UKRAINIAN JOURNAL OF MECHANICAL ENGINEERING AND MATERIALS SCIENCE
Vol. 8, No. 3, 2022

Oksana Velyka[1], Myhajlo Bojko[2]

[1] Department of Equipment Design and Operation, Lviv Polytechnic National University, 12, S. Bandery Str., Lviv, Ukraine, e-mail: oksana.t.velyka@lpnu.ua, ORCID 0000-0003-1575-8829
[2] Department of Equipment Design and Operation, Lviv Polytechnic National University, 12, S. Bandery Str., Lviv, Ukraine, e-mail: mykhailo.v.boyko@lpnu.ua

THE RESEARCH ON MATRIX OF PRESS-FORM STIFFNESS AND STEADINESS

Received: June 14, 2022 / Revised: June 28, 2022 / Accepted: August 30, 2022

https://doi.org/10.23939/ujmems2022.03.054

Abstract. The formalized algorithm of calculation of rectangular matrix of press-form is developed on durability and inflexibility with the purpose of determination of optimum geometrical sizes which would provide it optimum operating capacity. For this purpose CAD and CAE systems were used, in particular Solidwork, Matcad. A three-dimensional model of the mold matrix was designed. Research and analysis of the stress-strain state of the matrix was carried out and its optimal parameters were determined. As a result, it makes it possible to optimize the structure of the matrix in terms of stiffness and stability and ensure its optimal performance.

Keywords: press-form, matrix, finite element method.

Introduction and Problem Statement

The main manufacturing process of plastics is injection blow molding. Metal press-forms are used for injection blow molding. The form may have one or more forming cavities; its configuration is a negative reflection of the product being produced. The essence of injection blow molding lies in the following: plastic is preheated in the cylinder of the injection molding machine (thermoplastic machine) and is poured into the forming cavity of the mold under the pressure of the machine's piston.

Plastic acquires the configuration of the product and cools down. Then the form is opened and the product is taken out. Forms are made of steel that must withstand repeated exposure to high pressures and temperatures and can be well processed by cutting [4–6]. The configuration of the form depends on the product design being manufactured.

The matrix is the part of the press form participating directly in the formation of the outer surface of the product. It works under the influence of a variable load (which occurs at the moment of injection of plastic into the form), and the construction of the matrix is subjected to increased requirements regarding its strength. Therefore, the calculation of the strength and stiffness of the matrix[1] is a mandatory stage in the process of constructing the press-form.

One of the major tasks in designing press forms for injection blow molding and optimization of the project process is the development of a formalized methodology for technological and construction calculations. The method of construction calculation includes the calculation of all press-form components. Matrix is one of the main details in which the process of formation of plastic products is carried out. Thus, the calculation of its stiffness and steadiness represents practical interest. Matrix construction is viewed as a box-shaped structure of the parallel form. Based on the conditions of uniform strength and geometric limitations, we resolved the task of the rational use of the material in matrix construction. Matrix is conventionally divided into N elements (substructures) of a given geometry with M constructive controlled parameters that determine the consumption of the material.

Separate parts of the structure (side walls, lintels) are considered elements, and their thicknesses are considered as defining structural parameters [2–3].

It is necessary to develop an algorithm for calculating the rectangular matrix of press-form for steadiness and stiffness and conduct research related to the optimal choice of geometric parameters of the press-form, which would ensure its efficiency.

The research on the necessity of calculation confirms Fig. 1, where the crack in the central part of the matrix is visible and was formed in the process of operation and incorrect manual construction calculation.

