

A METHODOLOGICAL APPROACH TO ASSESSING THE DURABILITY OF WELDED STRUCTURES OF SCREENS USING SOLIDWORKS SIMULATION SOFTWARE

Purpose. To develop methodological recommendations for studying the characteristics of screen welded assemblies of ГВЧ-31 C type using specialized tools of the SolidWorks Education Edition software package.

Methodology. The methodology for analyzing welds when designing a vibrator fixation system is based on the use of current software tools, in particular SolidWorks Simulation in combination with SolidWorks Motion, as well as the HOT SPOT STRESS technology. To analyze the accuracy of determining the dynamic loads arising during the screen operation, the results of modeling using the SolidWorks Motion program are compared with those obtained by traditional methods using the Lagrange equation of the second kind. The computation discrepancy is within 10 %.

Findings. It has been determined that it is rational to analyze welds using the SolidWorks Education Edition software shell, taking into account the least profitable option, namely, when their edges are not finished. Furthermore, during solid-state modeling, the formation of a cavity in the weld is simulated in the form of a gap between the welded elements. The stress-strain state analysis of welded constructions using the Finite Element Method has revealed that when the computational mesh cell size is reduced, the stress value increases significantly, which indicates the result singularity and the need for further application of the HOT SPOT STRESS method. In the course of research, it has been determined that the highest stresses in welds occur on the central rib. The duration of use of welded joints is more than $2 \cdot 10^6$ cycles.

Originality. For the first time, the methodology has been substantiated for determining the service life of welded elements of screen constructions using the latest computer analysis tools. The method makes it possible to calculate the dynamic forces arising in constructions performing oscillatory movements, as well as to analyze the stress-strain state of welds using the HOT SPOT STRESS technology and assess their service life.

Practical value. It has become possible to implement the method for calculating the stress-strain state of welds in the design of a vibrator fixation system and assessing their service life, which is important for analyzing and determining the parameters of various vibrating machines. The results derived from the investigation were incorporated by the local entity, Ukrprommineral, to modernize the screens.

Keywords: *screen, SolidWorks, SolidWorks Motion, SolidWorks Simulation, welded joints, stresses, Hot Spot Stress*

Introduction. Researchers and engineers pay great attention to the development of scientific methods for determining the design characteristics of technological machines. A great contribution to the development of this subject has been made by the staff of the National Technical University Dnipro Polytechnic.

Thus, specialists from the Department of Engineering and Design in Machinery Industry are constantly studying the subject of analysis of thin-walled elements of machines with rib reinforcement, using computer modeling methods and physical experiments to test constructions [1, 2]. Professor V. P. Franchuk, together with his students, conducts research into the field of vibration engineering, in particular, he solves the issue of modernizing the designs of screen feeders for heavy industry [3, 4]. Particular attention in scientific research at the department is paid to the calculation of design parameters, the development of calculation methods and the theory of technical characteristics of the operation of vibrating machines [5, 6].

Under the guidance of Professor K. S. Zabolotnyi, a draft program has been developed based on modern CAD systems to calculate the characteristics of screens that are used for dewatering of coal sludge. During the project development, models were created using the SolidWorks software product, and the movement of elements and dynamic changes were simulated using the SolidWorks Motion application.

As a result of one of the research stages, an application has been created that helps the designer automate the screen model development, as well as conduct the necessary research into its dynamics in a short period of time. Among the technologies used, besides the SolidWorks API software product, mention should be made of the use of the C# programming language and the Godot engine.

It turned out that the joints in the construction are stress concentrators, and special attention should be paid to the as-

semblies of vibrating machines where there are welded elements, since the characteristic alternating-sign loads cause fatigue metal destruction.

Today, the difficulty in calculating such assemblies is that the applied load is usually neglected, and the calculation result depends on the thickness of the welded parts, while the weld dimensions are found in the reference book.

At present, computer modeling is one of the mandatory stages of creating machines, which avoids the need to manufacture physical prototypes of mechanisms during the production process.

The use of computer technologies for calculating the parameters of welded constructions necessitates the development of methods for a proper transition from the calculation using the laws of material resistance to the analysis of properties using the Finite Element Method. Next, consider the method for studying welded joints called Throat Shear Method (TSM). In the process of its application, based on the calculated forces occurring in the welded elements, it is possible to determine the optimal dimensions of the weld. TSM is based on a model in which one of the parts must necessarily have the form of a shell, and the geometric dimensions of the weld are not specified. Therefore, after calculation, we obtain the values of the internal forces arising in the construction, in particular, the bending moment along the weld, one force perpendicular to the weld and two tangential ones. Considering the values of internal forces, it is possible to determine the weld dimensions. There is also another method to analyze welded joints, and this is Hot Spot Stress [7]. In contrast to TSM, here it is necessary to specify the weld dimensions, and after calculation, find out the value of stress occurring in it.

A difficult scientific-technical problem arises when it is necessary to calculate the durability and strength of vibrating equipment. Since, when using welded elements, it is necessary to understand that the weld is, of course, a stress concentrator,

which significantly reduces the service life of the construction, especially under the influence of alternating-sign loads.

