ARCH. MECH. TECH. MATER.VOL. 39, (2019) 1-6

DE GRUYTER OPEN

ARCHIVES OF MECHANICAL TECHNOLOGY AND MATERIALS
WWW.AMTM.PUT.POZNAN.PL

Development of a computational model of a lightweight vertical lathe with the use of superelements

Bartosz Powałka[a], Michał Dolata[a]*

[a]*Institute of Mechanical Technology, Department of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology, al. Piastów 19, 70-310 Szczecin, Poland*

* *Corresponding author, Tel.: +48-91-449-43-70, Fax: +48-91-449-44-42,e-mail address: Michal.Dolata@zut.edu.pl*

ARTICLE INFO

Received 30 January 2018
Received in revised form 07 April 2018
Accepted 14 December 2018

KEY WORDS

Lathe, FEM, superelements, weak links

ABSTRACT

This article presents a process of developing a computational model of a light construction vertical lathe. The model is made using the finite elements method. Due to the complexity of the machine tool and the optimization of the computation time superelements have been used in the model. The application of this method has reduced the computation time allowing the analysis of many variants of the mutual position of the machine components. The results of the analyzes led to the indication of weak links of the machine tool. After the improvement of the machine tool construction the movement of the tool tip and the workpiece was much smaller.

1. INTRODUCTION

The use of advanced simulation tools in the process of developing modern technological machines is a standard procedure. Those producers who want to play a significant role on today's market have to optimize their constructions. The process of designing a modern machine, along with simulation analysis, allows the avoidance of making expensive preliminary prototypes often containing erroneous solutions. Despite the undoubted advantages of such an approach, professional programs used to conduct simulation analyzes of complex machine tool models are extremely expensive. Not all companies are able to invest considerable funds in the process of computations. Moreover, the development of effective and reliable calculation models requires knowledge and experience.

2. COMPUTATIONAL MODEL

Thanks to the new, better optimized software, a design department can indicate the weak links of the machine tool. The use of dynamic analyzes and the use of dynamic models of the cutting process in machine tool models [1, 3, 5, 15] gives the opportunity to determine maximum parameters of the technological process in which a virtual machine can work.

This approach to the design process has been applied in the case of designing a modern lightweight vertical lathe, which an illustrative geometric model is shown in the figure below (Fig. 1.).

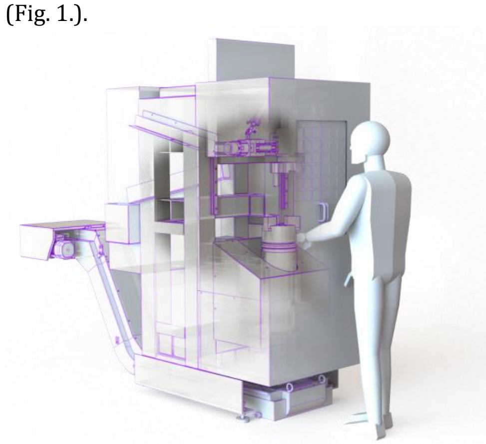

Fig. 1. Geometrical model of a light weight vertical lathe

DOI: 10.2478/amtm-2019-0001

The design is based on welded steel profiles. A proper rigidity and stability of the machine tool is to be achieved by filling the profiles with concrete mass. The development of the method of proper modeling of this type of connection is a separate issue and has been raised by many authors [4, 8, 10, 11, 16, 20]. The outer dimensions of the lathe are 1000 mm wide, 1600 mm long (without chip conveyor) and 2250 mm high. In the initial phase of the work, our task was to determine the weak links of the analyzed machine tool. For this purpose, it was necessary to build a computational model of the analyzed lathe prepared for the FEM software which in this case was Midas NFX. The first step was to simplify the geometric model to a form in which unnecessary elements - not affecting the technological process will not increase the dimensionality of the computational model (Fig. 2). Therefore, the casing and feeder were removed. The screws and their holes were also removed. The screw connections were replaced with welded contact.

