Problems of Tribology, V. 28, No 3/109-2023, 41-48

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

DOI: https://doi.org/10.31891/2079-1372-2023-109-3-41-48

Simulation model of contact interaction during surface strengthening of steel parts

K. Holenko*, V. Dytyniuk, M. Dykha

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

Received: 20 August 2023: Revised 10 September 2023: Accept 20 September 2023

Abstract

In the processes of surface strengthening of steel parts, the stress-strain state is decisive for explaining the physical processes of strengthening, forming the dimensions of the contact area. Analytical dependences of contact parameters are quite approximate. In this work, based on the Ansys software complex, a simulated model of the contact of a truncated torus with a cylinder is proposed, which demonstrates the kinetics of the process of pressing a hard alloy tool into a steel workpiece - a cylinder. The experiment was conducted for 4 seconds in order to determine the maximum level of stresses, the distribution of stresses and the amount of residual stresses after removing the load. The clamping force was applied mainly in the zone of elastic deformations. The results showed an uneven stress distribution with a maximum in the center of the contact spot of 1082 MPa. After changing the load direction, small residual deformations at the level of 0.00311 μm were observed in the center of the contact patch. This indicates a violation of the elastic region on a small contact area, which does not affect the general nature of the stress distribution and can be removed during the finishing process. The results of simulation of the stressed state are used for the correlation with the observed structural changes of the material during the action of thermal and power stresses. The stress peak was formed at a distance of 200 μm, which contributes to the formation of maximum values of microhardness at this depth.

Key words: stress-strain state, surface, strengthening, contact processing, tool, truncated torus, cylinder, elastic deformations

Introduction

The method of discrete strengthening of steel parts of the "shaft" type requires the need to assess the depth of the strengthened wear-resistant layer. A reinforced layer is a layer characterized by the formation of a so-called white layer. This layer is formed in the volumes of the material, the heating temperature of which exceeds the phase transformation temperature. A high-temperature volume can be defined as a volume in which the temperature is above 600 °C.

Experimental studies show that the width and height of the high-temperature volume in working hardening modes are close to the width and height of the contact of the tool with the part. Therefore, determining the contact surface of the tool and the processed part is one of the first steps necessary for the correct selection of technological parameters.

In addition, knowledge of the geometric parameters of the contact surface is required to determine other characteristics of the technological process, in particular, the current density, which ensures the necessary temperature on the surface of the contact zone.

The purpose of the task is to create a calculation method for assessing the influence of the load of the run-in roller on the resulting stresses and deformations of the shaft when simulating their mutual rotation in the environment of the finite element method (Ansys Static Structural).

Literature rewier

In paper [1] describes the development of a 3D tyre–pavement interaction model to predict the tyre–pavement contact stress distributions for future use in the mechanistic analysis of pavement responses. The steady-state tyre rolling process was simulated using an arbitrary Lagrangian Eulerian formulation. The model results are consistent with previous measurements and validate the existence of non-uniform vertical contact stresses and localised tangential contact stresses. The analysis results show that the non-uniformity of vertical contact stresses decreases as the load increases, but increases as the inflation pressure increases. The model results provide valuable insights into understanding the realistic tyre–pavement interaction for analysing pavement responses at critical loading conditions.

In article [2] have formulated a computational theory based on the 'soft contact' approach that models contact through localized non-permanent deformation in the vicinity of contact. The model of mechanical contact between polyhedral objects that we propose, strikes a balance between realism and computability. The main cost associated with our model is the need for small time steps during contact, which slows down the simulation. The final sections explain the mathematical details of our contact model.

In paper [3], a simple nonlinear contact model is presented for use in computer simulation. The nonlinear model is shown to maintain the computational simplicity of the linear model while addressing many of its deficiencies. One such advantage is that contact forces vary continuously over time. A new phase plane solution for the nonlinear model is obtained which reveals many previously unnoted properties. These include proper variation of the coefficient of restitution with impact velocity over a wide range of impact velocities, independence of model parameters, and lack of tensile (sticking) forces in simple impacts. An example is presented which demonstrates the use of the contact model in simulating the foot-ground interaction during the locomotion cycle of a walking machine.