Literature review. By employing up-to-date numerical analysis techniques, it is possible to determine the stresses and deformations arising under the action of loads in mechanical constructions. For example, the Finite Element Method (FEM) can be used for this purpose. In this case, the study on the stress-strain state of welded joints is of particular importance. Since, when analyzing the construction state using the FEM method with parts separated by a finite-element mesh, an effect has been identified in which a decrease in the size of the mesh cells causes a tendency to an unlimited increase in stresses, which indicates the problem singularity.

The accuracy of results for welded joints obtained using the Finite Element Method can be increased by implementing HOT SPOT STRESS (HSS) technology [7], which is based on the principle of data interpolation. In this method, the place of junction of metal joints is modeled in the form of shell, three-dimensional and square elements. When modeling welds, stress concentrators, called maximum stress points (HOT SPOT), are placed in the lower part of each joint, and the mesh cell size is selected so that it does not exceed half the thickness of the metal sheet in the zone of welding. In addition, the mesh step should be such that it is possible to measure the stresses in different mesh sections. In this case, the interpolation lines run perpendicular to the weld. Subsequently, by linear interpolation, the nominal stress value at the specified points is found. The results obtained are then corrected taking into account the coefficient of material expansion and the coefficient of effective stress concentration calculated on the basis of experimental data.

Calculated stress data in hot spots using HSS technology should be compared with the values reflected in literature sources [8, 9]. It is known that four methods can be used to analyze the results obtained [10, 11].

Thus, in particular, a welded joint between a rib and a deck (RD) made of steel (OSD) has been investigated. These methods serve to calculate the stresses determined by the HSS method at this critical point of the construction using Siemens software based on the analysis of the results obtained by the Finite Element Method (FEA) HX.12. Possible limitations and level of accuracy of the mentioned results are observed for different conditions, varying the types of elements and ways of constructing the mesh.

In general, when studying shell elements, the use of HSS technology produces consistent stress calculation results compared to those using solid-state models. The use of SolidWorks Simulation software necessitates an additional assessment of the service life of the construction operation, since the program only records the occurrence of the singularity effect. To determine this indicator, it is necessary to take into consideration the type of deformation occurring in the weld and the stress value.

When creating finite-element computational meshes, SolidWorks Simulation divides the model into tetrahedrons for using spatial finite elements in the calculation and into two-dimensional triangles for shell finite-element calculation. The latter are usually used to identify sheet constructions with rib reinforcement. Moreover, the use of shell elements in calculation compares favorably with the use of spatial models, requiring significantly less computer resources.

Using the “Butt-Weld” program function, the user has the opportunity to choose between two main categories of welded joints. The first category is the T-butt weld or fillet weld, which is ideal for joining faces placed at a certain angle relative to each other. This type of welding provides reliable junction of these complex constructions. The second category is a butt weld for joining faces placed parallel to each other. This method is suitable for welding simple and straightforward elements.

At the same time, both welding methods allow for one-sided or two-sided welding, providing engineers with flexibility

in design. The “Butt-Weld” function is ideal for modeling real butt joints.

Using the “Butt-Weld” function, engineers can automatically configure the required welded joint parameters, taking into account its specified size. Thanks to this technology, the weld will withstand the loads predicted by the calculation, while taking into account the safety factor and ultimate strength of the joint.

This unique function also allows you to select a junction type that complies with American and European welding standards. Each of these standards has its own specification and information requirements that must be included in the program to achieve the best welding results.

Therefore, in order to correctly calculate the weld parameters, it is necessary to take into account the specific design of the machine when drawing up the calculation scheme.

The scientific idea of the research is to use modern computer technologies to analyze the operation of vibrating machines. Most computer programs use Finite Element Method, and the Dassault Systemes SolidWorks Corp. program is no exception. The National Technical University Dnipro Polytechnic has licenses for SolidWorks Education Edition with the ability to use applications for calculating static and dynamic loads when analyzing the vibrating equipment operation.

Purpose is to develop and substantiate the methodology for assessing the service life of the HVCH-31 C screen welded metal constructions.

Methods. The main concept of the research is to use the SolidWorks Education Edition software package to study the stress-strain state that occurs in the joints of construction elements. To accomplish this task, careful preparation is required, in particular, the creation of a modeling methodology, the determination of optimal parameters for the finite-element mesh, as well as the selection of the correct calculation scheme. Additionally, it is necessary to develop an algorithm for creating models of assemblies in which stress-strain states may occur.

Results. Development of a fastening assembly computer model and analysis of its parameters is a key point for understanding the reasons for the occurrence of the stress-strain state in the screen welded elements.

To analyze the mentioned phenomenon, it is important to construct a correct model of the junction between different construction components, that is, to take into account all the peculiarities of welded joints. Finite-element analysis in programs such as SolidWorks makes it possible to determine the distribution of stresses and deformations in the fastening assembly and draw conclusions about its reliability and durability.

The fastening assembly (Fig. 1) is a base for placing vibrators. This assembly interacts with vertical and horizontal plates 2, which ensures the rigidity and stability of the construction. Pipe 3 is the main connecting element of these

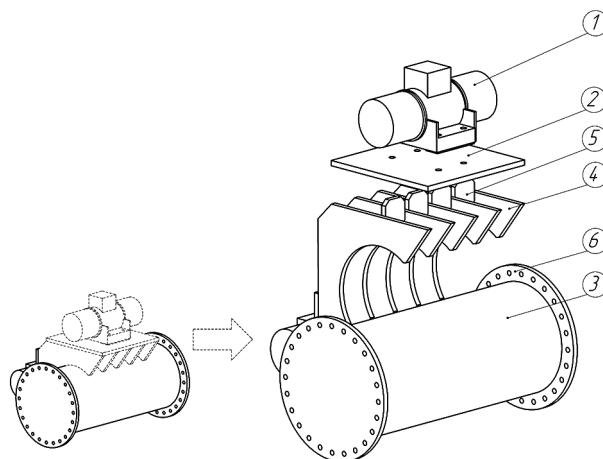


Fig. 1. Vibrating screen design

plates, which is welded to them. Additional strength and rigidity of the construction is achieved by setting transverse 4 and longitudinal 5 ribs.