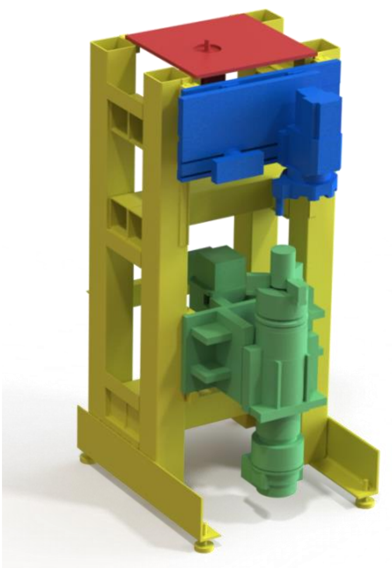

Fig. 2. Geometrical model of a lathe after simplifications

The process of developing a computational model was based on the finite elements method. In this method, the dimensionality of the model affects the effectiveness of obtaining results. Authors decided to use superelements [12, 14, 17, 18, 19]. The contact layer of the model, in other words, the elements responsible for the movements of individual machine parts, were replaced by superelements with reduced dimensionality. The method of determining the parameters of such elements has been described in several works [2, 6, 7, 9, 13]. Generally, the determination of substitute stiffness of superelements is based on Hertz's theory. In the software used by the authors of this work, the best results are obtained by using substitute elements in the form of rods with analytically determined stiffness. In the machine tool model, superelements replace bearings, roller guides, and ball-screw mechanisms.

The use of replacement elements significantly reduced the dimensionality of the model (number of nodes). Due to the specific construction consisting of thin-walled profiles, the size of a single finite element is small, which leads to a large model (Fig. 3) despite all simplifications.

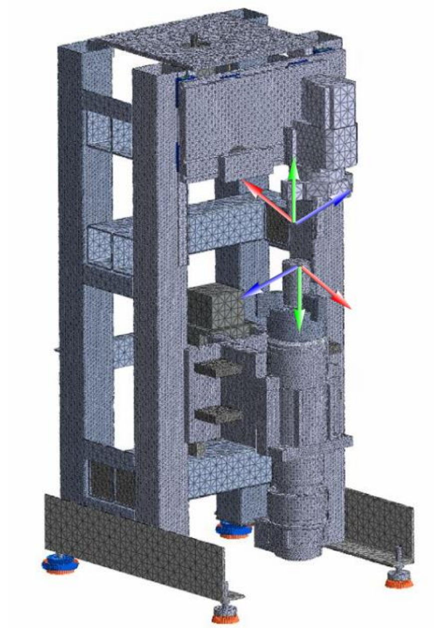

Fig. 3. A meshed model with constraints and loads

The developed model consisted of 2,72 million nodes and 4,52 million finite elements. The figure above (fig. 3.) shows constraints and loads applied to the model.

The feet of the machine tool are fixed. The load was derived from the forces of gravity and additionally from the load resulting from the cutting process. The forces simulating the cutting process have values determined on the basis of experimental research. These forces are applied to the tool and to the workpiece. Their values that correspond to rough turning process (experimental data) are presented in the table below (Table 1).

Table 1. Forces applied to the model

Direction	Fx	Fy	Fz
Cutting tool	-2384N	1294 N	-2395N
Workpiece	2384 N	-1294 N	2395 N

3. STATIC ANALYSIS

The first step was to perform a static analysis including only gravity. The following figure shows the displacement graph (Figure 4).

In Fig .4, a significant displacement of the upper part of the structure is visible. After observing significant displacements, stress and strain diagrams were analyzed. The authors concluded that the angle beams that are the basis of the structure are too weak. The displacements of the working part exceeded 0.3mm. The design department of the manufacturer required some time to change the structure of the foundation. Our team decided not to waste time and to subject the construction to further analyzes without the angle beams of the foundation.

The next study was carried out on a model with no problematic basis (Figure 5).

The model without susceptible elements of the foundation consisted of 4.49 million nodes and 2.66 million finite elements. In the next stage of the research, the model was loaded with the force of gravity and by the forces simulating the cutting process (Table 1). As a result of the calculations, a displacement graph was obtained as shown in the figure below (Figure 6).