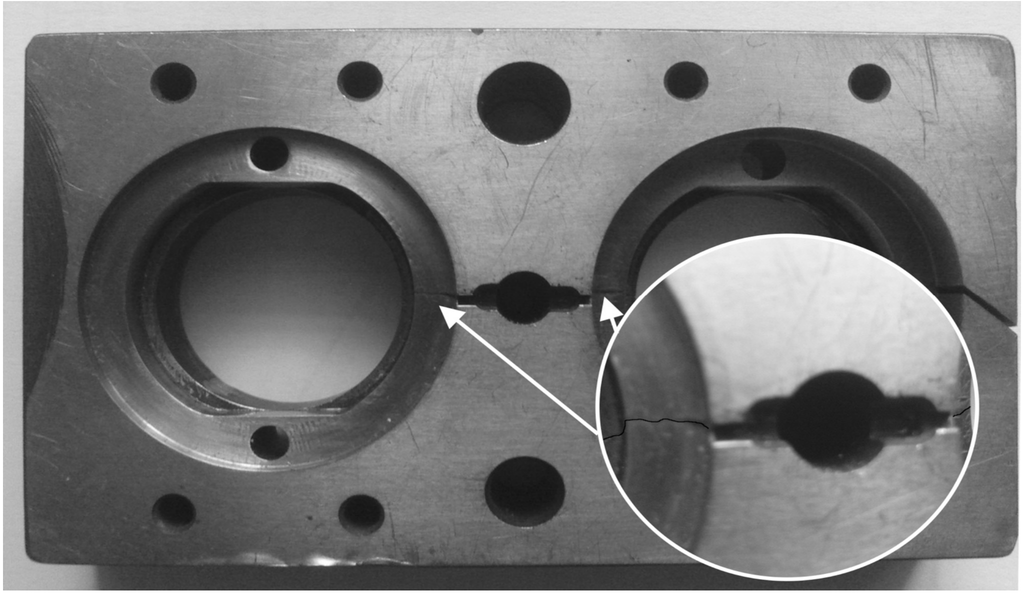

Fig. 1. Matrix of the press-form in which defects appeared during operation (cracks are indicated by the arrows)

It can be seen from the analysis of the detail that the dimensions of the permit between the forming nests are insufficient and a crack has formed, so it is relevant to determine their optimal dimensions.

Main Material Presentation

Formalized algorithm of automatic calculation of rectangular matrix for steadiness and stiffness.

While finite element construction, the algorithm for the automated calculation of the rectangular matrix for steadiness and stiffness in the environment of any functional construction system (CAE) includes the following steps:

1st stage. A three-dimensional model of the part (matrix) is constructed in the 3D module of the CAD system, for which a calculation must be made. We open the built model with the CAE module for calculations.

2d stage. For calculations, the following parameters of the constructed model are set:

a) material & details. The system enters a number of characteristics of the specified material: the strength limit of the material, the boundary of fluidity, the modulus of elasticity of the first and second kind, etc. in the outer data:

b) areas and shapes of cross-sections of individual elements of the detail;

c) fixing the detail. Based on the operating conditions of the future product, the engineer sets the limits of the degrees of freedom of the model, imposing standard restrictions and fastenings on its corresponding faces (surfaces), edges, and vertices – movable hinge, fixed hinge, rigid clamping;

d) stressing fittings. Similarly, based on the operating conditions of the future product, the engineer sets the forces acting on the part by applying force vectors to the corresponding faces (surfaces), edges, and vertices of the three-dimensional model of the part and indicates their absolute values. Load can be set both concentrated and distributed. The concentrated load is set at a specific point of the model and is set through force projections on the X, Y, Z axis (the corresponding numerical values of these projections are entered), or the absolute value of the force and the angle between the force vector and the X, Y, Z axes are set. Distributed load can be applied both to the edges of the model and to its faces. Force values are specified in the appropriate units of the used CAE system, preferably in the units of the SI system.

3rd stage. CAE system is needed to be set up for calculations: choose finite element analysis type (static or dynamic), select type and size of finite elements, specify the required accuracy of the future calculation, etc.

4th stage. Based on the entered parameters, the CAE system in its automatic mode, closed from the user calculates the strength of the model of the part (selects nodes detail volume, divides it into finite elements, numbers nodes, builds a matrix of elements, compiles calculation equations). This process could be quite long and depends on the technical calculation facilities, the complexity of the three-dimensional model, the specified accuracy of the calculation, etc. The results are displayed in the form of tabular data and visual charts.

