ISSN: 2754-4915

Journal of Material Sciences & Manufacturing Research

Scientific Research and Community

Research Article

Open Access

Generative Software for Automated Mechanical Simulations

Hostak Peter* and Soos Lubomir

Faculty of Mechanical Engineering, Slovak University of Technology in Bratislava, Nám Slobody 17, 812 31 Bratislava, Slovakia

ABSTRACT

Nowadays, process of automation is becoming increasingly prominent in nearly all areas of life. Its goal is to streamline routine processes that are time-consuming and can also lead to errors. This article presents an innovative approach to the design of spindle heads that combines the principles of generative programming and adaptive design. It introduces a method for replacing the traditional iterative design process with an automated process in which, using a custom-built software tool, it will be possible to autonomously generate, analyze, and optimize proposed design variants based on precisely defined input parameters and target requirements. The aim of this paper was to create a software tool for the design of spindles and their subsequent verification through automated simulations. The article presents sequence diagrams that the proposed software executes in the background during specific procedures. The main output of the practical part is the verification of the analytical calculation integrated into a separate module in the proposed software and the analysis of simulations performed in ANSYS software. The entire component testing process will also be demonstrated using a sample example of a machine tool spindle. The process will be described in full, covering the analytical method used to determine total deflection, the incorporation of the effect of changes in the stiffness of the bearing nodes, as well as the selection of the tool, the material being machined, and the associated machining conditions. The results for total deflection will be divided into two groups. The first group will consist of total deflection calculations using the presented analytical formulas for deflection calculation. The second group of results will consist of deflection values obtained through automated simulations using ANSYS simulation software, while maintaining the same input parameters as in the analytical calculation. At the end of the article, both sets of results are compared and evaluated, and further research objectives in this area are proposed.

***Corresponding author**

Hostak Peter, Faculty of Mechanical Engineering, Slovak University of Technology in Bratislava, Nám. Slobody 17, 812 31 Bratislava, Slovakia.

Received: June 02, 2026; **Accepted:** June 08, 2026; **Published:** June 15, 2026

Introduction

The spindle, as the carrier of the tool or workpiece, is a key component of all machine tools. It is often described as the „heart“ of the machine tool, its primary function is to ensure the precise and stable rotation of the tool or workpiece under specified cutting conditions. Spindle characteristics affect the behavior of the entire MACHINE – TOOL – FIXTURE – WORKPIECE system, whereby they have a direct impact on the accuracy of machined parts, their surface quality, and the overall productivity of the manufatruing process.

Designing this kind of complex system and the components integrated within it involves cooperation among several specialized departments. Process begins in the design departments, where initial versions of the CAD model are created on the basis of given requirements. Created design is then sent to the computational department, where simulations are performed on the component. During these simulations, the component is verified to ensure it meets the specified technological requirements. Usually, they repeat this cycle several times until they find a suitable solution which meets the customer’s specifications. For engineering companies the process is both time-consuming and costly, therefore, in this article we present a solution for testing variants using a generative tool programmed in MATLAB.

Calculation Methods for Spindle Machine Tools

Engineers use several approaches when designing spindle - bearing systems. Among the most commonly used methods are the transfer matrix method and the finite element method.

Our suggested software tool incorporates an initial parameter method that allows us to determine the total deflection of the spindle. Subsequently, we will compare the value of the total deflection (y_0) with the ANSYS software, which is based on the finite element method. We perform simulations using ANSYS through automated testing, which we discuss in more detail in the chapter 4.

Matrix Form of the Initial Parameters Method

Method of initial parameters is an analytical approach to solving the deformation of beams, based on the direct integration of the differential equation of the line of deformation. Instead of the classical solution for the entire beam, the state of the system - defined by state variables such as deflection, rotation, bending moment and shear force is expressed at any point as a function of its initial state (at the origin of the coordinate system).

Transition through individual sections of the shaft is mathematically represented by multiplying the state vector (v_0) by the corresponding transfer matrix of the given element. Deflection at the end of beam is obtained by sequentional multiplication of the input parameter vector, which according to, is as follows [1]:

$$v_0 = \begin{bmatrix} y_0 \\ \varphi_0 \\ M_0 \\ F_{y0} \\ q_0 \end{bmatrix} \qquad (1.1)$$

where:
y_0 -deflection[m]
φ_0 -rotation angle [°]
M_0 -bending moment [N.m]
Fy_0-shear force [N]
q_0-continuous loading [N.m^{-1}]

Vector (1.1) contains two unknown variables v_{01} and φ_{01}, which are carried forward to the end of the beam in matrix form. We determine the unknown parameters from the deformation conditions. The method is based on repetition of a simple computational algorithm in matrix form, which allows us to easily implement it in MATLAB software.

