How to cite this article:
Authors: Wit Grzesik
Title of article: „Optimisation methods for programming CNC machine tools"
Mechanik, No. 8–9 (2024)
DOI: https://doi.org/10.17814/mechanik.2024.8-9.16

Optimisation methods for programming CNC machine tools

ADAM ZALEWSKI
WIT GRZESIK
KRZYSZTOF JAROSZ *
https://doi.org/10.17814/mechanik.2024.8-9.16

* Dr inż. Adam Zalewski – adam.zalewski@pw.edu.pl, https://orcid.org/0000-0002-8698-5503 – Warsaw University of Technology, Faculty of Mechanical Engineering [Politechnika Warszawska], Warsaw, Poland
Prof. dr hab. inż. Wit Grzesik – wit.grzesik@gmail.com, https://orcid.org/0000-0003-3898-5119 – Opole, Poland
PhD Candidate Krzysztof Jarosz – Rochester Institute of Technology, USA

This paper presents some important methods of optimisation of CNC programming which covers algorithms of feed changes along the path of tool movement, the concept of feature-based recognition machining (FBM), numerical simulation (FEM/FEA), advanced virtual modelling of machining systems, real-time process simulations with the use of digital twins (DTs) and intelligent control strategies based on the STEP-NC standards. Selected examples of practical industrial applications are overviewed and future development trends are discussed.
KEYWORDS: CNC machine tool, control program, feed optimization, numerical simulation, virtual simulation, STEP-NC interface

Introduction

The key to understanding the essence of CNC machine tool programming optimization tasks is the analysis of research, development and implementation work on intelligent machine tools (IMT) conducted in the last decade. Their main task is to achieve an advanced level of virtualization of modeling and programming and related processing and analysis of information used to make decisions in the scope of optimization activities of the machining process and monitoring. This first optimization task therefore concerns strictly the programming of machining processes carried out on CNC machines with various degrees of intelligence.

From fig. 1, presenting the future structure of an intelligent machine tool, which was developed by Japanese researchers about 20 years ago, it is clearly visible that the most developed, not only in the sense of selecting the optimal motion trajectory, but also optimal technological conditions, which in turn determine the efficiency and quality of the machining process, is tool movement control. At the highest level, adaptive control (AC) and supervision necessary for intelligent monitoring of the machining process are located, the task of which is to detect the current state of machining regardless of the cutting conditions and the type of machining operation performed. For this purpose, it is necessary to develop effective feedback in the flow of information regarding the obtained process results (also based on measurements of the machined part) and their evaluation.

An additional condition of IMT functionality is the implementation of the autonomous process planning strategy as one of the functions, which guarantees the generation of a flexible and adaptive machining plan/program (technological route). This task can be realized by introducing several functions in the scope of process planning and information processing, including operational planning, selection of cutting tools, determination of cutting parameters and generation of the tool motion path for each machining operation. In the context of information processing, the results of product data analysis and recognition of cutting features are taken into account, which are the subject of the FBM (feature-based machining) concept [2].

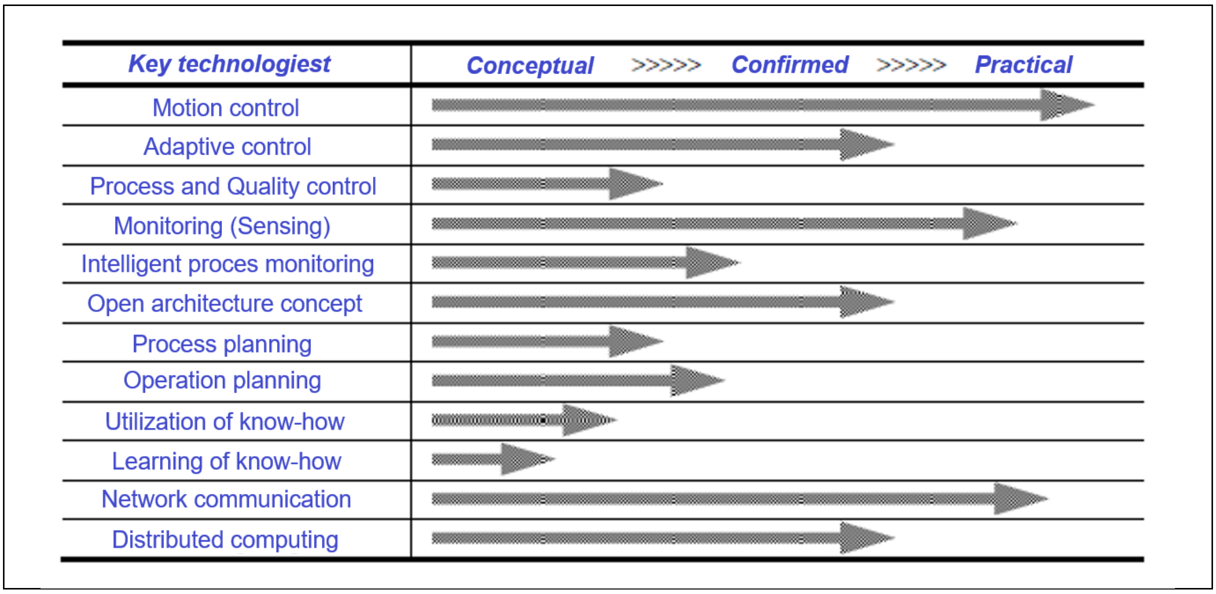

Fig. 1. Key techniques influencing the development of IMTs [1–3]

The key techniques are summarized in the diagram in fig. 1 and discussed in [3].

The concept of "machine tool 4.0" uses supporting information technologies such as: cyber-physical systems and the Internet of Things (IoT) [3, 7, 14], which increase the ability to collect information and use it in new process conditions and machining operations.

Advanced, intelligent machine tools use two-stage information processing, including close cooperation of edge computing and virtual cloud computing [4]. This creates a new IMT architecture, designated with the acronym IMT-ECC (edge-cloud collaboration), as shown schematically in fig. 2. The entire information system in the IMT-ECC machine consists of three layers – data acquisition, network communication and the ECC module.

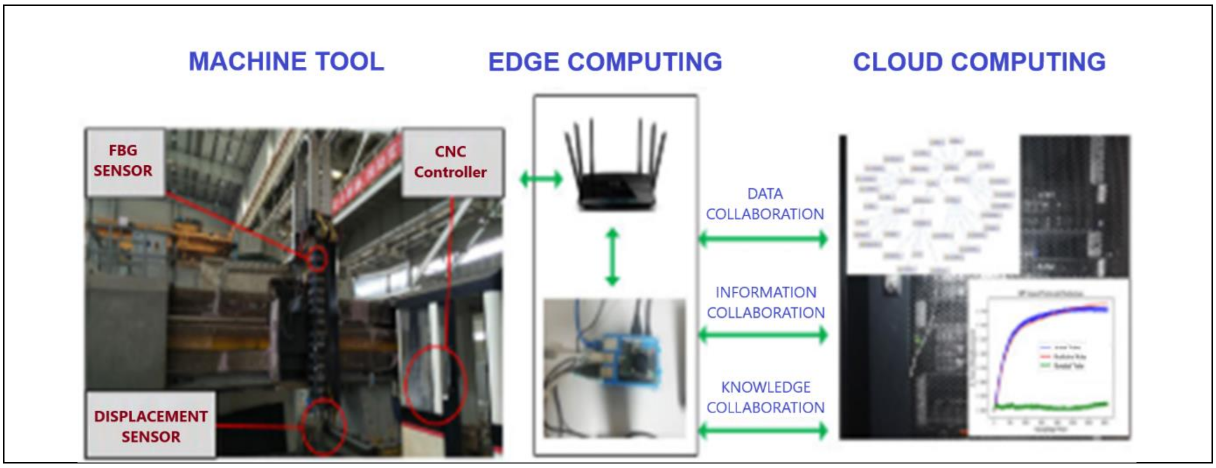

Fig. 2. Schematic representation of information collaboration in real IMT-ECC system [2, 8]. FBG – fiber optical sensor (fiber bragg grating)

An example of practical implementation of information cooperation in a CNC machine tool is shown in fig. 2. Data on the machine tool condition is obtained from numerous built-in sensors, including displacement, limit switches, current and voltage sensors. Data collected by these sensors is obtained from the CNC machine tool controller. Additional sensors are installed, including temperature, AE, piezoelectric accelerometers, which allow for expanding information on the machine tool condition based on data processing. In turn, edge processing platforms (close to the machine tool) and further cloud platforms are installed for this purpose. At the same time, the data is pre-processed in the edge computing platform of the EdgeX type Foundry IoT and then transmitted to the cloud computing platform. The collected data is displayed using the HMI interface.

