models of k-ε and k-ω types. The k-ω turbulence model behaves most reliably in the boundary layer, and the k-ε turbulence model shows itself best in the situation of transition of the liquid flow to a free flow. The HL modification, which is an addition to the SST turbulence model, allows the most accurate modeling of the separation of the liquid flow from the channel wall.

In the second approximation, the Reynolds stress model, namely the SSG (Reynolds-stress) was used as the design model. This model is used for calculations of more complex engineering problems, since the k-ε and k-ω models have significant disadvantages in real turbulent flows. One of the most important functions in the Reynolds stress model is the pressure-strain correlation, $\Phi_{ij}$. It is used to stimulate turbulence to an isotropic state by redistributing Reynolds stresses.

The pressure strain member can be divided into two parts:

$$\Phi_{ij} = \Phi_{ij,1} + \Phi_{ij,2}$$

where, $\Phi_{ij,1}$ is the "slow" component, also known as the "return to isotropy" term; $\Phi_{ij,2}$ – the “fast” component of the equation.

The SSG model, unlike other Reynolds stress models where the pressure-strain correlation is linear, uses quadratic pressure-strain correlation relationships:

$$\Phi_{ij,1} = -\rho \varepsilon [C_{s1}a_{ij} + C_{s2}\left(a_{ik}a_{kj} - \frac{1}{3}a_{mn}a_{mn}\delta_{ij}\right)]$$

$$\Phi_{ij,2} = -C_{r1} P a_{ij} + C_{r2} \rho k S_{ij} - C_{r3} \rho k S_{ij} \sqrt{a_{mn}a_{mn}} +$$

$$+ C_{r4} \rho k \left(a_{ik}S_{jk} + a_{jk}S_{ik} - \frac{2}{3}a_{kl}S_{kl}\delta_{ij}\right) + C_{r5} \rho k (a_{ik} \Omega_{jk} + a_{jk} \Omega_{ik})$$

where $\rho$ – density, $\Omega_{ij}$ – vorticity tensor, $a_{ij}$ – anisotropy tensor, $\Phi_{ij}$ – pressure-strain correlation, $\bar{u}_i \bar{u}_j$ – Reynolds stress tensor, $\varepsilon$ – dissipation rate of turbulent kinetic energy, $\rho$ – fluid density, $C_{r1}, C_{r2}, C_{r3}, C_{r4}, C_{r5}, C_{s1}, C_{s2}$ – constants, $S_{ij}$ – average strain rate, $\delta_{ij}$ – the Kronecker coefficient, $k$ –

![Figure 1. The hydraulic block of the open-pit dump truck: a – the geometric model; b – the hydraulic diagram of the studied section of the hydraulic block.](image-url)
average turbulent kinetic energy, $U_i$, $U_j$ – speeds.

In addition to the turbulence model, the cavitation model was also used in the design of the hydraulic block, in the channels of which cavitation may occur. When estimating cavitation, a preliminary calculation for a channel with a convergent solution without a cavitation model was carried out. In the AnsysCFX, the Rayleigh-Plesset model was used as a cavitation model, implemented in a multiphase structure as an interphase mass transfer model.

2.2. Description of the design model of the hydraulic block channels

Figure 2 shows a model of the studied section of the hydraulic block channels (see figure 1b). To build a better-quality design finite element grid, channels that do not affect the calculation results were removed from the geometric model, but these channels were used in the calculations to determine the pressure on the hydroblock's walls.

![Figure 2. The hydraulic block channels.](image)

When constructing the finite element grid (see figure 3), the “Inflation” function was used in the Ansys CFX software package along the channel walls to more accurately determine the fluid flow regime in the boundary layer and to obtain more accurate solutions using the selected SST-HL and SSG turbulence models.

![Figure 3. The finite element grid.](image)

Setting the fluid pressure parameters at the inlet and its flow rate at the outlet is one of the reliable ways to set the boundary conditions with the highest probability of convergence. The inlet pressure “TotalPressure” equal to 70 bar, and the “MassFlowRate” Outlet1 and Outlet2 (see figure 2) of mass flow rate were set as boundary conditions. The flow rate at the outlets was set to be the same, since the valves that are considered fully open in our model must open synchronously with each other, while having the same resistance at the outlet of the hydraulic block. To determine the hydro-mechanical losses in the channels of the hydraulic block, the value of channel wall roughness was considered in the settings of the design model. The calculation was performed in a stationary setting, the process was considered to be adiabatic.

As a result of the calculations, the pressure distribution along the channel wall, the flow line, the
pressure and speed in the cross section of the channels at the inlet and outlet are obtained for estimating hydro-mechanical losses.