In [4] the absence of transition curves at the entry and exit of the turnout, and the cant deficiency, leads to large wheel–rail contact forces and passenger discomfort when the train is switching into the turnout track. Two alternative multibody system (MBS) models of dynamic interaction between train and a standard turnout design are developed. The first model is derived using a commercial MBS software. The second model is based on a multibody dynamics formulation, which may account for the structural flexibility of train and track components (based on finite element models and coordinate reduction methods). The variation in rail profile is accounted for by sampling the cross-section of each rail at several positions along the turnout. Contact between the back of the wheel flange and the check rail, when the wheelset is steered through the crossing, is considered.

In [5] based on a numerical strategy previously developed, the present study introduces a numerical-experimental comparison of such occurrence. Attention is first paid to the review and analysis of existing experimental results. Good agreement with numerical predictions is then illustrated in terms of critical stress levels within the blade as well as final wear profiles of the abradable liner. Numerical results suggest an alteration of the abradable mechanical properties in order to explain the outbreak of a divergent interaction.

A general approach to simulate the mechanical behaviour of entangled materials submitted to large deformations is described in paper [6]. The main part of this approach is the automatic creation of contact elements, with appropriate constitutive laws, to take into account the interactions between fibres. The construction of these elements at each increment, is based on the determination of intermediate geometries in each region where two parts of beams are sufficiently close to be likely to enter into contact. Numerical tests simulating a 90% compression of nine randomly generated samples of entangled materials are given. They allow the identification of power laws to represent the evolutions of the compressive load and of the number of contacts.

Main material

The complexity of the experiment lies in the curvilinearity of the forms in contact: the cylinder, represented by the shaft, and the torus, which corresponds to the pressure roller. The geometric parameters of the model elements are as follows: a roller with a diameter of 56 mm and a radius of the working surface of 2.5 mm; shaft with a diameter of 25 mm.

The experiment presented in the current task is aimed at evaluating the influence of material nonlinearity on the results of stresses and strains. The current model simplified in terms of its components was adopted (truncated torus of the roller (segment) and a short segment of the shaft (Fig. 1).

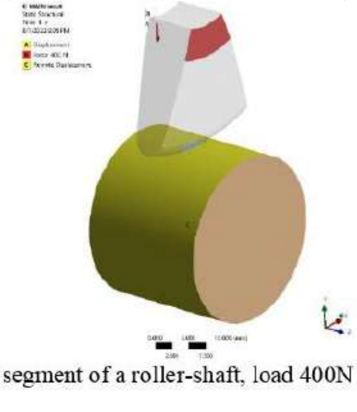
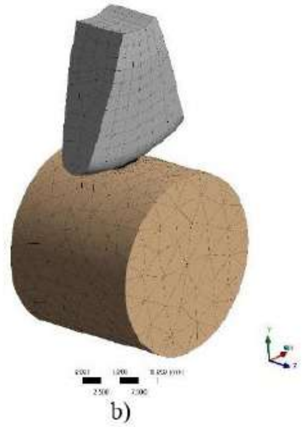


Fig. 1. Model of a truncated segment of a roller-shaft, load 400N

The task required high computational resources, the total calculation time was 1 hour. 42 min. The roller presses in a straight line (along the Y axis) on the surface of the shaft, having only one degree of freedom.

The MKE grid consists of 70,661 elements connected by 115,564 nodes, and the shape of the final elements is mainly Tetrahedrons (Fig. 2).



b)

Fig. 2.Mesh of FEM elements

At the point of contact, the body of the stain itself (area 4.2x1.8 mm and depth 0.3 mm) is modeled separately (Fig. 3.8). The size of the final elements here does not exceed 0.05 mm. Contact Sizing and Contact Match functionality of Ansys with the Tolerance indicator of 0.05 mm and 0.04 mm, respectively, was used to connect the nodes of the shaft model and the spot body.