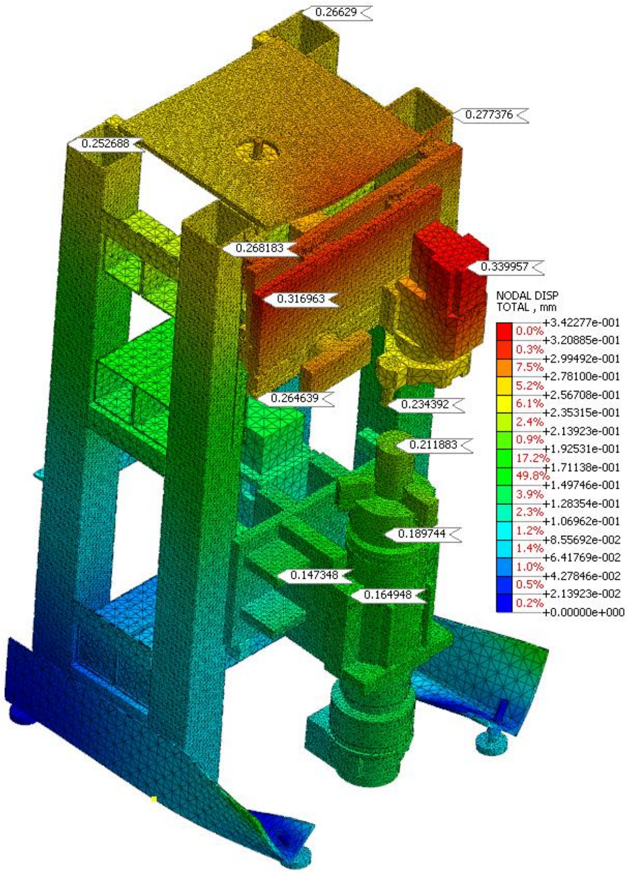

Fig. 4. Displacements after gravity applied to the model

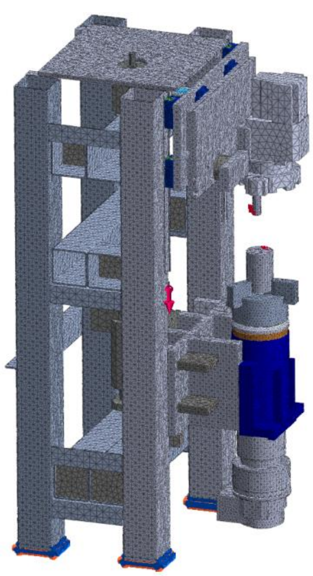

Fig. 5. Meshed model without its base beams

The graph above shows a significant relative displacement of the tool tip and the workpiece. Both the workpiece and the tool branches are too susceptible – in both cases displacements are over 0.2 mm. Finding the reasons for this condition – the machine subassemblies responsible for excessive displacement under loads required in-depth analysis of deformations in the contact zone.

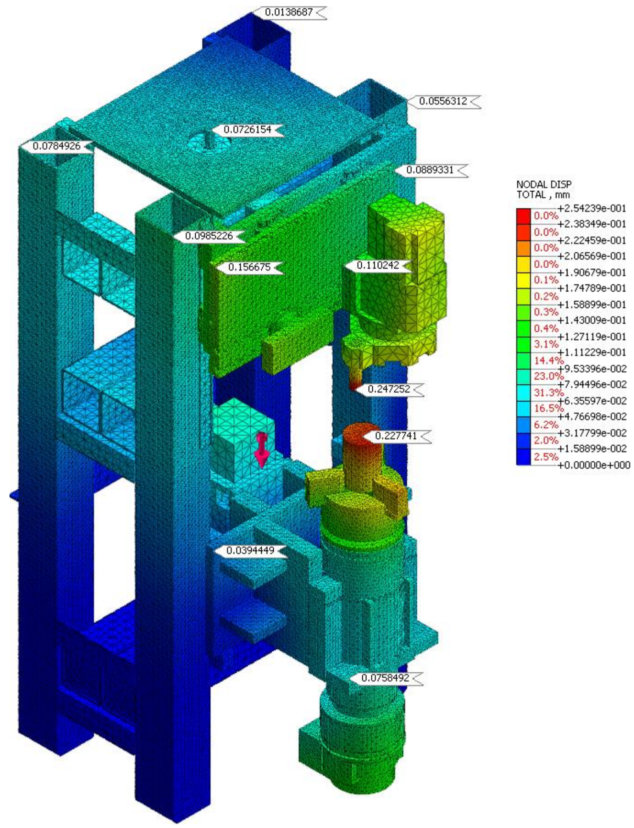

Fig. 6. Displacements after applying gravity and forces to the model without base

