

<https://doi.org/10.32056/KOMAG2021.2.7>

Concept of a Drill Press Rotary Tool Workstation Stand with a strength analysis

Received: 09.03.2021

Accepted: 18.06.2021

Published online: 30.06.2021

Author's affiliations and addresses:

¹ KOMAG Institute of Mining Technology, Pszczyńska 37, 44-100 Gliwice, Poland

² Ivano-Frankivsk National Technical University of Oil and Gas, 15 Karpatska Str., building № 4, Ivano-Frankivsk, 76000, Ukraine

*Correspondence:

e-mail: drupik@komag.eu

Dominik RUPIK ^{1*}, **Liubomyr ROMANYSHYN**  ²

Abstract:

Comparative analysis of the strength properties of a drill stand intended for a power tool made by the selected manufacturer and the stand of own design is the paper subject. The specifics of high-speed power tools as well as their capabilities, practical applications and limitations are described. The design of a commercially available drill stand for a drill press rotary tool is presented. Basic issues in the field of CAD modelling, as well as the most important concepts related to the finite element method, as well as the stages of strength analysis using the above-mentioned method, are presented. Based on the simplified model and the analysis of the presented stand, a CAD model of a drill press stand, designed in accordance with own alternative concept is given. Results of the analysis of the static stiffness of both models using the finite element method and the conclusions drawn on this basis are presented.

Keywords: drill stand, power tools, FEM analysis, static stiffness, CAD



1. Introduction

High-speed power tools are compact, hand-operated electric tools designed for minor machining work. Versatility is their most important feature. Wide range of auxiliary accessories extends the device's capability, so the following operations are possible [1]:

- drilling,
- cutting,
- carving,
- engraving,
- milling,
- grinding,
- sharpening,
- sanding,
- polishing.

Due to the manual control of drill pressed rotary tools, it is impossible to drill precisely without the use of additional accessories. For this reason, such power tools are usually combined with stands. An example of a drill stand is shown in Fig. 1.

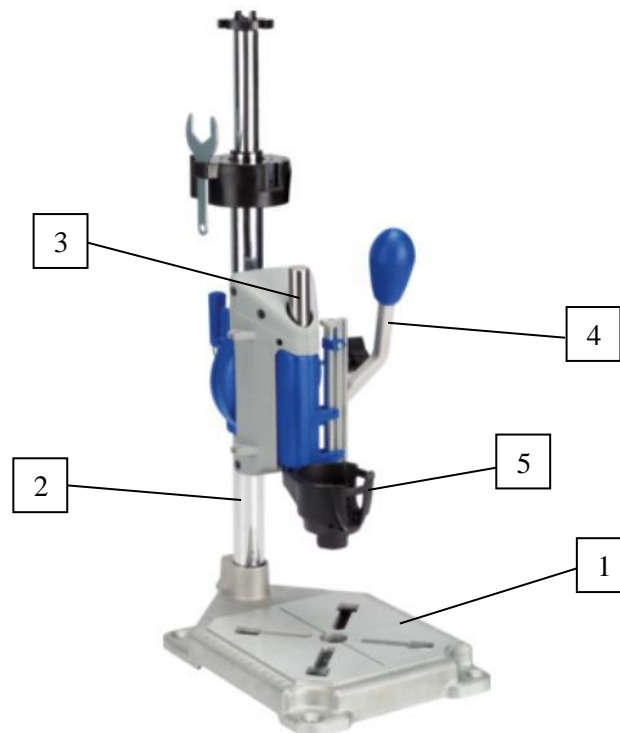


Fig. 1. The drill stand of 220 type [1]:

1 – base, 2 – telescopic column, 3 – rack and pinion mechanism,
4 – lever, 5 – tool holder

The 220-type stand consists of an aluminum base bolted to the base of a working area of dimensions 143x153 mm, on which the workpiece is mounted. The base is connected to a telescopic stand to adjust the height of the power tool by unscrewing the bolt, moving the handle and tightening it. The rack and pinion mechanism enables moving the drill. Pulling the lever down rotates the gears and lowers the tool holder. The return spring brings the tool to its original position, when the lever is released. Drill press rotary tool is screwed to the tool holder by a threaded head of the device. The maximum feed of the mechanism is 50 mm and it is adjustable. A large part of the construction components, including the tool holder, are made of plastic [1].