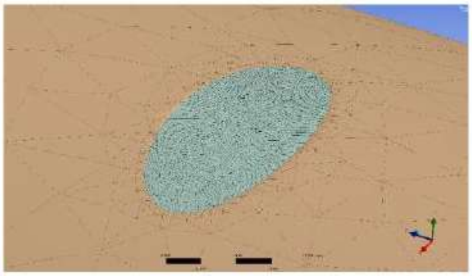
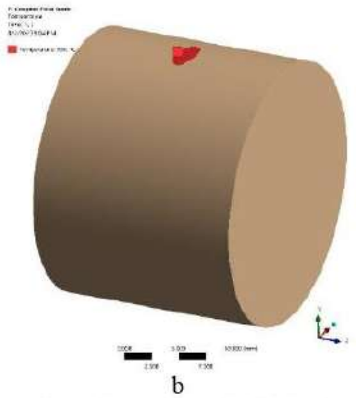


Fig. 3. Spot contact

Nonlinear properties of the Structural Steel material: beyond the yield point, the stress-strain graph abruptly changes its character: instead of being proportionally linear, it acquires a fracture (Bilinear Isotropic Hardering). In fact, this means that beyond the yield point, with the next slight increase in loads (and as a result, stresses), deformations (mm/mm) increase significantly - irreversible inertial plastic processes occur (the body "floats").



b

Fig. 4. Thermal load - 900 0C

In addition, the above-mentioned effect of plasticity is enhanced by the influence of temperatures - a thermal load (900°C). Convection of the medium is 25 W/m2°C at a temperature of 22°C.

Boundary conditions also include forceF_p= 400 N, directed opposite to the Y axis and applied to the sides of the roller. The reason is the heat load, the graph of the temperature distribution along the cross-section of the model is shown in Fig. 5.

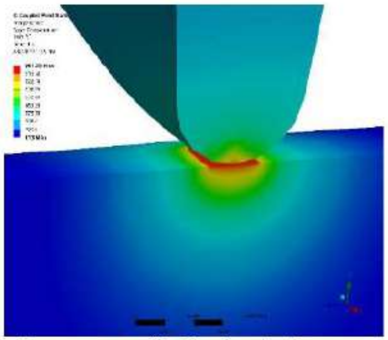


Fig. 5. Temperature distribution in the contact zone

The time of the experiment is 4.0 s, and the load application schedule is stepwise (Fig. 3.10):

- during the period of 0.0-1.0 s, the load increasesF_p from 0 N to 400 N;
- the interval of 1.0-2.0 s has a stabilization character -$F_p$keeps the value of 400 N;
- during the next second (2.0-3.0 s), the load is reduced to 0 N;F_p
- the last interval (3.0-4.0 s) passes at rest for the system - = 0 N.F_p

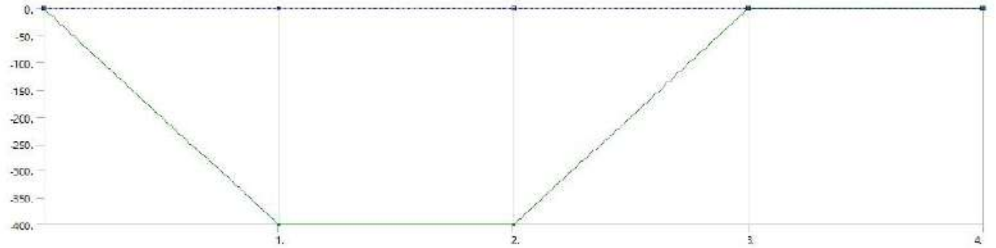


Fig. 6. Load application schedule

The purpose of simulating natural load according to such an algorithm is to identify peak stresses during the steady process of loading and residual stresses after releasing the system from the action of forces on it.

Traditionally, we make sure that the calculation reached a successful conclusion on the basis of the Force convergence graph, no abnormal jumps between iterations or gaps were recorded, and the time and force curves reached the specified limit - 4.0 c. Thus, we can proceed to the evaluation of model stresses: as expected, the largest value of stresseswas 1082.9 MPa andrecorded at a time of 2.0 s when the force$F_p$still kept the value of 400 N. This indicator significantly exceeds the yield point of the material and indicates the appearance of plastic deformations. The stress-strain state of the contact spot is shown in Fig. 3.11 - it visually shows a deflection in the central part, which is expected.