Three main reasons for significant displacements were found. The first concerned the mounting plate of the tool head. It turned out that the plate has a significant undercut necessary to avoid collision with the bearing node of the ball screw mechanism responsible for horizontal movement of the head support. This design causes considerable system susceptibility. The deformation plot for this part of the structure is shown in the figure 7.

The second problem is the spindle bearings. The forces act on the workpiece along the Z axis in a downward direction (the same as gravity). The bearings should be arranged in a way that they are stiffer in the opposite direction to the largest force. The upper spindle bearing node consisted of a set of three angular contact bearings, two of which were directed in the wrong way, which caused the spindle assembly to be too susceptible.

The third problem is the considerable load on the front profiles caused by the mass of the tool head assembly and the spindle exposed beyond the outline of the structure. Welding of the ribs or reinforcement of the front profiles were suggested. The second proposal was to redesign the structure so that the spindle was supported by an additional leg. The deformation plot of this fragment of the structure is shown in the figure below (Figure 8).

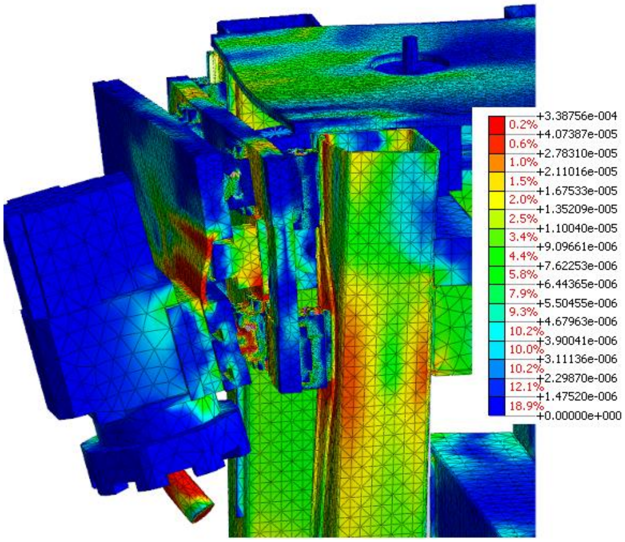

Fig. 7. Strains in the upper part of analyzed lathe

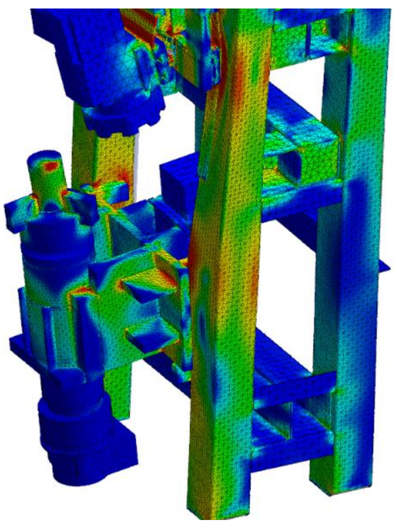

Fig. 8. Strains in lower part of analyzed lathe

4. MODIFIED MODEL

The report from the analyzes was provided to manufacturers design team which introduced changes to the lathe model. The base structure has been strengthened by welding the ribs in the form of plates carrying loads from the front vertical profiles towards the front feet. The spindle bearing system has been changed, the main drive motor mounting has been redesigned. The tool head has been attached to a much larger plate. The corrections introduced into the supporting structure are shown in the figure below (Fig. 9.).

As in the case of the first study, we also started the analysis from gravity loading of the model. The following figure presents the displacement graph (Figure 11).

The second version of the computational model (Figure 10) consisted of 4.21 million nodes and 2.51 million finite elements.

Comparing the graph above with one for the first model (Figure 4), a big improvement is noticeable. The reinforcement plates significantly stiffened the base angle beams.