The assembly construction is fastened to the screen using special flanges 6. When the vibrator generates a tensile force, it is transferred to plate 2, after which it propagates through the longitudinal and transverse ribs to the main pipe. In this case, the main force acts on the longitudinal ribs, while the greatest stress is observed in the middle rib.

The screen (Fig. 2) is equipped with a non-directional single-shaft inertial vibrating actuator 1. Its drive is not mounted in the gravity center of the screen. As a result of such a drive placement, the screen box 2 is able to move in three directions: horizontally (along the x-axis), vertically (along the z-axis) and to perform rotational oscillations around its axis (transversely, along the y-axis). Given the relatively low rigidity of elastic components 3, our analysis does not take into account their potential slight deviation from the vertical position relative to the gravity center.

During active operation, the screen is influenced by a force directed at an angle of 45°, which is a consequence of the operation of the vibrators. It concentrates gravity force, inertial forces and reactions from the springs. Disregarding inertial forces, one spring exhibits a reaction force of 9.65 kN. Simultaneously, the SolidWorks Motion software's dynamic evaluation reveals a maximum force of 7 kN at the connection point.

To verify the results obtained in SolidWorks Motion, the dynamic reaction forces in the springs are calculated using the analytical method and the Lagrange equation of the second kind, adapted to a specific screen operating scenario, namely

$$\frac{d}{dt} \left(\frac{\partial T}{\partial \dot{q}_i} \right) - \frac{\partial T}{\partial q_i} + \frac{\partial \Pi}{\partial q_i} + \frac{\partial D}{\partial \dot{q}_i} = Q_i,$$

where i is the box movement freedom degree; Q_i – summarized force value.

In this case, the kinetic energy of the system is

$$T = \frac{m\dot{x}^2}{2} + \frac{m\dot{z}^2}{2} + \frac{J_y\dot{\varphi}^2}{2} + \frac{m_0(\dot{x} + \dot{x}_0 + h\dot{\varphi})^2}{2} + \frac{m_0(\dot{z} + \dot{z}_0)^2}{2},$$

with m – weight of the screen casing, kg; m_0 – imbalance mass, kg; h – distance from the center of gravity of the box to the vibrator's axis, m; J_x – moment of inertia with respect to the y-axis, kg · m²; φ , x , z – generalized coordinates, m; $x_0 = r \sin(\omega t)$, $z_0 = r \cos(\omega t)$, if r – distance to the gravitational center of the imbalance mass; ω – rotational angular speed of the motor-vibrator rotor.

Then the potential energy of the system is

$$\Pi = \frac{c_x x^2}{2} + \frac{c_z(z - b\varphi)^2}{2} + \frac{c_z(z + b\varphi)^2}{2},$$

with c_x – the elastic element longitudinal rigidity; c_z – the elastic component vertical rigidity; b – distance from the gravity center to the axis of the elastic component.

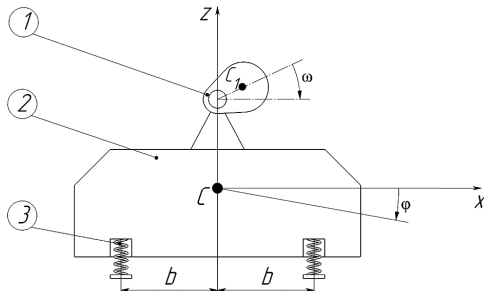


Fig. 2. Calculation model

Rayleigh function is

$$D = \frac{\mu c_x \dot{x}^2}{2} + \frac{\mu c_z (\dot{z} - b\dot{\varphi})^2}{2} + \frac{\mu c_z (\dot{z} + b\dot{\varphi})^2}{2},$$

where μ – dissipation factor.

Utilizing the aforementioned equation and considering $Q_i = 0$, subsequent manipulations yield the equation characterizing the screen movement, specifically

$$\begin{cases} (m + m_0)\ddot{x} + m_0\ddot{x}_0 + m_0h\ddot{\varphi} + c_x x + \mu c_x \dot{x} = 0 \\ (m + m_0)\ddot{z} + m_0\ddot{z}_0 + m_0\ddot{\varphi} + c_z z + \mu c_z \dot{z} = 0 \\ (J_y + m_0 h^2)\ddot{\varphi} + m_0 h \ddot{x}_0 + c_z b^2 \varphi + \mu c_z b^2 \dot{\varphi} = 0 \end{cases}.$$

The second equation in the presented scheme is independent, therefore it is solved separately, that is

$$(m + m_0)\ddot{z} + c_z z + \mu c_z \dot{z} = m_0 r \omega^2 \cos \omega t,$$

with $m_0 r$ – the kinetostatic moment, determined by the following dependence as: $m_0 r = \frac{F_v}{\omega^2}$, where F_v – coercive force, N; ω – vibrator shaft rotation velocity, s⁻¹.

