UDC 621.9.048.6
DOI: 10.32626/2308-5916.2026-29.28-38

Velyka O.
ORCID: 0000-0003-1575-8829,
PhD in Engineering, Lviv Polytechnic
National University, Lviv, Ukraine,
E-mail: oksana.t.velyka@lpnu.ua

Bojko M.
ORCID: 0000-0002-7955-5062,
Senior Lecturer, Lviv Polytechnic National University, Lviv, Ukraine,
E-mail: mykhailo.v.boyko@lpnu.ua

Velykyi N.-T.
ORCID: 0009-0001-8550-4974,
PhD student, Lviv Polytechnic National University, Lviv, Ukraine,
E-mail: nestor-taras.i.velykyi@lpnu.ua

MODELING AND RESEARCH ON THE STRESS-STRAIN STATE OF A PUNCH AND THE OPTIMIZATION OF GEOMETRIC PARAMETERS

Punch is one of the main components of the stamp. During use, punches are exposed to high force loads, so they are worn faster. It is necessary to optimize the geometrical parameters of the punch to localize the critical areas where deformation and destruction occur during the stamping process, thereby increasing the durability of the entire stamp design. The peculiarities of the stress-strain state of the punch for the cutting of oval openings in products from sheet material have been investigated. It is shown that ensuring reliability, durability and faultless operation of the punch directly depends on the level of the stress-strain state, which is constantly changing in the course of prolonged operation. Particular attention is given to the finite element analysis of the stress-strain state of the punch structure, which failed during changing operational conditions. A 3D model of the punch design in the AUTODESK INVENTOR environment was constructed using the finite element method. The parameters were calculated, and the critical areas of the punch were identified. This article describes various stages of punch failure. The optimization of the punch's geometric parameters is presented, along with a rational choice of punch model that can withstand op-

Стаття надійшла до редакції: 17.03.2026
Рекомендовано до друку: 02.04.2026
Оприлюднено (online): 15.05.2026

erational loads, extend its service life, and ensure efficient performance under excessive loading during the process of making oval holes. The effectiveness of the proposed design decisions was verified on a modernized punch model.

Key words: *punch, stress-strain state, stamping die, finite element method.*

Introduction. Sheet metal stamping is widely used in many branches of mechanical engineering. Sheet stamping is the forming of parts of various shapes and sizes from sheet, strip or coil material. Alloy steels, low carbon steels, titanium, copper, aluminum, etc. are used as feedstocks for sheet stamping. The main advantages of sheet stamping are:

1) the ability to manufacture durable, lightweight, and rigid thin-walled products of simple or complex shapes that are difficult or impossible to produce in other ways;
2) high productivity and economical metal consumption;
3) wide possibilities for automation of stamping operations using relatively simple devices;
4) interchangeability of parts and high surface finish. A stamp consists of technological (working) and structural parts (block) and ensures the performance of technological operations. Stamps include punches, guide bars, ejectors, dies, clamps, and others. Stamps are classified by technological features into sequential, simple, and combined action stamps.

To perform several technological operations or technological transitions on several positions in a certain number of strokes of the moving part of the die, sequential action dies are used. Single-action stamps are used when performing one or more identical technological operations at a single position in one stroke of the moving part of the die. In a combined action die, different technological operations or transitions are combined in a single stroke of the moving part of the die, such as cutting and drawing, among others.

Problem statement. In the production of instrumentation products, a key technological operation is cold sheet metal forming (CSM). These operations are carried out in the forging and pressing departments of enterprises and account for 30-40% of the overall technological process. The widespread use of sheet metal stamping in industry is due to several positive attributes:

1. High productivity: Up to 30,000-90,000 parts per shift.
2. Utilization of low-skilled labor: The process requires minimal specialized skills.
3. Part precision: Parts are produced with high precision, ensuring interchangeability and typically eliminating the need for further machining.
4. Automation potential: The process is well-suited for automation.

Cold sheet metal stamping is a complex and labor-intensive process. It demands significant labor and physical resources. Key aspects of successful cold sheet stamping include the precise feeding of the workpiece into the die, timely cutting, punching, and removal of the finished part.

