Multibody system simulation as a predictive tool for possible estimation of negative effects caused by vibrations of FDM device

J Kaščak, M Kočiško, L Knapčíková, T Coranič, J Török

Technical University of Košice, Faculty of Manufacturing Technology with a seat in Prešov, Slovakia

E-mail: jakub.kascak@tuke.sk

Abstract. The design of fused deposition modeling (FDM) devices in their current form is associated with many negative effects, which result mainly from their construction deficiency. Due to the constant effort to increase the accuracy and speed of these devices, we often encounter the emergence of various negative factors. One of the most significant factors is the presence of negative vibrations of the frame and individual components. These are directly linked to several shortcomings that FDM devices come with. Efforts to make the structures as simple as possible, their low weight or the use of filament extruders placed directly on the printheads and axis travels, are perceived as well-known shortcomings. These negative phenomena are subsequently manifested by the emergence of specific defects visible on the surface of the manufactured models. The article presents the possibility of predicting the occurrence of these negative phenomena, with the use of multibody simulation. This simulation analyzes the movements of a specific device at different print speeds. The article then presents the results of these simulations and analysis of the transmission of negative oscillations at specific critical points of the FDM device. Finally, the article examines the influence of possible regulation of devices acceleration rates caused by electrical motors in individual axes and their influence on the final surface quality of the manufactured model. The article points out the possibilities of using this type of simulation processes and analysis in the design process of new types of frames and translation mechanisms for FDM devices.

Keywords. Fused deposition modeling, motion analysis, additive manufacturing, print quality

1. Introduction

Finite element analysis is currently an essential tool for simulating the behavior of virtual models in specific situations. This article introduces the application of its advanced method. This allows us to reduce the analysis of external influences acting on the model to a minimum. The so-called motion analysis enables the analysis of complex assemblies. These are dedicated as functional sets of models that allow the integration of actuators, stepper motors, springs, bearings of connecting elements and the creation of various types of contacts. These processes result in a simulation of the design in individual operations, based on relatively real operating parameters. The article presents a possible application of this method presented on the FDM device Ender 3. In specific cases e.g. in the continuous printing of thin-walled models or non-planar printing, it actively involves the Z-axis in the process. The active
involvement of the third axis causes various types of negative phenomena. These cause defects that reduce the surface quality of the model, its geometry and dimensions.

The aim of the article is to create a digital twin of FDM device. Subsequent analysis of the structure as a whole, will present individual unwanted vibrations that cause oscillations of the print bed arising during operation. The construction must not contain any form of intersections and other shortcomings.

The article presents the possibility to analyse the structure as a whole and read the results characteristic of the whole construction, or its individual subsystems. This process is currently widely used, especially in combination with topological optimization. Model topology optimization is widely used, for example, to minimize the amount of material used and the deformation energy of structures while maintaining their mechanical properties. It is a mathematical method that spatially optimizes the distribution of material in a defined structure and taking into account predefined limiting parameters and minimizing costs. The main elements in the optimization are design variables, force load and constraint parameters. [1]

Topological optimization is divided into two parts.

- The first is LO (Layout Optimization) optimization of the layout, focusing mainly on small-volume structures with rod, frame or grate design.
- The second is GSO (Generalized Shape Optimization) generalized shape optimization, focusing mainly on high-volume structures. In this section, the shape and topology are proposed. The FEM finite element method is used to solve these constructions. [5]
- Isotropic Solid or Empty element (ISE)
- Anisotropic Solid or Empty element (ASE)
- Isotropic Solid, Empty or Porous element (ISEP). Rigid, resp. a solid element is one that is filled with a single material.
- An empty element is one that is not filled with any material.
- A porous element is one that is filled with a single material that contains void cavities.

The method using isotropic solid or empty elements (ISE-topology) is most often used for shape optimization. ISE topology uses three basic computational strategies:

- SIMP - (Solid Isotropic Microstructure with Penalization) solid isotropic microstructures with penalties
- OMP - (Optimal Microstructure with Penalization)
- NOM - (NonOptimal Microstructures) suboptimal microstructures. [1]

SIMP, or solid isotropic microstructures with a penalty, is a method that optimizes the distribution of material in an optimized object. The material is distributed so that the volume is minimized resp. maximized rigidity that reflects specific loads, performance requirements, limitations, and manufacturing limitations. [2] Usually, topology optimization is solved by discretizing the domain into a network of finite elements (isotropic solid microstructures). Subsequently, each element is either filled with material (in areas where material is needed) or is stripped of material (in areas where removal is possible, i.e. where there will be cavities). Within the proposed domain \( p \), the density distribution of the material is discrete. Each element has an assigned binary value:

- \( p = 1 \) where material is added (black),
- \( p = 0 \) where material is removed (white).
The relative density assigned to each element can range from the minimum value of the element density $P_{min}$ up to 1. $P_{min}$ is the minimum permissible value of the relative density for empty elements, which has a value greater than 0. This makes it possible to assign the correct binary value for elements whose binary density value could not be determined with obviousness. Elements between $P_{min}$ and 0 have an assigned binary value of 0. [6]

2. Materials and methods

When creating any technical structure, it is necessary to use among many other preparations, also a series of mathematical calculations based on functionality of the geometry that is designed. Durability, efficiency but also the selection of a suitable material all these properties are necessary. Previously analytical methods were mainly used for calculations, when the technical construction of a physical model, element or system was simplified so that the simplest possible mathematical equations could be used for the calculation. [4] Gradual development of computer technology currently allows the creation of mathematical as well as physical models of solved technical structures, which are very similar to real structures in terms of material composition, geometry but also function. The algorithm of these computer-oriented numerical calculation methods is adapted for computer software. Using these calculations, it is possible to analyse in a virtual environment complex linear, nonlinear but also stationary and non-stationary reactions of the technical equipment, which is proposed to normal operating conditions but also to critical loads. [7], [8]

The calculations are performed using various numerical methods. Numerical methods reform the structure of partial as well as common differential equations and rewrite the given example into a system of algebraic equations, which is solved by a computer. [10] The main idea of several numerical methods solving differential equations (least squares method, collocation, variational, differential, finite element method) will be a suitable substitute function for reference coordinates, which consist of free coefficients and a given type of function of the algebraic system of equations. It is possible to determine free parameters for the substitute function, and therefore it is possible to determine the overall shape of the function.

Numerical methods differ individually by the physical nature of the free coefficients and the method of their determination, as well as by the choice of the substitute function. In FEM, unlike other
numerical methods, substitute functions are not chosen for the whole investigated area, but it is divided (discretized) into sub-areas - finite elements. Finite elements usually form the shape of finite dimensions (line, area, volume). The accuracy of the solution increases with increasing fineness of the finite element network. [11, 12]

FEM based simulations are incorporated in virtually every branch of technical practice. At present, the software environment using FEM is commonly accessible and it is relatively easy to simulate real-life problems (e.g. analysis of construction and machinery, seismic analysis of buildings, impact analysis of vehicles, flow of various fluids, etc.) [12]

![Figure 3 Demonstration of the component of the FDM device analysed as a subsystem during simulated operation [13]](image)

2.1. Methodology

Due to the differences caused in the analysis of the structure as whole, and not only its individual parts, it is necessary to perform a series of steps that are necessary for this process. [14] In the following points, we define the individual sub-objectives that will allow us to achieve the desired results. Chronological sequence of steps required for the analysis of a complex structure are:

- creation of an assembly model with appropriate constraints,
- import of the assembly into specialized software,
- definition of contacts, pins, screw connections, springs and bearings,
- definition of base and rigid groups,
- addition of drives, setting of starting speeds,
- final analysis and interpretation of results.

2.2. Hardware

In order to apply and verify the theory, the Ender 3 device was chosen. The advantage of this FDM device is its availability and relatively undemanding design. This allows us to create an almost identical virtual model. [15] As can be seen in Figure 4., it is then imported into the simulation software.
The Ender 3 Pro FDM device from the Creality3D company is capable of printing in a space of 220 x 220 x 250 mm. It is sold as a kit that needs to be assembled into a functional set with an innovative and compact design. Thanks to its compactness, quiet and smooth operation, the Ender 3 Pro can also be placed on a desk. It is an easy-to-use 3D printing device that works with G-code, which can be imported directly from a computer via USB 2.0 or by uploading a file to an SD card. [16]

2.3. Software

Due to the number of components in the assembly and the need for its components to interact, Inspire software was chosen for this application. As we can see in Figure 5, the software uses the already mentioned SIMP method in its work. The example shows the analysis and optimization of the shape of the door support of the Fairchild Dornier 728-10 aircraft. [17] The analysed model was later optimized with the intention of reducing the weight in order to maintain the rigidity of the model. Due to the optimization, the weight of the model was reduced from 9.16 to 7.5 kg, which represented a 20% weight saving. As for the model design process itself, it was reduced by 3 weeks as a result. [18]

2.4. Field of application

As already mentioned, the subject of a possible application is FDM technology. Due to its constant development and demand, especially in the field of research of new materials and the way of their layering, we often encounter innovative designs of construction solutions. These often require the efforts of entire design teams, which can detect and resolve many shortcomings through a lengthy process. This article interprets the possibility of contributing to the identification of these deficiencies during the design phase.