The solution is assumed to be as follows

$$z = A_z \sin \omega t + B_z \cos \omega t.$$

Equating the coefficients with each other under the assumption that the harmonics of oscillations are the same, the following system of equations can be obtained after transformations

$$\begin{cases} (c_z - M\omega^2)A_z - \mu\omega c_z B_z = 0 \\ \mu\omega c_z A_z + (c_z - M\omega^2)B_z = m_0 r \omega^2 \end{cases}$$

From here equations are compiled by which the amplitude of oscillations can be determined, namely

$$A_z = \frac{m_0 r \omega^2 \mu \omega c_z}{(c_z - M\omega^2)^2 + (\mu \omega c_z)^2};$$

$$B_z = \frac{m_0 r \omega^2 (c_z - M\omega^2)}{(c_z - M\omega^2)^2 + (\mu \omega c_z)^2},$$

where $M = m + m_0$ – total weight, kg.

Adopt the equation's solution as follows

$$z = a_z \cos(\omega t + \vartheta_z).$$

Then, the amplitude of oscillations is

$$a_z = \sqrt{A_z^2 + B_z^2} = \frac{m_0 r \omega^2}{\sqrt{(c_z - M\omega^2)^2 + (\mu \omega c_z)^2}},$$

phase angle

$$\vartheta_z = \arctg \frac{B_z}{A_z} = \arctg \frac{c_z - M\omega^2}{\mu \omega c_z}.$$

Transform the first and third equations of the system

$$\begin{cases} (m + m_0)\ddot{x} + m_0 h \ddot{\varphi} + c_x x + \mu c_x \dot{x} = m_0 r \omega^2 \sin \omega t \\ (J_y + m_0 h^2)\ddot{\varphi} + m_0 h \ddot{x}_0 + c_z b^2 \varphi + \mu c_z b^2 \dot{\varphi} = m_0 r h \omega^2 \sin \omega t \end{cases}$$

Now the solution takes the following form

$$x = A_x \sin \omega t + B_x \cos \omega t;$$

$$\varphi = A_\varphi \sin \omega t + B_\varphi \cos \omega t.$$

Substitute these values into a system of differential equations that describe the box motion, and equating the coefficients with each other under the assumption that the harmonics of the oscillations are the same, a system of algebraic equations is composed, from which the unknown values A and B can be determined

$$\begin{cases} (c_x - M\omega^2)A_x - \mu\omega c_x B_x - m_0 h\omega^2 A_\varphi = m_0 r\omega^2 \\ \mu\omega c_x A_x + (c_x - M\omega^2)B_x - m_0 h\omega^2 B_\varphi = 0 \\ -m_0 h\omega^2 A_x + (c_z b^2 - (J_y + m_0 h^2)\omega^2)A_\varphi - \mu\omega c_z b^2 B_\varphi = m_0 r h\omega^2 \\ -m_0 h\omega^2 B_x + \mu\omega c_z b^2 A_\varphi + (c_z b^2 - (J_y + m_0 h^2)\omega^2)B_\varphi = 0 \end{cases}$$

From this system of equations, the unknown coefficient values are determined, namely

$$A_x = \frac{\Delta A_x}{\Delta}; \quad B_x = \frac{\Delta B_x}{\Delta}; \quad A_\varphi = \frac{\Delta A_\varphi}{\Delta}; \quad B_\varphi = \frac{\Delta B_\varphi}{\Delta},$$

where

$$\Delta = \begin{vmatrix} (c_x - M\omega^2) & -\mu\omega c_x & -m_0 h\omega^2 & 0 \\ \mu\omega c_x & (c_x - M\omega^2) & 0 & -m_0 h\omega^2 \\ -m_0 h\omega^2 & 0 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) & -\mu\omega c_z b^2 \\ 0 & -m_0 h\omega^2 & \mu\omega c_z b^2 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) \end{vmatrix};$$

$$\Delta A_x = \begin{vmatrix} m_0 h\omega^2 & -\mu\omega c_x & -m_0 h\omega^2 & 0 \\ 0 & (c_x - M\omega^2) & 0 & -m_0 h\omega^2 \\ m_0 r h\omega^2 & 0 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) & -\mu\omega c_z b^2 \\ 0 & -m_0 h\omega^2 & \mu\omega c_z b^2 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) \end{vmatrix};$$

$$\Delta B_x = \begin{vmatrix} (c_x - M\omega^2) & -m_0 r\omega^2 & -m_0 h\omega^2 & 0 \\ \mu\omega c_x & 0 & 0 & -m_0 h\omega^2 \\ -m_0 h\omega^2 & m_0 r h\omega^2 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) & -\mu\omega c_z b^2 \\ 0 & 0 & \mu\omega c_z b^2 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) \end{vmatrix};$$

$$\Delta A_\varphi = \begin{vmatrix} (c_x - M\omega^2) & -\mu\omega c_x & -m_0 r\omega^2 & 0 \\ \mu\omega c_x & (c_x - M\omega^2) & 0 & -m_0 h\omega^2 \\ -m_0 h\omega^2 & 0 & m_0 r h\omega^2 & -\mu\omega c_z b^2 \\ 0 & -m_0 h\omega^2 & 0 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) \end{vmatrix};$$

$$\Delta B_\varphi = \begin{vmatrix} (c_x - M\omega^2) & -\mu\omega c_x & -m_0 h\omega^2 & m_0 r\omega^2 \\ \mu\omega c_x & (c_x - M\omega^2) & 0 & 0 \\ -m_0 h\omega^2 & 0 & (c_z b^2 - (J_y + m_0 h^2)\omega^2) & -m_0 r h\omega^2 \\ 0 & -m_0 h\omega^2 & -\mu\omega c_z b^2 & 0 \end{vmatrix}.$$