The table data shows the values of stresses that occur in the nodes of the model, that is, the values of stresses throughout the volume of the detail depending on its material, fastening, and applied load as well as the deformation of the detail (indicated in relative or absolute movements of its individual nodes). Color epures visually show the distribution of stresses, zones of the largest and smallest stresses, and places where the detail will be destroyed according to the model of the detail through the connection map of the corresponding color with the corresponding stress value indicated on the picture. Another type of epur shows a deformed (depending on the initial conditions) detail. Upon completing the calculations, the CAE system also shows the coefficient of the strength reserve of the detail. As result, one can come to a conclusion about the possibility of using the designed part in the given operating conditions.

5th stage. This stage can be called the engineering analysis stage. Based on the given results of the strength calculation and the given task, the user works in a dialog mode with the CAE/CAD system: he changes the shape, the type, the conditions of fastening and loading of the detail if necessary, conducts new finite element calculations, analyzes to get the necessary result to ensure the reliability of the future product operation.

Reference example

The necessary strength calculation for the designed three-dimensional matrix model of press-form in the SolidWorks CAD system is shown in Fig. 2

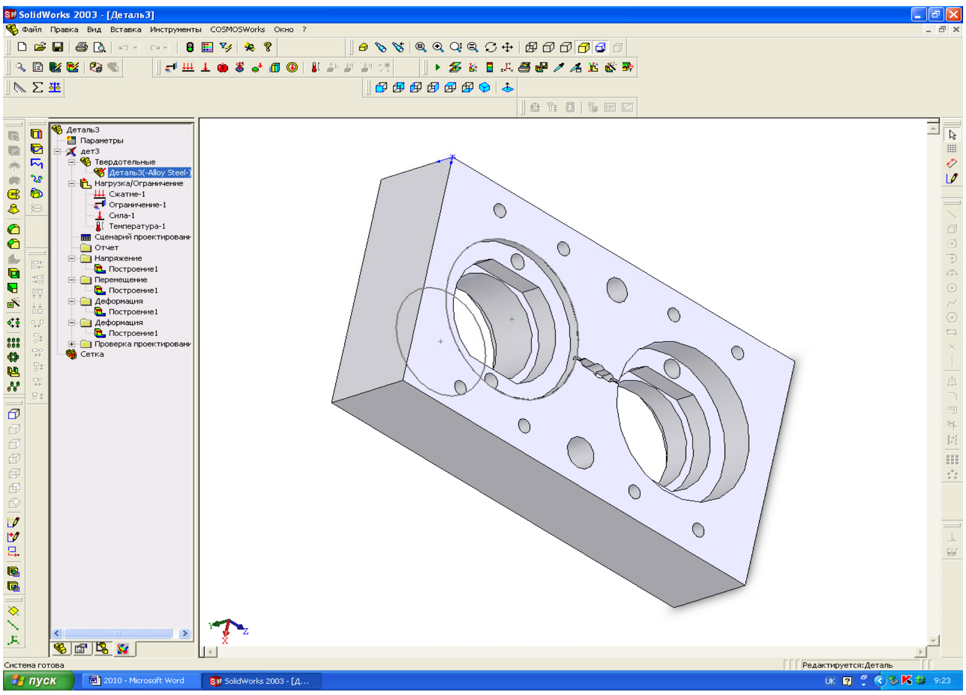

Fig. 2. Matrix designed in the SolidWorks system.

The next step lies in using its COSMOSWorks finite element calculation module in the system for construction studies. To do so, we set the operating temperature and forces acting on the part based on the operating conditions of the future product: the specific pressure of liquid plastic into the cavity formulation (normal evenly distributed force). By applying force vectors to the forming faces (surfaces), edges, and vertices of the three-dimensional model, we indicate their absolute values in Fig. 3.