A simplified CAD model of the discussed drill stand (Fig. 1), developed for the purpose of static analysis is shown in Fig. 2. Structure and geometrical features of the 220 type drill stand, were measured and recreated in the model.

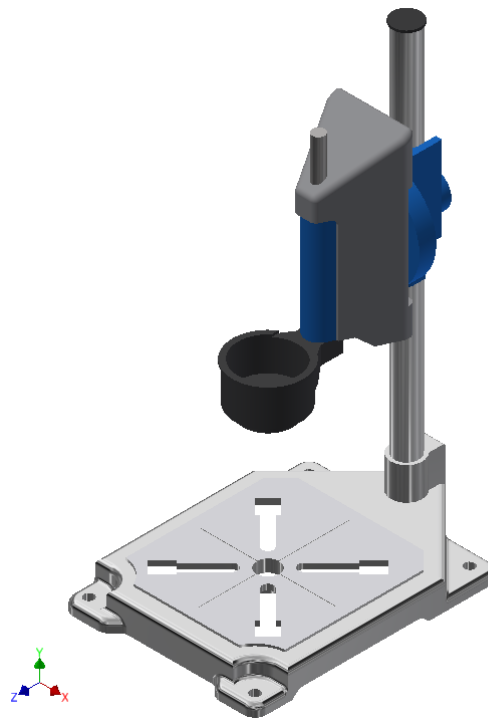


Fig. 2. Simplified CAD model of 220-type stand [12]

2. Materials and Methods

Development of advanced computer technologies led to creation of Computer Integrated Manufacturing (CIM), which resulted in shortening of production cycles and an increase in the degree of utilisation of manufacturing potential. Computer Aided Design (CAD), which supports technical design processes through the use of specialized computer software is one of the basic CIM subsystems. Following processes are aided [2,3]:

- geometric modelling,
- strength analysis,
- creation and development of technical documentation based on the model,
- development of technological documentation,
- prototyping, simulations.

CAD software allows the creation of digital geometric models. The model is expressed by geometric elements consisting of a series of points. Each point has specific coordinates in the adopted coordinate system. The geometric element can be a single point, line, area, or a solid. Digital models present structural form of the product. The product may be a simple element consisting of one object or an extended unit consisting of many interconnected objects [3].

The finite element method is one of the basic tools used in engineering calculations. It is an advanced method of numerical solving of boundary problems, used already at the designing stage for structural strength analysis and simulation of displacements, deformations, stresses, heat and fluid flow. The method consists in using a digital CAD model of a structure to build a discrete model for the needs of strength analysis [4,5]. There are the following stages of the method [5,6]:

- division of the analyzed area into finite elements connected by nodes,

- specification of the element and approximate solution for each of the elements (selection of approximate method, selection of interpolating functions, determination of the algebraic equation system of the tested element),
- creating one system of algebraic equations of the tested element based on the equations of elements taking into account continuity between them,
- equation of boundary conditions of the entire system,
- solution of the created system of equations,
- calculation of values dependent on the determined discrete values.

A model which surface is divided into confined sub-areas called finite elements is known as a discrete model. Finite elements form a mesh that covers the entire model. The mesh density depends on the number of finite elements it has and it significantly impacts the results accuracy. As the mesh density increases, the accuracy of analysis also increases, but also time required for calculation and the demand for computing power increases. It is possible to densify only a certain area of the mesh in the key places of the structure. Some simplifications of the model structure details, which are irrelevant to the results of analysis and significantly extending process duration are often applied to the discrete model [7, 8, 9].

A finite element is a separated fragment of a structure that connects with other finite elements in nodes. Nodes are the points that define the geometry of each finite element. The number of nodes in each finite element depends on the geometrical shape of the element, and each node may belong to several finite elements at the same time [4,10]. There are many types of finite elements. The criteria that characterise the features of each type of element are as follows [11]:

- element dimension: one-dimensional, two-dimensional, three-dimensional,
- geometric shape: point, segment, triangular, quadrilateral, polygonal, tetrahedral, pentagonal, cuboid, axisymmetric,
- type and polynomial degree of the assumed shape function,
- number of nodes in the element,
- general constraints imposed on the element.