It should be emphasized that in the intelligent machine tool with the IMT-ECC architecture, the collection and processing of data in the cloud is intended to optimize the algorithms of virtual models [3], while the edge processing performs the function of fusion, preprocessing of source data and analysis of transmitted data on the machine state as a response in a real-time feedback loop. In this way, the intelligence of the machine tool, and therefore also the efficiency of programming, is increased as a result of the cooperation of ECC systems at the level of data, information and knowledge.

Principles of feed optimization in CAD/CAM programs

A CNC machine tool operates based on a CNC program. The process parameters described in the CNC program (such as feed, spindle speed) can be changed based on feedback signals during machine operation (e.g. adaptive control) or adapted to the expected machining conditions based on process simulation in the CAD/CAM program and saved to the CNC program.

Technological processes performed on CNC machines include various stages of processing, which means that in the creation of CAD/CAM software it may be necessary to modify the parameters of specific operations, change their sequence and other aspects. In the Mastercam program, such a role is played by a tool called *the Operation Manager*, which allows, among others, editing, adding, deleting, copying processing stages [2]. An important role is played by the associativity of CAD/CAM software, which allows the linking of geometry with tool movement paths in such a way that in the case of a change in the tool path geometry, they can be adjusted automatically or controlled by the user who introduces changes to the geometry of the machined part. The scope of the interoperability can concern 2D and 3D geometry. It can occur only within the CAD/CAM program or also concern geometry imported from external files. In the latter case, a change in the part model in the original CAD file can cause a change in both the part model and the tool paths in the CAD/CAM file.

Each element of the tool path has a programmed feed, but for auxiliary movements it can result from the settings on the CNC machine. During cutting, the working conditions of the tool can change due to the current cross-section of the cutting layer and the angle of the tool contact. There are various CAM programs, e.g. Vericut, ICAM or iMachining, which in such cases have the ability to adjust the cutting parameters to the machining stability criterion [2]. The MasterCAM program, popular in the industry, allows for feed optimization during roughing and finishing cutting for three-axis milling. During roughing, optimization means increasing the feed where the volume of material removed per unit of time (*material removal rate* – MRR) is small and a reduction where the MRR value is large [2]. In the case of finishing, the volume of material removed is not important and then the feed rate is reduced or increased due to the change in the machining direction (in practice this often happens when machining the corners of pockets in moulds and dies), as shown in fig. 3*a* and *b*.

Feed optimization in MasterCAM can be ineffective when machining that is characterized by frequent changes in the direction of tool movement, such as dynamic, HSM, trochoidal machining. This is due to the design of the tool path algorithm designed to generate tool movement under sufficiently stable load conditions, which eliminates the need to adjust the feed.

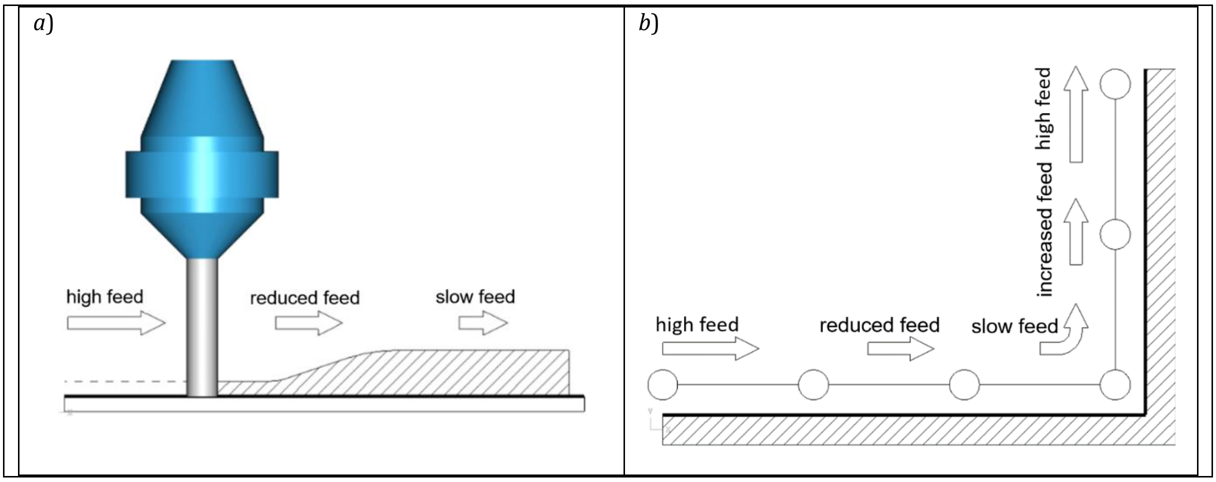

Fig. 3. Change of feed rate in relation to the uncut chip thickness (UCT) (*a*) and resulting from variable direction of the tool movement (*b*) [2]

Changing the feed at constant milling cutter rotational speeds changes the feed per tooth. When it is necessary to maintain a constant feed per tooth, the tool rotational speed should also be changed along with the change in feed. This functionality is used, among others, in the ICAM program [19]. It should be noted, however, that not every machine tool can dynamically change the spindle rotational speed. Another interesting functionality of this program is the optimization of auxiliary movements, which not only changes the tool travel speeds in areas where the tool is not in contact with the workpiece, shortening the operation time, but also automatically changes the path of auxiliary movements based on the virtual model of the machine, tool, holder, workpiece and semi-finished product. This happens already at the stage of processing the intermediate file (e.g. CL-Data) from the CAD/CAM program to the NC program of the machine. The effect is not only the optimization of auxiliary movements on the programmed CNC machine, but also the possibility of automatically adapting them using the so-called "intelligent" postprocessor to the kinematic system of the new machine (fig. 4).

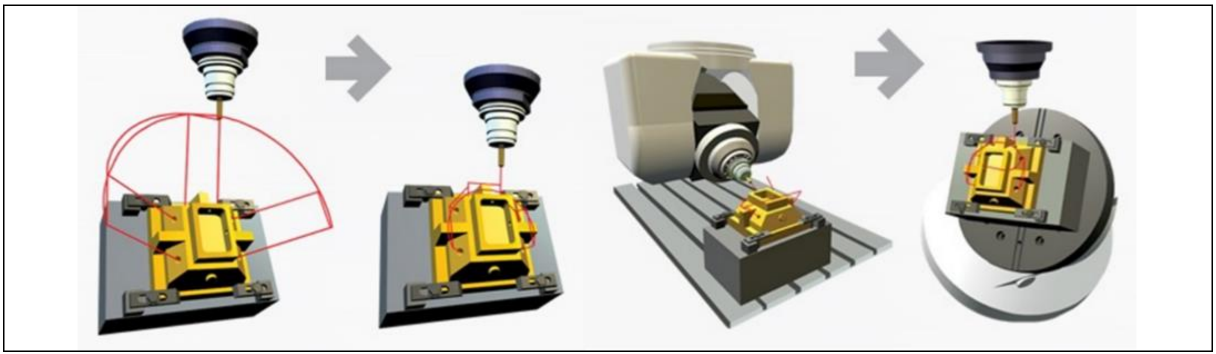

Fig. 4. Implementation of the same technological machining process on different machines after processing by "ICAM VR Simulation, Adapted NC program"