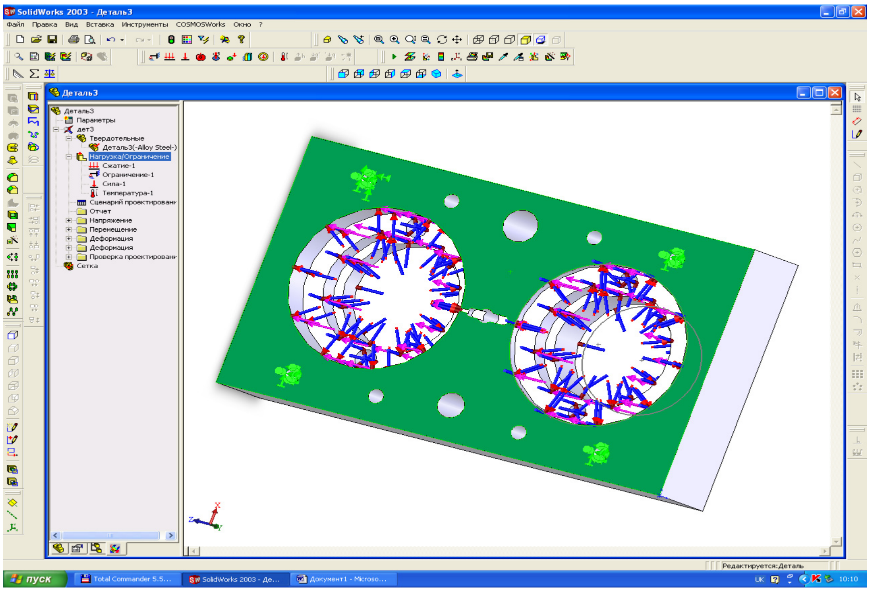

Fig. 3. The forces falling on the matrix and the fixing options

We specify matrix material (steel of a certain type), select the static analysis, the type of finite elements, and indicate the required accuracy of the future calculation. Then we carry out discretization – construction of finite-element mesh (Fig. 4). Adjusting the mesh density ensures the accuracy and time of calculations.

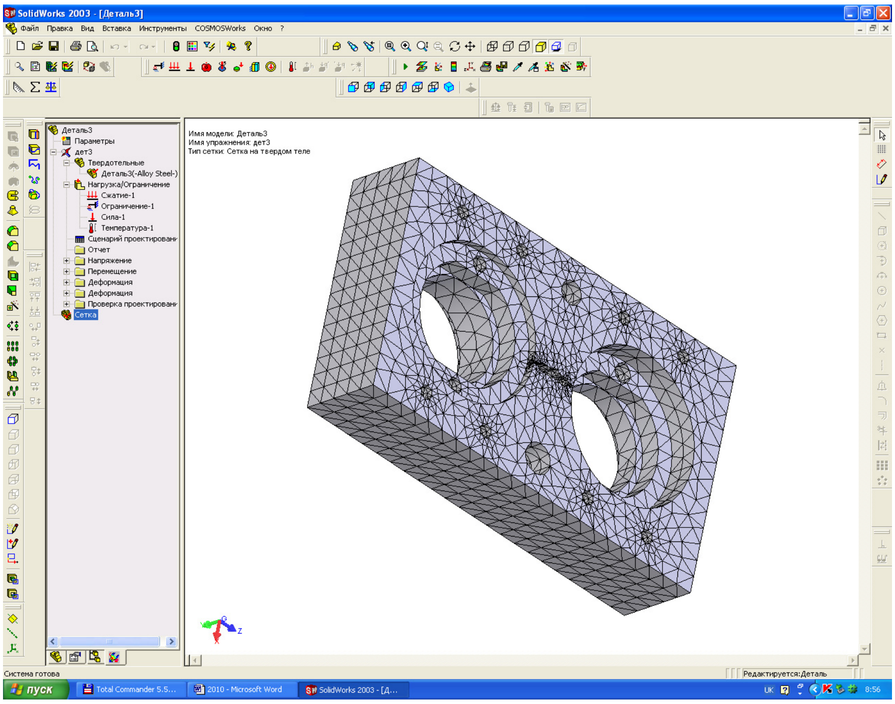

Fig. 4. Finite-element mesh

We make calculations for different model parameters and choose the optimal model construction form (Fig. 5).