For the calculation, we consider vector 1.1 in its transposed form. Transfer matrix for the "i"-th section of the spindle is defined as follows:

$$P_i = \begin{bmatrix} 1 & i & -\frac{l_i^2}{2EJ} & -\frac{l_i^3}{6EJ} & \frac{l_i^4}{24EJ} \\ 0 & 1 & -\frac{l_i}{EJ} & \frac{l_i^2}{2EJ} & \frac{l_i^3}{6EJ} \\ 0 & 0 & 1 & l_i & \frac{l_i^2}{2} \\ 0 & 0 & 0 & 1 & -l_i \\ 0 & 0 & 0 & 0 & - \end{bmatrix}$$

Vector of output parameters for the „i-th" section of the spindle is denoted as v_{hh}, and in transposed form it will look as follows:

$$v_{hh}^T = [y_{hh} \quad \varphi_{hh} \quad M_{hh} \quad T_{hh} \quad q_{hh}]$$

Addition of this vector and the vector of load parameters z_{hi} at point "h" provides the vector of input parameters for the new section:

$$v_{hi}^T = v_{hh}^T + z_{hi}$$

, where the vector z_{hi} takes the following form when transponed:

$$z_{hi}^T = [0 \quad 0 \quad M_h \quad -F_h \quad 0]$$

Finite Element Method – Computational Model in ANSYS Software
The method is a „fundamental" building block in simulation software. In this paper, we will use ANSYS software to simulate the structural design of the spindle. Software i sused to simulate the mechanical behavior of the system under various operating conditions, thereby significantly reducing the need for experimental verification in the early stages of development.

Numerical model is created by dividing a real geometric model into a finite number of elements (mesh). Within the mathematical model, material properties, boundary conditions and loads are defined, which represent real operating conditions of the spindle, such as cutting forces, moment, or bearing preload.

Testing Change of Cutting Conditions on a Simplified Spindle
As model example for performing an automated test, a spindle from the literature was chosen [1]. The diagram of this simplified spindle can be found on Figure 1.

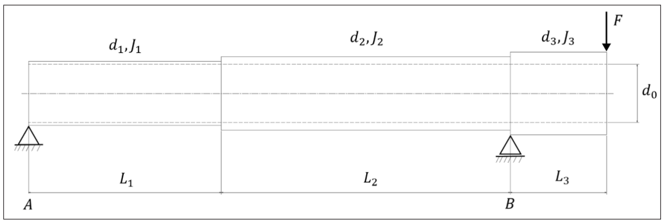

Figure 1: Simplified Spindle for Automated Test

The given spindle will be analyzed and verified using mulptiple analytical systems. The first analytical verification system is MATLAB software and the interpretation of initial parameter method through this computational software.

Table 1: Input Parameters of the Spindle from Figure 1

Material Properties	Diameters	Section Lengths
E=2,1 .10^5 MPa	d_0=6 .10^{-2} m	L_1=0,2 m
	d_3=6,50.10^{-2} m	L_2=0,3 m
	d_2=7,50 .10^{-2} m	L_3=0,1 m
	d_3=8,50 .10^{-2} m	

Determination of Cutting Conditions
During the designing process of the spindle, it's very important to accurately determine the technology for which it will be intended, as well as the nature of the load to which it will be exposed during operation. Acting load is in practice influenced by the selection of the material to be machined and the definition of cutting conditions for the chosen technologies.

In the presented paper, we will perform only a static analysis, which focuses exclusively on milling operations. Decisive factor in this case is determining the load resulting from these operations, defined through the cutting forces that arise when machining the chosen material under specific cutting conditions.

As the machined material was chosen material S235J1G3 according to EN 10025-2:2004. This material according to the classification of the tool manufacturer Sandvik Coromant belongs to the group P2.1.Z.AN [2].

Range of milling operations for which this spindle will be designed is detailed in Table 2. Table 2 also presents the abbreviations of operations and tools indicated for the machined material. Cutting conditions and the calculation of individual parameters were calculated using the ToolGuide software application from the tool manufacturer. For face milling was chosen the CoroMill 245 face mill (N1), and for contour milling CoroMill Plura (N2) from Sandvik Coromant. The cutting conditions for the mentioned tools and operations are described in more detail in Table 3 [3].