Overall, cold sheet metal stamping is a progressive and high-performance technological process. It involves plastic deformation of the material, changing the shape and size of the workpieces using a strip or sheet of thin metal as the raw material.

A punch (French *poinçon*) is one of the main components of any stamp. During stamping, the punch directly exerts pressure on the workpiece material. Depending on the purpose, it can be piercing, punching, slitting or notching punch. In operation, the punch transfers pressure to the sheet metal workpiece, cuts and pushes the finished product through the die. It plays an important role in the process of stamping or applying marking data to the surface of the part and is one of the main elements of the technological equipment of any stamp. During operation, the punches are exposed to high force loads, so they are made of wear-resistant steel with increased strength and high hardenability.

A stamp has a certain service life, with its main components, the punch and die, wearing out more quickly. The service life is limited by the number of times the punch strikes the surface of the part. When the operating conditions of the die change, the loads also change, leading to the potential destruction of its main components. Therefore, it is essential to study and analyze the stress-strain state and changes in the geometric parameters of the punch during operation. This analysis allows for the design of optimized punch structures that ensure effective operation under excessive loads during the cutting process. It is necessary to optimize the geometric parameters of the punch in such to localize the critical areas in which deformation and fracture occur during stamping, which will increase the efficiency of the entire die structure.

Analysis of recent research and publications. Recent trends in sheet metal stamping have highlighted its widespread adoption due to its high material efficiency and production productivity [1, 2]. Increasing demands for product performance are placing greater requirements on both the tools and the materials used for blanks [3, 4].

Research in [5, 6] explored the optimal steel grades for stamping equipment and sheet blanks across various industries. The research problem addressed in [7] involves improving the methodology for calculating stress states and residual stresses during the bending of sheet blanks using finite element modeling. The calculation of the stress-strain state of sheet billets is discussed in [8-10], while the study, calculation, and analysis of the stress-strain state of stamping equipment are detailed in [11-12].

Given the high cost of stamp manufacturing, ensuring long-term operation, reliability, and durability is crucial. This can be achieved by:

- Developing a punch design with an optimal shape.

- Selecting appropriate punch materials.
- Providing surface treatment to enhance surface hardening during manufacture.

This work addresses these challenges and proposes a new punch design characterized by high reliability and performance.

Material and Method. A stamping tool is specialized equipment used to manufacture sheet metal stamping parts. This tool comprises a set of dies and punches that shape the workpiece.

A die is a tool designed to impart a specific configuration to a part through the plastic deformation of the workpiece or by dividing it into parts (stamping). Each part requires a distinct die, with its design depending on factors such as the type of billet (e.g., long or flat products), the type of stamping machines used (e.g., hammer, press), the nature of the operations performed, and the production batch size.

A key and universal component in the instrumentation case is the «Plank» part (Fig. 1). This component is used for securing and positioning the device components.

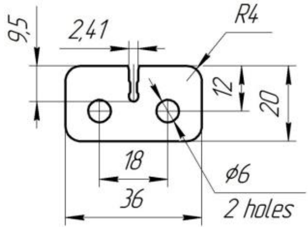

Fig. 1. *Working drawing of the «Plank» part*

The manufacturing sequence for the «Plank» part is illustrated in Fig. 2. The design feature of the part is an open narrow groove. The part is manufactured in three technological steps:

- In the first pass, two holes are punched into the sheet strip.
- In the second pass, an open narrow groove is cut.
- In the third pass, the part is cut along the outer contour.

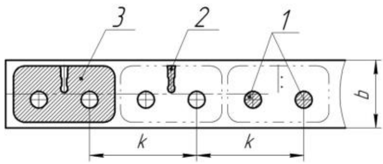

Fig. 2. *Cutting of the part «Plank» cutting strip: 1 – hole punch; 2 – groove punch; 3 – contour punch; k – strip movement value; b – strip width*

The process of punching the contours of the part occurs when the punch moves from top to bottom (stroke) with punches. The punching force acts on punches 1 and 3, which depends on the thickness and mechanical properties of the material, the perimeter of the cut, the shape of the cutting edges of the punch and die, and the size of the gap between the punch and die:

$$P = L\,s\,\tau,\ [N], \tag{1}$$

where L – the perimeter of the contour that is cut down by the punch, *mm*; s – the thickness of the workpiece (strip), *mm*; τ – resistance of the material, N/mm^2.