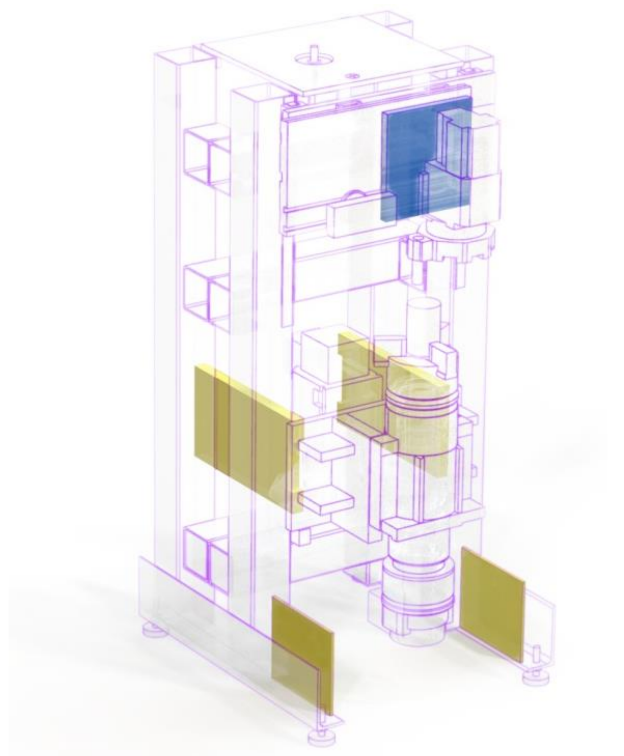

Fig. 9. Modifications in geometrical model

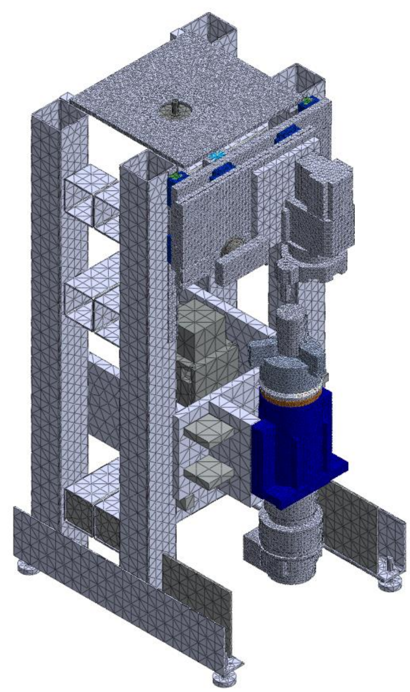

Fig. 10. Mesh of a second version model

The displacements of the upper ends of the structure pillars decreased from approx. 0.26 mm to approx. 0.08 mm. Due to such a large improvement in further research, the authors decided to include the basis in the model. For this reason, the comparison of the results of the first and second versions cannot be done directly - one should check the displacements with respect to the specific point chosen.

