*ISSN 2079-1372 **Problems of Tribology**, V. 30, No 2/116-2025, 42-48*

Problems of Tribology
Website: http://tribology.khnu.km.ua/index.php/ProbTrib
E-mail: tribosenator@gmail.com

DOI: https://doi.org/10.31891/2079-1372-2025-116-2-42-48

Modeling of contact interaction and wear of the trolleybus contact insert-wire tribopair

K.E. Holenko0000-0002-6140-4573**, O.S. Kovtun**0000-0002-1430-6479**, O.V. Dykha***0000-0003-3020-9625

Khmelnytskyi national University, Ukraine
**E-mail: tribosenator@gmail.com*

Received: 02 April 2025: Revised 20 April 2025: Accept: 10 May 2025

Abstract

The article presents the results of numerical modeling of the contact interaction of the tribopair "contact insert - contact wire" of a trolleybus using the Ansys software environment. The main attention is paid to the analysis of the stress state and the prediction of wear of friction surfaces when modeling a stable frictional contact. Based on the constructed solid-state models of the contact insert and the MF-100 copper wire, a finite element mesh was formed and real conditions of operational loading were simulated. The distributions of contact pressure, frictional and equivalent stresses, as well as the nature of the contact spot were determined. Linear wear was calculated using the energy criterion, taking into account local frictional stresses and material hardness. The obtained results were compared with the available experimental data, which confirms the adequacy of the proposed model. The article also outlines the prospects for interpolation of short-term simulations into long-term operating modes by mathematical modeling of wear under variable load. The material is useful for predicting the service life of current collection devices and optimizing their geometry and materials.

Keywords: tribocouple, contact insert, contact wire, ANSYS, wear, friction stress, contact zone, modeling.

Introduction

Wear of contact elements of electric transport, in particular contact inserts of trolleybus pantographs, remains one of the main reasons for the decrease in reliability and increase in operating costs of urban transport. During the movement of the trolleybus, the insert is constantly in contact with the wire network, which leads to intensive abrasive and adhesive wear. Change in the insert profile, loss of contact surface geometry and increase in sliding resistance reduce the quality of current collection, cause electrical sparks and deterioration of the operation of electrical equipment.

Traditionally, wear resistance assessment has been performed experimentally in laboratory or field conditions, but such methods are laborious, expensive and do not always allow to take into account the complex variability of operating modes characteristic of real conditions. Therefore, it is relevant to introduce numerical analysis tools, in particular the finite element method (FEM), which allow to model not only the mechanical interaction, but also the processes that directly affect wear, such as contact pressure, friction, local stresses, the shape of the contact spot and the geometry of the elements.

Of particular interest is the possibility of quantitative assessment of linear wear based on the distribution of frictional stresses obtained in Ansys, with the subsequent application of energy approaches to the calculation. This approach allows not only to take into account local friction characteristics, but also to predict the service life of the insert depending on the length of the run and the modes of movement of the trolleybus. This opens up prospects for integrating numerical modeling into regulatory procedures for assessing the wear resistance of current-collecting components and unifying approaches to their constructive optimization.

Literature review

The problem of wear of tribosystem elements is key in transport engineering, in particular for current-collecting pairs of electric transport. A systematic study of wear mechanisms is given in the work P. Blau [1], where transient friction modes and the influence of loading conditions on the nature of wear of materials are considered. G. Stachowiak and A. Batchelor [2] in their fundamental monograph classify in detail the types of wear — adhesive, abrasive, fretting, etc. — and consider mathematical models that describe the dependence of wear on contact pressure, sliding velocity and material properties.

Particular attention in the study of wear is paid to energy criteria. Meng and Ludema [3] summarized the main approaches to building wear models and emphasized their limitations under conditions of variable loading and unsteady contact. Their analysis of the forms and content of predictive equations highlights the need to use models that take into account local contact parameters. At the same time, modern software environments, as shown in [4], allow the use of these equations based on numerical modeling data - in particular, friction stresses and contact pressure.

Han et al. [5] presented a numerical analysis of a wear-sensitive contact problem using a fully discrete scheme to model an elastic body under friction and wear conditions. Their results demonstrate the effectiveness of the approach in predicting the evolution of the contact surface. Ravitej and Kumar [6] conducted a comparative study on the prediction of wear rate of hybrid composites by combining experimental analysis, finite element modeling, and machine learning, which allowed to improve the accuracy of predictions.