The solution results can be presented in the following form

$$x = a_x \sin(\omega t + \vartheta_x); \quad \varphi = a_\varphi \sin(\omega t + \vartheta_\varphi),$$

where

$$a_x = \sqrt{A_x^2 + B_x^2}; \quad a_\varphi = \sqrt{A_\varphi^2 + B_\varphi^2};$$

$$\vartheta_x = \arctg \frac{B_x}{A_x}; \quad \vartheta_\varphi = \arctg \frac{B_\varphi}{A_\varphi}.$$

Using the well-known equation describing the screen body motion along the Z-axis, and using the D'Alembert's principle, the dynamic loads acting on the springs can be easily calculated. In operational scenarios, these loads roughly equate to 90 % of the forces determined through the SolidWorks Motion software. As can be seen, the importance of using this method for modeling the dynamic characteristics of the screen and for determining the loads occurring on elastic elements is confirmed.

To calculate the stress-strain state parameters in screen components using SolidWorks Simulation, the computer model of the latter can be simplified by extracting parts that do not have a significant impact on the calculation result. Thus, the model does not reflect bolt joints and holes for them, the mesh with supporting elements for it, as well as elastic parts. The vibrator is also removed, but using the "Parting Line" function, the points where it interacts with the plate are shown (Fig. 4).

Consider the most critical case, when the welds are made without edge finishing and a "lack of fusion" zone is formed in them. In this case, a gap is formed between the welded parts and the weld, which is a stress concentrator, and this reduces the joint durability.

Vibrational forces act on plate 2 (Fig. 2) and are transferred through longitudinal 4 and transverse 5 ribs to pipe 3.

Most of the forces are directed to the longitudinal ribs, and the greatest load falls on the central rib. As the simplest model of interaction between a plate and a longitudinal rib, a T-formed welded joint with lack of fusion is considered.

There are two approaches for modeling the lack of fusion, as provided by SolidWorks. One of the options is to represent this defect as a rectangular crack with a width of 1 mm. According to the second option, it may be a linear crack (Fig. 4).

In the SolidWorks Simulation application, the crack is reproduced taking into account the contact between adjacent surfaces, but such modeling causes increased computation time. In the studied case, the forces in the welds arise under the influence of tension, so it is possible to consider a contact using "penetration is allowed" function.

In the scheme of a welded joint, developed by the two described methods, it is provided that a force is applied to the upper end of the rib, while the lower part of the plate is fastened.

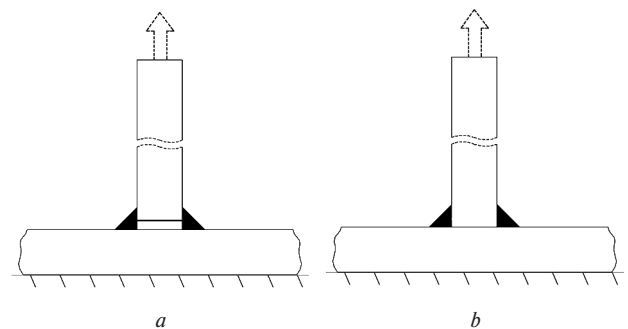
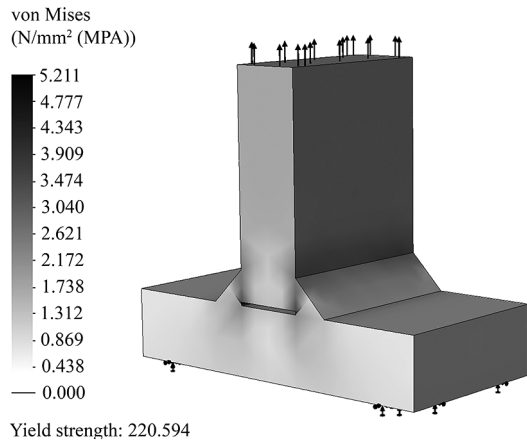
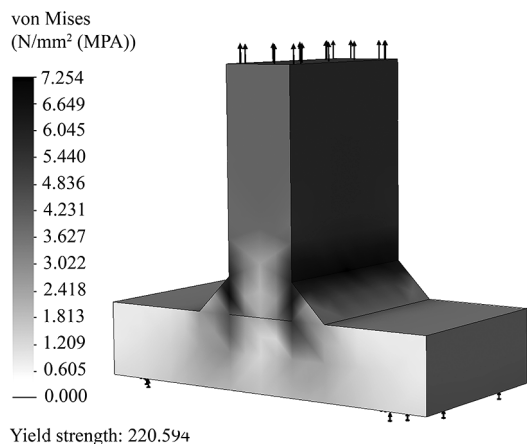


Fig. 3. Model for representing "lack of fusion" as – gap (a); tangency (b)



a



b

Fig. 4. Dependence of the field of equivalent stresses in a welded joint on the mesh cell size – gap (a); tangency (b)

tened (Figs. 3, a, b). To employ the FEM method in analyzing a welded T-shaped structure, one must ascertain the parameters of the computational mesh. It is stipulated that the average size of each mesh cell is selected from a range of values of 10, 5, 1 mm. In addition, additional analysis is performed for setting mesh parameter, and the average element size is specified to 1 mm and a layer-to-layer mesh transition rate is 1.5. Next, define two types of calculations: “problem with a gap” for the model (Fig. 3, a) and “problem with a tangency” for the model (Fig. 3, b). The findings of the study can be found in Table 1 and Figs. 4, a, b. In this context, mesh quality refers to the highest ratio of the dimensions of its cell sides.