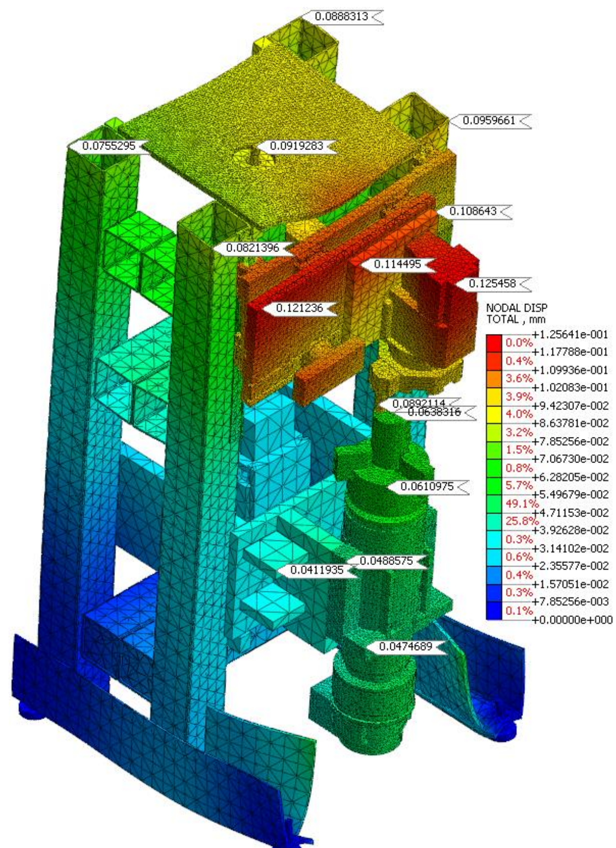

Fig. 11. Displacements after applying gravity force to the second version model

In the next study, a load simulating the forces generated in the cutting process was applied to the model beyond the force of gravity (Table 1). The graph below presents the displacements graph (Figure 12).

The maximum displacement of the tool tip in this case was 0.18 mm, the maximum displacement of the workpiece is 0.19 mm. Referring to the first version, it should be noted that the improvement is significant. The displacements of the tool tip and the workpiece are smaller in this case than before, despite the inclusion of a susceptible base in the construction.

5. SUMMARY

The developed computational model of a vertical lightweight lathe allowed to detect structural defects at the design stage. The initial analyzes carried out concerned the issues of static simulations. The research was aimed at verifying the initial assumptions. Subsequent works will be conducted based on the computational model already built. Due to its specificity, i.e. the use of thin-walled profiles in the construction, the model requires significant mesh resulting in a large model.

Dynamic machine properties should also be analyzed. The application of the finite element method in combination with the dynamic cutting force model will give the possibility to obtain stability lobes graphs. These charts give the opportunity to determine the technological parameters of the designed device. The design team thus has the ability to select the components of the machine and modify the structure to optimize its parameters. This approach enables the company to be competitive on the market.

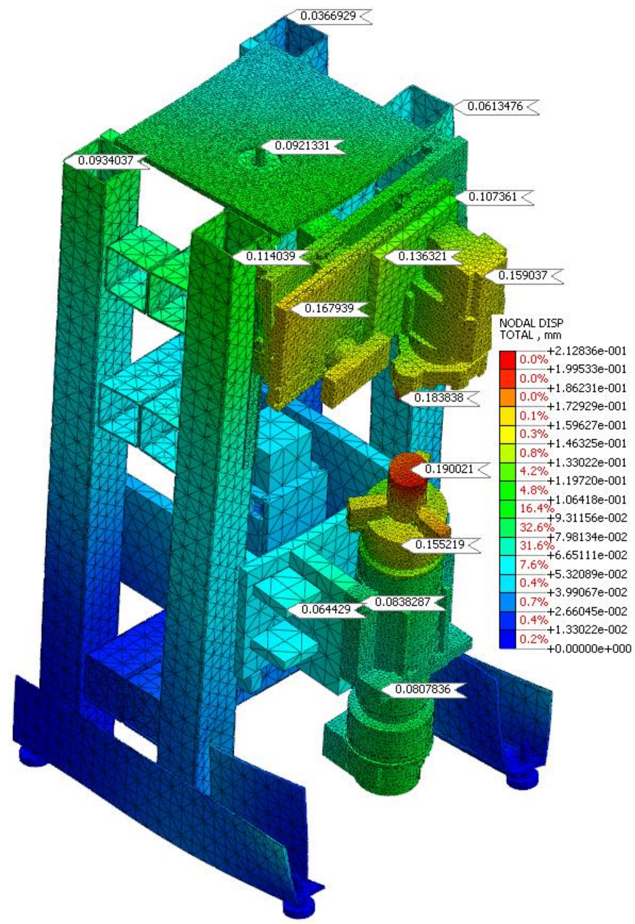

Fig. 12. Displacements in second model after applying gravity and loads

ACKNOWLEDGEMENTS

This work is partially funded by grant Młoda Kadra 2017 from Department of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology and EU grant „Tokarka pionowa o lekkiej konstrukcji" POIR.04.01.02-00-0078/16. The authors would like to thank AFM Defum S.A. company for their help.