The punch 2 (Fig. 3), which cuts an open contour, is subjected to a cutting force P1 and a lateral force P2 directed perpendicular to the punch (stroke), which bends the punch towards the open section of the contour (Fig. 4).

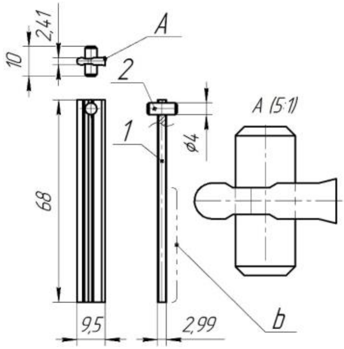

Fig. 3. *Punch for cutting a groove: 1 – punch body; 2 – pin shoulder; b – area where the punch is deformed (or destroyed)*

Under load the punch operates in compression and bending. This significantly increases the stresses that cause the punch to break (zone *b*), leading to production shutdown.)

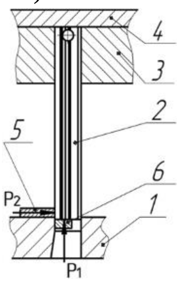

Fig. 4. *The scheme of cutting a groove: 1 – matrix; 2 – punch; 3 – punch holder; 4 – backing tile; 5 – workpiece; 6 – guide; P1 – cutting force; P2 – lateral force*

To address this issue, the geometric parameters of the punch were studied, and it was proposed to change their values, namely, to increase the cross-section of the punch. This is taken into account in the modernized model of the punch design (Fig. 5). and Fig. 6, 7 shows the technological scheme of cutting with a modernized punch.

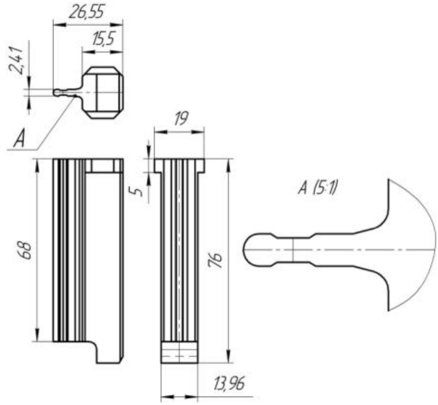

Fig. 5. *Modernized groove punch*

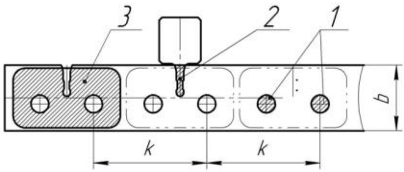

Fig. 6. *Cutting of the cutting strip of the «Plank» part: 1 – holes punching punch; 2 – modernized groove punch; 3 – contour cutting punch*

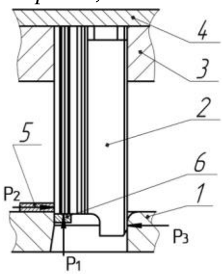

Fig. 7. *Scheme of groove cutting with a modernized punch: 1 – matrix; 2 – punch; 3 – punch holder;.4 – support tile; 5 – workpiece; 6 – guide; P1 – cutting force; P2 – lateral force; P3 – counterpressure force*

Research results. The CAE finite element method was used to study and analyze the stress-strain state of the punch [13].

The finite element method (FEM) is widely used to solve practical problems of deformable solid mechanics, in particular, to perform strength calculations during the 3D design phase.

The finite element method requires a specific calculation algorithm:

1. Creating a solid model of the object under study.
2. Creating a mesh breakdown of the model into finite elements.
3. Setting initial and boundary conditions.

4. Performing the calculation.
5. Analyzing the results.

Modern CAD systems automate the strength calculation of a part model using input data. The system selects nodes within the part, divides it into finite elements, numbers the nodes, constructs the element matrix, and formulates calculation equations. Results are presented as tabular data and visual diagrams.

Having calculated the given punch design by the finite element method, we will determine the critical areas of the punch in which deformation and fracture occur.