Ansys Inc. [7] provided an example of modeling the wear of a contact surface using the Archard wear model, demonstrating the process of sliding a hemispherical ring on a flat ring and evaluating the strains and normal stresses before and after wear. Lopez [8] provided an overview of contact modeling methods in Ansys Mechanical, including different contact types such as bonded, non-separated, frictionless, and rough, and discussed the contact algorithms used in Ansys to accurately model surface interactions.

Thus, modern literature confirms the effectiveness of combining classical energy models with numerical FEA data for wear analysis in tribopairs, which is a relevant and promising approach for modeling the insert and contact wire of a trolleybus.

Purpose and objectives of the study

The aim of the work is to model the stress state and assess the wear of the tribopair "contact insert - contact wire" of a trolleybus using the Ansys environment.

To achieve the goal, the following tasks were set: to create geometric models of tribopair elements; to construct a finite element mesh taking into account the parameters of contact interaction; to simulate the distribution of pressure and stresses in the contact zone; to determine wear parameters using the energy approach; to compare the obtained results with experimental data; to formulate directions for further research taking into account variable load modes.

Main material

Output data

Based on the working drawings, the following solid models were created: contact insert (Fig. 1) and cross-section of the MF-100 contact wire (Fig. 2).

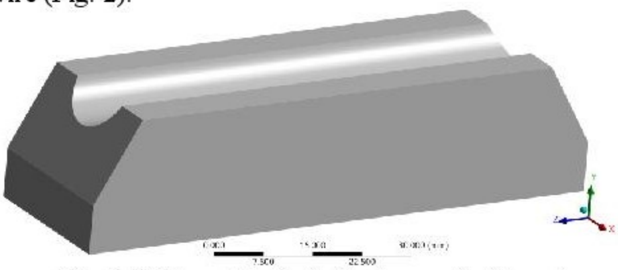

Fig. 1. Solid model of a trolleybus contact insert

The models were created in SolidWorks with working dimensions. The body length is 300 mm (solid model extrusion).

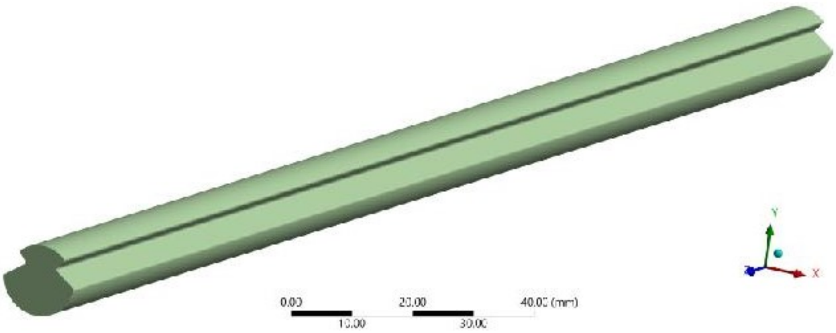

Fig. 2. Solid model of the contact wire

Based on the solid-state models of the insert and the contact wire, a finite element FEA mesh was built in the Ansys environment. The FEA mesh parameters are as follows: Body Sizing = 2.0 mm – the maximum size of the finite elements of the model bodies; Face Sizing = 1.0 mm – the size of the finite elements on the contact surfaces of the bodies; Inflaction = 9 for the wire and 12 for the insert. The parameters also determine the number of layers of finite elements with the size Face Sizing deep into the wire body and insert.

Body materials. From the available Granta EduPack library, the only possible materials that are close to the target ones were selected: Carbon (CY) for the insert; Cooper C10100 for the wire. The mesh view is presented in Fig. 3.

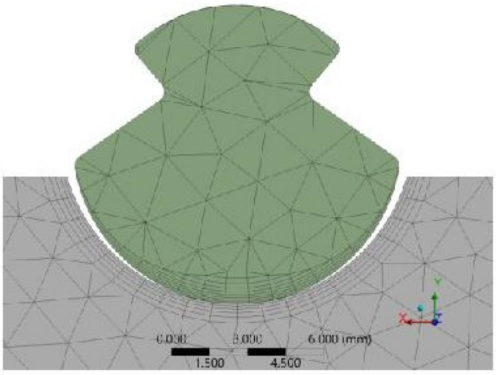

Fig. 3. FEA mesh of the contact model of the trolleybus and contact wire insert

To simulate the contact conditions for this pair, a frictional contact mode Frictional with a friction coefficient of 0.13 was specified (Fig. 4).