The concept of FEM is based on the use of a discrete model divided into sub-areas defining the sets of continuous functions to approximate any continuous function. This translates into a significant simplification of analytical processes as differential equations are replaced by algebraic equations [4]. By the use of finite elements during the analysis, it is possible to replace complicated calculations with algebraic functions of the displacement field. Most often they are low-degree polynomials [10].

It is assumed that the measure of the structure stiffness - in the case of a drill stand - significantly affecting the accuracy of machining, is the stiffness index - j , determined from the relationship (1):

$$j = \frac{P}{f_{max}} \quad (1)$$

where:

- j – stiffness index [N/ μ m],
- P – force [N],
- f_{max} – maximum displacement [μ m].

The concept of stiffness index was used in this publication for a comparative analysis of drill stands of various designs.

3. Results: the concept of a drill stand

Analysis of designs and functional features of drill stands available on the market, in particular the drill stand shown in Fig. 1, enabled developing the own concept of this device, presented in detail in [12]. When designing the drill stand, the following assumptions were made:

- The stand is made of as many as possible commercially available components.

- The power tool feed mechanism is designed to ensure smooth movement of the required length.
- The materials selected for each component of the stand are to ensure the highest possible stiffness of the entire structure, which affects the accuracy of the drilled holes.

Final design is presented in Fig. 3.

The base of the stand is made of milled aluminium sheet. As in the 220 type stand (Fig. 1), it is possible to screw the base to the floor. There are also mounting holes on the surface of the metal sheet, needed to screw the stand column in the form of a square tube. The work table, of dimensions 195x130 mm, has grooves for fixing the workpiece and a hole for drilling that prevents the drill from contacting with the base when drilling outlets. The feed mechanism is based on the cooperation of a trapezoidal screw with a nut. The end of the screw is seated in a flange bearing bolted to the base

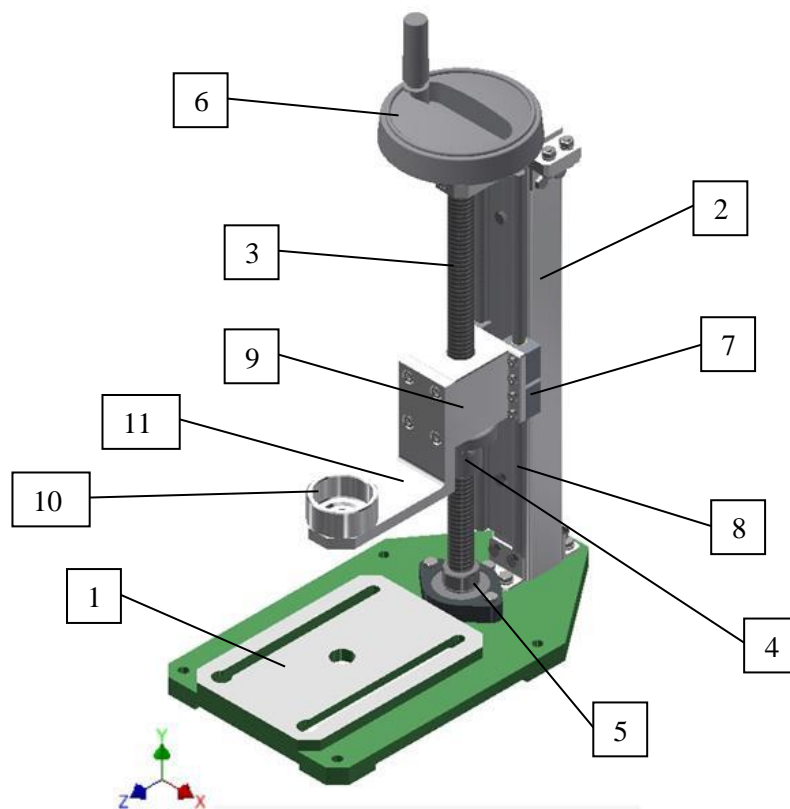


Fig. 3. Model of the drill stand of own design [12]:

1 – base, 2 – stand, 3 – trapezoidal screw, 4 – nut with a trapezoid thread, 5 – flange bearing, 6 – hand wheel, 7 – linear bearing, 8 – double-sided guide, 9 – connector, 10 – tool holder, 11 – holder arm

The base of the stand is made of milled aluminium sheet. As in the 220 type stand (Fig. 1), it is possible to screw the base to the floor. There are also mounting holes on the surface of the metal sheet, needed to screw the stand column in the form of a square tube. The work table, of dimensions 195x130 mm, has grooves for fixing the workpiece and a hole for drilling that prevents the drill from contacting with the base when drilling outlets. The feed mechanism is based on the cooperation of a trapezoidal screw with a nut. The end of the screw is seated in a flange bearing bolted to the base. There is a handwheel at the other end. By turning the wheel, the operator rotates the bolt, as a result of which the nut mounted on the bolt moves linearly up or down, depending on the direction of the screw rotation. Such a movement of the nut is possible thanks to four linear bearings, cooperating with a double-sided guide of a circular profile, which were screwed with the nut by an aluminium element. As the stand is supposed to work together with power tools of a given brand, a threaded hole in the handle of the stand is provided for mounting the device, matching the thread in the head of the tool. The tool handle is screwed into the hole of the arm made of a fragment of the angle bar. The maximum

height at which the drill can be placed is 245 mm. The overall dimensions of the structure are 190x340x523 mm [12].

Rigidity is an important property of a drill stand, significantly affecting its functional features. Taking the above into account, the own stand and the 220 type stand models were compared for the static stiffness. The calculations included the materials of both structures. In the case of a own stand, most of the elements were made of aluminium, while the structure of the type 220 stand consisted of steel and plastic elements. It was assumed that the drill is at a height of 115 mm from the working area of the stand base, and the force loading the model is 10N. The place and direction of the force applied is shown in Fig. 4 [12].

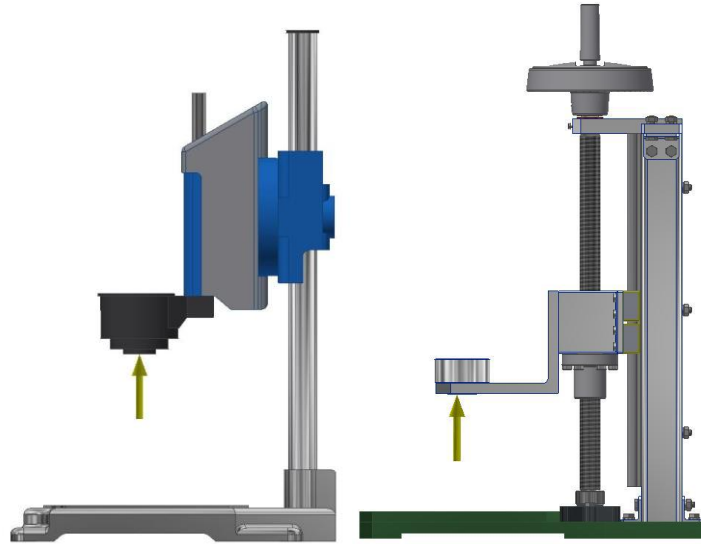


Fig. 4. Place and direction of applying force to both models [12]

In Fig. 5 the removed degrees of freedom of the stand models are schematically marked.