Figure 4 shows a section of distribution block channels. This section describes an area outlined in green that has undergone several changes during development, such as a change in the channel diameter (2 and 3 mm) and a change in roughness. Calculations were also made for mass expenditures of 0.25 and 1 kg/s. Such modes of operation are dangerous, because there is a high probability of cavitation or even supercavitation, that is a mode in which cavitation caverns completely fill the channel, and the pressure characteristic is greatly deteriorated, the process of destroying the walls of channels and clogging the hydraulic system with small metal particles are also possible.

Figure 4. The section of the hydraulic block channel.

The channel with a high probability of cavitation was calculated separately using a finer finite element grid. Other sections of the hydraulic block do not have a significant impact on the processes taking place in the channel, but their accounting would significantly complicate calculations and require large computing resources.

2.3. Analysis of cavitation process occurring in hydraulic channels
During the initial analysis of the cavitation process, taking into account the temperature equal to 100°C, the mass flow rate of 1 kg/s, the wall roughness value of 6.3 microns, and the saturated steam pressure value equal to 1250 PA, the values of the volume fraction of gas in the hydraulic channel were obtained, which are shown in figure 5. Areas with a large value of the volume fraction of gas indicate the presence of a cavitation process.

Figure 5. The calculation results for estimating the cavitation process.
The influence of geometric parameters of the channel on the occurrence of cavitation is also calculated. The temperature of the hydraulic fluid was assumed to be 100 °C, the pressure was 70 bar and the roughness was 6.3 microns with a reduced mass flow rate of 0.25 kg/s.

Initially, when designing, the diameter of the channel leading to the valve was chosen equal to 2 mm. During the calculation, the value of the volume fraction of gas in the supply channel indicated the presence of cavitation in it (figure 6).

![Figure 6. Cavitation in a channel with a diameter of 2 mm.](image)

In order to reduce the formation of cavitation caverns, the diameter of the supply channel was increased to 3 mm based on the results of calculations. The nominal flow rate was limited to 30 l/min, based on the operating parameters of the valve used, which will be described in more detail in paragraph 2.4.

2.4. Findings

The calculation was made for working channels (see figure 2) with channel wall roughness of 3.2, 6.3 and 12.5 microns, at temperatures of minus 10, plus 30, 60 and 100°C, with a set inlet pressure of 70 bar and outlet flow of 30 l/min. The maximum temperature equal to 100 °C is taken into account in the calculation, despite the fact that this temperature regime is not working for the hydraulic system of the open-pit dump truck. The values of viscosity and density of the hydraulic fluid at different temperatures were established empirically and are presented in table 1.

| Roughness, mcm | Speed, m/s | Pressure, bar. |
|---------------|-----------|---------------|
|               | Inlet     | Outlet1 | Outlet2 | Inlet | Outlet1 | Outlet2 |
| −10           | 3.2       | 4.92    | 10.28   | 10.29 | 69.87   | 43.20   | 43.29   |
|               | 6.3       | 4.92    | 10.28   | 10.29 | 69.87   | 43.19   | 43.29   |
|               | 12.5      | 4.92    | 10.28   | 10.29 | 69.87   | 43.17   | 43.27   |

Based on the above-given conditions, the speed and pressure values were obtained, averaged over the area of the input and output sections depending on the temperature, that is, the fluid viscosity and the wall roughness, which are presented in table 2.
Based on the results obtained, it is possible to estimate how much influence a certain roughness value has on the output parameters at the appropriate temperature and dynamic viscosity.

Figure 7 shows the flow lines with the distribution of the fluid speed along the channels and the pressure along the wall for the selected roughness at different temperature conditions and a roughness value of 6.3 microns.

|    | 3.2  | 4.84 | 10.31 | 10.31 | 69.89 | 48.92 | 48.46 |
|----|------|------|-------|-------|-------|-------|-------|
| 60 | 3.2  | 4.82 | 10.49 | 10.42 | 69.89 | 48.66 | 48.55 |
|    | 6.3  | 4.82 | 10.49 | 10.42 | 69.89 | 48.61 | 48.50 |
|    | 12.5 | 4.82 | 10.50 | 10.43 | 69.89 | 48.51 | 48.38 |
| 100| 3.2  | 4.81 | 10.70 | 10.57 | 69.89 | 48.68 | 48.67 |
|    | 6.3  | 4.81 | 10.60 | 10.55 | 69.89 | 48.39 | 48.63 |
|    | 12.5 | 4.81 | 10.63 | 10.69 | 69.89 | 48.35 | 48.19 |

**Figure 7.** The pressure distributions on the wall and the flow lines.
The speed value indicated on a scale related to the flow lines, it can be concluded that the value of the maximum fluid flow speed can be reached at the maximum temperature in the working channel, i.e. at the minimum viscosity value.

Below are the results of calculating the cavitation of the selected channel at a saturated steam pressure of 1250 Pa. Figure 8 shows that the intensity of cavitation formation on the channel walls is significantly higher at a roughness of 12.5 microns than at 6.3 and 3.2 microns, and the differences between 6.3 and 3.2 microns are insignificant. For these reasons, the choice of roughness in the design was made in favor of 6.3 microns.