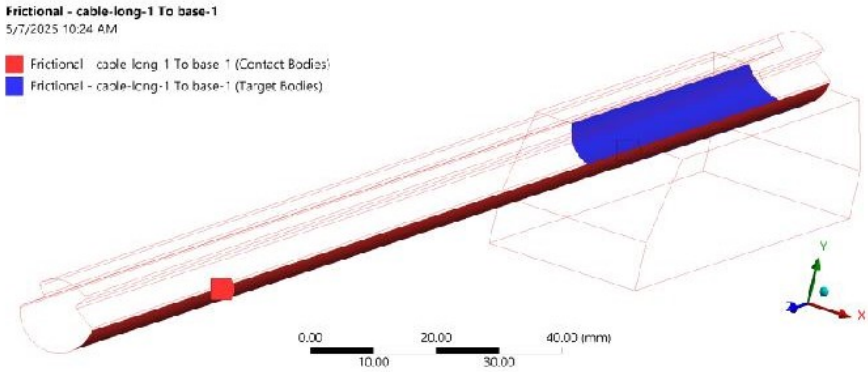

Fig. 4. Frictional contact in the insert-wire pair

To simulate contact pressure from the insert side, a uniformly distributed force of 0.5 MPa was applied to the wire perpendicular to the lower surface of the insert (Fig. 5).

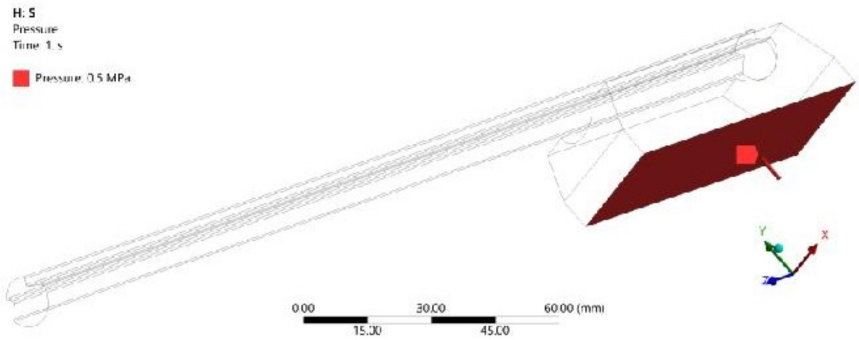

Fig. 5. Modeling the load on the contact pair

A steady state loading regime of 0.5 MPa was analyzed. The insert was displaced 200 mm along Z in both L- and S-modes.

Results of calculation of tribocontact parameters

The equivalent Mises stresses for the insert and wire are shown in Fig. 6-7, respectively. For the insertmaximum stresses σ are observedmax=16.236 MPa at the moment of time t=2.e-002 s; for wire σmax=14.33 MPa at time t=0.12 s.

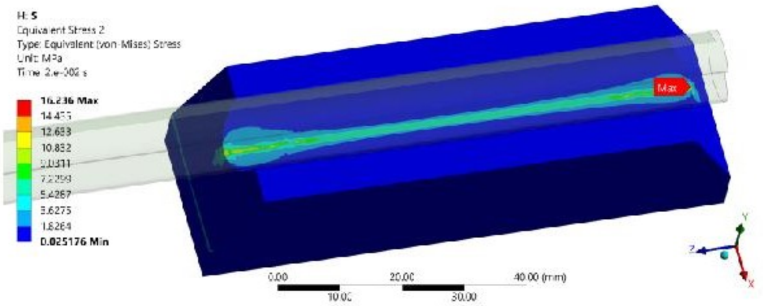

Fig. 6. Equivalent Mises stresses for the insert

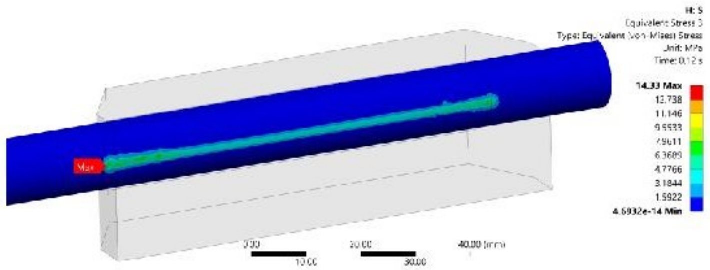

Fig. 7. Equivalent Mises stresses for a wire

Next, the characteristics of the dimensions of the contact zone (Contact Tool) were determined. A slip spot was recorded in the contact area, further from it the nature of the contact changes to a transitional one, and at the edges – to a gap (Fig. 8).