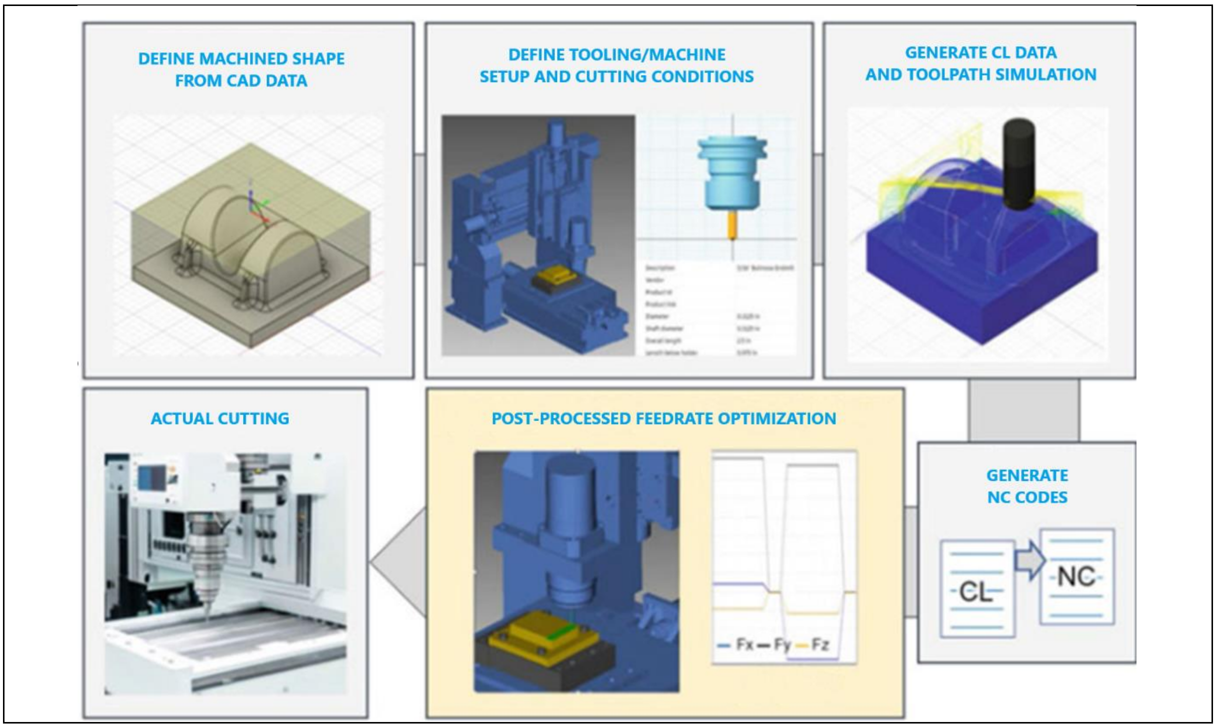

Fig. 5. Schematic diagram of process planning for CNC cutting process [4]

Fig. 5 shows a process planning scheme currently used in CNC machining. The planning cycle includes defining the machined contour based on data from the CAD system, selecting tool and technological equipment and machining conditions, generating geometric data in the CL-Data format in combination with tool path simulation, and then generating appropriate *G*-codes. It should be added that the CAM software used enables automatic optimization of the feed recorded in the form of *G*-codes through post- processing available in specialized modules, e.g. Production Module in the package Advant Edge by Third Wave Systems, VERICUT companies CGTech, or MACHpro by MAL Inc. [4]. As a result, the maximum/average values of the cutting force components (F_z, F_y, F_x) or cutting torque remain smooth, without peaks. In addition, the feed optimization performed is independent of other high-level functions available in the CAM system and is usually not accompanied by interaction.

An interesting direction of solving the optimization problem is the approach used in the IMachining program [2]. It uses the recognition of geometric features of the 3D model (like many modern 3D machining strategies), but its distinguishing feature is the automatic adjustment of cutting parameters. The program automates the preparation of single 2D or 3D roughing, shaping or finishing operations for solid parts and uses an updated blank model from previous operations or operations to detect areas requiring machining.

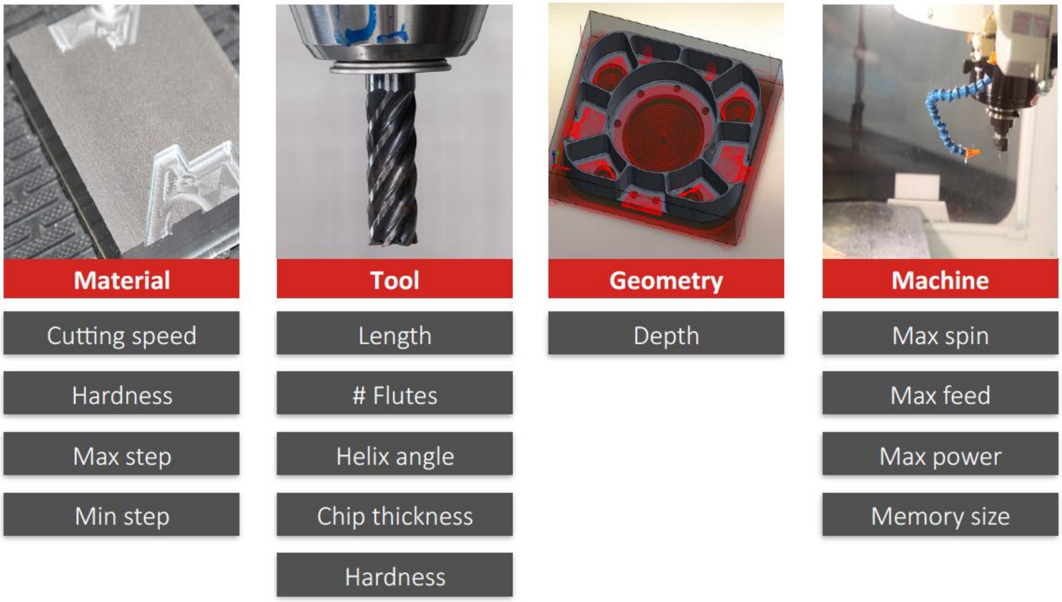

Fig. 6. Groups of input data using in so-called Technology Creator in Imachining program [2]

Fig. 6 shows the input parameters used by the Technology Creator. After indicating the data of the machined model and selecting the tool, the so-called Technology Creator automatically selects the cutting parameters, such as: feed, cutting speed (equivalent spindle revolutions), axial depth of cut, angle of engagement and undeformed chip thickness, with the possibility of their manual modification (which is not recommended). All values of these parameters are calculated based on the mechanical properties of the material, technical limitations of the CNC machine (spindle revolutions, power, maximum feeds and stiffness), the model of the machined part, the blank and the properties of the tool.

Modern CAM systems use various techniques to support the automation of machining process design based on automatic feature recognition (feature-based recognition machining – FBM). This is especially important if the production is diverse and requires programming a large number of cases in a short time. FBM drilling and milling functions are available in Mastercam 2024 PL. For example, the hole drilling procedure can generally be defined based on the selection of the hole axis and the direction of the tool feed movement (in three-axis machining it is the vertical direction of the Z axis). However, these settings can be based on a geometric feature of the solid part, which allows for the identification of, for example, the drilling depth and the analysis of the collision of movements.