![Figure 8. Cavitation on the channel walls of the distribution block.](Image)

When modeling fluid flow in the hydraulic block, cavitation in the operating temperature range from minus 10 to 60°C was not observed. Besides, the formation of cavitation near the channel walls at the temperature of 100°C was minimized by selecting the optimal roughness values and geometric dimensions of the channel.

3. Conclusion
As a result of the design hydrodynamic analysis of the developed distribution hydraulic block of the open-pit dump truck, the intensity of cavitation caverns occurrence was revealed at various parameters, such as surface roughness, temperature regime, fluid flow rate and the supply channel diameter. The dependence of losses on the values of roughness and temperature (viscosity) is also established. The obtained data being taken into consideration, it was possible to significantly reduce cavitation at 100 °C, thereby providing a greater cavitation margin in the open-pit dump truck hydraulic system, to avoid the phenomenon of supercavitation and to exclude it at other temperature conditions, as well as to select the necessary geometric parameters of the hydraulic block channels and the hydraulic fluid of the open-pit dump truck hydraulic system.

Acknowledgements
This study was conducted at the Bauman Moscow State Technical University financially supported by the Russian Ministry of Education and Science (Agreement about federally funded grants provided in the form of subsidies No. 14.577.21.0287 (Unique identifier of the work: RFMEFI57718X0287).

References
[1] Tamburrano P, Plummer A R, Distaso E and Amirante R 2018 A Review of Direct Drive Proportional Electrohydraulic Spool Valves: Industrial State-of-the-Art and Research Advancements *Journal of Dynamic Systems Measurement and Control* **141**(2) 020801
[2] Jerry Boza 2016 Design and Validation of an Electro-Hydraulic Pressure-Control Valve and Closed-Loop Controller *Western Michigan University* **12** p 128
[3] Jin-gang Liu, Gao-sheng Wang, Tian-heng Peng and Sheng-qiang Jiang 2019 Numerical
Simulation of Solid Particle Erosion in Aluminum Alloy Spool Valve *Multiscale and Multiphase Computational Particle Technology* vol 2019, 9465406 DOI: 10.1155/2019/9465406

[4] Jasionowski R, Polkowski W and Zasada D 2016 The Destruction Mechanism of Titanium SubJECTED to Cavitation Erosion *Key Engineering Materials* 687 117–122

[5] Yuan-Jian Yang, Peng Weiwen, Debiao Meng and Shun-Peng Zhu 2014 Reliability analysis of direct drive electrohydraulic servo valves based on a wear degradation process and individual differences *Proceedings of the Institution of Mechanical Engineers Part O Journal of Risk and Reliability* 228(6) 621–630

[6] Kayakol N 2016 Cavitation modelling in micro channels *Conference Paper* May 2016 p 7

[7] Lomakin V O, Kuleshova M S and Kraeva E A 2015 Fluid flow in the throttle channel in the presence of cavitation *Dynamics and Vibroacoustics of Machines, Procedia Engineering* 106 27–35

[8] Rudolf P, Juliš M, Klakurková L, Gejdoš P and Hudec M 2019 *IOP Conf. Series: Earth and Environmental Science* 240 062057

[9] Rooze J *at al.* 2012 Hydrodynamic cavitation in micro channels with channel sizes of 100 and 750 micrometers *Microfluid Nanofluid* 12 499–508

[10] Araz Sheibani Aghdam, Morteza Ghorbani, Gokberk Deprem, Fevzi Çakmak Cebeci and Ali Koşar 2019 A New Method for Intense Cavitation Bubble Generation on Layer-by-Layer Assembled SLIPS *Scientific Reports* 9 11600

[11] Ji Pei, Majeed Koranteng Osman, Wenjie Wang, Desmond Appiah, Tingyun Yin and Qifan Deng 2019 A Practical Method for Speeding up the Cavitation Prediction in an Industrial Double-Suction Centrifugal Pump *Energies* 12 2088

[12] Saeed Sh, Aboul-Fotouh T M and Ashour I 2016 A Current Viscosity of Different Egyptian Crude Oils: Measurements and Modeling Over a Certain Range of Temperature and Pressure *J. Pet Environ Biotechnol* vol 7 6 1000305

[13] Knežević D M, Lovrec D, Mitar J and Karanovic V 2009 Determination of Pressure Losses in Hydraulic Pipeline Systems by Considering Temperature and Pressure *Strojinski Vestnik* 55(4) 237–243

[14] Knežević D and Savić V 2006 Mathematical modeling of changing of dynamic viscosity, as a function of temperature and pressure, of mineral oils for hydraulic systems *Mechanical Engineering* vol 4 1 27–34

[15] Bair S and Michael P 2010 Modelling the pressure and temperature dependence of viscosity and volume for hydraulic fluids *International Journal of Fluid Power* vol 11 2 37–42