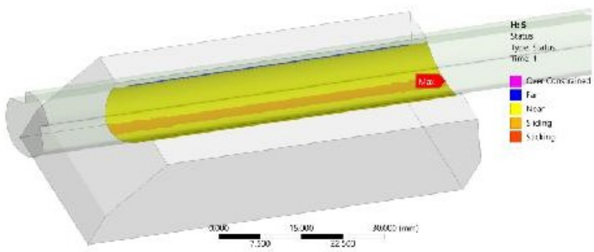

Fig. 8. Qualitative characteristic of the contact mode of the insert and wire

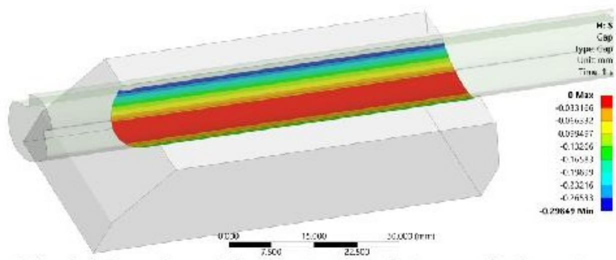

Fig. 9. The value of the contact gap between the insert and the wire

The contact gap varies from 0 to 0.29849 mm (Fig. 9). The maximum penetration is 0.0035762 mm at t=0.42 s.

The maximum pressure in the contact area is 23.951 MPa at t=0.42 s. The maximum average pressure value is 1.063 MPa at t=0.12 s. The fluctuations of the average pressure are shown in the graph (Fig. 10).

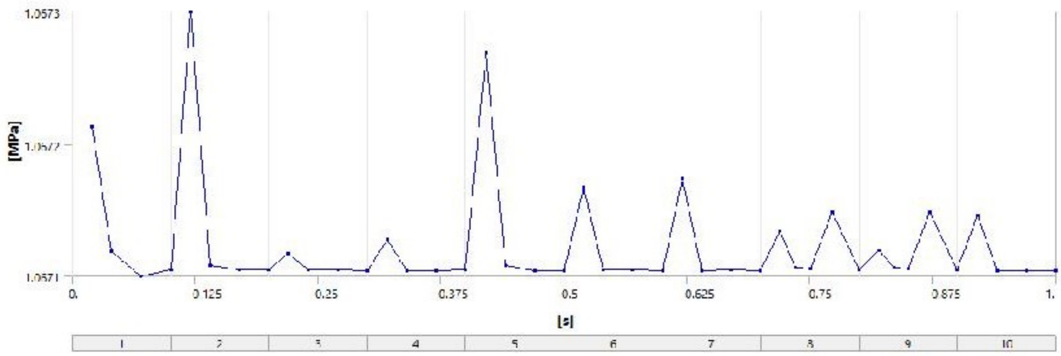

Fig. 10. Fluctuations in average pressure

Frictional stresses were used to calculate the wear characteristics in the insert-contact wire pair. Accordingly, the values of frictional stresses were further determined using the Ansys application package (Fig. 11).

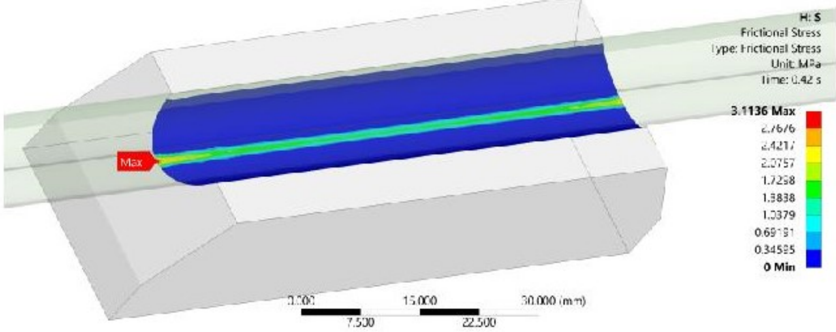

Fig. 11. Map of friction stresses on the surface of the contact insert

The maximum frictional stresses were σmax=3.1136 MPa at time t=0.42 s.
The maximum value of the average frictional stress is σmax = 0.13874 MPa at t=0.12 s. The fluctuations of the average frictional stress are shown in Fig. 12.