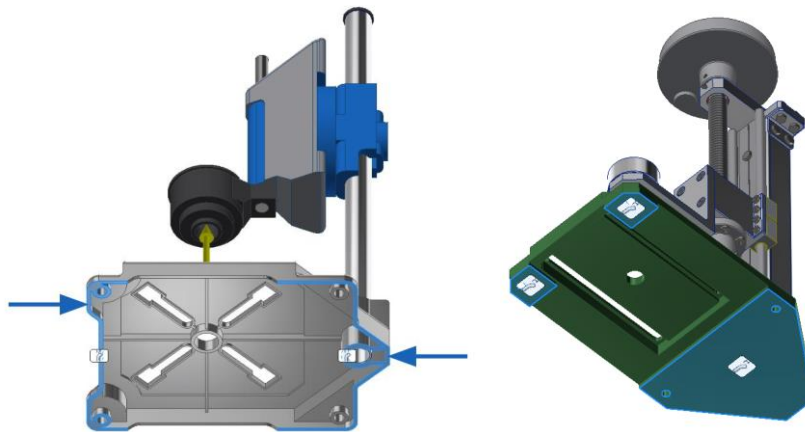


Fig. 5. Places of removing all degrees of freedom in the models are marked in blue [12]

The next stage of the static analysis was to discretize the models. Too low density of the mesh could result in a significant error, while with the increase in density, the duration of the calculations would significantly increase. For this reason, several calculation attempts were made for various mesh parameters to estimate its optimal compaction. Additionally, in the places, where the greatest stresses are predicted, the mesh was compacted. The mesh consists of triangular three-node finite elements (Fig. 6) [4,8,12].

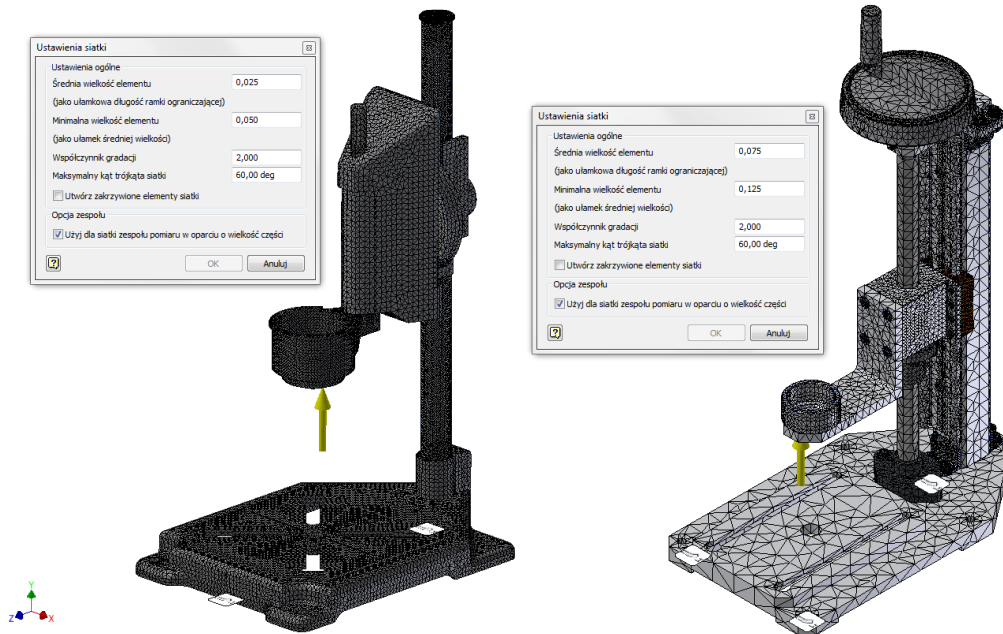


Fig 6. Finite elements mesh of the stand model [12]

As a result of the analysis, it was found that in both models the highest stresses caused by the given force arose within the tool holder arm. In the case of the 220-type stand model, the maximum stress was 3.2 MPa, while for our own design, it was 5.1MPa (Fig. 7-8) [12].