Table 2: Proposed Operations for Automated Simulation Test

Operation	Operation code	Tool
Face milling – roughing	OP1	N1
Face milling – finishing	OP2	N1
Contour milling – roughing	OP3	N2
Contour milling – finishing	OP4	N2

Table 3: Specified Tool Parameters using the Manufacturer's Tool Application

Parameter	OP1	OP2	OP3	OP4
Cutting torque (M_a)	92,9 N.m	4,01 N.m	18,8 N.m	1,13 N.m
Cutting force (F_c)	3260 N	159 N	3133 N	188 N
Axial force (F_a)	1793 N	88 N	685 N	41 N

Automated Testing of Changes in Cutting Conditions

The current paper arises as a continuation of paper in which a software tool for designing machine tool spindles was introduced. Within it, a sub-module was described that allows defining and changing its geometric parameters such as diameters, lengths, material and other aspects that need to be considered in the design of a machine tool spindle [4].

Major advantage of the solution we propose is the possibility of using external applications to perfrom certain tasks, such as generating a part through CATIA software as it was described in previous article. In the current article, simulation was integrated using ANSYS software. Both CAD software and ANSYS software can be automated through an API interface. User interface of the software tool through which we perform automated variant analysis is shown on Figure 2.

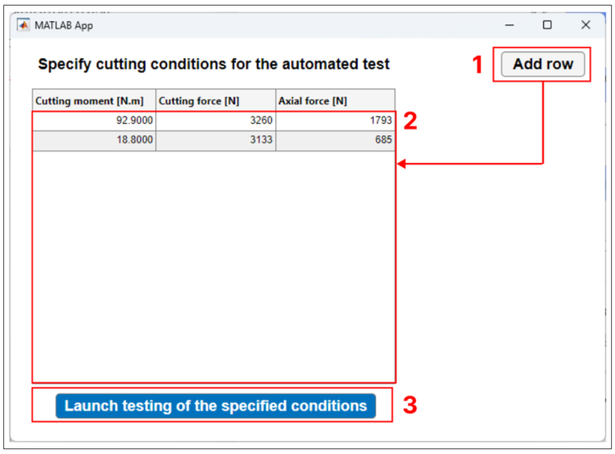

Figure 2: Software Tool used for Automated Mechanical Simulations

Main part of the application are shown on the user interface (Figure 2). On this figure by using the action button (1) We add test variants to the table (2) Test can be started by pressing the action button (3) Which is displayed uder table with the test variants (2).

When performing automated testing, it is first necessary to select the cutting force parameters in the user interface shown on Figure 2. Currently, this testing does not reflect the actual behavior of the machine tool – fixture – workpiece system, as the test assumes a constant value for the forces acring on the front part of the spindle. However, our goal was to demostrate and perform an automated test on one design variant of the spindle head shown in Figure 1 without manual intervention, with the results exported from the simulation. Figure 3 illustrates diagram that is describing the background processes in more detail after pressing button "3".

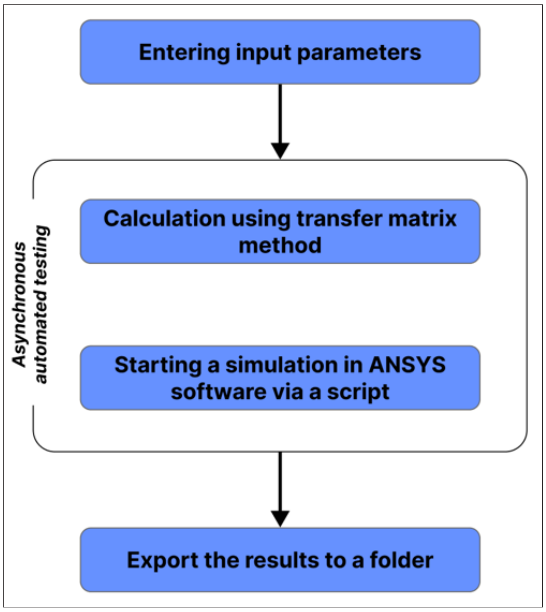

Figure 3: Diagram of Automated Simulations Shown on Figure 2

Diagram (Figure 3) illustrates the automated testing running in the background. When button (3) is pressed, the boundary conditions specified by the user in the application interface shown on Figure 2 are assigned to both computational models. These are used as input parameters for processing within the programmed functions in MATLAB, which integrate the computational methods from the chapter 2 of this article.