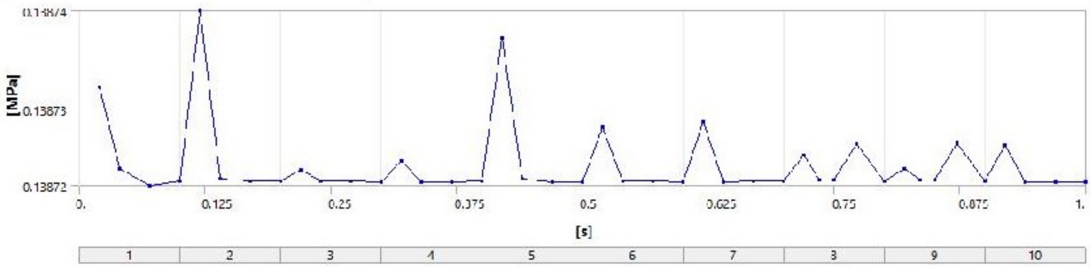

Fig. 12. Fluctuations of average friction stresses on the surface of the contact insert

Calculation of insert wear

According to the energy approach, wear is the result of the expenditure of part of the mechanical friction energy on plastic deformation and detachment of particles from the contact surface.

The general equation:

$$V = \alpha \cdot E_{frict} = \alpha \cdot \int p \cdot \Delta u dA, \quad (1)$$

where: α is the global efficiency coefficient of energy conversion into wear, mm³/J; E_{frict} is the friction energy (total or local), N m or J; p is the contact pressure or normal force per area, Pa; Δu is the relative shear displacement, m; dA is the contact area element, m².

Simplified local formula for linear wear:

$$d(x,y) = \frac{\eta \cdot \tau(x,y) \cdot s(x,y)}{H}, \quad (2)$$

where: $d(x,y)$ is the local wear depth, mm; η is the local efficiency coefficient of energy conversion into wear (≈0.01–0.1); τ is the tangential stress, Pa; s is the sliding distance, m; H is the material hardness, Pa.

Physical content:$\tau \cdot s$ is the friction work per unit area, J/m²); $\eta \cdot \tau \cdot s$ is the energy effectively spent on wear and tear(losses); H is the normalization of this energy due to the resistance of the material (the higher the hardness, the less wear).

Format for manual calculation:

$$d = \frac{\eta \cdot \tau_{avg} \cdot s_{total}}{H}, \quad (3)$$

where: τ_{avg} is the average frictional stress value from Ansys, Pa; s_{total} is the total sliding distance, m. According to FEA tests = 0.13873 MPa for the entire surface, and the max value is 3.1136 MPa. However, we will be interested only in the central part, where the permanent contact is fixed - the red area with a gap value of 0 mm (Fig. 9). The approximate value in the red area is 0.4 MPa. For the hardness of the insert material HRB = 50...75, the value varies between $H = 3.45 \cdot 50 = 172.5$ MPa and $H = 3.45 \cdot 75 = 258.75$ MPa. We take the average value $2.15 \cdot 10^8$ Pas. We consolidate the boundary conditions: $\tau_{avg} = 0.4$ MPa; $s_{total} = 0.2$ m; $H = 2.15 \cdot 10^8$ Pa; Mileage = 100 km.

Wear per 1 sliding cycle (0.2 m):

$$d = \frac{\eta \cdot \tau_{avg} \cdot s_{total}}{H} = \eta \cdot \frac{0.4 \cdot 10^6 \cdot 0.2}{2.15 \cdot 10^8} = \eta \cdot 0{,}37209 \text{mm}. \quad (4)$$

Number of cycles per 100 km: 100,000/0.2=500,000 cycles
Total wear:

$$d_{100km} = 0{,}37209 \cdot 500{,}000 \cdot \eta = 186{,}045 \cdot \eta \text{ mm}. \quad (5)$$

The result under the condition is: $\eta = 10^{-6} d_{100km} = 0.186$ mm, which correlates with experimental data on wear "Performance characteristics of VKT-M inserts when working in conjunction with MF-100 copper wire": linear wear of inserts, mm/100 km of run is +(0.1…0.2) mm under dry sliding contact conditions. Given the value of mm $\eta = 10^{-5} d_{100km} = 1.86$.