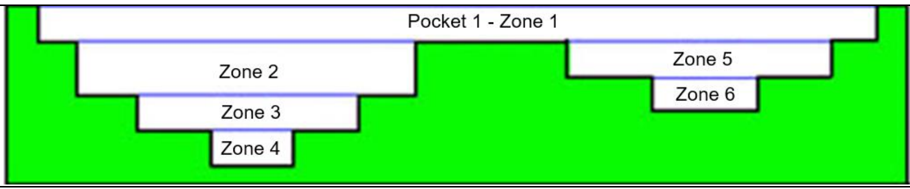

Fig. 7. Division of machining zones in the application of function FBM milling [2]

Recognition of geometric features of the object using the FBM Milling function is based on a detailed analysis of the three-dimensional solid model. Taking into account the data concerning the stock, this function allows the development of a comprehensive milling plan. The machining process is divided into subsequent stages: preliminary, shaping and finishing, with the possibility of controlling machining allowances between individual procedures. The program uses selected 2.5D machining strategies (3-axis milling) and is therefore applicable to simple 3D models (Mastercam 2024). The process is designed automatically, which refers to the strategy of machining thin-walled elements. The machining area is divided into zones, which are related to the position of subsequent flat elements of the part in the *XY* plane. In the case of complex, highly nested pockets, the FBM Milling function creates a separate zone for each depth as in fig. 7.

Another approach to designing the machining process is the analysis of the geometric features of the finished part model using artificial intelligence (AI). An example is the CAM Assist program working in the Autodesk Fusion 360 environment [20]. Based on effective machining strategies, a complete process is built, reducing programming time. Offering a simple user interface, based on typical data related to tools, the machined material automatically generates a machining process for a specific CNC machine tool, proposing cutting parameters. In this case, the recognition of shapes assigned to technological features (to which appropriate machining operations are further assigned) is based on the principle of AI training, which gives a chance for, for example, the minimum number of tool changes or a more advantageous division of machining zones.

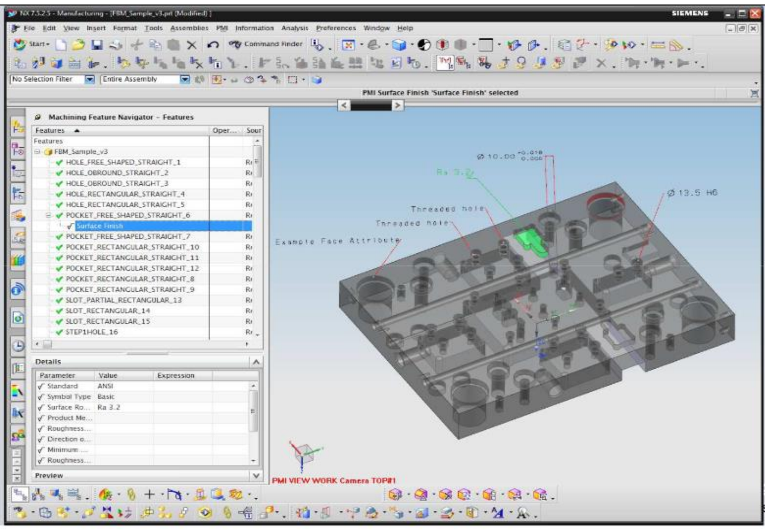

Fig. 8. Recognition of geometrical features of the workpiece (mold base) in NX CAM program [6]

In the NX CAM program from Siemens PLM Software, the FBM functions cover turning, milling (planes, plunging of cavities, threads and holes) and WEDM [6]. For the defined part geometry (dialog box in fig. 8), the program automatically recognizes and groups geometric features with the option of learning according to the user's needs (feature teaching), creates the necessary machining operations with setting the cutting parameters, generates tool paths and performs verification simulations. It is possible to tolerate dimensions and mark surface roughness based on reading the data by the NX PMI (Product and Manufacturing Information) program and use the knowledge base about cutting (MKE – Machining Knowledge Editor). The main application of the program concerns 2.5D prismatic parts. In the example of the injection mold plate shown in fig. 7, 115 characteristic geometric features were recognized, which resulted in programming 84 various machining operations.

Optimization taking into account the dynamics and kinematics of the process

The previously discussed methods of changing the feed rate do not take into account the dynamic properties of the machine tool and changes in the process over time (time-dependent processes – TDPs) in tool path planning and therefore may adversely affect the properties of the machined surface. [2, 4, 5]. For these reasons, it is necessary to locally correct the tool path and plan an appropriate change in the feed rate. In particular, this problem concerns the generation of tool motion paths for surfaces composed of irregular traces. The issue of classification of tool motion paths and their matching to scanned surfaces is described in [4].

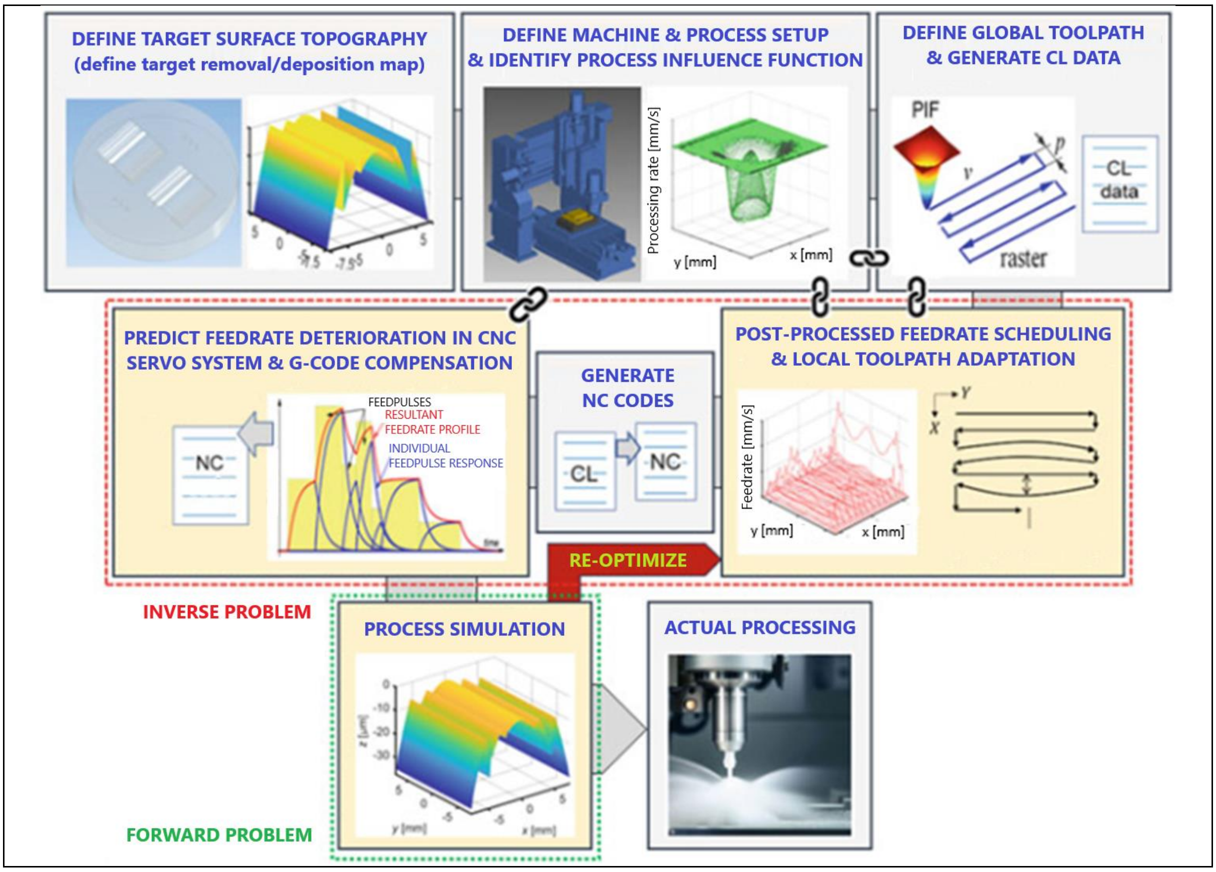

Fig. 9. Schematic diagram of process planning for CNC cutting process according to TDP strategy [4]

Fig. 9 shows a flowchart of the time-dependent process planning (TDP) with the definition of the required surface topography and target parameters such as volumetric efficiency, deviations and changes from the initial topography. This can in principle be achieved after analyzing the data in a CAD system before and after the process or based on shape corrections after direct 3D surface measurements.