The subject of the study is to reveal the possible negative influence of vibrations of the supporting frame of the structure. This can be caused by the active involvement of the Z-axis movement.
in specific cases, e.g. non-planar printing, or printing of thin-walled models. For this purpose, a digital assembly of the Ender 3 device was created, which can be seen in Figure 4. Due to the fact, that the analysis of entire assemblies requires the correct performance of the virtual model, the software requires the definition of contact surfaces and connecting elements, as can be seen in Figure 6.

![Figure 6 Demonstrating of adding screws, pins and defining contacts](image)

After defining all contacts, screws, pins, etc., the formation of rigid parts of the assembly respectively its subsystems and the definition of “ground”. As we can see in Figure 7. (a), the ground of the machine is its base. We further divided the assembly into 3 smaller subsystems (b, c, d).

![Figure 7 Definition of ground and rigid groups of equipment](image)

The next step is to define other elements such as springs and bearings, the software has a generator that allows their precise placement and definition. Figure 8 shows the location of the springs used to define the position of the print bed. It is this design solution that is one of those that significantly contributes to the spread of negative vibrations of the print bed. This has, as already mentioned, a negative effect on the quality of the prints.

![Figure 8 Definition and embedding of springs](image)

Figure 9 represents the most important part of the definition needed for a successful simulation. The creation of contacts, respectively, components of the assembly that are capable of interacting or colliding with each other. As can be seen in Figure 9. From the right side, the interactive
elements are the lead screw, the lead nut and supporting part of the subsystem, which is shown in Figure 7c. The second part is the location of the stepper motor. The software has an extensive configurator for stepper, electrical motors and actuators, allowing us to choose the right type. The software also has some sophisticated options for their adjustment. In our case, the stepper motor was placed on the lower part of the flexible coupling, which forms an interactive part of the assembly. Subsequent adjustment of the settings defines the maximum engine speed during the movement of the Z axis (set to 35 rpm) and the acceleration to the maximum speed. There is a possibility of intervention in the acceleration process. Time for which the stepper motor reaches the maximum value, or decelerates from the maximum speed until complete cessation can be adjusted via settings. This allows to create conditions for simulating the effect of acceleration and deacceleration of the engine and the emergence of negative phenomena. Due to the fact that it is possible to perform this changes on the FDM device, it is possible to verify the results in a real environment.

The process that requires the definition of all common assemblies and individual subsystems is, of course, extensive. The examples used in the article point to some of its supporting parts. Of course, when creating a digital twin and all the necessary definitions, few problems occurred. The majority representation was the mutual intersection of parts, which made it practically impossible for the simulation to take place. All these shortcomings had to be eliminated before dividing the assembly into individual subsystems.

2.5. Results and discussion

Despite the fact that the definition of a moving assembly is laborious and the conditions for the geometric correctness of the assembly model are relatively demanding, this method of definition and subsequent analysis has its advantages. The main positive factor is that, unlike the FEM method, the analysis of forces, loads and other influences on a particular component is not necessary. All these definitions result from the mutual interaction of the assembly subsystems and the simulation of operation according to the load of the drives in the assembly. The software also allows a thorough display of the resulting loads on the assembly itself.

Figure 10, due to better readability, shows us a simplified version of results and allows us to view the negative effects arising from the movement of the Z axis according to the specified settings. As we can see, due to the negative vibrations, the individual areas of the print bed are significantly shifted. This negative phenomenon was assumed due to the design of the print bed mounting. Figure 10b, shows the reduction of these displacements due to the change in acceleration and deacceleration, i.e. the already mentioned "jerk settings" available to FDM devices.
Figure 10 Interpretation of the displacement of locations on the printing plate (a) 35 rpm, jerk value 10, b) 35 rpm, jerk value 5

The results of the simulation point to the real possibility of using this type of software just to create a behavior preconditions of a specific design solutions. The next step in the verification of these data, resp. another subject of research will be the verification of results by performing vibrodiagnostic measurements. Their subsequent comparison and determination of the degree of correlation of simulated data and data from measurements will allow verification of the obtained data and the accuracy of the simulation.

3. Conclusion

FEM analysis as such, is a supporting part of every designers work. However, we are currently witnessing the emergence of new types of user-friendly specialized software that allow alternative access to this type of technology. The article presents the application and possibilities of motion analysis of a complex set of FDM devices. It specifically applies theoretical knowledge from the areas of FEM analysis and topological optimization to a specific case of an FDM device called Ender 3, whose virtual model is part of the simulation. The virtual model itself, its properties and the way of defining individual subsystems, transformation mechanisms and components subsequently enable its participation in a comprehensive analysis. The expected result of the simulation of the negative effects arising from the vibration of supporting structure, which results from the predefined movement of the Z-axis, indicates the real possibility of using this kind of software to predict these phenomena during the design phase.