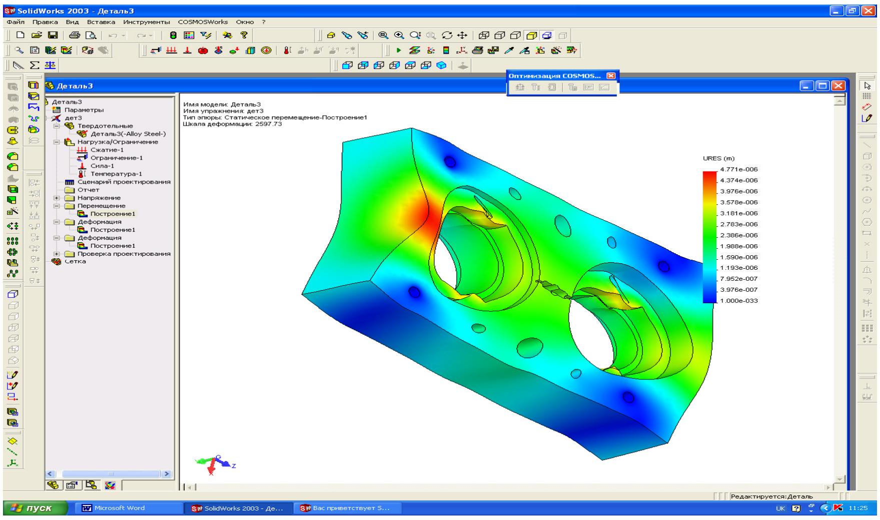

Fig. 5. Virtual results of the calculation of the deformed matrix of press-form

Analyzing the isofield of equivalent stresses and deformations of the models along the Oz axis (Fig. 5), we can conclude the following:

1. The distribution of equivalent stresses is observed on the lateral short sides of the matrix, the stress concentrators are located on the bridge between the formative cavity and the side panel.
2. The distribution of equivalent stresses on the longer sides of the matrix is smaller because the fastening nodes (four holes from bolts) are situated in this zone.
3. The lowest stresses and the smallest deformations were obtained by the finite element method on “CosmosWorks”, where all structural elements of the matrix are taken into account with attention to detail, which indicates the expediency of highly detailed modeling.

Conclusions

1. As a result of three-dimensional models of the matrix of press-form in the SolidWork system, we obtained its construction optimized by stiffness and steadiness.

2. Based on the research results, we identified the matrix parameters that will ensure optimum operating capacity.

References

[1] Dubenets V. H., Khilchevskyi V. V., Savchenko O. V. Osnovy metodu skinchennykh elementiv: navch. posib. Chernihiv: ChDTU, 2007. 288 s. [in Ukrainian].

[2] Dobrianskyi S. S. Tekhnolohichni osnovy mashynobuduvannia. Kyiv: KPI im. Ihoria Sikorskoho, 2020. 379 s. [in Ukrainian].

[3] Petrenko A. M., Semenov O. Y. Osnovy postroenyia system avtomatyzyrovannoho proektyrovanyia. K.: Vyshcha shkola, 1984. 296 s. [in Russian].

[4] Boiko M. V., Velyka O. T., Topilnytskyi V. H., Lampika R. V. Rozrakhunok priamokutnoi matrytsi pres-formy na mitsnist i zhorstkist, Avtomatyzatsiia vyrob. protsesiv u mashynobud. ta pryladobuduvanni, NULP 2010. Vyp. 44. S. 74–79. [in Ukrainian].

[5] Velyka O., Boiko M. Rozrakhunok priamokutnoi matrytsi pres-formy na mitsnist i zhorstkist. Teoriia ta praktyka ratsionalnoho proektuvannia, vyhotovlennia i ekspluatatsii mashynobudivnykh konstruktsii. Materialy konferentsii NULP, Lviv, 2010. S. 38. [in Ukrainian].

[6] 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 / Nats. un-t “Lviv. Politekhnika” [ta in.]. L., 2009. S. 155–156. [in Ukrainian].