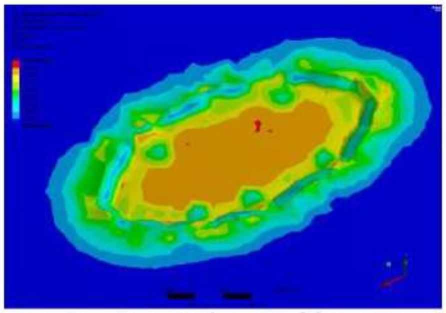


Figure 3.11 – Stress-strain state of the contact patch

The nature of the stress change during loading is the most significant, so let's analyze the key time points of fig. 3.7.

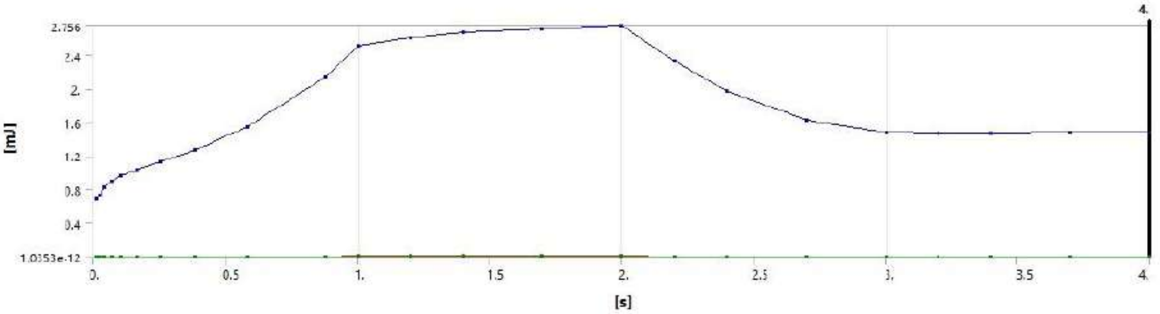


Fig. 7. The graph of changes in stress energy in the contact zone
depending on time

t =0.0116 s: at the initial moment of contact between the roller and the shaft (0.0116 s), the stress value was 996.19 MPa and was recorded on the surface of the shaft (max tag). The fact of the location of the max tag is interesting: it is not in the center of the contact patch, as one might expect, but on the periphery. What actually happens: the roller "captured" a certain part of the shaft surface with its contact (the dimensions of the contact surface are 0.8×0.3 mm) and transmits the load to the rest of the shaft body through it. As a matter of fact, maximum stresses are formed at their junctions. Conventionally, this process can be called instant "sticking" of the roller to the shaft in the micro region. Such sticking can also be detected by the contact status graph (Contact Tool > Status) – the spot shows a Sticking-type contact with a sharp contour, which is further smeared, filling with Sliding and Near statuses.

t =0.0406 s: stress drops to the lowest value throughout the experiment (957.83 MPa) with movement closer to the central part of the contact patch. At the same time, the stress extreme is still on the surface of the shaft. This indicates the end of the momentary process of "sticking" the roller to the shaft established in the previous step - the contact has stabilized: the roller begins to act on the shaft as an independent body, bending it (the deformations of the spot body during loading will be analyzed below).

t =1.0 s: starting from the previous characteristic moment of time, there is a gradual increase in stresses up to 1058.7 MPa, when the load reached 400 N. The extremum of stresses migrated to a depth of the order of 0.1-0.2 mm, where, under the conditions of temperature load, the corresponding white layer began to form (Fig. 8) , which corresponds to the defamation process of the metal surface (the typical depth of the defamation layer is 0.2 mm or more).F_p