Obviously, there is a question of interpolation of FEA test results and selection of a real value of the coefficient . Also, the question of the correspondence of the physical and mechanical properties of Granta EduPack materials in Ansys to real materials in operation. The subject of future research is interpolation of the insert loading modes: with constant and linearly increasing pressure. It can be assumed that when starting from stops, the pressure value changes (increases or decreases); under the conditions of linear movement of the trolleybus and a flat road surface, the pressure value can maintain a relatively constant value; in turns, the contact spot of the insert with the wire migrates to the inner sidewalls of the insert recess, so the wear area shifts, providing a greater overall resource. Interpolation of short FEA tests to long-term operating conditions requires the creation of a mathematical prediction model, which is an integral of various driving modes during the trolleybus working shift. Undoubtedly, studying the totality of these factors can help actualize the meaning of wear and tear and bring it closer to real conditions η.

Conclusions

1. Numerical modeling of the contact interaction of the insert–wire tribopair was carried out, which allowed determining the stress-strain state and friction characteristics.
2. According to the results of FEA analysis, the values of stresses, pressures, and penetration in the contact zone were obtained, indicating the presence of a stable slip spot and a gap zone.
3. The application of the energy criterion allowed us to calculate the wear depth of the contact insert per cycle and per full mileage, which is consistent with experimental data.
4. It has been established that modeling in the Ansys environment provides a reliable assessment of wear conditions provided that the physical and mechanical parameters are correctly selected.
5. The need to develop a mathematical model for interpolation of short-term FEA tests for long-term operation, taking into account changes in load during trolleybus movement, is shown.

References

1. Blau, PJ (1989). Friction and Wear Transitions of Materials: Break-In, Run-In, Wear-In. Noyes Publications. ISBN: 9780815511960.
2. Stachowiak, GW, & Batchelor, AW (2013). Engineering Tribology (4th ed.). Butterworth-Heinemann. ISBN: 9780123970473.
3. Meng, HC, & Ludema, KC (1995). Wear models and predictive equations: their form and content. Wear, 181–183, 443–457.https://doi.org/10.1016/0043-1648(95)90158-2
4. Bhushan, B. (2013). Introduction to Tribology (2nd ed.). Wiley. ISBN: 9781119943925.
5. Han, D., Han, W., Jureczka, M., & Ochal, A. (2019). Numerical Analysis of a Contact Problem with Wear. arXiv preprint arXiv:1905.05541.https://arxiv.org/abs/1905.05541arXiv
6. Ravitej, YP, & Kumar, R. (2025). Wear Rate Prediction of Hybrid Composites: A Comparative Study Using Experimental Analysis, Finite Element Simulation, and Machine Learning. SSRN Electronic Journal.https://doi.org/10.2139/ssrn.5177817SSRN
7. Ansys Inc. (2024). Contact Surface Wear Simulation. PyMechanical Examples.https://examples.mechanical.docs.pyansys.com/examples/technology_showcase/example_07_td_043.htmlpymechanical
8. López, S. (2024). Contact Modeling in Ansys Mechanical for Beginners. Ozen Engineering Blog.https://blog.ozeninc.com/resources/understanding-contacts-in-ansys-mechanicalOzen Inc Blog+1LinkedIn+1

Голенко К.Е., Ковтун О.С., Диха О.В. Моделювання контактної взаємодії та зносу трибопари контактна вставка-провід тролейбуса

У статті представлено результати чисельного моделювання контактної взаємодії трибопари «контактна вставка – контактний провід» тролейбуса із використанням програмного середовища Ansys. Основну увагу приділено аналізу напруженого стану та прогнозуванню зношування поверхонь тертя при моделюванні сталого фрикційного контакту. На основі побудованих твердотільних моделей контактної вставки та мідного дроту МФ-100 сформовано кінцево-елементну сітку та змодельовано реальні умови експлуатаційного навантаження. Визначено розподіли контактного тиску, фрикційних та еквівалентних напружень, а також характер контактної плями. Розраховано лінійний знос за енергетичним критерієм із урахуванням локальних напружень тертя та твердості матеріалів. Отримані результати порівняно з наявними експериментальними даними, що підтверджує адекватність запропонованої моделі. Стаття також окреслює перспективи інтерполяції короткочасних моделювань на довготривалі режими експлуатації шляхом математичного моделювання зносу при змінному навантаженні. Матеріал є корисним для прогнозування ресурсу струмознімальних пристроїв та оптимізації їх геометрії й матеріалів.

Ключові слова: трибопара, контактна вставка, контактний провід, ANSYS, знос, напруження тертя, контактна зона, моделювання.