Table 1

Relationship between maximum equivalent stress and mesh quality

	Cell size, mm	σ_{max} , N/m ²	Number of cells	Mesh quality
Problem with a gap	10	5.1	$2.3 \cdot 10^3$	14
	5	5.3	$1.5 \cdot 10^4$	6
	1	7.3	$1.2 \cdot 10^6$	5
	from 5 to 1	7.2	$2.2 \cdot 10^4$	8
Problem with a tangency	10	2.4	$2 \cdot 10^3$	7
	5	3.1	$1.4 \cdot 10^4$	5
	1	5.2	$1.2 \cdot 10^6$	5
	from 5 to 1	4.9	$2.8 \cdot 10^4$	5

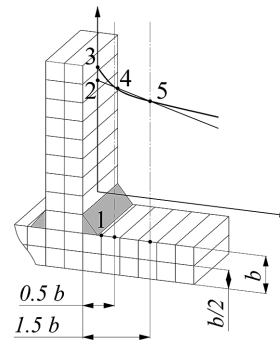


Fig. 5. Approach for ascertaining stresses in the “hot spots” of the connection through the HSS technique

Thus, based on the data in Table 1 and Figs. 4, a, b, the following conclusions can be drawn: the size of the finite-element mesh cells should correspond to the distance between the welded parts of the weld. In this context, the number of cells approaches a million. The use of “mesh control” function causes a decrease in the accuracy of stress determination by 2 %, but significantly simplifies the task. An experiment in the calculation of the “problem with a gap” demonstrates an increase in stresses by 40 %. Reducing the size of the mesh cells leads to an infinite increase in stress values, prompting the need for interpolation techniques in calculations.

Using the Hot Spot Stress method [9], on the edge of the modeled element, two measurement lines are set at a distance of $0.5b$ and $1.5b$ from the point where the base metal passes to the welded joint – 1 (Fig. 5) (where b is the thickness of the welded plates). Calculate the average stresses at points 4 – σ_1 and 5 – σ_2 , determined during edge modeling, and calculate the stresses at the point of singularity 2 σ_{HSS} by linearly interpolating the values σ_1 and σ_2 . The scheme for determining these stresses is shown in Fig. 5 and included in Table 2.

Subsequently, formulate a method to simulate the stress-strain condition in the screen’s welded components via the SolidWorks Simulation software. So, in the upper part of the screen body, the stresses in the welded joints correspond to the process of perturbation of the vibrators, while in its lower part, when inertial forces are not taken into account, but the static operating mode is considered, the stresses will be 10–15 % higher than the dynamic ones.

The main task of calculating a welded construction to determine the SSS parameters is to find the stress values at the point of fastening the vibrating devices. In this case, to simplify the task, the influence of inertia can be ignored when performing a static calculation of forces in the screen body depending on its position on stationary supports. As part of the static analysis, it is taken into account that the screen body is rigidly fastened to the base. Therefore, the interaction of vibrating devices with the fastening point is modeled under the influence of the perturbation force, FP , which is equal to 12.3 kN, as provided by the vibrator specification.

The next research step is to simulate the occurrence of a possible defect in the fastening point of vibrating devices if there is a gap between the construction parts.

After that, the solid-state models of welded joints are built between the following elements:

Table 2

Findings from the stress evaluation executed with the HSS technique

Task	σ_{max} , N/m ²	σ_{HSS} , N/m ²	Number of freedom degrees	Mesh quality
with a gap	2.27	1.298	$2.3 \cdot 10^3$	14
with a tangency	2.07	1.271	$2 \cdot 10^3$	7

- 1) a plate – a transverse rib;
- 2) a longitudinal rib – a pipe;
- 3) a transverse rib – a pipe;
- 4) a transverse rib – a pipe;
- 5) a flange – a pipe.

We assume that in a model containing less than four circular zones of a bolted joint with a vibrating device, tensile forces are applied, the value of each of which is equal to 3.075 kN. To model the fastening of the construction, the “Fixed geometry” function is used to fix the vibrators, in which the model is allowed to move, and the construction is fastened to flange cylindrical surfaces.

In order to find points in the weld with the highest equivalent stress, a study is conducted using a finite-element model, in which the average mesh cell size is 5 mm (Fig. 6).

Based on Fig. 6 data analysis, the following conclusions can be drawn:

- the highest stresses are observed at the points of fastening the vibrators to the plate, which became known based on the standard calculation of forces in bolted joints;

- the highest stress in the welded joints has been found where the central rib joins the plate and pipe. Therefore, in order to determine the service life of the welds, it is necessary to conduct an additional study using parting lines drawn at a distance of 10 and 30 mm from the weld edge, using the HSS method.