The next step includes the selection of the process conditions and the prediction of the PIF (Process Influence Function) and then the generation of data in the CL-Data format and the preliminary definition of the entire tool path. These activities can in principle be performed in a commercial CAM system.

Thus, the feed rate represented by the response time in the servo system (dwell time) is optimized in post-processing mode, but the implications (entanglements) are different than in cutting. This is due to the fact that the KPI (Key Performance Indicator) in cutting refers to the cutting force (i.e. it has an indirect character), while in the case of the TDP strategy it refers to the surface topography (shape or roughness itself), i.e. a direct effect of the process. Since the feed optimization is achieved by deconvolution (separating noise, "unraveling" the signal in order to filter out interference from the recorded data and obtain undistorted dynamics data) of the modeled TDP, its results strongly depend on the process conditions and the shape of the tool path. Although these interactions complicate the machining planning process, local path adaptation is achieved in order to improve the process control capability in conjunction with the feed control. In currently available software both the dynamics of feed drives and CNC machine tool interpolators, which cause signal response distortion, are not taken into account, but their identification/prediction can help to compensate *G*-codes and obtain a result closer to the one required by solving the inverse problem. In addition, taking into account the dynamics increases the accuracy of the simulation, i.e. solving the forward problem.

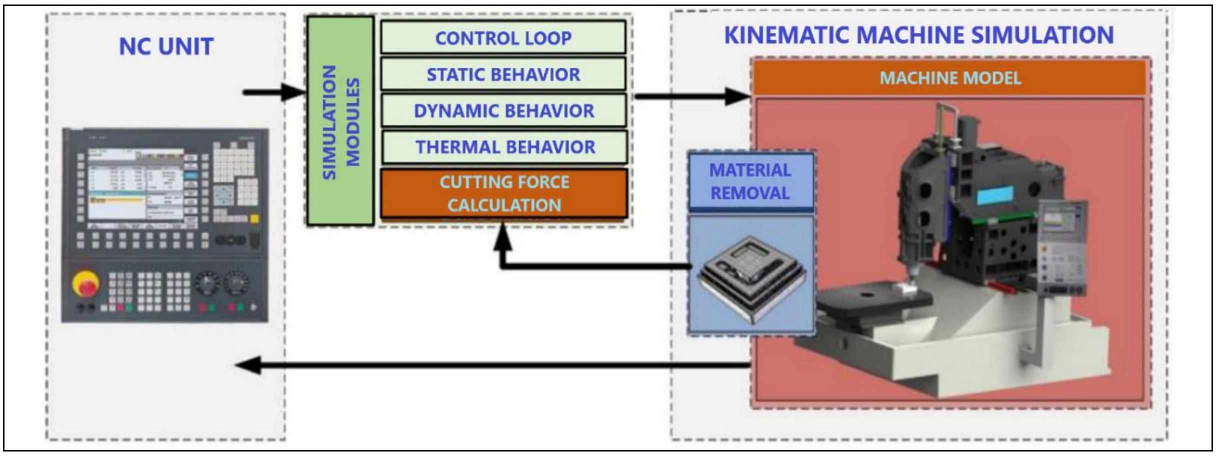

Fig. 10. Scheme of real-time machining simulation [2, 3, 5]

Modern CAD/CAM systems include built-in modules for kinematic machine tool simulation with visualization of workpiece material removal and simplified tool and workpiece/machining fixture collision detection.

In the case of the simulation of machining kinematics presented in fig. 10, the workpiece is composed of spatial elements, the so-called voxels, which are successively removed as the machining progresses. During the simulation, changes in the position of the controlled axes resulting from the removal of voxels are recorded and transferred to the simulation model of the machine tool kinematics, and on this basis a new center point of the tool is determined (tool center point – TCP). The TCP is then used in the simulation of material removal from the workpiece to determine new cutting force values and cutting parameter changes. The simulation module predicts the current static, dynamic and thermal state of the machine tool. According to the concept presented in fig. 10, the simulation of the machine tool operation is performed with feedback from the physical unit, where the CNC controller transmits data on the machine axis positions, as well as sending other signals from sensors to the appropriate simulation modules. This information is used for real-time calculations, which allows the machining parameters and tool movements to be adjusted to the actual cutting conditions. After making appropriate corrections, the final resulting axis positions and corrected machining parameter values are transmitted back to the real CNC control unit [5]. The advantage of simulation conducted in parallel with machining is the prediction of process conditions and correction of machine operating parameters before the expected exceedance of limit values.

Optimization using numerical and virtual simulation

Selected commercial software packages enable more advanced simulation, including virtual representation of the entire machine tool, simulation of workpiece material removal, and advanced collision detection [1, 3, 5]. Therefore, current research on machining simulation focuses on real-time process simulation and the combination of virtual machine tool models, simulation modules, and coupling with data from sensors installed on the physical machine tool. Three main areas are crucial in real-time process simulation: forecasting of cutting conditions (cutting force components, temperature distribution, cutting tool load, etc.), visualization of workpiece material removal, and virtual modeling of CNC machine tool kinematics and dynamics. The approach based on numerical simulation FEM (FEM/FEA) is schematically shown in fig. 11.