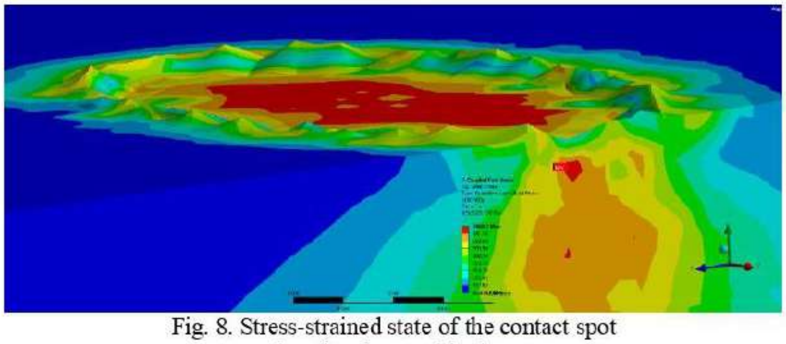


Fig. 8. Stress-strained state of the contact spot
when forming a white layer

t =2.0 s: the force is maintained at 400 N - the system has accumulated maximum energy (2.756 mJ), which can be seen on the Strain Energy graph, and the stresses have increased to 1082.9 MPa. The location of the stress extreme has changed only minimally (within 0.05 mm it has sunk into the shaft body) F_p.

t =2.2 s: the force begins to linearly decrease to zero during the period 2.0-3.0 s, therefore turbulent processes appear in the structure of the outer layer - the maximum stress has decreased to 970.05 MPa and moved to the surface of the shaft, but inside the body there is still a zone of high stresses F_p.

t =4.0 s: the force remains zero during the last second of the experiment, so the shaft is free and not subjected to loads. The plastic deformation has stabilized, and the residual stresses are 1036 Mpa F_p.

Confirmation of the presence of plastic deformation can also be found on the graph of the vertical movement of the roller (along the Y axis) - as can be seen, the contact surface of the roller (Fig. 9) did not return to the initial position that corresponded to the beginning of the experiment (0.0 s). The value of the movement of the roller at the end of the experiment was 0.00311 mm, that is, the model did not restore its original location, which means that the shaft received irreversible deformations t =4s.

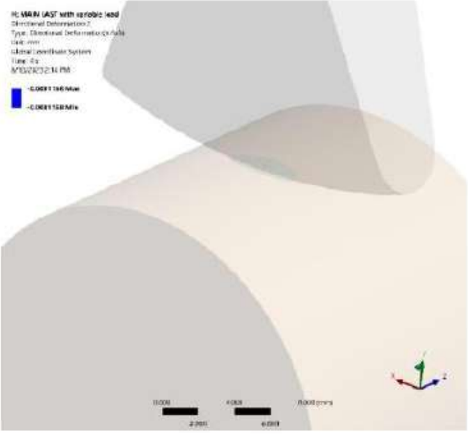


Fig. 9. Stress-strained state of the contact spot when after removing the load

The specified dynamics of the movement of maximum stresses from the surface to the body of the model can also be observed on the displacement maps: at the initial moment of contact (t =0.0116 s) the maximum value was 0.0059 mm, at the time of 1.0 s - 0.0126 mm, and at the end of the experiment - 0.0104 mm. The distribution of heat fluxes over the contact area is expected and adequate in nature - the extremum falls on the central part with a value of 166.06 W/mm2. Deformations at the peak moment of the experiment (2.0 s) were 0.01384 mm/mm.t = t =4s.

Summarizing the conducted research, the following conclusions can be reached:

Non-linearity of the material significantly affects the magnitude of the stresses in the model, and therefore the resulting plastic deformations. The key factor in the case of the graph of bilinear isotropic hardening (Bilinear Isotropic Hardening) of the Structural Steel material used in Ansys the angle of inclination of the straight line, starting from the point of the yield point, protrudes. The closeness of simulated FEM calculations to full-scale tests of material surface slander depends on the veracity of the strengthening schedule entered into the model (Multilinear, Kinematic, Nonlinear, Chaboche and other types). In fact, each experimental laboratory forms original graphsstresses and strainsbased on their own physical research of samples of material that is their intellectual property. Our task is to create a universal technique to which any graph created in Ansys could be applied.