To calculate the punch by the finite element method, we used the Stress Analysis module, which is designed to perform calculations of solid objects in the Autodesk Inventor system [14] and visualize the results. This module helps you find the best designs for parts or assemblies. It provides tools for rapid stress-strain analysis, including the determination of stresses, strains, displacements, safety margins, and resonant frequencies.

This module features tools for preparing parts and assemblies for calculations, setting boundary conditions and loads, and generating finite element meshes (with constant or variable spacing). This software allows you to automatically calculate the strength model of a part in accordance with the input data (the system selects nodes in the volume of the part, divides the part into finite elements, numbers the nodes, builds an element matrix, and compiles calculation equations). Results are shown in tabular form and visual diagrams of the required parameters. This functional set allows you to model a solid object and comprehensively analyze the behavior of the calculation model for various operational tasks in terms of statics, determination of natural vibration frequencies, stability, and thermal load, etc.

To study the stress-strain state of the punch, a three-dimensional solid model of the punch was created in Autodesk Inventor (Fig. 8).

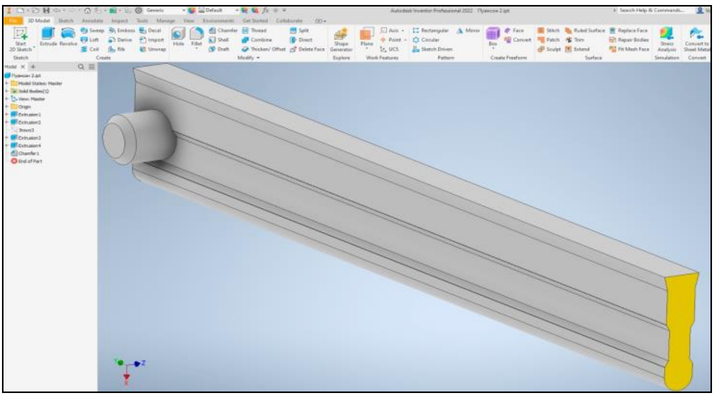

Fig. 8. *Three-dimensional solid model of the punch*

In the study, the input parameters included geometric parameters and a cutting force of P = 150000 N. Fig. 9 displays the results of the study, specifically the distribution of the safety factor across the volume of the punch (dimensionless value). The lowest safety factor of 0.1 is observed on the side surfaces of the punch. Since this value is less than 1, it indicates that these areas are prone to guaranteed destruction due to the applied active and reactive forces.

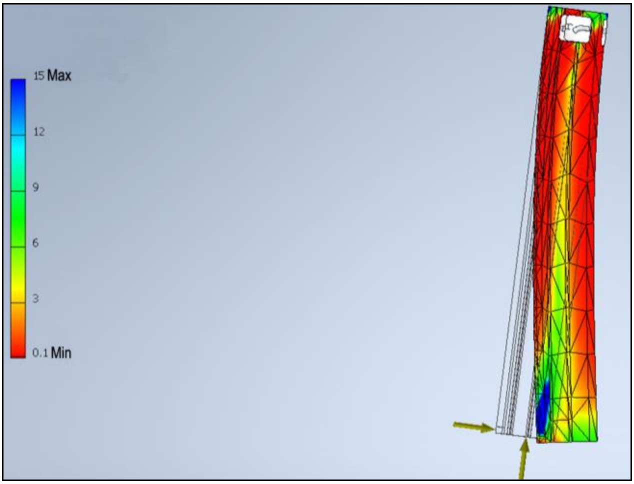

Fig. 9. *Results of the strength calculation for the 3D model of the punch before modernization*

The correctness of the design solutions was verified for the modernized punch (Fig. 10). The resulting punch features a larger cross-sectional surface area due to the addition of an extra lateral surface, which enhances its transverse stiffness.

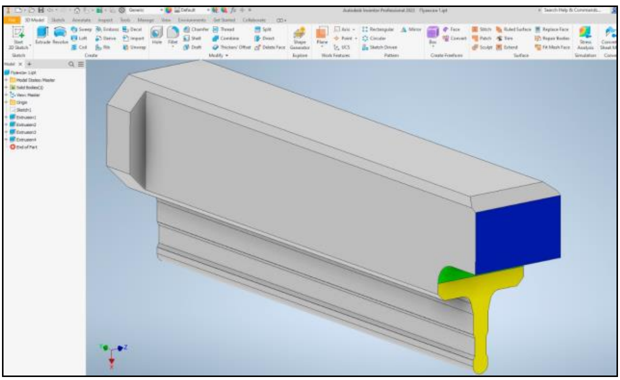

Fig. 10. *3D model of the modernized punch*

In dialog mode with the CAE/CAD system, new finite element calculations were performed to verify the required reliability of the punch operation (Fig. 11).