The conducted analysis indicates that the highest stress is observed in the weld connecting the plate and the rib.

In the scheme for calculating equivalent stresses based on a solid-state screen model, tensile forces equal to 3.075 kN are applied to the area on which the vibrator is set, fixed with bolts. The four circular zones on the lower screen parts indicate the points where this construction is fastened to the springs.

According to the calculation results (Fig. 7), the place where the vibrators are fastened to the screen is the most heavily loaded.

All the results obtained were carefully analyzed earlier when studying the point where the vibrator is fastened to the screen. It is known that the ultimate strength of steel 45 under the influence of tensile forces σ_{-1P} is 220 N/m² under normalization conditions. Next, determine the permissible stress in the welded joint in the fastening point, namely, MPa

$$[\sigma] = \frac{\sigma_{-1P}}{k \cdot K_{eff}} = \frac{220}{1.5 \cdot 3.5} = 41.9,$$

where K_{eff} is effective value of the stress concentration factor, $K_{eff} = 3.5$; k – safety factor, $k = 1.5$ [8].

Consequently, if the stress in the material structure remains below the $[\sigma]$ value, the service life of welded joints will exceed $2 \cdot 10^6$ cycles. Given that the highest stress in welded joints, determined using the HSS method, is equal to 2.5 N/m², it can

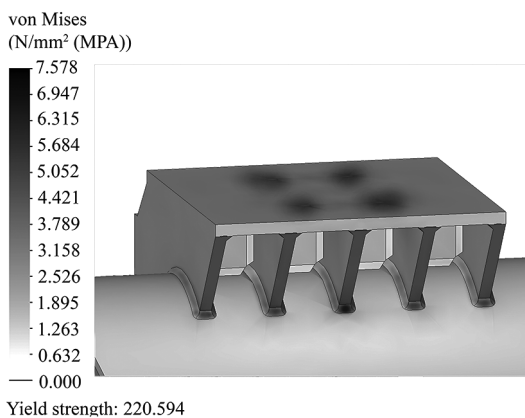


Fig. 6. Curve for determining the equivalent stresses in a weld based on a 5-millimeter computational mesh

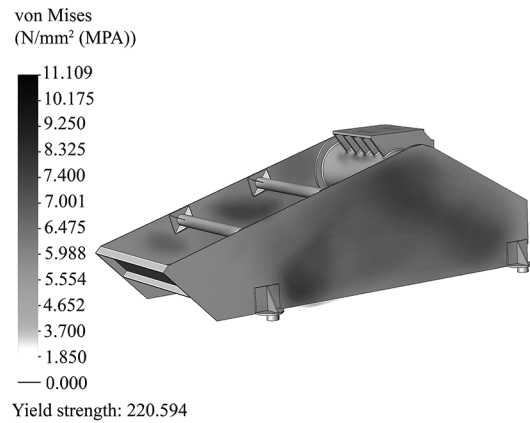


Fig. 7. Curve for determining the stress-strain state of the studied construction

be stated that the screen will be suitable for operation for at least $2 \cdot 10^6$ cycles [10].

Conclusions. When analyzing the screen welded construction, it was revealed that the welds in such a construction are stress concentrators. The need to use the SolidWorks software package in the study arose due to the fact that the analytical method of analysis considers the construction of vibrating machines as a solid body, and this limits the possibilities of determining the stresses arising there, as well as calculating the operating time of this machine.

The authors have developed a method for modeling the stress-strain state of a construction that fixes the vibrators using advanced software technologies, in particular SolidWorks Simulation, together with the SolidWorks Motion application and the HOT SPOT STRESS method. This method was used to calculate the construction welded joints fixing the vibrators in the HVCH-31 C type screen.

It has been found that welds that do not have full edge penetration should be analyzed by modeling this defect as a gap between the welded parts. The study on the stress-strain state has revealed that if the computational mesh cell size decreases, the stress increases sharply, which indicates the specificity of the problem and raises the need for further use of the HOT SPOT STRESS method. The highest stress in the welded joints is observed precisely in the middle rib, which joins the plate and pipe. It has been determined that the service life of the screen precisely with such a design of fixing the vibrators will be at least $2 \cdot 10^6$ cycles. The described research results were implemented during the modification of screens at the LLC Ukropromineral enterprise. The investigation is undertaken within the parameters of the designated scientific topic, which is presented at the Department of Engineering and Design in Machinery Industry.

References.

1. Zabolotnyi, K., Zhupiiiev, O., Panchenko, O., & Tipikin, A. (2020). Development of the concept of recurrent metamodeling to create projects of promising designs of mining machines. *E3S Web of Conferences*, 201, 01019. <https://doi.org/10.1051/e3sconf/202020101019>.
2. Zabolotnyi, K., & Panchenko, O. (2019). Development of methods for optimizing the parameters of the body of a fixed jaw crusher. *E3S Web of Conferences*, 209, 00120. <https://doi.org/10.1051/e3sconf/201910900120>.
3. Franchuk, V. P., & Antsyferov, O. V. (2022). Separation of bulk material under working surface vertical vibrations. *Fundamental and applied problems of ferrous metallurgy*, 36, 466-475. <https://doi.org/10.52150/2522-9117-2022-36-466-475>.
4. Hankevich, V., Moskalova, T., Kabakova, L., & Livak, O. (2019). The feasibility evaluation of using cyclic thermal effect in the rock-cutting tools during drilling hard rock. *E3S Web of Conferences*, 109, 00026. <https://doi.org/10.1051/e3sconf/201910900026>.
5. Iqbal, N., Fang, H., Naseem, A., Muhammad Kashif, M., & De Backer, H. (2020). A Numerical Evaluation of Structural Hot-Spot

Stress Methods in Rib-To-Deck Joint of Orthotropic Steel Deck. *Department of Civil Engineering, Ghent University, Technologiepark 60, B-9052 Zwijnaarde, Belgium, 10(19), 6924.* <https://doi.org/10.3390/app10196924>.