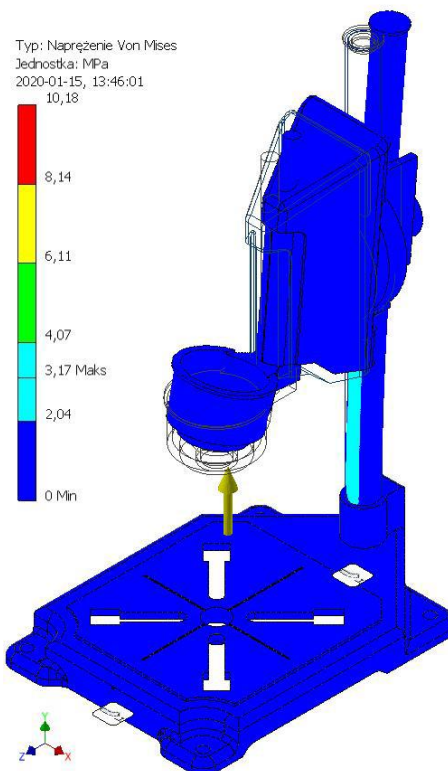


Fig. 7. Distribution of stresses on 220-type stand model according to the Huber hypothesis [12]

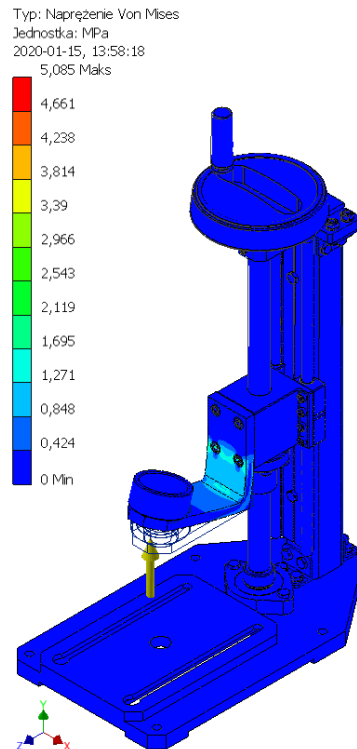


Fig. 8. Distribution of stresses on own developed stand model according to the Huber hypothesis [12]

In the case of the displacement analysis for the 220-type stand model, the greatest total displacements were recorded at the tool holder and in the upper part of the stand. The maximum total displacement was 0.156 mm (Fig. 9). For the stand model of own design, displacements appeared only on the tool holder and the holder arm. The maximum total displacement was 0.03 mm (Fig.10) [12].

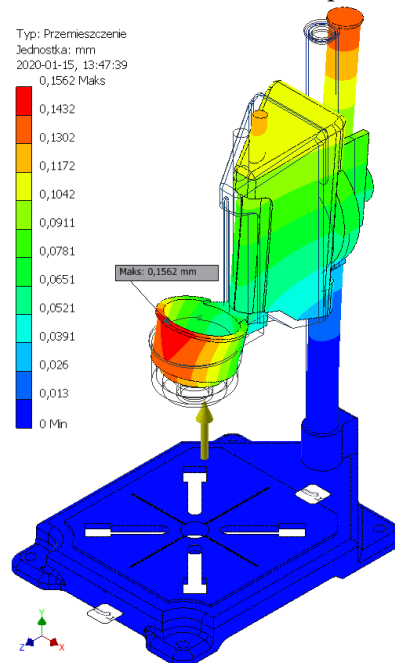


Fig. 9. Total displacements on 220-type stand model [12]

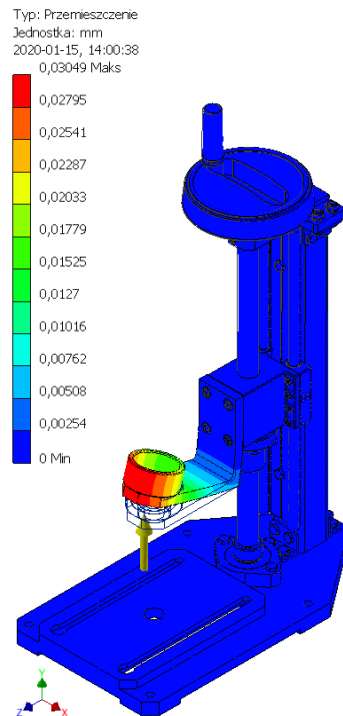


Fig. 10. Total displacements on the stand model of own design [12]

Using the formula (1), the static stiffness index in the direction of the loading force along the Y axis of the coordinate system for both analysed models of the stand was determined. For the 220-type stand model, the maximum displacement in relation to the Y axis was 0.146 mm, and the stiffness index was 0.068 N/ μ m (Fig. 11). For the stand of own design, the maximum displacement in relation to the Y axis was 0.03 mm and the stiffness index was 0.332 N/ μ m (Fig. 11) [12].

