

ANALYSIS OF DMU40 MACHINE CENTRE BY CAE SOFTWARE

RÓBERT KISS – ATTILA SZILÁGYI

University of Miskolc, Department of Machine Tools
3515 Miskolc-Egyetemváros
robert.kiss9405@gmail.com, szilagyi.attila@uni-miskolc.hu

Abstract: This article is part of a longer research-analytical work, because it outlines the results and conclusions of the study according to the main topic of the research and the applied method. The central theme of the research is the dynamic stiffness of machine tools and the various methods for their determination. After analytical testing, we will deal with another method, because we also use CAE software (ANSYS Workbench R19.1) to perform previous tests in the form of simulations. The results obtained are compared with the values previously determined analytically.

Keywords: ANSYS R19.1, CAE, dynamic stiffness, modal analysis, natural frequency, FEM

1. INTRODUCTION

One of the most commonly used finite element method (FEM) for mechatronic problems is a structural analysis [1] [3]. In mechanical engineering we mean the examination of mechanical elements by the structural analysis under, which can be done e.g. a specific part (shaft) or subassembly (machine base of a lathe or a milling machine) of a machine, or even the machine itself (machine centre) (*Figure 1*). The main purpose of the calculations is to determine the linear displacement or angular of rotation, from which additional quantities may be derived, such as elongation, stress or reaction force.



Figure 1
Catalog image of the examined machine tool [2]

The CAE (Computer Aided Engineering) software, what we used during the simulations, is the ANSYS Workbench R19.1, which is generally an extremely complex engineering software package, that can be widely used in the industry due to its large number of simulation capabilities (*Figure 2*).

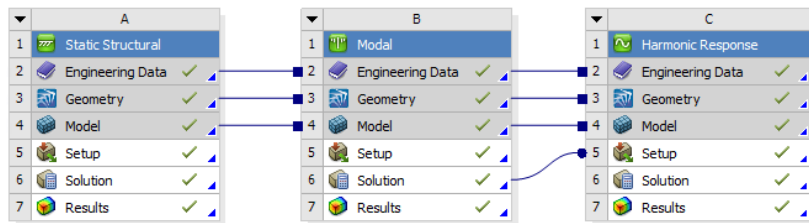


Figure 2

Types of simulations applied to the machine tool 3D model

Briefly the main types that can be examined using finite element software are:

- **Static analysis:** Method for the calculation of permanent deformation or stress due to time independent load. It does not take into account the elements causing possible damping and inertia, that is time-dependent loads.
- **Modal analysis:** Suitable for the determination of the vibration characteristics of the structure under simulation, as well as the individual natural frequencies and the representation of their oscillations. It can serve as a baseline for more detailed dynamic investigations (e.g. transient dynamic, harmonic or spectral analysis).
- **Harmonic response analysis:** Used to determine a steady-state response to a cyclically changing excitation over time. The test ignores transient phenomena occurring at the beginning of the load. It makes it possible to predict the long-term dynamic behaviour of the structure under investigation, to examine the phenomena of fatigue and resonance.

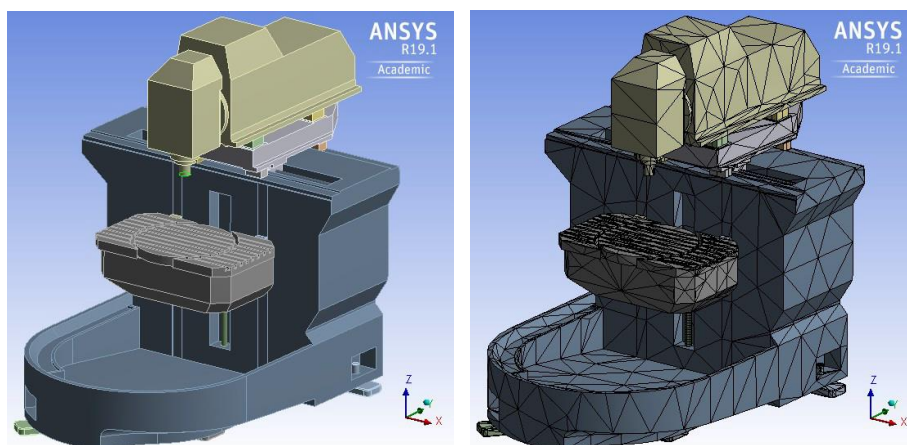


Figure 3

The studied 3D model after import and meshing

The (simplified) 3D model of the machine tool is essential for the test, what we made with a 3D design software, which called NX11, taking into account catalog data and a machine centre, which can be found in the department's workshop. *Figure 3* shows the 3D model of the machine, which is already in parasolid format, appears as an imported model in the ANSYS model space and in parallel with its post-meshed state.

2. MODAL ANALYSIS

Any physical system can vibrate or can do vibrating motion. In this case the natural frequencies of the possible free vibrations and the various vibrational forms (modes) are considered to be the properties of the system.

In an analytical case, after constructing the mechanical model of the system, solving the mathematical model (the differential equation describing the motion or the system of differential equations), the system's natural frequencies can be derived. Moreover, its associated eigenvalues, which is now executed by a finite element software based on the available information (e.g. 3D geometry, constraints, boundary conditions, mesh properties, element size, damping factors, material quality...).

During the analysis the built geometry of the model is completely broken into elements of finite size. Constructs stiffness, mass and damping matrices for the structure as a whole from matrices describing the characteristics of these elements.