6. Naumov, V., Taran, I., Litvinova, Y., & Bauer, M. (2020). Optimizing Resources of Multimodal Transport Terminal for Material Flow Service. *Sustainability, 12(16)*, 6545. <https://doi.org/10.3390/su12166545>.

7. Minieiev, S., Vasyliiev, L., Trokhymets, M., Vialushkin, Y., & Moskalova, T. (2022). Heading set of equipment for underground development galleries drivage in rocks prone to gas-dynamic phenomena. *IOP Conference Series: Earth and Environmental Science* this link is disabled, 970(1), 012044. <https://doi.org/10.1088/1755-1315/970/1/012044>.

8. Sydorenko, Y. M., Pashchyn, M. O., & Mykhodui, O. L. (2020). Effect of Pulse Current on Residual Stresses in AMg6 Aluminum Alloy in Electrodynamic Treatment. *Strength of Materials, 52(5)*, 731-737. <https://doi.org/10.1007/s11223-020-00226-2>.

9. Lobanov, L. M., Pashchin, N. A., Mikhodui, O. L., & Sidorenko, Y. M. (2018). Electric Pulse Component Effect on the Stress State of AMg6 Aluminum Alloy Welded Joints Under Electrodynamic Treatment. *Strength of Materials, 50(2)*, 246-253. <https://doi.org/10.1007/s11223-017-9862-810>.

10. Lobanov, L. M., Pashchyn, M. O., & Mikhodui, O. L. (2021). Modeling of stress-strain states of AMg6 alloy due to impact action of electrode-indenter in electrodynamic treatment. *The Paton Welding Journal, 6*, 2-11. <https://doi.org/10.37434/tpwj2021.06.01>.

11. Niemi, E., Fricke, W., & Maddox, S. J. (2018). *Fatigue Analysis of Welded Components: Designer's Guide to the Structural Hot-Spot Stress*. Springer Nature Singapore Pte Ltd. ISBN 978-1-84569-124-0. <https://doi.org/10.1007/978-981-10-5568-3>.

Методика визначення ресурсу зварних конструкцій грохотів у комплексі SolidWorks Simulation

А. Шкут*

Національний технічний університет «Дніпровська політехніка», м. Дніпро, Україна

* Автор-кореспондент e-mail: shkut.an.p@nmu.one

Мета. Розробка методичних рекомендацій з оцінювання ресурсу зварних вузлів грохоту типу ГВЧ-31 С з використанням у програмному комплексі SolidWorks Education Edition.

Методика. В основу методики дослідження зварних вузлів кріплення вібраторів покладено використання сучасних програмних продуктів, таких як SolidWorks Simulation у комбінації з SolidWorks Motion і технології HOT SPOT STRESS. Для оцінки точності визначення динамічних зусиль, що виникають під час роботи грохоту, зіставлялися результати моделювання за допомогою SolidWorks Motion і класичними методами на основі рівняння Лагранжа другого роду. Похибка складає 10 %.

Результати. Визначено, що дослідження зварних швів у програмному комплексі SolidWorks Education Edition доцільно проводити на найнесприятливішому випадку, тобто при непроварі кромок. У твердотільній моделі непровар моделювали зазором між зварювальними деталями. Дослідження напружено-деформованого стану зварних конструкцій з використанням методу скінченних елементів показав, що при зменшенні елементу сітки значно зростає напруження, а це свідчить про сингулярність задачі й необхідність подальшого застосування метода HOT SPOT STRESS. У результаті досліджень отримано, що максимальні напруження у зварних швах виникають там, де поздовжнє середнє ребро з'єднане із пластиною та трубою. Довговічність зварних з'єднань перевищуватиме $2 \cdot 10^6$ циклів.

Наукова новизна. Уперше обґрунтована методика визначення ресурсів зварних конструкцій грохотів, у якій використовуються сучасні методи комп'ютерного аналізу. Методика дозволяє визначати динамічні зусилля, що виникають у конструкціях під час здійснення коливального руху й визначати напружено-деформований стан зварних швів з використанням технології HOT SPOT STRESS, оцінювати їх ресурс.

Практична значимість. Застосування методики розрахунку напружено-деформованого стану вузлів кріплення вібраторів, розрахунку їх ресурсу можливо використовувати для аналізу й розрахунку інших вібраційних машин. Результати досліджень використовувалися при модернізації грохотів підприємством «Укрпроммінерал».

Ключові слова: грохот, SolidWorks, SolidWorks Motion, SolidWorks Simulation, зварні шви, напруження, Hot Spot Stress

The manuscript was submitted 11.04.23.