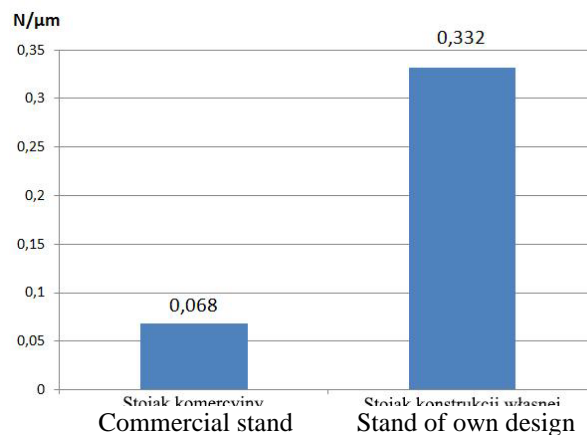


Fig. 11. Bar graph of the stiffness index in a direction of force action for both models [12]

4. Conclusions

The CAD models enabled analysis of the static stiffness, the results of which showed that the use of stronger materials and a stand of a bigger cross-section surface area resulted in significantly better strength parameters. The stiffness index was almost five times higher than in the case of existing stand. The analysis allowed to specify the areas with the greatest stresses and the elements with the greatest displacements when drilling, what would affect the machining accuracy. In the case of the custom-made stand, arm with the tool holder attached was the element most susceptible to displacement and greatest stress. However, the displacements and stresses were so small that it was not necessary to modify the structure. The use of the finite element method allowed to verify and

compare the strength of the structure of both models at the designing stage. If structural modifications were necessary, the method would reduce designing costs and time, as there is no need to make a test stand prototype model.

References

- [1] Dremel: User manual, 2019
- [2] Płuciennik P, Maciejczyk A.: Wykorzystanie systemów CAD w procesie konstruowania maszyn i urządzeń, *Logistyka* 2016 nr 6
- [3] Sydor M.: Wprowadzenie do CAD. Podstawy komputerowo wspomaganego projektowania, Warszawa, PWN 2009
- [4] Mater A.: Metoda elementów skończonych w wybranych zagadnieniach mechaniki konstrukcji inżynierskich, Politechnika Poznańska, 2003
- [5] Sadecka L.: Metoda różnic skończonych i metoda elementów skończonych w zagadnieniach mechaniki konstrukcji i podłoża, Politechnika Opolska 2010
- [6] Singiresu S. Rao: The finite element method in engineering, Elsevier Butterworth-Heinemann 2018
- [7] Józwick J., Włodarczyk M., Adamowicz D.: Analiza numeryczna MES wybranych rozwiązań konstrukcyjnych ścian bocznych kabin dźwigów osobowych. *Postępy Nauki i Techniki*. 2011, nr 7
- [8] Czabanowski R.: Numeryczna analiza wytrzymałościowa ustroju nośnego stojaka podnośnego, *Autobusy: technika, eksploatacja, systemy transportowe*. 2017 nr 11
- [9] Zienkiewicz O. C., Taylor R. L.: The finite element method for solid and structural mechanics, Elsevier Butterworth-Heinemann, 2006
- [10] Śródka W.: Trzy lekcje metody elementów skończonych: materiały pomocnicze do przedmiotu wytrzymałość materiałów, Oficyna Wydawnicza Politechniki Wrocławskiej, Wrocław 2004
- [11] Rusiński E., Czmochoński J., Smolnicki T.: Zaawansowana metoda elementów skończonych w konstrukcjach nośnych, Oficyna Wydawnicza Politechniki Wrocławskiej. Wrocław, 2000
- [12] Rupik D.: Projekt stojaka elektronarzędzia szybkoobrotowego, Politechnika Śląska, Wydział Mechaniczny Technologiczny, Katedra Budowy Maszyn, Gliwice 2020