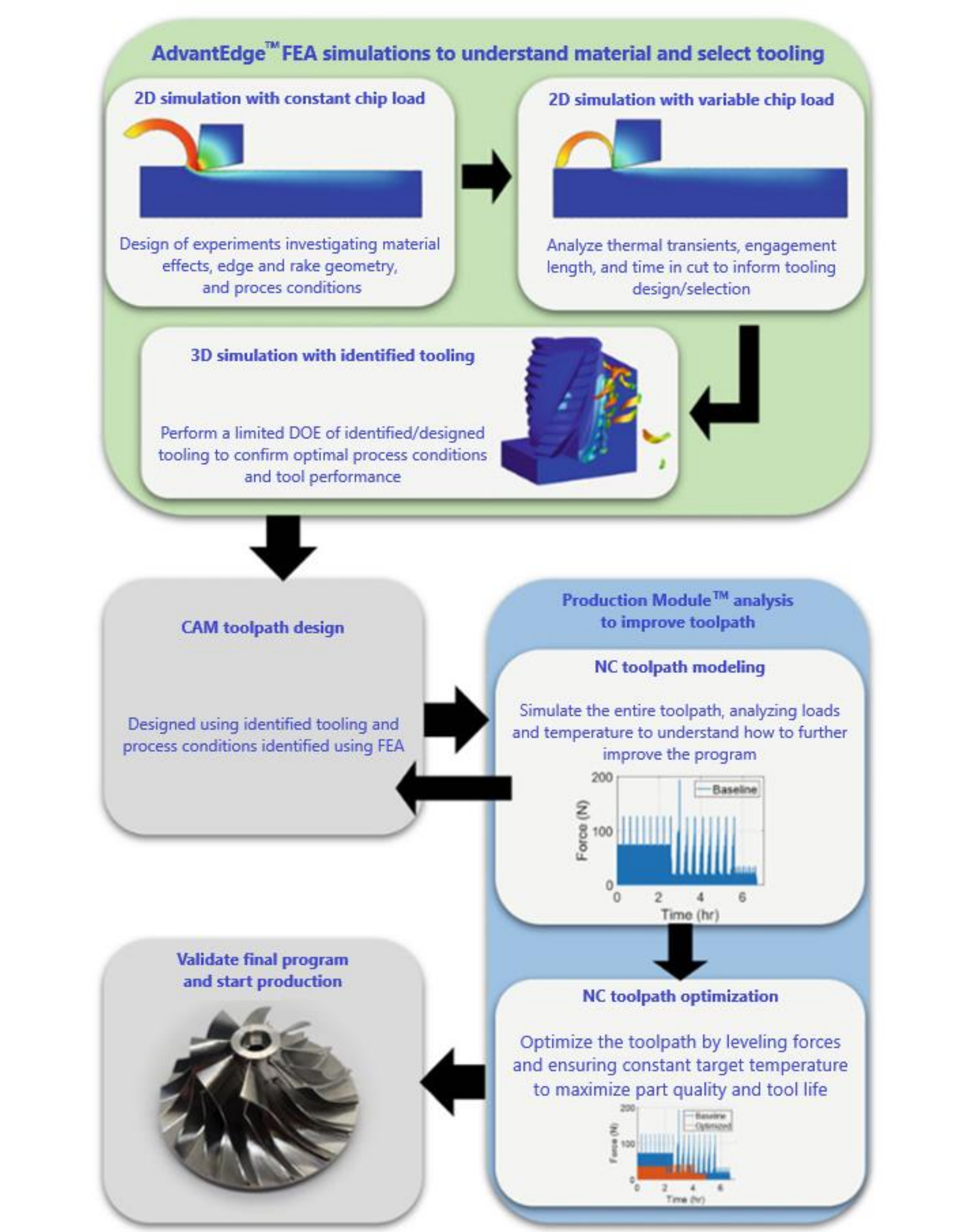

Fig. 11. Flow chart describing optimization run of NC tool path in Production Module™ Advant Edge simulation program [9]

In the first phase, routine 2D FEA (Finite Element Approach) simulations of the process with constant and variable tool/chip load are performed to check the material behavior and tool selection. In the second phase, a complete 3D simulation is performed for the selected tool and cutting parameters to verify the conditions determined in the previous 2D simulation. The parameterized data regarding the tool are imported to the STEP (Standard for the Exchange of Product Data) model. Since the cutting parameters have been selected, it is possible to program the tool path in the CAM system using the specialized Production Module. The experience gained [9] shows that the selection of tools and cutting parameters based on FEA modeling allows for the design of better CAM programs. Additionally, the current tool path can be better assessed in the context of the tool load, spindle power and cutting temperature.

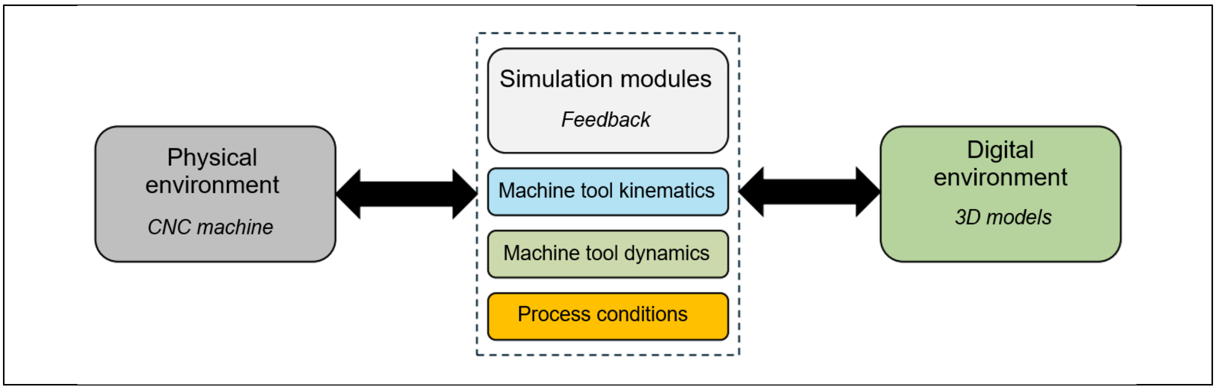

Fig. 12. Concept of a modified real-time simulation of CNC machine tools [2, 3]

In virtual modeling, a reverse exchange of information is realized between the digital (twin) and physical environment according to the scheme presented in fig. 12 [2, 3]. It concerns the kinematics and dynamics of the CNC machine tool and the current process conditions, i.e. factors that influence the execution of the control program.

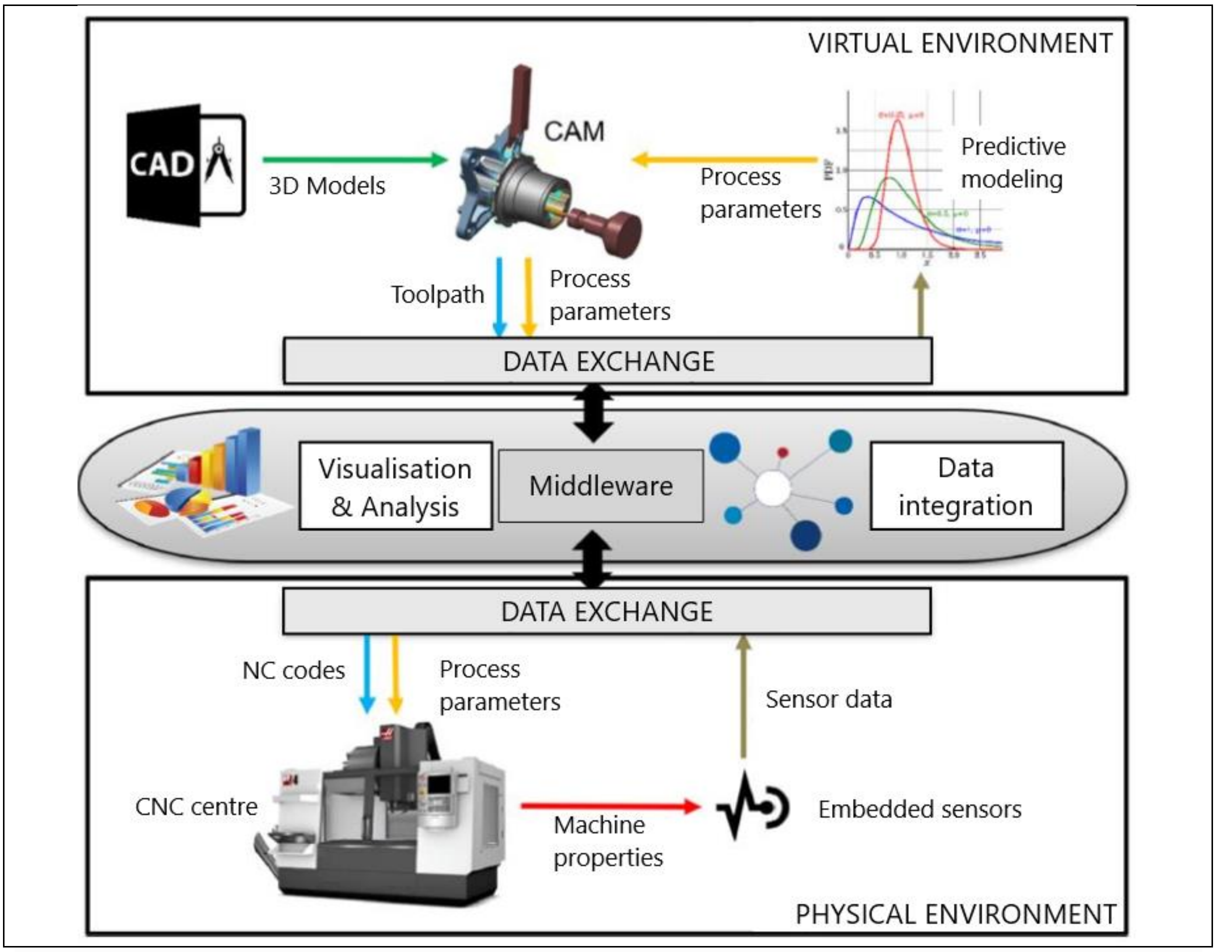

Fig. 13. A digital twin concept for process monitoring and control in chatter-free machining [10, 11]

Figure 13 shows another concept of a digital twin (DT) for monitoring and controlling the cutting process with an intermediate layer facilitating the exchange of data between the virtual and physical environment. DT consists of a digital environment, which is a replica of the physical environment connected by an interface with middleware that facilitates communication, collection and transfer of data, information and knowledge. Similarly to the case shown in fig. 10, CAD models for the workpiece, holder and tools are used in the CAD/CAM package to identify the tool movement trajectory for selected cutting parameters. Cutting parameters are selected based on information about the tool and the machined material, or based on physical predictive models [10, 11]. These activities are supported by data obtained from sensors in order to increase the process efficiency (MRR) and surface quality. Data obtained from the measurements of force components are used to predict parameters ensuring the conduct of the process without self-excited vibrations (chatter-free process). The model has been built into the digital twin for the process control system.