Conclusions

1. The Ansys Static Structural calculation module turned out to be a sufficient tool in terms of its calculation capabilities. The results of the problem are absolutely adequate within the scope of Hooke's law and carry valuable information about the geometric parameters of the body of the contact spot, which is the basis of the boundary conditions of the problem.

2. The heat load of the surface of the contact spot, together with the consideration of the non-linearity of the material, significantly affects the amount of stress in the upper layers of the shaft, including the formation of the so-called white layer at a depth of about 0.2 mm. The extremum of stress migrates during the loading-unloading process of the shaft from the surface to the body of the shaft and vice versa. The regularity is as follows: as the load on the surface of the shaft increases, the extremum of stress moves inward, starting the formation of the indicated white layer.

3. The applied boundary conditions made it possible to obtain an array of information on residual stresses, deformations and displacements of the model, strain energy graphs, a temperature distribution map, as well as data on the type and nature of contact of bodies during the experiment. Undoubtedly, such a multifactorial model in the form of various input parameters (load, temperature, experiment time, convection, etc.) is a promising object of future research on the analysis of surface strengthening of the outer layers of shafts, and the calculations described in the work can serve as a basis for the formation of original FEM simulation methods natural strengthening of the material, which is especially relevant for experimental laboratories in materials science.

References

1. Wang, H., Al-Qadi, I. L., & Stanciulescu, I. (2012). Simulation of tyre–pavement interaction for predicting contact stresses at static and various rolling conditions. *International Journal of Pavement Engineering*, *13*(4), 310-321.

2. Goyal, S., Pinson, E. N., & Sinden, F. W. (1994). Simulation of dynamics of interacting rigid bodies including friction I: General problem and contact model. *Engineering with computers*, *10*, 162-174.

3. Marhefka, D. W., & Orin, D. E. (1996, April). Simulation of contact using a nonlinear damping model. In *Proceedings of IEEE international conference on robotics and automation* (Vol. 2, pp. 1662-1668). IEEE.

4. Kassa, E., Andersson, C., & Nielsen, J. C. (2006). Simulation of dynamic interaction between train and railway turnout. *Vehicle system dynamics*, *44*(3), 247-258.

5. Batailly, A., Legrand, M., Millecamps, A., & Garcin, F. (2012). Numerical-experimental comparison in the simulation of rotor/stator interaction through blade-tip/abradable coating contact.

6. Durville, D. (2005). Numerical simulation of entangled materials mechanical properties. *Journal of materials science*, *40*, 5941-5948.

Голенко К.Е., Дитинюк В.О., Диха М.О. Імітаційна модель контактної взаємодії при поверхневому зміцненні сталевих деталей

В процесах поверхневого зміцнення сталевих деталей напружено-деформований стан є визначальним для пояснення фізичних процесів зміцнення, формування розмірів площадки контакту. Аналітичні залежності контактних параметрів є досить наближеними. В даній роботі на основі програмного комплексу Ansys запропонована імітаційна модель контакту усіченого тора з циліндром, що демонструє кінетику процесу втиснення твердосплавного інструменту- тора в сталеву заготовку- циліндр. Експеримент проводився протягом 4с з метою визначення максимального рівня напружень, розподілу напружень і величини залишкових напружень після зняття навантаження. Зусилля притискання приймалось переважно в зоні пружних деформацій. Результати показали нерівномірниq розподіл напружень з максимумом в центрі плями контакту 1082 МПа. Після зміни напрямку навантаження в центрі плями контакту спостерігались невеликі залишкові деформації на рівні 0,00311 мкм. Це що свідчить про порушення пружної області на невеликій площі контакту, яка не впливає на загальний характер розподілу напружень і може бути видалена в процесі фінішної обробки. Результати моделювання напруженого стану використані для співвідношення із спостерігаємими структурними змінами матеріалу в процесі дії термічних і силових напружень. Пік напружень формувався на відстані 200 мкм, що сприяє формуванню максимальних значень мікротвердості на цій глибині.

Ключові слова: напружено-деформівний стан, поверхня, зміцнення, контактна обробка, інструмент, усічений тор, циліндр, пружні деформації