Using the matrix method for dynamic systems leads to the following differential equation system [4] [6]:

$$[M]\{\ddot{x}\} + [D]\{\dot{x}\} + [K]\{x\} = \{f\}, \quad (1)$$

where $\{f\}$ is the forcing term, $[M]$, $[D]$, $[K]$ are the matrices of masses, dissipation and stiffness, while $\{\ddot{x}\}$, $\{\dot{x}\}$, $\{x\}$ are the coordinates of acceleration, velocity and displacement in that order.

Main steps of the simulation:

1. Import the 3D geometry of the machine centre into ANSYS.
2. Selecting the types of simulations, establishing the relations between the tests, as shown in *Figure 2*.
3. Check the relations and connections, what the ANSYS automatically created before, and if it is necessary correct that. After, define and assign material quality to each component.
4. Define meshing settings, methods and then meshing the model (*Figure 3*).
5. Specifying boundary conditions and constraints on the model.
6. Specify simulation settings: number of modes to look for, method selection, damping consideration, damping factors, and other settings.
7. Run the simulation and evaluate the results.

Following the above-mentioned steps, we have obtained the following results, which are the roots of (1) at the same time (*Table 1*):

Table 1
The natural frequencies computed numerically

mode	1	2	3	4	5	6	7	8
f [Hz]	108.16	132.98	159.49	229.01	325.65	379.22	538.34	663.45

After running the simulation, in addition to determining the values of the natural frequencies the software capable for displaying the degree of deformation of the individual natural frequencies in the simulation assembly as well. That is, the program displays the vibration images for each mode. After specifying the damping factors for each material type and, in the modal analysis settings with the damping option selected it is possible to determine the natural frequencies in the damped case. However, no significant change can be observed in the results obtained, since the effect of damping is primarily manifested in a decrease in the amplitude [5]. The software can display the oscillations shown in *Figure 4* below at some of the machine's natural frequency locations, that is for each mode:

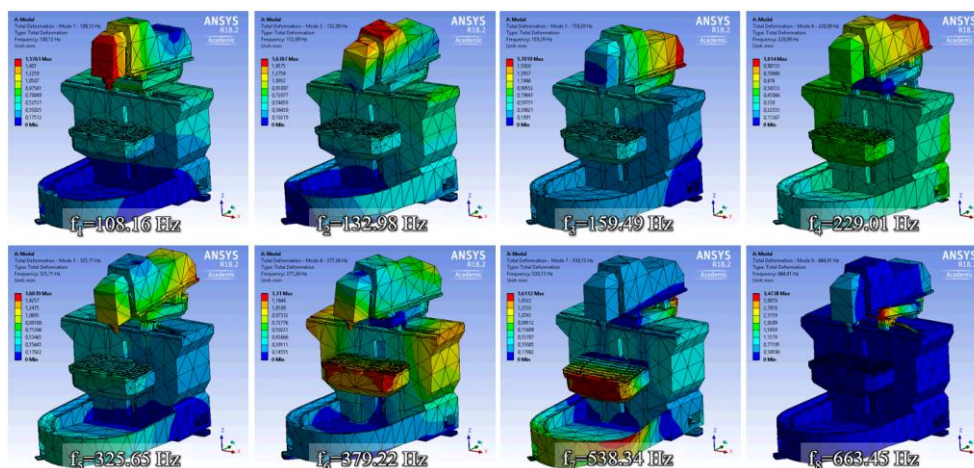


Figure 4
Results of modal analysis – vibration images

3. SUMMARY

In this article, we have presented a possible method for determining the dynamic stiffness (natural frequencies) of a machine centre. Testing with CAE (Computer Aided Engineering) software enables fast and accurate computation, if we have a right 3D model and the material quality of each element which information necessary to run the simulation. In terms of results, the previously analytically calculated natural frequency values approximate the software-calculated data. In addition, the vibration images show that the dynamic stiffness of the machine in the z-direction is the weakest.

ACKNOWLEDGEMENT

The described article/presentation/study was carried out as part of the EFOP-3.6.1-16-2016-00011 *Younger and Renewing University – Innovative Knowledge City – institutional development of the University of Miskolc aiming at intelligent specialisation* project implemented in the framework of the Szechenyi 2020 program. The realization of this project is supported by the European Union, co-financed by the European Social Fund.

REFERENCES

- [1] Kiss, R., Szilágyi, A. (2019). Analysis of the dynamic behaviour of the CNC machine centre by FEM. *DMS Journal*, Miskolc, Vol. 9, No. 1, pp. 24–28.
- [2] *DMU monoBLOCK Series catalogue*. <https://www.dmg.com>, downloaded: 2018. 01. 26.
- [3] Kiss R. (2019). *CNC megmunkáló központ dinamikai viselkedésének vizsgálata végelelem-módszerrel*. Diplomaterv, Miskolc.
- [4] Csernák G., Stépán G. (2012). *A műszaki rezgésstan alapjai*. Egyetemi jegyzet, Budapest, BME.
- [5] Dömötör F. (2008). *Rezgésdiagnosztika I*. Dunaújváros, Dunaújvárosi Főiskola.
- [6] Pascal, M. (2012). *Parallelization of Design and Simulation: Virtual Machine Tools in Real Product Development*. Doctoral Thesis, ETH Zürich.