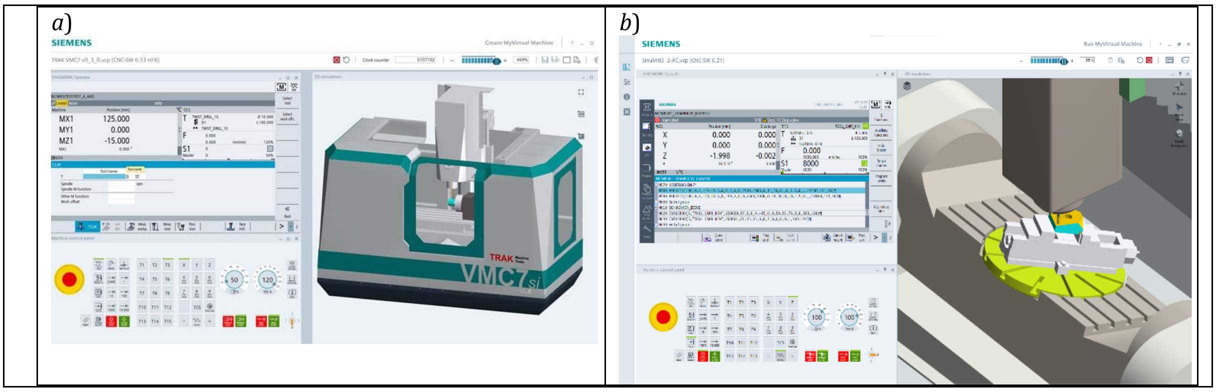

Fig. 14. Virtual representation of a CNC machine tool (*a*) and an example of an integrated virtual model of a titling rotary including machining simulations (*b*) [9]

An example of commercial application of the DT concept in CNC machine tool programming are the Create software packages MyVirtual Machine and Run My Virtual Machine for the virtual Sinumerik One digital control offered by Siemens [13]. In 2023, the "native" SINUMERIK ONE control system was tested in the context of creating and using a digital twin on a 5-axis training machine SLV EDU from SolidVision [13]. As a result, the machine operator has its full virtual representation on a PC in the form of a digital twin, as shown in fig. 14.

Software integrity is ensured at the stage of prototyping, commissioning, programming and servicing of the machine tool and in cooperation with CAD-CAM-CNC support systems. In such cases, the digital twin is created in the create environment MyVirtual Machine. It is also possible to configure the virtual part of the CNC system, PLC controllers and HMI interface, as well as the mechanical part of the machine. The level of virtual representation of the machine reaches 99%. As a result of using DT, the time of implementing a new machine in production is shortened by 30% and the time of its launch by 50%. These activities fully correspond to the implementation of the INDUSTRY 4.0 strategy in the Polish industry.

In the case of performing operator tasks, the ability to simulate collision risk assessment and object measurements taking into account the zero point shift is important. In turn, using the Run MyVirtual Machine software means that programming, production preparation and operator training can significantly replace tasks performed on a real 5-axis machine.

Optimization using the STEP-NC interface

The *G*-code format described by the ISO standard [ISO 6983-1 2009] is still the standard for describing control programs for CNC machines. CAM systems use their own method of describing a technological procedure and then transform it into a tool path (usually saved in the form of CL-Data). Each CAD/CAM/CNC software manufacturer has its own format for a digital product model. However, STEP [ISO 10303-11 1994] is a universal format that allows the transfer of product geometry between different CAD systems. In the case of describing technological processes of machining, the STEP-NC standard, which has been developed for years, offers similar possibilities [ISO 14649-1 2003]. The STEP-NC standard was originally developed as a model for exchanging data between CAD/CAM systems and CNC machines. Unlike ISO 6983, it describes machining operations rather than machine tool movements, using an object-oriented approach [18]. Its application may also involve direct use in the control of a CNC machine tool.

The role of the STEP-NC interface is mainly focused on the implementation of intelligent control, which causes the machine tool/CNC controllers to be subject to intelligent actions, such as machining optimization. An example is the autonomous STEP-compliant CNC (ASNC) in which data stored in ISO 14649 format is converted to ASNC format and then processed into a process sequence graph (PSG) [2, 3, 14]. The use of the universal XML markup language (ISO 10303 AP28) facilitates data transfer in e-manufacturing. The data model allows for the location, extraction/selection and archiving of tool paths in extensive markup language XML format. A set of instructions is created to format texts in a machine-readable way. STEP-NC programs are created by the data input module (XML DATA INPUT) and the data interpreter (INTERPRETER) based on the supplied CAD files.

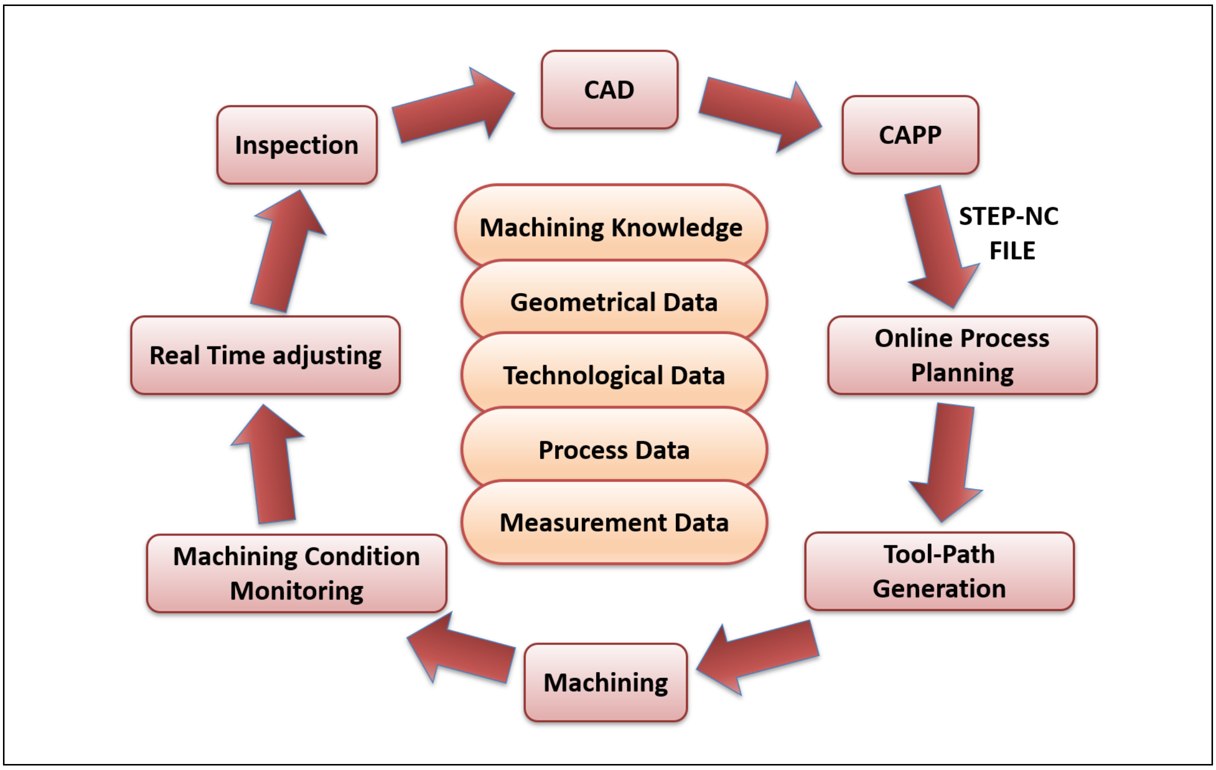

Fig. 15. An example of a closed-loop self-learning STEP-NC machining system [3, 14]