Results of the tests can be monitored in real time in the computer's memory. When the button (3 on Figure 2), folder named „TEST_" is created in the computer's memory with the date and time at the moment when test started. Results of the automated tests are shown on Figure 4

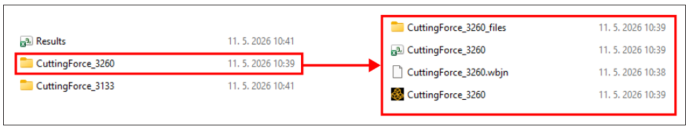

Figure 4: Output of the Automated Test

Results are divided into two categories. Results from the ANSYS analysis are saved in a separate folder named „CuttingForce_ VALUE", where „VALUE" represents the force value entered in the „Cutting Force [N]" column in Figure 2. Subsequently, the output file from the analysis in *.csv format and the ANSYS analysis project are saved to this folder, which can then be used after the analysis is complete.

After successful completion, all results are automatically extracted and provided to the designer for evaulation in the form of an output file named „Results" in the main folder of the oberall test. Subsequently, we evaluated the values from the output file, and the results are shown in Table 4.

Table 4: Comparison of Total Deflection Using the Transfer Matrix Method (MATLAB) and Analysis Using ANSYS Software

Testing Case	ANSYS [m]	TRANSFER MATRIX METHOD [m]	Comparision of results TMM – ANSYS [%]
OP1	$6{,}74 \cdot 10^{-05}$	$7{,}19 \cdot 10^{-05}$	6,29
OP2	$3{,}29 \cdot 10^{-06}$	$3{,}46 \cdot 10^{-06}$	4,88
OP3	$5{,}94 \cdot 10^{-05}$	$6{,}25 \cdot 10^{-05}$	4,87
OP4	$3{,}57 \cdot 10^{-06}$	$3{,}75 \cdot 10^{-06}$	4,89

As we can from the results obtained after succsessfully running the automated test that the differences between the two methods ranged from 4,87% to 6,29%, results are shown on Figure 5.

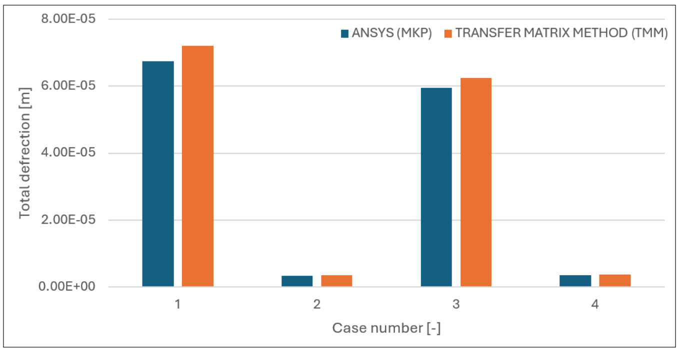

Figure 5: Graphical Comparison of the Results from Both Methods

Conclusion and Future Research Posibilities

My article aimed to present the current state of the development of design variants for machine tool spindles, while also highlighting the possibility of automating the design process in line with current industry trends. Presented software under development is a continuation of and supplements it with mathematical modules for calculating the total deflection of spindles [4].

This article presented changes in cutting conditions during machining; our process is modular, and we plan to expand the mathematical model to include dynamic loads on the spindle, which is also a focus of my research. We plan to measure loads during spindle operation and we will use the measured data as input for an automated test. Subsequently, we will compare the deflection and natural frequency values obtained from the automated test with the values measured using laboratory equipment under laboratory conditions.

Acknowledgements

This article was published thanks to the support of the Agency for Science and Research for the project APVV-23-0619 “Research of progressive technology of decomposition of glued layered materials”.

References

1. (2025) Vadovic Fridrich Selected States of Elasticity - Stiffness and strength of production machines. 1st edition Undated 69-78.
2. (2026) Machinable Materials. Sandvik Coromant https://www.sandvik.coromant.com/cs-cz/knowledge/materials/workpiece-materials.
3. (2026) CoroMill® 245. Sandvik Coromant https://www.sandvik.coromant.com/cs-cz/tools/milling-tools/face-milling-tools/coromill-245.
4. Hostak Peter, Lubomir Soos (2025) Adaptive Design of Spindle Machine Tool. Journal of Mechanical Engineering 75: 35-42.