REFERENCES

[1] **Y. Altintaş and E. Budak**, Analytical Prediction of Stability Lobes in Milling, *CIRP Ann. - Manuf. Technol.*, vol. 44, no. 1, pp. 357–362, 1995.

[2] **M. Dolata**, Model zastępczy mechanizmu śrubowo-tocznego i jego weryfikacja doświadczalna, Zachodniopomorski Uniwersytet Technologiczny,

2015.
[3] **R. P. H. Faassen, N. Van de Wouw, J. A. J. Oosterling, and H. Nijmeijer**, Prediction of regenerative chatter by modelling and analysis of high-speed milling, *Int. J. Mach. Tools Manuf.*, vol. 43, no. 14, pp. 1437–1446, 2003.
[4] **L. Fang, B. Zhang, G. F. Jin, K. W. Li, and Z. L. Wang**, Seismic behavior of concrete-encased steel cross-shaped columns, *J. Constr. Steel Res.*, vol. 109, pp. 24–33, 2015.
[5] **V. Gagnol, B. C. Bouzgarrou, P. Ray, and C. Barra**, Model-based chatter stability prediction for high-speed spindles, *Int. J. Mach. Tools Manuf.*, vol. 47, no. 7–8, pp. 1176–1186, 2007.
[6] **T. A. Harris**, *Rolling bearing analysis*, 2nd ed. John Wiley and sons, 1984.
[7] **H. Hertz**, *Miscellaneous papers*. Macmillan, 1896.
[8] **A. Le Hoang and E. Fehling**, Numerical study of circular steel tube confined concrete (STCC) stub columns, *J. Constr. Steel Res.*, vol. 136, no. June, pp. 238–255, 2017.
[9] **L. Houpert**, An engineering approach to Hertzian contact elasticity-part I, *Trans. Soc. Mech. Eng. J. Tribol.*, vol. 123, no. 3, pp. 582–588, 2001.
[10] **L. Hu, P. Feng, and X. L. Zhao**, Fatigue design of CFRP strengthened steel members, *Thin-Walled Struct.*, vol. 119, no. June, pp. 482–498, 2017.
[11] **B. Isojeh, M. El-Zeghayar, and F. J. Vecchio**, High-cycle fatigue life prediction of reinforced concrete deep beams, *Eng. Struct.*, vol. 150, pp. 12–24, 2017.
[12] **D. Jastrzębski, P. Pawełko, and G. Szwengier**, Modeling the effect of geometric errors on the static characteristics of guide rail systems, *Adv. Manuf. Sci. Technol.*, vol. 34, no. 4, pp. 23–33, 2010.
[13] **K. L. Johnson and K. L. Johnson**, *Contact mechanics*, Ninth. Cambridge university press, 2003.
[14] **L. Kania**, Modelling of rollers in calculation of slewing bearing with the use of finite elements, *Mech. Mach. theory*, vol. 41, no. 11, pp. 1359–1376, 2006.
[15] **K. Marchelek, M. Pajor, and B. Powałka**, Vibrostability of the Milling Process Described by the Time-Variable Parameter Model, *Modal Anal.*, vol. 8, no. 4, pp. 467–479, Apr. 2002.
[16] **N. H. Nguyen and A. S. Whittaker**, Numerical modelling of steel-plate concrete composite shear walls, *Eng. Struct.*, vol. 150, pp. 1–11, 2017.
[17] **H. Prashad**, Relative comparison of stiffness and damping properties of double decker high precision and conventional rolling-element bearings, *Tribol. Int.*, vol. 35, no. 4, pp. 265–269, 2002.
[18] **E. Rusiński, J. Czmochowski, and T. Smolnicki**, *Zaawansowana metoda elementów skończonych w konstrukcjach nośnych*. Oficyna Wydawnicza Politechniki Wrocławskiej, 2000.
[19] **G. Szwengier**, Modelowanie i obliczania projektowe układów prowadnicowych obrabiarek, *Pr. Nauk. Politech. Szczecińskiej*, vol. 13, no. 512, 1994.
[20] **J. B. Yan and W. Zhang**, Numerical analysis on steel-concrete-steel sandwich plates by damage plasticity model: From materials to structures, *Constr. Build. Mater.*, vol. 149, pp. 801–815, 2017.