Given the area and equipment on which the analysis was performed, we can predict further steps related to the verification of results. In the next phase, it is necessary to compare the achieved results with a real vibrodiagnostic measurement. Such verification of results will allow us to define the degree of correlation of the results simulated and obtained by measurement. Thus, the expected result will be the prediction of the negative consequences of the design solutions of the FDM device subsystems already during the design phase, which can significantly contribute to the final quality of new or alternative constructions and rapidly increase the quality of produced prints.

Acknowledgements
This work was supported by the projects KEGA 004TUKE-4/2020, VEGA 1/0051/20, granted by the Ministry of Education, Science, Research and Sport of the Slovak Republic.

References
[1] Joga Ch S, Haber R B, 1996 Stability of finite element models for distributed-parameter optimization and topology design, Computer Methods in Applied Mechanics and Engineering, Vol. 130, I 3–4, P. 203-226
[2] Araújo A F, Varela M L R, Gomes M S, Barreto R C C, Trojanowska J, 2018 Development of an intelligent and automated system for lean industrial production, adding, maximum productivity and efficiency in, the, production process, Lecture Notes in Mechanical Engineering, (201519), pp. 131–
[3] Pavlenko I, Saga M, Kuric I, Kotliar A, Bassova Y, Trojanowska J, Ivanov V, 2020 Parameter identification of cutting forces in crankshaft grinding using artificial neural networks *Materials*, 13(23), pp. 1–12, 5357

[4] Coranic T, Gaspar S, Pasko J 2021 Utilization of Optimization of Internal Topology in Manufacturing of Injection Moulds by the DMLS Technology, *Applied Sciences* 11/1, pp. 1-13

[5] Židek K, Pitel J, Pavlenko I, Lazorík P, Hošovsky A 2020 Digital twin of experimental workplace for quality control with cloud platform support *EAI/Springer Innovations in Communication and Computing*, pp. 135–145

[6] Benča Š, 2006 Výpočtové postupy MKP, *Bratislava: STU Bratislava* pp.150

[7] Židek K, Lazorík P, Pitel’ J, Hošovský A, 2019 An automated training of deep learning networks by 3D virtual models for object recognition *Symmetry*, 11(4), 496

[8] Kaščák J, Gašpár Š, Paško J, Husář J, Knapčíková L 2021 Polylactic acid and its cellulose based composite as a significant tool for the production of optimized models modified for additive manufacturing *Sustainability* 13(3),1256, pp. 1-20

[9] Szczepaniak M, Trojanowska J 2020 Preventive maintenance system in a company from the printing industry *Lecture Notes in Mechanical Engineering* pp. 351-358

[10] Benča Š, 2009 Riešenie nelineárnych pevnostných úloh pomocou MKP. *Bratislava: STU Bratislava*, pp. 205

[11] Murín J, Hrabovský J, Kutiš V, 2014 Metóda konečných prvkov: Vybrané kapitoly pre mechatronikov. *Bratislava: STU Bratislava*, pp. 142

[12] Kaščák J, Gašpár Š, Paško J, Knapčíková L, Husář J, Baron P, Török J. 2021 Design of an Atypical Construction of Equipment for Additive Manufacturing with a Conceptual Solution of a Printhead Intended for the Use of Recycled Plastic Materials. *Applied Sciences*, 11.7: 2928

[13] Židek K, Pitel’ J, Adámek M, Lazorík P, Hošovský A 2020 Digital twin of experimental smart manufacturing assembly system for industry 4.0 concept *Sustainability* 12(9),3658

[14] Ivanov V, Pavlenko I, Liaposhchenko O, Gusak O, Pavlenko V. 2021 Determination of contact points between workpiece and fixture elements as a tool for augmented reality in fixture design *Wireless Networks*, 27(3), pp. 1657–1664

[15] Knapčíková L, Balog M, Ruggiero A, Husář J, 2019 Composites Manufacturing: A New Approaches to Simulation, *EAI/Springer Innovations in Communication and Computing*, 2019, pp. 319–325

[16] Guo X, Cheng G-D 2010 Recent development in structural design and optimization. *Acta Mechanica Sinica*, 26.6, pp. 807-823

[17] Židek K, Maxim V, Pitel J, Hosovsky A, 2016 Embedded vision equipment of industrial robot for inline detection of product errors by clustering-classification algorithms. *Int. J. Adv. Robot. Syst.* 13, pp.1–10

[18] Michalík P, Hatala M, Straka L, Petrus M, Macej J, Jusko J, Tirpak P 2020 Optimizing Component Production with Multiaxis Turning Technology, *4th EAI International Conference on Management of Manufacturing Systems*, pp. 273–283