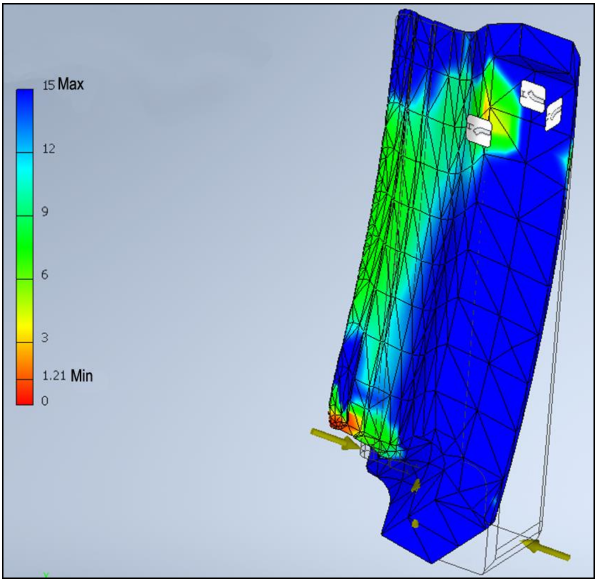

Fig. 11. *Results of the strength calculation for the 3D model of the modernized punch*

After analyzing the new calculation results, it is evident that the problem areas of the punch now show improved strength and resistance to deformation and destruction during operation. Specifically, the minimum safety factor is 1.21, which is above 1. Therefore, the punch is expected to withstand operational loads effectively. Consequently, this modernization is anticipated to increase the punch's service life in the die.

Conclusions. A three-dimensional model of the punch was developed.

Finite element analysis of the punch in the punching die initially indicated that the design was not reliable.

To address this, a modernized punch design with optimized geometric parameters was proposed.

Subsequent finite element analysis confirmed the operational reliability and efficiency of the new design. The modernized punch is capable of achieving nearly six times the number of workpiece cuts compared to the previous design.

The economic benefit of using the new punch, despite its slightly increased mass, significantly outweighs the cost of the additional materials required for its manufacture.

The study of the stress-strain state of stamping components, which evolve during their long-term operation, is crucial for ensuring the reliability, durability, and trouble-free operation of the technological equipment.

The research findings have been implemented in production.

References:

1. Gronostajski Z., Pater Z., Madej L. et al. Recent development trends in metal forming. *Archives of Civil and Mechanical Engineering.* 2019. Vol. 19. P. 898-941
2. Dobrianskyi S. S. Tekhnolohichni osnovy mashynobuduvannia. Kyiv: KPI im. Ihoria Sikorskoho, 2020. 379 p.
3. Cao J., Brinksmeier E., Fu M. et al. Manufacturing of advanced smart tooling for metal forming. *CIRP Annals.* 2019. Vol. 68. P. 605-628.
4. Lim Y., Venugopal R., Ulsoy A. G. Process Control for Sheet-Metal Stamping: Process Modeling, Controller Design and Shop-Floor Implementation (Advances in Industrial Control). Springer, 2014. 140 p.
5. Paunoiu V., Saadatou M. A., Nedelcu D., Octavian M. Experimental and numerical investigations of sheet metal circular bending. *Indian Journal of Engineering and Materials Sciences*. 2015. Vol. 22. P. 487-496.
6. Koehler W., Plege B., Sahm K. F., Padmapriya N. Metal Forming: Specialized Procedures for the Aircraft Industry. *Reference Module in Materials Science and Materials Engineering.* 2017. 8 p.
7. Nandanwar P. S., Bajaj P. S., Patil P. D. Analysis of spring back defect in right angle bending process in sheet metal forming. *Pratibha: International Journal of Science, Spirituality, Business and Technology.* 2014. Vol. 3. № 1. P. 18-22.
8. Fu Z., Mo J., Zhang W. Study on multiple-step incremental air-bending forming of sheet metal with springback model and FEM simulation. *The International Journal of Advanced Manufacturing Technology.* 2009. Vol. 45. P. 448-458.
9. Kuhar V. V., Anishchenko O. S., Nagnibeda O. M. Metodyka rozrahunku napruzeno-deformovanogo stanu lystovoi zagotovky pry gnutti z urahuvanniam zmicnennia materialu. *Visnyk Nac. Teh.un-tu «HPI»*: zb. nauk. prac. Harkiv, 2017. Vyp. 37 (1259). P. 36-41.
10. Viorel P., Saadatou A. S., Nedelcu D., Octavian M. Experimental and numerical investigations of sheet metal circular bending. *Indian Journal of Engineering and Materials Sciences.* 2015. Vol. 22. P. 487-496.
11. Boiko M., Velyka O., Topilnytskyi V. Rozrakhunok ta optymizatsiia matrytsi vyrubnykh shtampiv z umovy mitsnosti ta zhorstkosti. *9-y Mizhnarodnyi sympozium ukrainskykh inzheneriv-mekhanikiv u Lvovi: prysviach. 165-richchiu Nats. un-tu «Lviv. politekhnika», 20-22 trav. 2009 r.*: pratsi. Lviv, 2009. P. 155-156.
12. Velyka O. T., Bojko M. V., Velykyj N.-T. I. Computation of Tense-Deformed State of the Die Mold Matrix. *International Scientific and Theoretical Conference «Modeling and Computer Engineering in Mechanical Engineering: Theory, Practice, and Innovation» (MCEME-2022) 28/09/2022-21/10/2022.* Lviv.
13. Topilnytskyy V., Liaskovska S., Rebot D., Velyka O., Boiko M. Basic principles of the finite element method in mechanical engineering field: navchalnyj posibnyk. Drohobych: POSVIT, 2021. 154 p.
14. Dmytriv V., Dmytriv I., Gorodniak R., Карр M., Кlimov E. Systemy avtomatyzovanogo proektuvannia. AUTODESK INVENTOR: navchalnyj posibnyk. Lviv: Spolom, 2022. 284 p.

МОДЕЛЮВАННЯ І ДОСЛІДЖЕННЯ НАПРУЖЕНО-ДЕФОРМОВАНОГО СТАНУ ТА ОПТИМІЗАЦІЯ ГЕОМЕТРИЧНИХ ПАРАМЕТРІВ ВИРУБНОГО ПУАНСОНА

Пуансон відноситься до однієї з основних деталей штампу. В процесі роботи пуансони піддаються впливу високих силових навантажень, тому зношуються швидше. Необхідно оптимізувати геометричні параметри пуансона таким чином, щоб локалізувати критичні області, в яких виникає деформація та руйнування в процесі штампування деталей, що дозволить збільшити працездатність конструкції цілого штампу. Проведено дослідження особливостей напружено-деформованого стану пуансона для вирубування відкритого паза у виробах із листового матеріалу. Показано, що забезпечення надійності, міцності та безвідмовної роботи пуансона прямо залежить від рівня напружено-деформованого стану, який постійно змінюється в процесі тривалої експлуатації. Особливу увагу приділено скінченно-елементному аналізові напружено-деформованого стану конструкції пуансона, який в процесі зміни умов експлуатації руйнувався. Побудовано тривимірну модель конструкції пуансона в середовищі AUTODESK INVENTOR за допомогою методу скінченних елементів, виконано розрахунки його параметрів, виявлено критичні області режимів роботи пуансона, в яких виникає деформація та руйнування в процесі експлуатації.. Запропоновано змінити геометричні параметри конструкції пуансона та здійснити раціональний вибір типу моделі пуансона, який витримує прикладені експлуатаційні навантаження, збільшує термін його експлуатації та забезпечує ефективну роботу при надмірному навантаженні в процесі вирубування овальних отворів у деталях. Результативність прийнятих проектних рішень перевірено на модернізованій моделі пуансона та впроваджено на виробництві.

Ключові слова: *пуансон, напружено-деформований стан, штамп, метод скінченних елементів.*