Fig. 15 shows the concept of a self-learning system based on STEP-NC operating in a closed loop, which uses high-level information, including data on the machining process conditions and the results of on-line and real-time control. The STEP-NC interpreter adjusts the technological parameters of the machining on a multi-axis CNC machine tool according to the knowledge about the workpiece model and the blank, the process, the machine tool and the tool status. On this basis, a machining plan and tool paths are generated remotely. On the other hand, sensors built into the feedback system provide up-to-date knowledge about the machining process and on this basis, the optimal technological parameters are determined using an adaptive control algorithm. After completing the working step of the machining, they are verified based on the on-line control of the required dimensional tolerances [11]. Based on the STEP-NC platform, the concept of sustainable manufacturing can be developed, which involves creating an energy-efficient manufacturing environment and intelligent manufacturing, which requires the introduction of intelligent controllers at the industrial level [3, 15, 16].

Summary

In technological activities related to programming modern, now largely intelligent, CNC machine tools, appropriate steps should be taken regarding their optimization, which results in the proper selection of the tool path and machining parameters. The optimization criteria are selected as threshold values of cutting force components as an indirect criterion or the required state of the machined surface topography as a direct criterion.

Engineers-programmers can use many functions available in commercial CAD/CAM programs, e.g. automation of feed changes depending on the configuration of the workpiece and changes in process conditions over time or recognition of characteristic features of the workpiece and grouping of turning, drilling and milling operations. More advanced methods are based on concurrent simulation of the process using numerical methods or creation of complete virtual models, so-called digital twins.

A development issue is the functional STEP-NC interface, or ultimately an IT platform that streamlines calculations leading to the inclusion in the programming process, in adaptive mode, of knowledge about the current kinematics and dynamics of the CNC machine tool. Research works leading to the expansion of practical knowledge and development of production capabilities of CNC machine tools that can be used by less experienced machine tool operators are also important [17].

REFERENCES

[1] Moriwaki T., Shirase K. "Intelligent machine tools: Current status and future direction". *International Journal of Manufacturing Technology and Management.* 9, 3/4 (2006): 204–218, https://www.researchgate.net/publication/220572721.

[2] Zalewski A., Grzesik W. (red.), Deja M., Jarosz K., Ruszaj A. *„Podstawy programowania i funkcjonowania obrabiarek CNC. Procesy ubytkowe, przyrostowe i hybrydowe".* Warszawa: PWN (2024).

[3] Jarosz K., Grzesik W. "Development of techniques improving functional and programming abilities of intelligent CNC machine tools/Rozwój technik wspomagania funkcjonowania i programowania inteligentnych obrabiarek CNC. *Mechanik*. 7 (2024): 6–12, https://doi.org/10.17814/mechanik.2024.7.12.

[4] Yamato S., Sencer B., Beaucamp A. *Tool path planning and feed scheduling for time dependent processes.* Tolio T. (red.). "*CIRP novel topics in production engineering*". Lecture Notes in Mechanical Engineering. 1 (2024): 185–231, https://doi.org/10.1007/978-3-031-54034-9_3.

[5] Witt M., Schumann M., Klimant P. "Real-time machine simulation using cutting force calculation based on a voxel material removal model". *International Journal of Advanced Manufacturing Technology*. 105 (2019): 2321–2328, https://doi.org/10.1007/s00170-019-04418-2.

[6] van't Erve T. "*NX CAM Feature-Based Machining (FBM) Introduction*". NX CAM Development.

[7] „Obróbka, planowanie części i generowanie NC z wykorzystaniem PMI", www. https://cador.pl/inzynieria-w-praktyce/nx-pmi-w-cam/ (dostęp: lipiec 2024).

[8] Lou P., Liu S., Hu J., Li R., Xia Z., Yan J. "Intelligent machine tool based on edge-cloud collaboration". *IEEE Access* (2020): https://doi.org/10.1109/ACCESS.2020.3012829.

[9] Roth T., Heigel J., Marusich K., Thimm B. "Leveraging Transients and Material Properties to Improve Machining Material Removal". Tolio T. (red.). *CIRP Novel Topics in Production Engineering.* Lecture Notes in Mechanical Engineering. 1 (2024), https://doi.org/10.1007/978-3-031-54034-9_3.

[10] Afazov S., Scrimieri D. "Chatter model for enabling a digital twin in machining". *International Journal of Advanced Manufacturing Technology*. 110 (2020): 2439–2444.

[11] Grzesik W. "Digital Twin in manufacturing. Part II. Case studies in removal and additive machining processes/Cyfrowy bliźniak w procesach wytwórczych. Część II. Przykłady zastosowań w skrawaniu i obróbce przyrostowej". *Mechanik.* 2 (2023): s. 6–14, https://doi.org/10.17814/mechanik.2023.2.4.

[12] Jarosz K., Özel T. "Machine learning approaches towards digital twin development for machining systems". *International Journal of Mechatronics and Manufacturing Systems.* 15, 2–3 (2022): 127–148, https://doi.org/10.1504/IJMMS.2022.124922.

[13] „SINUMERIK – Inteligentne rozwiązania dla obrabiarek", www.siemens.pl/sinumerik (dostęp: lipiec 2024).

[14] Jeon B., Yoon J.S., Um J., Suh S.H. "The architecture development of Industry 4.0 compliant smart machine tool system (SMTS)". *Journal of Intelligent Manufacturing*. 31, 8 (2020): 1837–1859.

[15] "STEP/STEP-NC as a Solution for Data Interoperability in the CAx Chain", https://ukdiss.com/examples/step-step-nc-data-interoperability-cax-chain.php (dostęp: lipiec 2024).

[16] Zhang Y., Yu X., Sun J., Zhang Y., Xu X., Gong Y. "Intelligent STEP-NC-compliant setup planning method". *Journal of Manufacturing Systems.* 62 (2022): 62–75, https://doi.org/10.1016/j.jmsy.2021.11.002.

[17] Jarosz K., Chen Y.-T., Liu R. "Investigating the differences in human behavior between conventional machining and CNC machining for future workforce development: A case study". *Journal of Manufacturing Processes*. 96 (2023): 176–192, https://doi.org/10.1016/j.jmapro.2023.04.037.

[18] Habel J. „Cyfrowa reprezentacja procesu technologicznego obróbki na podstawie rozszerzonego standardu STEP-NC w kontekście Przemysłu 4.0”. Knosala R. (red.): *„Inżynieria zarządzania: cyfryzacja produkcji. Aktualności badawcze 3*”. Warszawa: Polskie Wydawnictwo Ekonomiczne, ISBN 978-83-208-2456-8 (2021).
[19] ICAM, https://www.icam.com (informacje o programach firmy ICAM – postprocesory, symulacja i optymalizacja parametrów procesu obróbki skrawaniem na obrabiarkach CNC) (dostęp: lipiec 2024).
[20] https://www.cloudnc.com/cam-assist/fusion-360 (informacje o programie CAM Assist firmy CloudNC) (dostęp: lipiec 2024).

■